



Model created in COMSOL Multiphysics 6.4

Axially Rotating Pipe and Swirling Jet Turbulent Flows

Introduction

The Single-Phase Flow, SSG-LRR interface is employed to investigate fully developed turbulent flow in a rotating pipe, as well as subsequent development of the swirling jet, emanating from the pipe outlet, into a drastically expanded nonrotating section. Properties of the flow regimes are thoroughly analyzed and illustrated.

Model Definition

The influence of rotation on turbulent flows is not naturally accounted for by eddy-viscosity models. Partially, this drawback can be healed by activating **Include rotation-curvature correction**. For the case of an axially rotating pipe this would shift axial velocity profile in a qualitatively correct direction. However, tangential (swirl) velocity would retain linear (solid-body) profile, which contradicts experimental observations, direct numerical simulations (DNS) and large-eddy simulations (LES), see [Ref. 1](#) and [Ref. 2](#). In contrast to RANS-EVM, differential Reynolds stress models (RANS-RSM) of turbulence self-consistently respond both to the in-frame flow vorticity and to the frame rotation.

Here, turbulent rotating pipe flow is investigated first. The pipe Reynolds number,

$$\text{Re} = \frac{U_b D_p}{\nu}$$

is $\text{Re} = 20,000$. Here, U_b is the streamwise bulk velocity and D_p is the pipe diameter. The flow is investigated at rotation number (or swirl number),

$$N_s = \frac{\Omega D_p}{2 U_b}$$

being $N_s = 0$ and $N_s = 0.5$. Ω is the angular rotation rate.

The development of a fully developed swirling state, starting from an isotropic turbulent state with solid-body rotation, requires the streamwise length of order $L = 30D_p$. **Periodic Flow Condition** in a streamwise-short geometry is employed to save computational resources.

Subsequently, the rotating pipe flow emanates into a large nonrotating region of very slow flow ($U_b/30$). The incoming angular momentum of the swirling jet fades away quite quickly, but nevertheless has profound effect on the jet development. Again, $N_s = 0$ and $N_s = 0.5$ cases are being considered.

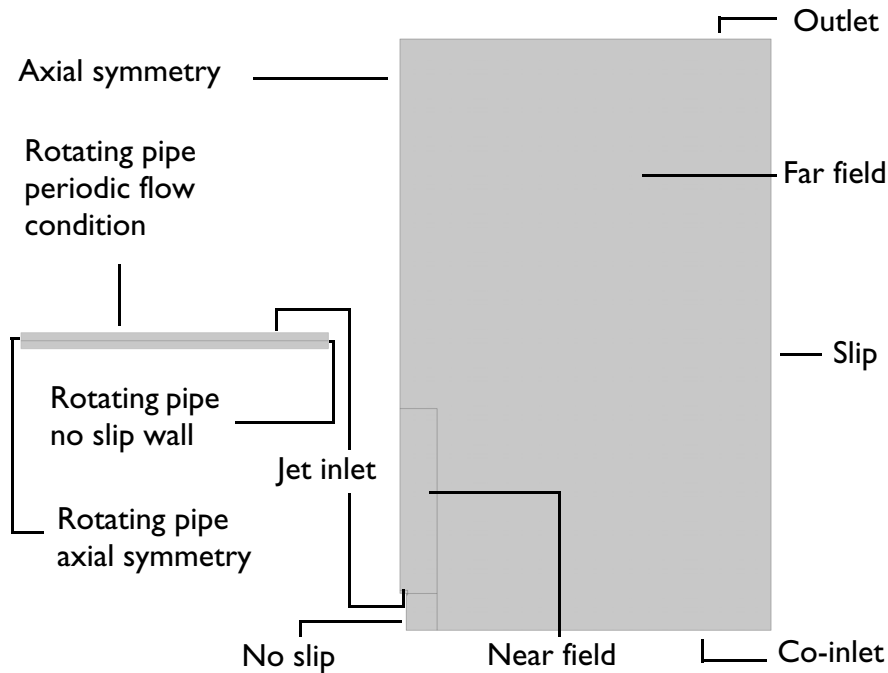


Figure 1: The model geometry of the rotating pipe (left) and swirling jet (right). The converged fully developed state of the rotating pipe study is used as an inlet condition for the swirling jet study.

Implementation in COMSOL Multiphysics

Only axisymmetric flow behavior is in the focus of this investigation, thus a 2D-axisymmetric component is used. **Swirl flow** has to be activated. Figure 1 shows geometries, along with boundary conditions, for both studies, while Figure 2 demonstrates the corresponding meshes. The rotating pipe study is performed using a moving wall condition; the same results (accounting for the shift in tangential velocity) would be obtained if **Rotating frame** (instead of **Wall movement**) was used instead. The swirling jet flow is originated via inlet conditions taken from the converged state of the rotating pipe flow analysis. The near-wall cells of the pipe should be meshed exceptionally well, and the development region of the jet (so called near field) should have good meshing. In contrast, the far-field region is used only to relieve the influence of the outlet and slip wall boundary

conditions, and the results there are not very reliable. An **Auxiliary sweep** with continuation is employed for efficient computation of cases with consequent rotation numbers.

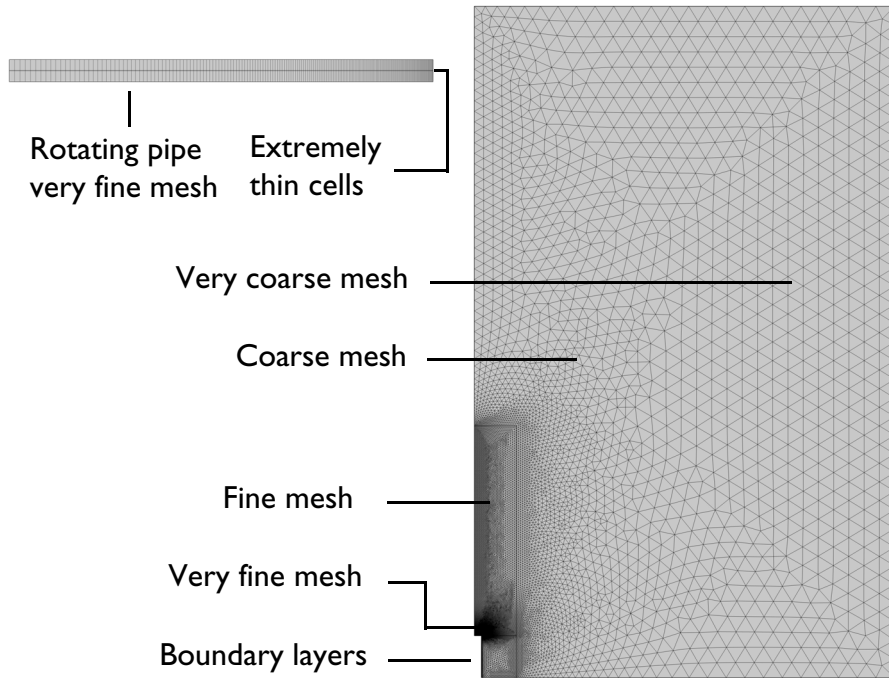


Figure 2: Meshes used for the studies of the rotating pipe (left) and swirling jet (right). Extremely thin cells at the rotating wall of the rotating pipe are used to ensure accurate evaluation of traction. The near field of the jet features really fine mesh cells, while the poorly meshed far field serves mostly to ensure that the development region of the jet is not much affected by the distant boundary conditions.

The details of the implementation of the Single-Phase Flow, SSG-LRR interface can be found in the *CFD Module User's Guide*; see the section “Theory for the Turbulent Flow Interfaces”.

Results and Discussion

Figure 3 illustrates how the axial velocity profile becomes modified due to rotation with $N_s = 0.5$. Apparently, the profile appearance shifts from the typical “round nose bullet-like” turbulent form to an “inverted paraboloid” laminar form, which is consistent with the general laminarizing effect of rotation. The swirl velocity profile clearly resembles a

paraboloid form. Figure 4 details these observations. It can be shown that the axial velocity in the $N_s = 0.5$ case has a clearly distinguishable logarithmic region, although it is shorter than in the $N_s = 0$ case. The swirl velocity curve is placed somewhat below pure parabola, the latter being considered as a canonical (experimentally confirmed) form in the rotating pipe.

