Model created in COMSOL Multiphysics 6.4

Secondary Flow in a Semicircular Duct

Introduction

In this example, the Single-Phase Flow, SSG-LRR interface is used to compute fully developed turbulent flow in a semicircular duct. The characteristics of the resulting secondary flow are analyzed and visualized in detail.

Model Definition

The properties of turbulent flows cannot be fully described by eddy-viscosity models. For more exact and complete predictions, differential Reynolds stress models are needed, since they self-consistently account for the turbulence anisotropy, system rotation, and flow curvature. A convenient practical example is given by duct flows with noncircular cross section. In such ducts, turbulence Reynolds stresses lead to secondary circulation in the cross-stream plane with typical velocities of approximately 1%–2% of the bulk streamwise velocity. More convex walls (and sharper corners) attract the secondary flow streamlines, while more concave walls (and blunter corners) repel them.

Here, the semicircular duct from Ref. 1 is investigated. Its height and width are $H = 33$ mm and $W = 94$ mm, respectively. The Reynolds number,

$$\mathrm{Re} = \frac{U_\mathrm{b} D_\mathrm{h}}{\nu}$$

is $\mathrm{Re} = 80{,}000$. Here, U_b is the streamwise bulk velocity and D_h is the hydraulic diameter of the duct.

Computing a fully developed solution (including accurately resolved secondary flow) from an isotropic turbulence state would require the streamwise length to be at least $L = 100\ D_\mathrm{h}$. In this study, only a stationary solution is of interest, so it is sufficient to use a Periodic Flow Condition with a very short streamwise geometry dimension.

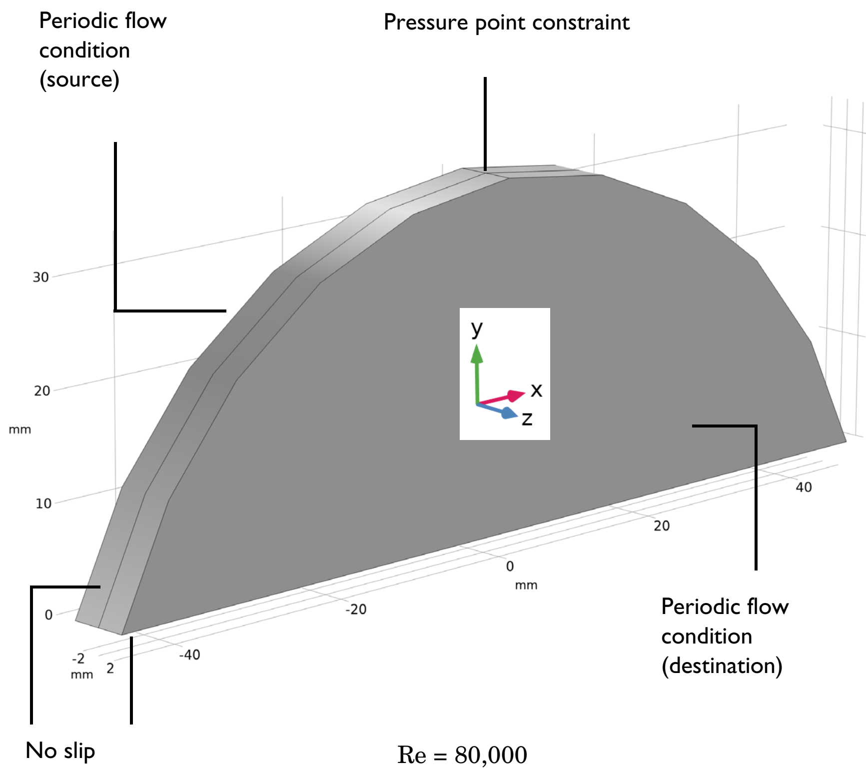

Figure 1: The computational domain of the semicircular duct, which is at least 700 times shorter than would be needed to achieve a fully developed state from the inlet with isotropic turbulence. Positive z-axis points in the streamwise direction.

Implementation in COMSOL Multiphysics

Figure 1 shows the geometry with the streamwise length l, which is 700 times shorter than L defined above, as well as the boundary conditions. For a duct, the number of numerical iterations needed to propagate high turbulent viscosity from the duct walls to the duct center is approximately independent of the mesh (coarse or fine). Thus, the solution is obtained subsequently on *Mesh 1*, *Mesh 2*, and *Mesh 3* shown in Figure 2 (continuing from the previously converged solution). Notice that skipping *Mesh 1* or *Mesh 2* or both increases the overall solution time.