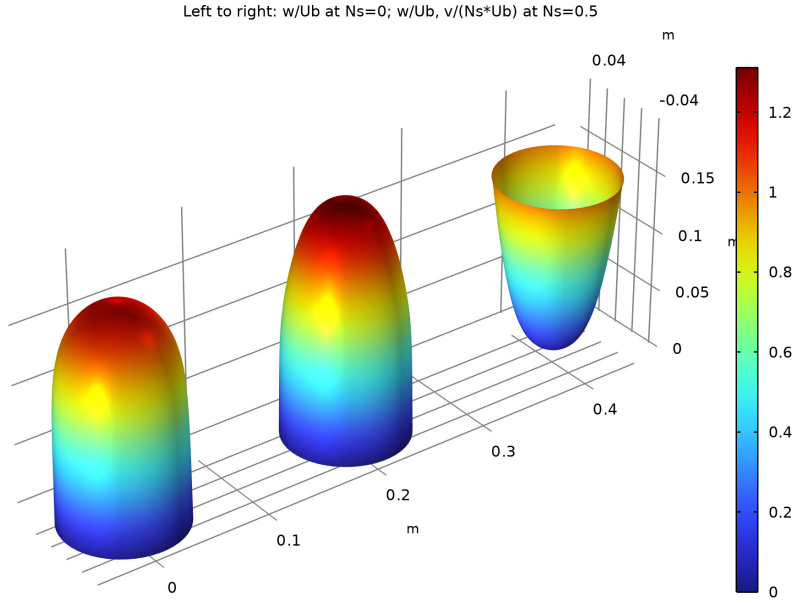


Figure 3: The pipe flow axial (nonrotating versus rotating) and tangential (rotating) velocity profiles.

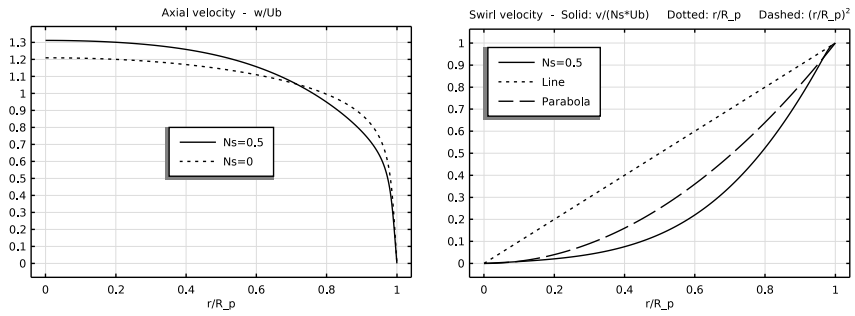


Figure 4: The pipe flow axial and swirl-velocity profiles.

Absolute system. Streamlines at $r=0.9 R_p$ and arrow surfaces. Color: $v/(N_s*U_b)$

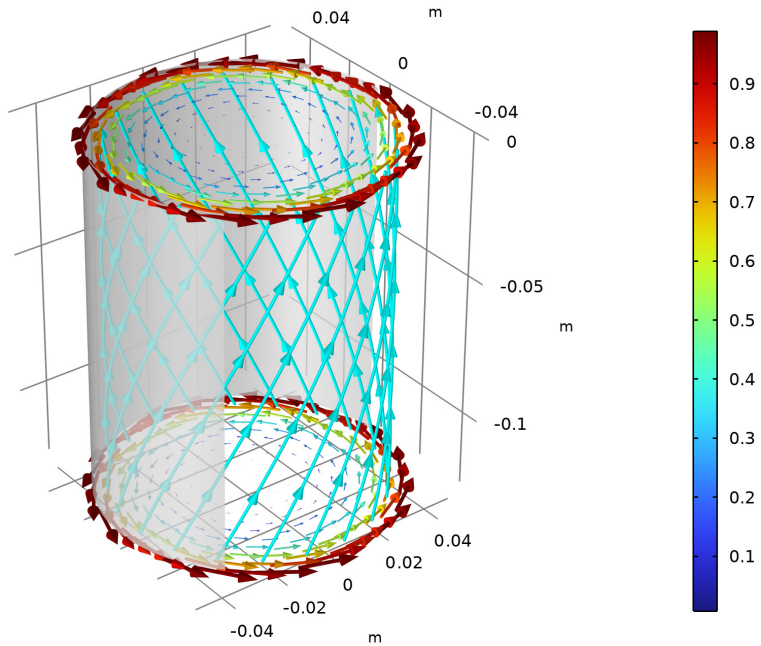


Figure 5: Rotating pipe flow in absolute system: streamlines on a cylindrical surface ($v / (N_s U_b) = 0.76$) and arrow surfaces on horizontal planes ($v/(N_s U_b)$) according to the color legend).

The contrast between [Figure 5](#) and [Figure 6](#) is due to slower-than-linear growth of the swirl velocity v with radius: in rotating system the streamlines are spiraling in clockwise (negative) direction, and the maximum in-frame swirl velocity $(\omega r - v)/(N_s U_b) = 0.38$ at $r/R_p = 0.587$. [Figure 6](#) can be obtained directly if using the **Rotating frame** feature for computation.

[Figure 7](#) compares $N_s = 0$ and $N_s = 0.5$ cases, and demonstrates that rotation reduces the shear stress, although profiles normalized by the friction velocity $u_\tau = \tau_w/\rho$ are almost the same. The friction coefficient, defined as

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho U_b^2}$$

falls from $6.7187 \cdot 10^{-3}$ ($N_s = 0$) to $5.342 \cdot 10^{-3}$ ($N_s = 0.5$). Meanwhile, the turbulence kinetic energy level grows just a bit and its maximum shifts away the wall. Profiles normalized by u_τ show significant growth due to the rotation.

Rotating system. Streamlines at $r=0.9 R_p$ and arrow surfaces. Color: $(\omega r - v)/(N_s U_b)$

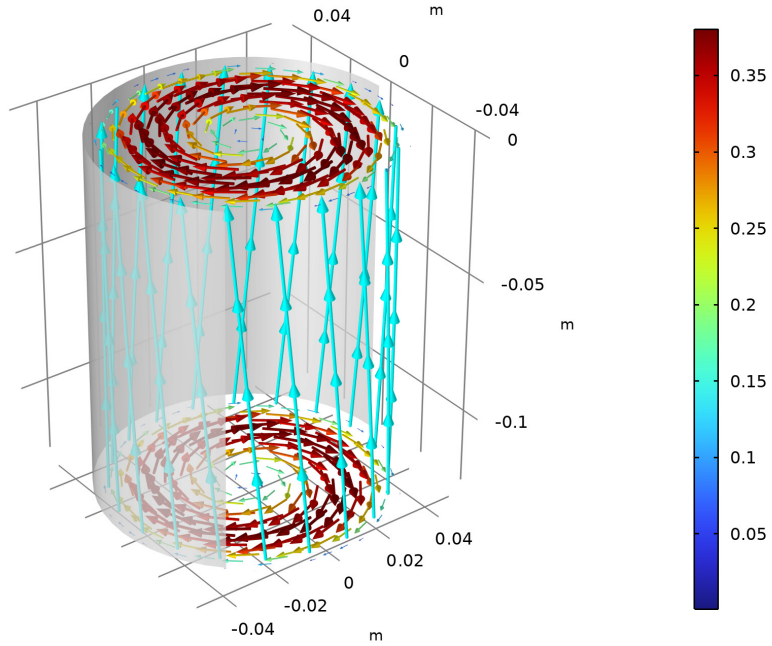


Figure 6: Rotating pipe flow in rotating frame: streamlines on a cylindrical surface $((\omega r - v)/(N_s U_b) = 0.15)$ and arrow surfaces on horizontal planes $((\omega r - v)/(N_s U_b)$ according to the color legend).

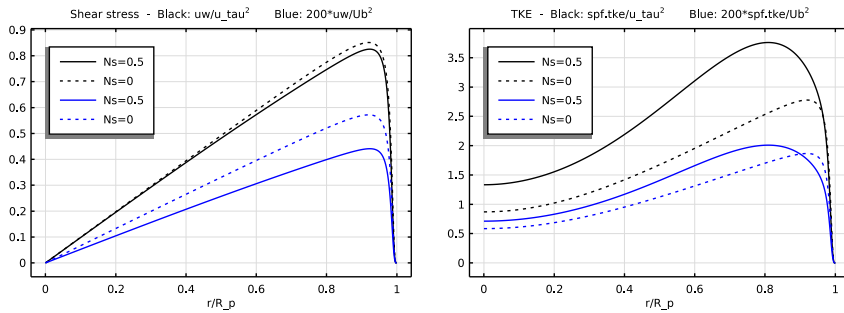


Figure 7: The pipe flow: shear stress and turbulence kinetic energy, normalized both by the friction velocity and by the bulk velocity.

Streamlines starting at a diameter $y=0$. Color: $v/(Ns*U_b)$

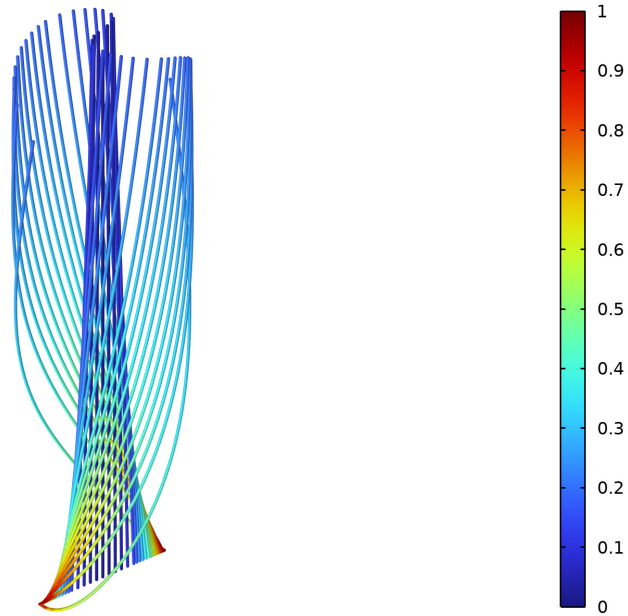


Figure 8: Streamlines of the swirling jet.

Figure 8 demonstrates the streamlines of the swirling jet, emanating from the outlet of the rotating pipe. Apparently, swirl velocity falls rapidly in the axial direction and already at $3 D_p$ ($D_p = 0.1$ m) maximum swirl velocity is no more than 0.25 of its original maximum.

Figure 9 confirms the intuitive expectation that the swirl leads to faster loss of axial momentum on the axis due to increased mixing with the surrounding fluid (upper left). The form of the curve and characteristic values have good correspondence to the experimental data, although do not reproduce it exactly. At the same time, in the rotating case the axial velocity falls less rapidly with radius at all axial cross sections (upper right). In a wide range of axial coordinates, maximum positive radial velocity happens at approximately the same radius and has approximately the same value (lower left). Although far away from the source the swirl velocity is small, it falls off very slowly with radius (lower right). Notice that negative swirl velocities near the jet axis are not observed.

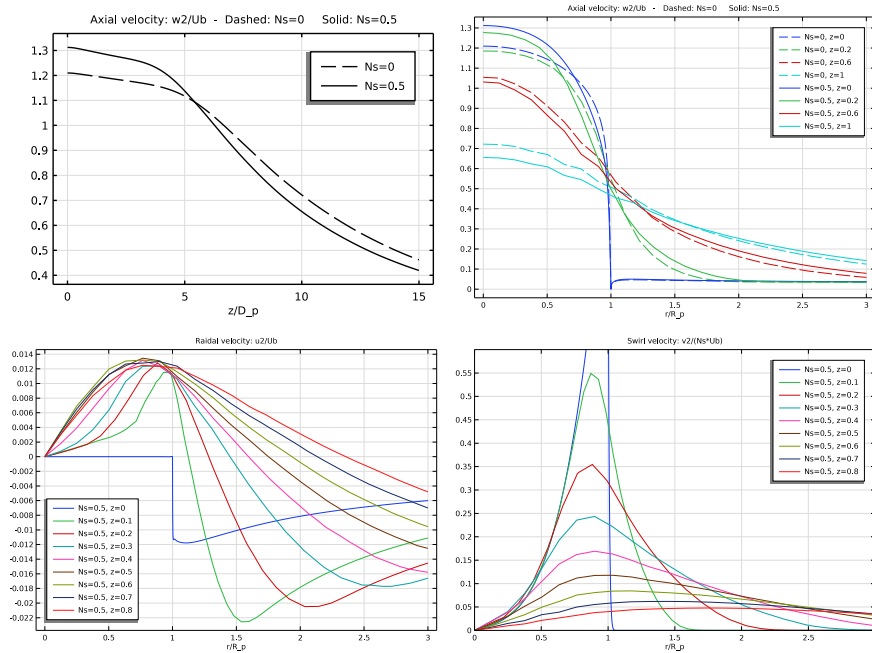


Figure 9: Top row: axial velocity (without and with swirl) at the jet axis (left) and at various axial cross sections (right). Bottom row — various axial cross sections in swirling case: radial (left) and tangential (right) velocities.

Figure 10 shows that shear stress uw is consistently larger in the $N_s = 0.5$ case (upper left), except at the central portion of the jet at quite high z/D_p (approximately 10). Axial velocity rms w demonstrates the same trend (upper right). Axial-tangential stress vw attains strong positive values at small z/D_p (say, 2), but quickly decreases further in the axial direction (lower left). Radial-tangential stress uv exhibits extremely high values at the same locations as vw , and rapidly becomes attenuated with increasing z as well.

[

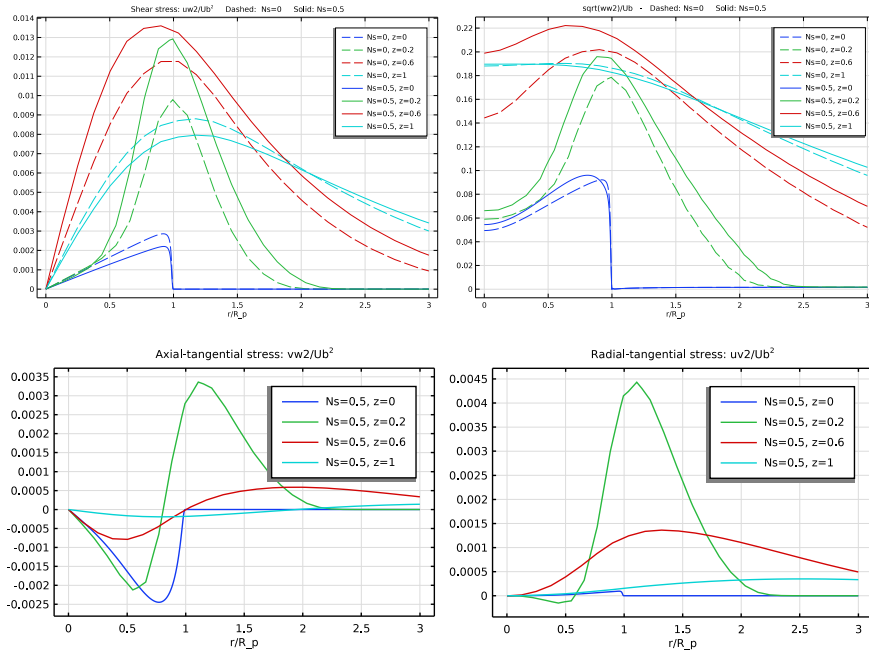


Figure 10: Top row: shear stress (left) and axial velocity rms (right) in $N_s = 0$ and $N_s = 0.5$ cases. Bottom row: axial-tangential stress (left) and radial-tangential stress (right) in the swirling case.

Summary and Outlook

Major consequences of turbulence anisotropy induced by rotation in rotating pipe flow and swirling jet flow can be described by the SSG-LRR Reynolds stress model implemented in COMSOL Multiphysics. Most of the characteristics obtained by the RSM, like friction reduction with growing rotation rate, are compatible with more precise experimental, DNS and LES studies, see [Ref. 1](#) and [Ref. 2](#).

However, in analysis based on SSG-LRR, tangential velocity deviates from the parabolic profile in the rotating pipe, as well as counter-rotating region is absent in the swirling jet. Thus, although correct trends of turbulence response to swirl have been obtained both in rotating pipe and swirling jet, it is desirable to use more reliable tools of analysis in case exact predictions are required.

References


1. L. Facciolo “A Study on Axially Rotating Pipe and Swirling Jet Flows,” *Doctoral thesis*, Department of Mechanics, Royal Institute of Technology, 2006.
2. N.D. Castro “Large-Eddy Simulation of Axially-Rotating, Turbulent Pipe and Particle-laden Swirling Jet Flows,” *Doctoral thesis*, Mechanical Engineering, Old Dominion University, 2012; doi.org/10.25777/mpdk-zd57.

Application Library path: CFD_Module/Single-Phase_Flow/
rotating_pipe_swirling_jet




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** > **Turbulent Flow** > **Turbulent Flow, SSG-LRR (spf)**.
- 3 Click **Add**.
- 4 Click **Add**.
- 5 Click  **Study**.
- 6 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Stationary with Initialization**.
- 7 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 Click the **Load** button. From the menu, choose **Load from File**.