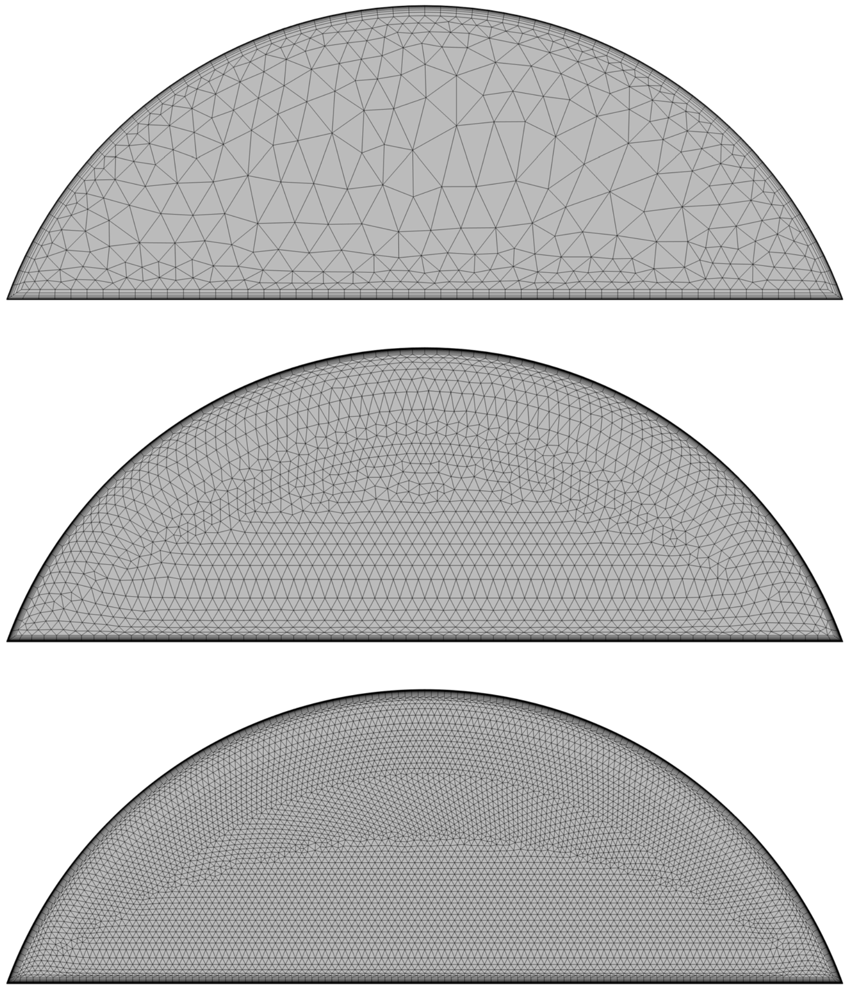

Figure 2: Mesh 1 (coarse), Mesh 2 (normal), and Mesh 3 (fine) used in the model (only two cells in the streamwise direction).

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".