- 4 Browse to the model's Application Libraries folder and double-click the file `rotating_pipe_swirling_jet_parameters_1.txt`.

Parameters 2

- 1 In the **Home** toolbar, click  **Parameters** and choose **Add > Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click the **Load** button. From the menu, choose **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `rotating_pipe_swirling_jet_parameters_2.txt`.

DEFINITIONS

Variables 1


- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Unit	Description
u_tau	$\sqrt{\text{spf.pfc1.delta_p} * \text{R_p} / (2 * \text{rho_w} * \text{dL})}$		Friction velocity in pipe

ADD MATERIAL FROM LIBRARY


In the **Home** toolbar, click  **Windows** and choose **Add Material from Library**.

ADD MATERIAL

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-in > Water, liquid**.
- 3 Click the **Add to Component** button in the window toolbar.
- 4 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.

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 `R_p`.
- 4 In the **Height** text field, type `dL`.


- 5 Locate the **Position** section. In the **z** text field, type $-dL$.
- 6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	$dL/2$


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 $E_j * R_p$.
- 4 In the **Height** text field, type L_j .


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 $(E_j - 1) * R_p$.
- 4 In the **Height** text field, type L_p .
- 5 Locate the **Position** section. In the **r** text field, type R_p .
- 6 In the **z** text field, type $-L_p$.

Rectangle 4 (r4)

- 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 $1.25 * R_p$.
- 4 In the **Height** text field, type $0.75 * R_p$.
- 5 Locate the **Position** section. In the **z** text field, type $-0.25 * R_p$.

Rectangle 5 (r5)



- 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 R_p .
- 4 In the **Height** text field, type L_p .
- 5 Locate the **Position** section. In the **z** text field, type $-L_p$.

Rectangle 6 (r6)



- 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 $2*L_j$.
- 4 In the **Height** text field, type $3*L_j+L_p$.
- 5 Locate the **Position** section. In the **z** text field, type $-L_p$.


Difference 1 (dif1)

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **r4** only.
- 3 In the **Settings** window for **Difference**, locate the **Difference** section.
- 4 Click to select the  **Activate Selection** toggle button for **Objects to subtract**.
- 5 Select the object **r5** only.
- 6 Select the **Keep objects to subtract** checkbox.

Difference 2 (dif2)

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **r6** only.
- 3 In the **Settings** window for **Difference**, locate the **Difference** section.
- 4 Click to select the  **Activate Selection** toggle button for **Objects to subtract**.
- 5 Select the object **r5** only.

Polygon 1 (pol1)


- 1 In the **Geometry** toolbar, click  **Polygon**.
- 2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.
- 3 In the table, enter the following settings:

r (m)	z (m)
R_p	0
$1.01*R_p$	0
$1.03*R_p$	$-dL/2$
R_p	$-dL/2$



- 4 Click  **Build All Objects**.

DEFINITIONS


Explicit 1

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 Select Domains 1 and 2 only.

Complement 1

- 1 In the **Definitions** toolbar, click  **Complement**.
- 2 In the **Settings** window for **Complement**, locate the **Input Entities** section.
- 3 Under **Selections to invert**, click  **Add**.
- 4 In the **Add** dialog, select **Explicit 1** in the **Selections to invert** list.
- 5 Click **OK**.

TURBULENT FLOW, SSG-LRR (SPF)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, SSG-LRR (spf)**.
- 2 In the **Settings** window for **Turbulent Flow, SSG-LRR**, locate the **Domain Selection** section.
- 3 Click  **Clear Selection**.
- 4 Select Domains 1 and 2 only.
- 5 Locate the **Physical Model** section. Select the **Swirl flow** checkbox.

Fluid Properties 1

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Turbulent Flow, SSG-LRR (spf)** click **Fluid Properties 1**.
- 2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.
- 3 From the ρ list, choose **User defined**. In the associated text field, type ρ_w .
- 4 From the μ list, choose **User defined**. In the associated text field, type μ_w .

Initial Values 1

- 1 In the **Model Builder** window, click **Initial Values 1**.
- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 Specify the **u** vector as

$0 \cdot \omega_p \cdot r$	ϕ
U_b	z


Wall 1

- 1 In the **Model Builder** window, click **Wall 1**.
- 2 In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.
- 3 From the **Translational velocity** list, choose **Manual**.


4 Specify the \mathbf{u}_{tr} vector as

$$\omega_p r \quad \phi$$

Periodic Flow Condition 1


- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Periodic Flow Condition**.
- 2 Select Boundaries 2 and 6 only.
- 3 In the **Settings** window for **Periodic Flow Condition**, locate the **Flow Condition** section.
- 4 From the **Flow condition** list, choose **Mass flow**.
- 5 In the \dot{m} text field, type Mf .

Pressure Point Constraint 1

- 1 In the **Physics** toolbar, click  **Points** and choose **Pressure Point Constraint**.
- 2 Select Point 10 only.

MESH 1

Mapped 1

- 1 In the **Mesh** toolbar, click  **Mapped**.
- 2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domains 1 and 2 only.

Distribution 1

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundaries 2, 4, and 6 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution type** list, choose **Predefined**.
- 5 In the **Number of elements** text field, type 200.
- 6 In the **Element ratio** text field, type 10.
- 7 From the **Growth rate** list, choose **Exponential**.

Distribution 2

- 1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundaries 16 and 17 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 1.


5 Click  **Build All**.

STUDY 1

Step 1: Wall Distance Initialization




- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Wall Distance Initialization**.
- 2 In the **Settings** window for **Wall Distance Initialization**, locate the **Physics and Variables Selection** section.
- 3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Turbulent Flow, SSG-LRR 2 (spf2)**.

Step 2: Stationary

- 1 In the **Model Builder** window, click **Step 2: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Turbulent Flow, SSG-LRR 2 (spf2)**.
- 4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 5 Click  **Add**.
- 6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Ns (Swirl number)	0 0.5	

Solution 1 (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Compile Equations: Wall Distance Initialization**.
- 3 In the **Settings** window for **Compile Equations**, locate the **Geometric Entity Selection** section.
- 4 From the **Use entities** list, choose **Selected**.
- 5 Under **Selections**, click  **Add**.
- 6 In the **Add** dialog, select **Explicit 1** in the **Selections** list.
- 7 Click **OK**.
- 8 In the **Study** toolbar, click  **Compute**.

TURBULENT FLOW, SSG-LRR 2 (SPF2)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, SSG-LRR 2 (spf2)**.
- 2 Select Domains 3–8 only.
- 3 In the **Settings** window for **Turbulent Flow, SSG-LRR**, locate the **Physical Model** section.
- 4 Select the **Swirl flow** checkbox.

Fluid Properties 1


- 1 In the **Model Builder** window, under **Component 1 (comp1)** > **Turbulent Flow, SSG-LRR 2 (spf2)** click **Fluid Properties 1**.
- 2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.
- 3 From the ρ list, choose **User defined**. In the associated text field, type rho_w.
- 4 From the μ list, choose **User defined**. In the associated text field, type mu_w.

Initial Values 1

- 1 In the **Model Builder** window, click **Initial Values 1**.
- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 Specify the **u** vector as

Ub/10	z
-------	---

Inlet 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Inlet**.
- 2 Select Boundary 6 only.
- 3 In the **Settings** window for **Inlet**, locate the **Velocity** section.
- 4 Click the **Velocity field** button.
- 5 Specify the **u₀** vector as

withsol('sol1',u,setval(Ns,Ns))	r
withsol('sol1',v,setval(Ns,Ns))	phi
withsol('sol1',w,setval(Ns,Ns))	z


- 6 Locate the **Turbulence Conditions** section. Click the **Specify turbulence variables** button.
- 7 From the list, choose **Symmetric**.

8 Specify the \mathbf{R}_0 matrix as


<code>withsol('sol1',uu, setval(Ns,Ns))</code>	<code>withsol('sol1',uv, setval(Ns,Ns))</code>	<code>withsol('sol1',uw, setval(Ns,Ns))</code>
<code>withsol('sol1',uv,setval(Ns, Ns))</code>	<code>withsol('sol1',vv, setval(Ns,Ns))</code>	<code>withsol('sol1',vw, setval(Ns,Ns))</code>
<code>withsol('sol1',uw,setval(Ns, Ns))</code>	<code>withsol('sol1',vw,setval(Ns, Ns))</code>	<code>withsol('sol1',ww, setval(Ns,Ns))</code>

9 In the ω_0 text field, type `withsol('sol1',spf.om_global,setval(Ns,Ns))`.

Inlet 2


- 1** In the **Physics** toolbar, click  **Boundaries** and choose **Inlet**.
- 2** Select Boundaries 13 and 26 only.
- 3** In the **Settings** window for **Inlet**, locate the **Velocity** section.
- 4** In the U_0 text field, type $U_b/30$.