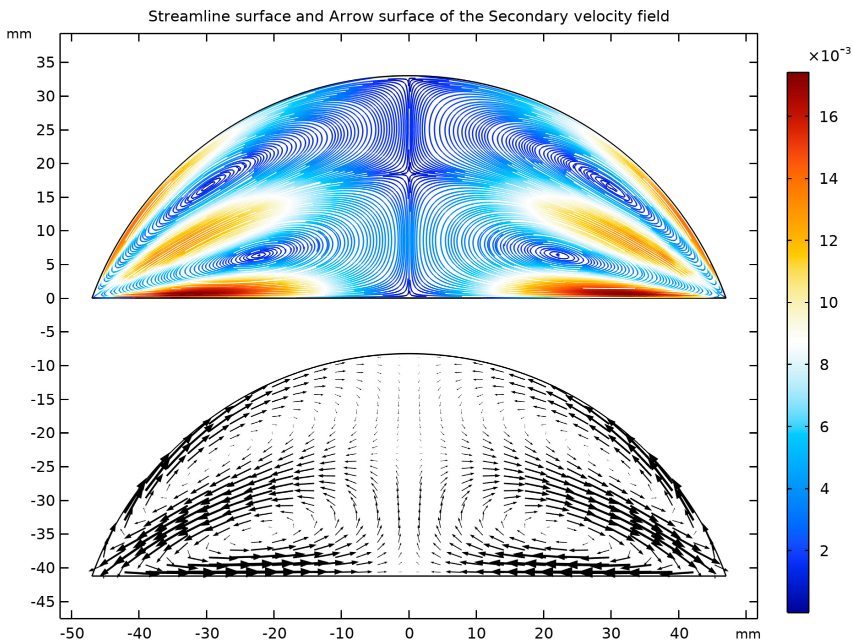

Figure 3: Secondary flow pattern in the cross-stream plane. Streamlines (top) and arrow surface (bottom).

All the results shown below are those obtained on *Mesh 3*. Mesh dependence can be analyzed using the model file.

Figure 3 illustrates the cross-stream streamlines in the model. Four vortices are formed with those near the flat wall being stronger. The streamlines are attracted by the corners and repelled by the bottom and top walls. The maximum magnitude of the cross-stream velocity is 1.6% of the bulk velocity (near the bottom wall), while the maximum far away from the walls is 1.3% of the bulk velocity (along the common streamline of the lower and upper vortices).

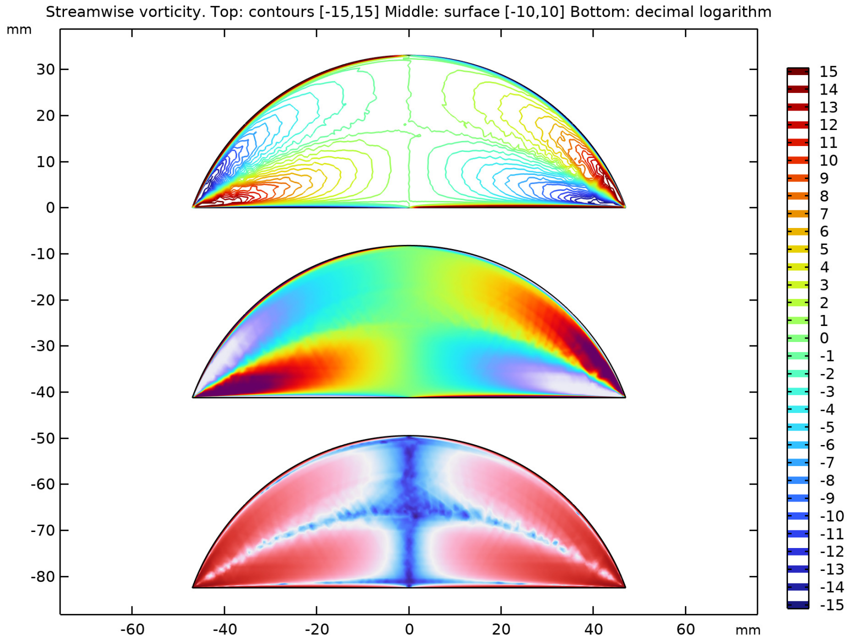

Figure 4: Vorticity produced by the secondary flow. Very high near-wall values (due to the no-slip condition) are beyond the color range for the top and middle plots.

Figure 4 illustrates the vorticity due to the secondary flow. Typical values of ω_z in the bulk of the duct are 10–$15\ \mathrm{s}^{-1}$, while in the wall boundary layer it reaches values above $200\ \mathrm{s}^{-1}$.

Figure 5 shows logarithmic axial velocity profiles on the lines normal to the bottom wall of the duct (friction velocity at the base of each line is used, $u_\tau(x, 0, z)$). All the lines except the one closest to the corner exhibit the same logarithmic region. The cross-stream components of the velocity are not characterized by any wall friction scaling. Indeed, the average values of the axial and cross-stream friction coefficients

$$C_\mathrm{f}^z = \frac{\tau_\mathrm{w}^z}{\frac{1}{2}\rho U_\mathrm{b}^2}, \qquad C_\mathrm{f}^{xy} = \frac{\tau_\mathrm{w}^{xy}}{\frac{1}{2}\rho U_\mathrm{b}^2}$$