Wall 2


- 1** In the **Physics** toolbar, click  **Boundaries** and choose **Wall**.
- 2** Select Boundaries 16 and 17 only.
- 3** In the **Settings** window for **Wall**, locate the **Wall Movement** section.
- 4** From the **Translational velocity** list, choose **Manual**.
- 5** Specify the \mathbf{u}_{tr} vector as

<code>(1+z/dL)*omega_p*r</code>	<code>phi</code>
---------------------------------	------------------

Wall 3

- 1** In the **Physics** toolbar, click  **Boundaries** and choose **Wall**.
- 2** Select Boundary 28 only.
- 3** In the **Settings** window for **Wall**, locate the **Boundary Condition** section.
- 4** From the **Wall condition** list, choose **Slip**.

Outlet 1

- 1** In the **Physics** toolbar, click  **Boundaries** and choose **Outlet**.
- 2** Select Boundary 11 only.
- 3** In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.
- 4** Clear the **Suppress backflow** checkbox.

MESH 2

In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

Distribution 1

Right-click **Mesh 2** and choose **Distribution**.


Size

- 1 In the **Settings** window for **Size**, locate the **Element Size** section.
- 2 From the **Calibrate for** list, choose **Fluid dynamics**.

Distribution 1

- 1 In the **Model Builder** window, click **Distribution 1**.
- 2 Select Boundary 6 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution type** list, choose **Predefined**.
- 5 In the **Number of elements** text field, type 200.
- 6 In the **Element ratio** text field, type 10.
- 7 From the **Growth rate** list, choose **Exponential**.
- 8 Select the **Reverse direction** checkbox.

Mapped 1

- 1 In the **Mesh** toolbar, click  **Mapped**.
- 2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 8 only.

Distribution 1

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundary 19 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution type** list, choose **Predefined**.
- 5 In the **Number of elements** text field, type 20.
- 6 In the **Element ratio** text field, type 40.
- 7 From the **Growth rate** list, choose **Exponential**.

Distribution 2


- 1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

- 2 Select Boundary 18 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution type** list, choose **Predefined**.
- 5 In the **Number of elements** text field, type 20.
- 6 In the **Element ratio** text field, type 5.
- 7 From the **Growth rate** list, choose **Exponential**.
- 8 Select the **Reverse direction** checkbox.


Distribution 3

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundaries 17 and 20 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution type** list, choose **Predefined**.
- 5 In the **Number of elements** text field, type 20.
- 6 In the **Element ratio** text field, type 2.
- 7 From the **Growth rate** list, choose **Exponential**.

Free Triangular 1

- 1 In the **Mesh** toolbar, click  **Free Triangular**.
- 2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domains 3 and 7 only.

Size 1


- 1 Right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 Click  **Clear Selection**.
- 4 Select Domain 3 only.
- 5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.
- 6 From the **Predefined** list, choose **Extremely fine**.

Size 2


- 1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.

- 4 Select Boundary 16 only.
- 5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.
- 6 From the **Predefined** list, choose **Extremely fine**.

Size 3

- 1 Right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 Click  **Clear Selection**.
- 4 Select Domain 7 only.
- 5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.
- 6 From the **Predefined** list, choose **Extra fine**.

Free Triangular 2

- 1 In the **Mesh** toolbar, click  **Free Triangular**.
- 2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 4 only.

Size 1


- 1 Right-click **Free Triangular 2** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Calibrate for** list, choose **Fluid dynamics**.
- 4 From the **Predefined** list, choose **Finer**.
- 5 Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section.
- 7 Select the **Maximum element growth rate** checkbox. In the associated text field, type 1.03.

Size 2


- 1 In the **Model Builder** window, right-click **Free Triangular 2** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 7 only.
- 5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.
- 6 Click the **Custom** button.

- 7 Click the **Predefined** button.
- 8 From the **Predefined** list, choose **Extremely fine**.
- 9 Click the **Custom** button.
- 10 Locate the **Element Size Parameters** section.
- 11 Select the **Maximum element size** checkbox. In the associated text field, type 0.0075.

Free Triangular 3

- 1 In the **Mesh** toolbar, click  **Free Triangular**.
- 2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domains 5 and 6 only.

Size 1

- 1 Right-click **Free Triangular 3** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 Click  **Clear Selection**.
- 4 Select Domain 6 only.
- 5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.
- 6 From the **Predefined** list, choose **Finer**.

Size 2

- 1 In the **Model Builder** window, right-click **Free Triangular 3** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 12, 13, and 25 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section.
- 7 Select the **Maximum element size** checkbox. In the associated text field, type 0.01.

Size 3


- 1 Right-click **Free Triangular 3** and choose **Size**.
- 2 Select Domain 5 only.
- 3 In the **Settings** window for **Size**, locate the **Element Size** section.
- 4 From the **Calibrate for** list, choose **Fluid dynamics**.
- 5 From the **Predefined** list, choose **Extremely coarse**.

- 6 Click the **Custom** button.
- 7 Locate the **Element Size Parameters** section.
- 8 Select the **Maximum element growth rate** checkbox. In the associated text field, type 1.1.


Size 4

- 1 Right-click **Free Triangular 3** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 9 only.
- 5 Locate the **Element Size** section. From the **Predefined** list, choose **Coarse**.
- 6 Click the **Custom** button.
- 7 Locate the **Element Size Parameters** section.
- 8 Select the **Maximum element size** checkbox. In the associated text field, type 0.075.



Boundary Layers 1

- 1 In the **Mesh** toolbar, click  **Boundary Layers**.
- 2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domains 6 and 7 only.

Boundary Layer Properties

- 1 In the **Model Builder** window, click **Boundary Layer Properties**.
- 2 Select Boundaries 12 and 14 only.
- 3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.
- 4 In the **Number of layers** text field, type 7.
- 5 Click  **Build All**.

ADD STUDY

- 1 In the **Home** toolbar, click  **Add Study** to open the **Add Study** window.
- 2 Go to the **Add Study** window.
- 3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces > Stationary with Initialization**.
- 4 Click the **Add Study** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Wall Distance Initialization



- 1 In the **Settings** window for **Wall Distance Initialization**, locate the **Physics and Variables Selection** section.
- 2 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Turbulent Flow, SSG-LRR (spf)**.

Step 2: Stationary

- 1 In the **Model Builder** window, click **Step 2: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Turbulent Flow, SSG-LRR (spf)**.
- 4 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 5 Click **+ Add**.
- 6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Ns (Swirl number)	0 0.5	

Solution 3 (sol3)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Compile Equations: Wall Distance Initialization**.
- 3 In the **Settings** window for **Compile Equations**, locate the **Geometric Entity Selection** section.
- 4 From the **Use entities** list, choose **Selected**.
- 5 Under **Selections**, click **+ Add**.
- 6 In the **Add** dialog, select **Complement 1** in the **Selections** list.
- 7 Click **OK**.
- 8 In the **Study** toolbar, click  **Compute**.

RESULTS

- 1 In the **Model Builder** window, click **Results**.
- 2 In the **Settings** window for **Results**, locate the **Update of Results** section.
- 3 Select the **Only plot when requested** checkbox.

Evaluation Group 1

In the **Results** toolbar, click  **Evaluation Group**.

Global Evaluation 1

- 1 Right-click **Evaluation Group 1** and choose **Global Evaluation**.
- 2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
$\text{spf.pfc1.delta_p} \cdot R_p / (\rho_w \cdot U_b^2 \cdot d_L)$	1	

Surface Average 1

- 1 In the **Model Builder** window, right-click **Evaluation Group 1** and choose **Average > Surface Average**.
- 2 Select Domains 3 and 4 only.
- 3 In the **Settings** window for **Surface Average**, locate the **Data** section.
- 4 From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.
- 5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
w2	m/s	Velocity field, z-component

Surface Average 2

- 1 Right-click **Surface Average 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface Average**, locate the **Expressions** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
v2	m/s	Velocity field, phi-component

Surface Average 3


- 1 Right-click **Surface Average 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface Average**, locate the **Expressions** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
u2	m/s	Velocity field, r-component

Surface Average 4

- 1 Right-click **Surface Average 3** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface Average**, locate the **Expressions** section.
- 3 In the table, enter the following settings:


Expression	Unit	Description
spf2.muT/spf2.mu	1	

- 4 In the **Evaluation Group 1** toolbar, click  **Evaluate**.