are $4.535\cdot 10^{-3}$ and $7.832\cdot 10^{-5}$, respectively. Here, the streamwise and cross-stream wall frictions are defined as

$$\tau_w^z = \mu\frac{\partial w}{\partial n}, \qquad \tau_w^{xy} = \mu\frac{\partial}{\partial n}\sqrt{u^2+v^2}$$

where n is inward-pointing wall normal. Figure 5 uses wall friction velocity defined using the streamwise component of the wall friction vector

$$u_\tau = \sqrt{\tau_w^z/\rho}$$

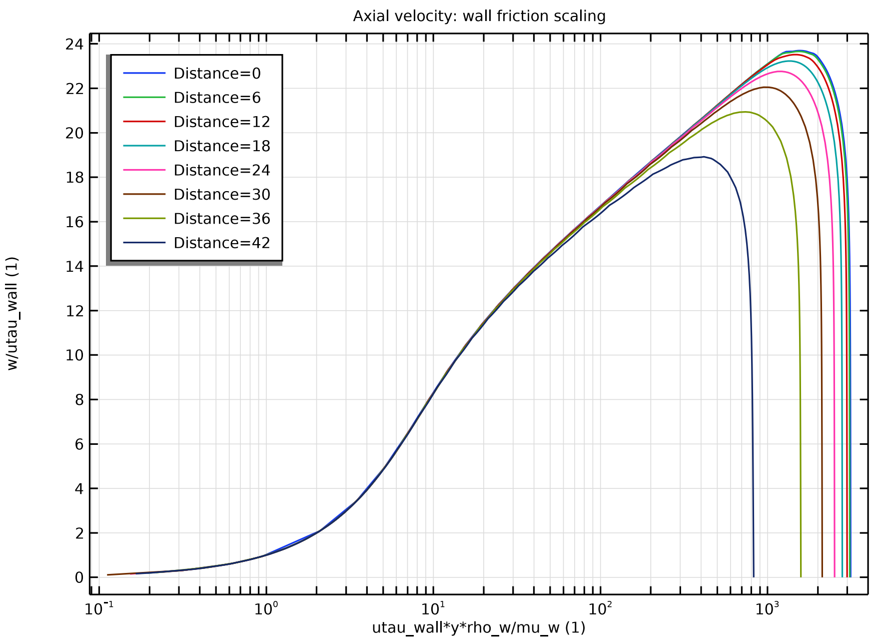

Figure 5: Axial velocity in log-law form. Wall friction taken at the base of each line. The legends indicate the distance of the lines from the symmetry axis in the cross-stream plane.

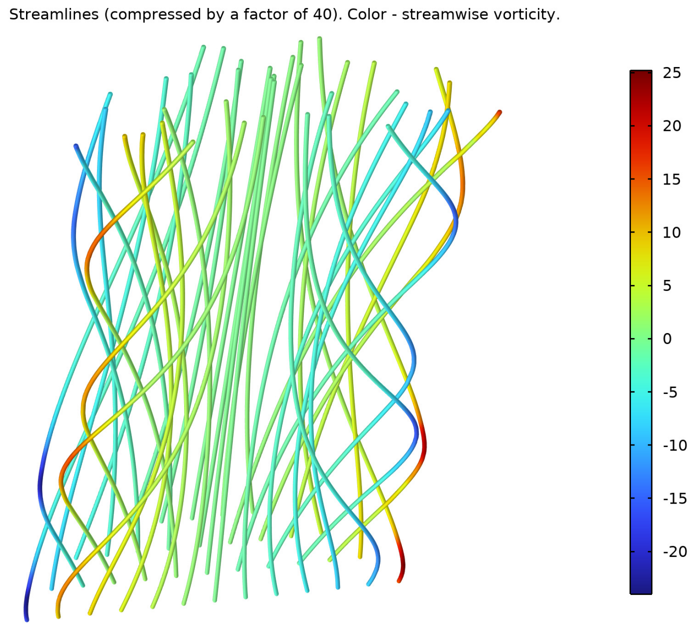

Figure 6: Three-dimensional streamlines compressed 40 times in the streamwise direction to compensate for the low cross-stream velocity.

Figure 6 is the three-dimensional view of the streamlines in the semicircular duct. Significant compression in the streamwise direction is needed to make the bending of the streamlines visually discernible.