EVALUATION GROUP 1

- 1 Go to the **Evaluation Group 1** window.
- 2 Click the **Full Precision** button in the window toolbar.

RESULTS

- 1 In the **Model Builder** window, click **Surface Average 4**.
- 2 In the **Evaluation Group 1** toolbar, click  **Evaluate**.


Revolution 2D

- 1 In the **Model Builder** window, expand the **Results > Datasets** node, then click **Revolution 2D**.
- 2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution Layers** section.
- 3 In the **Start angle** text field, type 0.
- 4 In the **Revolution angle** text field, type 360.

Edge 2D 1

- 1 In the **Results** toolbar, click  **More Datasets** and choose **Edge 2D**.
- 2 Select Boundaries 16 and 17 only.

Revolution 2D 3


- 1 In the **Results** toolbar, click  **More Datasets** and choose **Revolution 2D**.
- 2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Edge 2D 1**.
- 4 Locate the **Revolution Layers** section. In the **Revolution angle** text field, type 225.
- 5 In the **Model Builder** window, expand the **Results** node.

Revolution 2D 4


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

2 Right-click **Revolution 2D** and choose **Duplicate**.


Selection

- 1 In the **Results** toolbar, click  **Attributes** and choose **Selection**.
- 2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 4 only.


Parametric Surface 1

- 1 In the **Results** toolbar, click  **More Datasets** and choose **Parametric Surface**.
- 2 In the **Settings** window for **Parametric Surface**, locate the **Parameters** section.
- 3 Find the **First parameter** subsection. In the **Maximum** text field, type R_p .
- 4 Find the **Second parameter** subsection. In the **Maximum** text field, type 2π .
- 5 Locate the **Expressions** section. In the **x** text field, type $s1 \cdot \cos(s2)$.
- 6 In the **y** text field, type $s1 \cdot \sin(s2)$.
- 7 In the **z** text field, type $-dL$.


Parametric Curve 3D 1

- 1 In the **Results** toolbar, click  **More Datasets** and choose **Parametric Curve 3D**.
- 2 In the **Settings** window for **Parametric Curve 3D**, locate the **Parameter** section.
- 3 In the **Maximum** text field, type 2π .
- 4 Locate the **Expressions** section. In the **x** text field, type $0.9 \cdot R_p \cdot \cos(s)$.
- 5 In the **y** text field, type $0.9 \cdot R_p \cdot \sin(s)$.
- 6 In the **z** text field, type $-dL$.

Transformation 3D 1

- 1 In the **Results** toolbar, click  **More Datasets** and choose **Transformation 3D**.
- 2 In the **Settings** window for **Transformation 3D**, locate the **Transformation** section.
- 3 Select the **Scale** checkbox.
- 4 In the **z** text field, type `pipe_plot_z_scale`.

Cut Line 2D 1

- 1 In the **Results** toolbar, click  **Cut Line 2D**.
- 2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.
- 4 Locate the **Line Data** section. In row **Point 2**, set **r** to $3 \cdot R_p$.

- 5 Select the **Additional parallel lines** checkbox.
- 6 In the **Distances** text field, type $2*D_p$ $6*D_p$ $10*D_p$.


Cut Line 2D 2

- 1 Right-click **Cut Line 2D 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.
- 3 In the **Distances** text field, type range ($D_p, D_p, 8*D_p$).

Revolution 2D 5

- 1 In the **Model Builder** window, under **Results** > **Datasets** right-click **Revolution 2D 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution Layers** section.
- 3 In the **Start angle** text field, type 0.
- 4 In the **Revolution angle** text field, type 360.

Pipe Axial Velocity

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Pipe Axial Velocity in the **Label** text field.
- 3 Locate the **Data** section. From the **Parameter selection (Ns)** list, choose **Manual**.
- 4 In the **Parameter indices (1-2)** text field, type 2, 1.
- 5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 6 In the **Title** text area, type Axial velocity - w/U_b .
- 7 Locate the **Plot Settings** section.
- 8 Select the **x-axis label** checkbox. In the associated text field, type r/R_p .
- 9 Select the **y-axis label** checkbox. Clear the associated text field.
- 10 Locate the **Legend** section. From the **Position** list, choose **Center**.

Line Graph 1


- 1 Right-click **Pipe Axial Velocity** and choose **Line Graph**.
- 2 Select Boundary 4 only.
- 3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 4 In the **Expression** text field, type w/U_b .
- 5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 6 In the **Expression** text field, type r/R_p .

- 7 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.
- 8 From the **Color** list, choose **Black**.
- 9 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 10 From the **Legends** list, choose **Manual**.
- 11 In the table, enter the following settings:

Legends
Ns=0.5
Ns=0

- 12 In the **Pipe Axial Velocity** toolbar, click  **Plot**.

Rotating Pipe Tangential Velocity

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Rotating Pipe Tangential Velocity in the **Label** text field.
- 3 Locate the **Data** section. From the **Parameter selection (Ns)** list, choose **From list**.
- 4 In the **Parameter values (Ns)** list box, select **0.5**.
- 5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.
- 6 Locate the **Title** section. From the **Title type** list, choose **Manual**.
- 7 In the **Title** text area, type Swirl velocity - Solid: $v/(Ns*U_b)$ Dotted: r/R_p Dashed: $(r/R_p)^2$.
- 8 Locate the **Plot Settings** section.
- 9 Select the **x-axis label** checkbox. In the associated text field, type r/R_p .

Line Graph 1

- 1 Right-click **Rotating Pipe Tangential Velocity** and choose **Line Graph**.
- 2 Select Boundary 4 only.
- 3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 4 In the **Expression** text field, type $v/(Ns*U_b)$.
- 5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 6 In the **Expression** text field, type r/R_p .
- 7 Locate the **Legends** section. Select the **Show legends** checkbox.
- 8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends

Ns=0.5

10 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

11 From the **Color** list, choose **Black**.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type r/R_p .

4 Locate the **Legends** section. In the table, enter the following settings:

Legends

Line

Line Graph 3

1 Right-click **Line Graph 2** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type $(r/R_p)^2$.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends

Parabola

Pipe Shear Stress

1 In the **Model Builder** window, right-click **Pipe Axial Velocity** and choose **Duplicate**.

2 In the **Settings** window for **ID Plot Group**, type Pipe Shear Stress in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type Shear stress - Black: u_w/u_{τ}^2
Blue: $200 * u_w / U_b^2$.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type r/R_p .

Line Graph 1

- 1 In the **Model Builder** window, expand the **Pipe Shear Stress** node, then 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 uw/u_tau^2 .

Line Graph 2

- 1 Right-click **Results > Pipe Shear Stress > Line Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type $200*uw/Ub^2$.
- 4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle (reset)**.
- 5 From the **Color** list, choose **Blue**.

Pipe Shear Stress

- 1 In the **Model Builder** window, click **Pipe Shear Stress**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Legend** section.
- 3 From the **Position** list, choose **Upper left**.

Pipe Turbulence Kinetic Energy

- 1 Right-click **Pipe Shear Stress** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type Pipe Turbulence Kinetic Energy in the **Label** text field.
- 3 Locate the **Title** section. In the **Title** text area, type $TKE - Black: spf.tke/u_tau^{>2}</sup> \quad Blue: 200*spf.tke/Ub^{>2}</sup>$.


Line Graph 1

- 1 In the **Model Builder** window, expand the **Pipe Turbulence Kinetic Energy** node, then 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 $spf.tke/u_tau^2$.

Line Graph 2

- 1 In the **Model Builder** window, click **Line Graph 2**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type $200*spf.tke/Ub^2$.

Rotating Pipe 3D Velocity Profiles

- 1 In the **Results** toolbar, click  **3D Plot Group**.

- 2 In the **Settings** window for **3D Plot Group**, type Rotating Pipe 3D Velocity Profiles in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **None**.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 5 In the **Title** text area, type Left to right: w/U_b at $N_s=0$; $w/U_b, v/(N_s*U_b)$ at $N_s=0.5$.
- 6 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.
- 7 In the **Relative padding** text field, type 1.

Surface 1

- 1 Right-click **Rotating Pipe 3D Velocity Profiles** and choose **Surface**.
- 2 In the **Model Builder** window, expand the **Results > Views** node, then click **Surface 1**.
- 3 In the **Settings** window for **Surface**, locate the **Data** section.
- 4 From the **Dataset** list, choose **Revolution 2D 4**.
- 5 From the **Parameter value (Ns)** list, choose **0**.
- 6 Locate the **Expression** section. In the **Expression** text field, type w/U_b .

Deformation 1

- 1 Right-click **Surface 1** and choose **Deformation**.
- 2 In the **Settings** window for **Deformation**, locate the **Expression** section.
- 3 In the **z-component** text field, type w/U_b .
- 4 Locate the **Scale** section.
- 5 Select the **Scale factor** checkbox. In the associated text field, type 0.15.

Surface 2

- 1 In the **Model Builder** window, under **Results > Rotating Pipe 3D Velocity Profiles** right-click **Surface 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Parameter value (Ns)** list, choose **0.5**.
- 4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Surface 3

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

View 3D 3

- 1 In the **Settings** window for **View 3D**, locate the **View** section.

- 2 Clear the **Show axis orientation** checkbox.


Surface 3

- 1 In the **Model Builder** window, expand the **Results > Rotating Pipe 3D Velocity Profiles > Surface 3** node, then click **Surface 3**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type $v / (Ns*Ub)$.

Deformation 1

- 1 In the **Model Builder** window, click **Deformation 1**.
- 2 In the **Settings** window for **Deformation**, locate the **Expression** section.
- 3 In the **z-component** text field, type $v / (Ns*Ub)$.

Rotating Pipe Streamlines and Surface Arrows

- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type Rotating Pipe Streamlines and Surface Arrows in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Transformation 3D 1**.
- 4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.
- 5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 6 In the **Title** text area, type Absolute system. Streamlines at $r=0.9 R_p$ and arrow surfaces. Color: $v / (Ns*Ub)$.

Streamline 1

- 1 Right-click **Rotating Pipe Streamlines and Surface Arrows** and choose **Streamline**.
- 2 In the **Settings** window for **Streamline**, locate the **Expression** section.
- 3 In the **r-component** text field, type u .
- 4 In the **phi-component** text field, type v .
- 5 In the **z-component** text field, type $w/pipe_plot_z_scale$.
- 6 Locate the **Streamline Positioning** section. From the **Along curve or surface** list, choose **Parametric Curve 3D 1**.
- 7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.
- 8 Find the **Point style** subsection. From the **Type** list, choose **Arrow**.
- 9 From the **Color** list, choose **Cyan**.

Arrow Surface 1

- 1 In the **Model Builder** window, right-click **Rotating Pipe Streamlines and Surface Arrows** and choose **Arrow Surface**.
- 2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.
- 3 In the **r-component** text field, type u .
- 4 In the **phi-component** text field, type v .
- 5 In the **z-component** text field, type 0 .
- 6 Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D 4**.
- 7 Locate the **Coloring and Style** section.
- 8 Select the **Scale factor** checkbox. In the associated text field, type 0.2 .

Color Expression 1

- 1 Right-click **Arrow Surface 1** and choose **Color Expression**.
- 2 In the **Settings** window for **Color Expression**, locate the **Expression** section.
- 3 In the **Expression** text field, type $v / (Ns * Ub)$.

Arrow Surface 2

- 1 In the **Model Builder** window, under **Results > Rotating Pipe Streamlines and Surface Arrows** right-click **Arrow Surface 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Arrow Surface**, click to expand the **Inherit Style** section.
- 3 From the **Plot** list, choose **Arrow Surface 1**.

Transformation 1

- 1 Right-click **Arrow Surface 2** and choose **Transformation**.
- 2 In the **Settings** window for **Transformation**, locate the **Transformation** section.
- 3 In the **z** text field, type $-dL * pipe_plot_z_scale$.

Surface 1

- 1 In the **Model Builder** window, right-click **Rotating Pipe Streamlines and Surface Arrows** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Revolution 2D 3**.
- 4 Locate the **Expression** section. In the **Expression** text field, type 1 .
- 5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 6 From the **Color** list, choose **Gray**.

Transformation 1

- 1 Right-click **Surface 1** and choose **Transformation**.
- 2 In the **Settings** window for **Transformation**, locate the **Transformation** section.
- 3 Select the **Scale** checkbox.
- 4 In the **z** text field, type `pipe_plot_z_scale`.

Transparency 1

- 1 In the **Model Builder** window, right-click **Surface 1** and choose **Transparency**.
- 2 In the **Settings** window for **Transparency**, locate the **Transparency** section.
- 3 Find the **Transparency** subsection. In the **Transparency** text field, type `0.25`.

View 3D 4

- 1 In the **Model Builder** window, under **Results > Views** click **View 3D 4**.
- 2 In the **Settings** window for **View 3D**, locate the **View** section.
- 3 Clear the **Show axis orientation** checkbox.

Rotating Pipe Streamlines and Surface Arrows, Rotating Frame

- 1 In the **Model Builder** window, right-click **Rotating Pipe Streamlines and Surface Arrows** and choose **Duplicate**.
- 2 In the **Settings** window for **3D Plot Group**, type `Rotating Pipe Streamlines and Surface Arrows, Rotating Frame` 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 `Rotating system. Streamlines at $r=0.9 R_p$ and arrow surfaces. Color: $(\omega r - v) / (Ns * Ub)$.`

Streamline 1

- 1 In the **Model Builder** window, expand the **Rotating Pipe Streamlines and Surface Arrows, Rotating Frame** node, then click **Streamline 1**.
- 2 In the **Settings** window for **Streamline**, locate the **Expression** section.
- 3 In the **r-component** text field, type `u`.
- 4 In the **phi-component** text field, type `v - omega_p * r`.

Arrow Surface 1

- 1 In the **Model Builder** window, click **Arrow Surface 1**.
- 2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.
- 3 In the **r-component** text field, type `u`.
- 4 In the **phi-component** text field, type `v - omega_p * r`.

- 5 Locate the **Coloring and Style** section. In the **Scale factor** text field, type 0.4.

Color Expression 1

- 1 In the **Model Builder** window, expand the **Arrow Surface 1** node, then click **Color Expression 1**.
- 2 In the **Settings** window for **Color Expression**, locate the **Expression** section.
- 3 In the **Expression** text field, type $(\omega_p r - v) / (Ns * Ub)$.


Arrow Surface 2

- 1 In the **Model Builder** window, expand the **Results > Rotating Pipe Streamlines and Surface Arrows, Rotating Frame > Arrow Surface 2** node, then click **Arrow Surface 2**.
- 2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.
- 3 In the **r-component** text field, type u .
- 4 In the **phi-component** text field, type $v - \omega_p r$.

Color Expression 1

- 1 In the **Model Builder** window, click **Color Expression 1**.
- 2 In the **Settings** window for **Color Expression**, locate the **Expression** section.
- 3 In the **Expression** text field, type $(\omega_p r - v) / (Ns * Ub)$.

Swirling Jet Axial Velocity

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Swirling Jet Axial Velocity in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 5 In the **Title** text area, type Axial velocity: $w2/Ub$ - Dashed: $Ns=0$ Solid: $Ns=0.5$.
- 6 Locate the **Plot Settings** section. Select the **x-axis label** checkbox.
- 7 Select the **y-axis label** checkbox.
- 8 In the **x-axis label** text field, type r/R_p .
- 9 Clear the **y-axis label** text field.

Line Graph 1

- 1 Right-click **Swirling Jet Axial Velocity** and choose **Line Graph**.
- 2 In the **Settings** window for **Line Graph**, locate the **Data** section.

- 3 From the **Dataset** list, choose **Cut Line 2D 1**.
- 4 From the **Parameter selection (Ns)** list, choose **From list**.
- 5 In the **Parameter values (Ns)** list box, select **0**.
- 6 Locate the **y-Axis Data** section. In the **Expression** text field, type $w2/U_b$.
- 7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 8 In the **Expression** text field, type r/R_p .
- 9 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
- 10 Locate the **Legends** section. Select the **Show legends** checkbox.
- 11 From the **Legends** list, choose **Manual**.
- 12 In the table, enter the following settings:

Legends
Ns=0, z=0
Ns=0, z=0.2
Ns=0, z=0.6
Ns=0, z=1

Line Graph 2

- 1 Right-click **Line Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Line Graph**, locate the **Data** section.
- 3 In the **Parameter values (Ns)** list box, select **0.5**.
- 4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Solid**.
- 5 From the **Color** list, choose **Cycle (reset)**.
- 6 Locate the **Legends** section. In the table, enter the following settings:

Legends
Ns=0.5, z=0
Ns=0.5, z=0.2
Ns=0.5, z=0.6
Ns=0.5, z=1

Swirling Jet Axial Velocity Fluctuations

- 1 In the **Model Builder** window, right-click **Swirling Jet Axial Velocity** and choose **Duplicate**.