Figure 7, Figure 8, and Figure 9 demonstrate different components of the Reynolds stress tensor as well as the turbulence kinetic energy; ww (the axial diagonal component of $\mathbf{R}$) and k are substantially larger than uu and vv (cross-stream diagonal components), which in turn are much larger than the nondiagonal components uv, uw, and vw.

The above results are in good qualitative and quantitative agreement with the experimental data in Ref. 1.

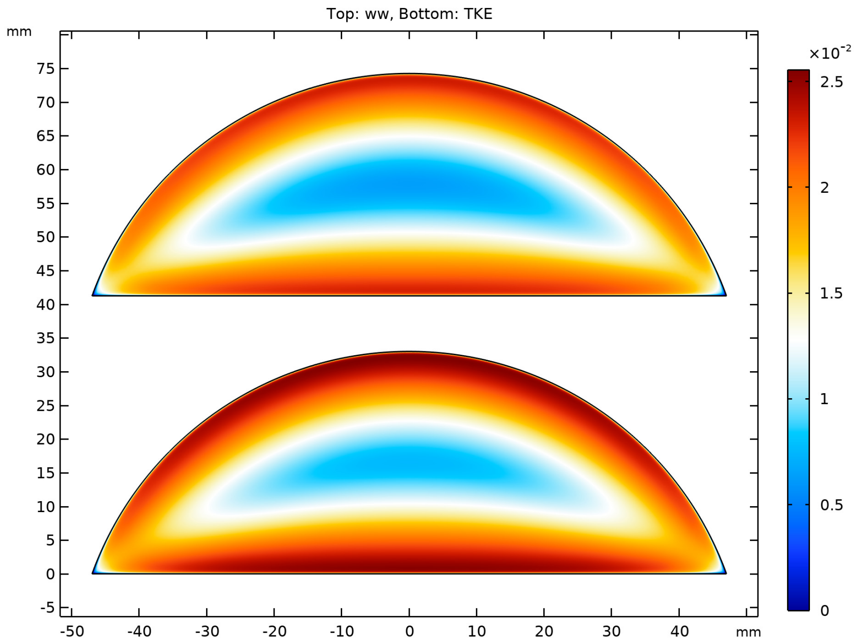

Figure 7: Reynolds stress component ww and turbulence kinetic energy k.