- 2 In the **Settings** window for **ID Plot Group**, type Swirling Jet Axial Velocity Fluctuations in the **Label** text field.
- 3 Locate the **Title** section. In the **Title** text area, type $\sqrt{w^2}/U_b$ - Dashed: $N_s=0$ Solid: $N_s=0.5$.


Line Graph 1

- 1 In the **Model Builder** window, expand the **Swirling Jet Axial Velocity Fluctuations** node, then 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 $\sqrt{w^2}/U_b$.

Line Graph 2

- 1 In the **Model Builder** window, click **Line Graph 2**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type $\sqrt{w^2}/U_b$.

Swirling Jet Axial Velocity on the Axis

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Swirling Jet Axial Velocity on the Axis in the **Label** text field.
- 3 Locate the **Title** section. From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type Axial velocity: w^2/U_b - Dashed: $N_s=0$ Solid: $N_s=0.5$.
- 5 Locate the **Plot Settings** section.
- 6 Select the **x-axis label** checkbox. In the associated text field, type z/D_p .
- 7 Select the **y-axis label** checkbox. Clear the associated text field.
- 8 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.

Line Graph 1

- 1 Right-click **Swirling Jet Axial Velocity on the Axis** and choose **Line Graph**.
- 2 In the **Settings** window for **Line Graph**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.
- 4 From the **Parameter selection (Ns)** list, choose **From list**.
- 5 In the **Parameter values (Ns)** list box, select **0**.
- 6 Select Boundaries 5 and 7 only.
- 7 Locate the **y-Axis Data** section. In the **Expression** text field, type w^2/U_b .

- 8 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 9 In the **Expression** text field, type z/D_p .
- 10 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
- 11 From the **Color** list, choose **Black**.
- 12 Locate the **Legends** section. Select the **Show legends** checkbox.
- 13 From the **Legends** list, choose **Manual**.
- 14 In the table, enter the following settings:


Legends
Ns=0

Line Graph 2

- 1 Right-click **Line Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Line Graph**, locate the **Data** section.
- 3 In the **Parameter values (Ns)** list box, select **0.5**.
- 4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Solid**.
- 5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Ns=0.5

Swirling Jet Radial Velocity

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Swirling Jet Radial 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 **Radial velocity: u_2/U_b** .
- 5 Locate the **Plot Settings** section. Select the **x-axis label** checkbox.
- 6 Select the **y-axis label** checkbox.
- 7 In the **x-axis label** text field, type r/R_p .
- 8 Clear the **y-axis label** text field.
- 9 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 2D 2**.
- 10 From the **Parameter selection (Ns)** list, choose **From list**.

11 In the **Parameter values (Ns)** list box, select **0.5**.

Line Graph 1

- 1 Right-click **Swirling Jet Radial Velocity** and choose **Line Graph**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type $u2/U_b$.
- 4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 5 In the **Expression** text field, type r/R_p .
- 6 Locate the **Coloring and Style** section. From the **Color cycle** list, choose **Long**.
- 7 Locate the **Legends** section. Select the **Show legends** checkbox.
- 8 From the **Legends** list, choose **Manual**.
- 9 In the table, enter the following settings:

Legends
Ns=0.5, z=0
Ns=0.5, z=0.1
Ns=0.5, z=0.2
Ns=0.5, z=0.3
Ns=0.5, z=0.4
Ns=0.5, z=0.5
Ns=0.5, z=0.6
Ns=0.5, z=0.7
Ns=0.5, z=0.8

Swirling Jet Radial Velocity

- 1 In the **Model Builder** window, click **Swirling Jet Radial Velocity**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Legend** section.
- 3 From the **Position** list, choose **Lower left**.

Swirling Jet Swirl Velocity

- 1 Right-click **Swirling Jet Radial Velocity** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Legend** section.
- 3 From the **Position** list, choose **Upper right**.
- 4 In the **Label** text field, type **Swirling Jet Swirl Velocity**.
- 5 Locate the **Title** section. In the **Title** text area, type **Swirl velocity: $v2/(Ns*U_b)$** .
- 6 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

- 7 In the **x minimum** text field, type 0.
- 8 In the **x maximum** text field, type 3.
- 9 In the **y minimum** text field, type 0.
- 10 In the **y maximum** text field, type 0.6.

Line Graph 1

- 1 In the **Model Builder** window, expand the **Swirling Jet Swirl Velocity** node, then 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 $v^2 / (N_s * U_b)$.

Swirling Jet Shear Stress

- 1 In the **Model Builder** window, right-click **Swirling Jet Axial Velocity** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type Swirling Jet Shear Stress in the **Label** text field.
- 3 Locate the **Title** section. In the **Title** text area, type Shear stress: $uw^2 / U_b^{²}$
Dashed: $N_s = 0$ Solid: $N_s = 0.5$.


Line Graph 1

- 1 In the **Model Builder** window, expand the **Swirling Jet Shear Stress** node, then 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 uw^2 / U_b^2 .

Line Graph 2

- 1 In the **Model Builder** window, click **Line Graph 2**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type uw^2 / U_b^2 .

Swirling Jet Radial-Tangential Stress

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Swirling Jet Radial-Tangential Stress 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 Radial-tangential stress: $uv^2 / U_b^{²}$.
- 5 Locate the **Plot Settings** section. Select the **x-axis label** checkbox.
- 6 Select the **y-axis label** checkbox.

- 7 In the **x-axis label** text field, type r/R_p .
- 8 Clear the **y-axis label** text field.
- 9 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 2D 1**.
- 10 From the **Parameter selection (Ns)** list, choose **From list**.
- 11 In the **Parameter values (Ns)** list box, select **0.5**.

Line Graph 1

- 1 Right-click **Swirling Jet Radial-Tangential Stress** and choose **Line Graph**.
- 2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type uv^2/U_b^2 .
- 4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 5 In the **Expression** text field, type r/R_p .
- 6 Locate the **Legends** section. Select the **Show legends** checkbox.
- 7 From the **Legends** list, choose **Manual**.
- 8 In the table, enter the following settings:

Legends
Ns=0.5, z=0
Ns=0.5, z=0.2
Ns=0.5, z=0.6
Ns=0.5, z=1



Swirling Jet Axial-Tangential Stress

- 1 In the **Model Builder** window, right-click **Swirling Jet Radial-Tangential Stress** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type **Swirling Jet Axial-Tangential Stress** in the **Label** text field.
- 3 Locate the **Title** section. In the **Title** text area, type **Axial-tangential stress: vw^2/U_b^2** .

Line Graph 1

- 1 In the **Model Builder** window, expand the **Swirling Jet Axial-Tangential Stress** node, then 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 vw^2/U_b^2 .

Jet streamlines

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type `Jet streamlines` in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D 5**.
- 4 Locate the **Plot Settings** section. From the **View** list, choose **New view**.
- 5 In the **Jet streamlines** toolbar, click  **Plot**.
- 6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 7 In the **Title** text area, type `Streamlines starting at a diameter y=0. Color: v/(Ns*Ub)`.
- 8 Clear the **Parameter indicator** text field.
- 9 Click to collapse the **Title** section. Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Streamline 1

Right-click **Jet streamlines** and choose **Streamline**.

View 3D 5

- 1 In the **Settings** window for **View 3D**, locate the **View** section.
- 2 Clear the **Show grid** checkbox.
- 3 Clear the **Show axis orientation** checkbox.

Streamline 1

- 1 In the **Model Builder** window, under **Results > Jet streamlines** click **Streamline 1**.
- 2 In the **Settings** window for **Streamline**, locate the **Expression** section.
- 3 In the **r-component** text field, type `u2`.
- 4 In the **phi-component** text field, type `v2`.
- 5 In the **z-component** text field, type `w2`.
- 6 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.
- 7 In the **x** text field, type `range(-R_p,R_p/40,-3*R_p/4) range(-3*R_p/4,R_p/20,-R_p/2) range(-R_p/2,R_p/10,R_p/2) range(R_p/2,R_p/20,3*R_p/4) range(3*R_p/4,R_p/40,R_p)`.
- 8 In the **y** text field, type `0`.
- 9 In the **z** text field, type `0`.

- 10** Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.
- 11** In the **Tube radius expression** text field, type 1.
- 12** Select the **Radius scale factor** checkbox. In the associated text field, type 0.001.
- 13** Click to expand the **Advanced** section. In the **Maximum streamline length** text field, type 0.07.
- 14** Clear the **Allow backward time integration** checkbox.

Color Expression 1

- 1** Right-click **Streamline 1** and choose **Color Expression**.
- 2** In the **Settings** window for **Color Expression**, locate the **Expression** section.
- 3** In the **Expression** text field, type $\sqrt{2} / (U_b * N_s)$.