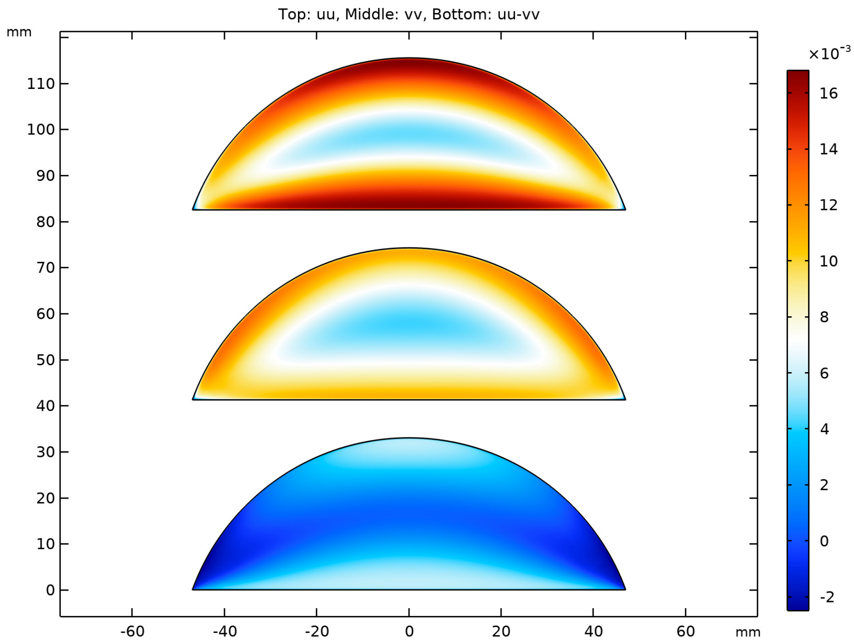

Figure 8: Reynolds stress components uu, vv, *and their difference.*

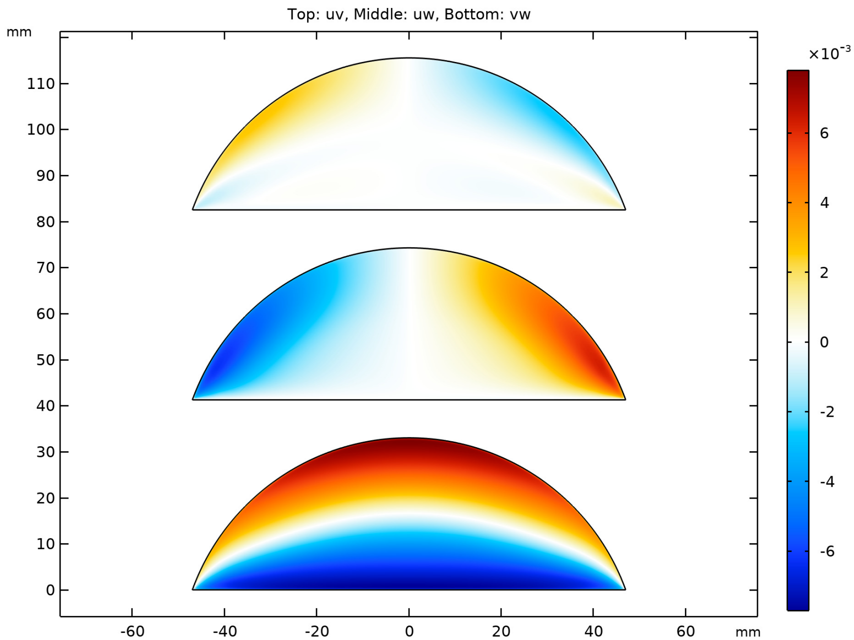

Figure 9: Reynolds stress components uv, uw, and vw.

Summary and Outlook

To summarize, the SSG-LRR Reynolds stress model implemented in COMSOL Multiphysics is able to reveal the main features of the turbulent flow in a duct with noncircular cross section.

Secondary patterns emerging in fully developed turbulent flow in various straight duct geometries can be quickly computed using the Periodic Flow Condition.

Fully developed flow in a curved duct can also be computed with a Reynolds-stress model, although a Periodic Flow Condition should be applied to account for the coordinate transformation between the source and the destination.

Reference

1. I.A.S. Larsson, E.M. Lindmark, T.S.Lundström, and G.J. Nathan "Secondary Flow in Semi-Circular Ducts," *J.Fluids Eng*, vol. 133, no. 10, p. 101206 (8 pages), 2011.

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

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, SSG-LRR (spf)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Load the model parameters. Notice that the density and dynamic viscosity of water are taken at the current temperature and need to be adjusted if the temperature changes.

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 **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `semicircular_duct_parameters_1.txt`.

Load geometry parameters given by expressions.

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 **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `semicircular_duct_parameters_2.txt`.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

4 From the **Angular unit** list, choose **Radians**.

Duct Cross Section

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, type `Duct Cross Section` in the **Label** text field.

Duct Cross Section (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Duct Cross Section (wp1) > Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `r_c`.

4 In the **Sector angle** text field, type `alpha_c`.

5 Locate the **Position** section. In the **yw** text field, type `h_duct-r_c`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `pi/2-alpha_c/2`.

Duct Cross Section (wp1) > Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `2*r_duct`.

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

5 Locate the **Position** section. In the **xw** text field, type `-r_duct`.

Duct Cross Section (wp1) > Intersection 1 (int1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Intersection**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Work Plane** toolbar, click **Build All**.

Since only the fully developed state of the flow is investigated, the flow domain can be taken very short in the streamwise direction and computations can be performed using a **Periodic Flow Condition**.

Extrude 1 (ext1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (mm)
`-l_duct`
`l_duct`

4 Click **Build All Objects**.

DEFINITIONS

Central Plane

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Central Plane` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 6 only.

Walls

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Walls` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 1, 2, 4, 5, 8, and 9 only.

Define a wall-averaging operator and variables for the streamwise and cross-stream friction coefficients, as well as wall friction velocity (pointwise and surface-averaged).

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Walls**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Walls**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
C_z	-spf.T_stressz/(0.5*rho_w*Ub^2)		Streamwise friction coefficient
C_xy	abs(spf.T_stressx*spf.nymesh-spf.T_stressy*spf.nxmesh)/(0.5*rho_w*Ub^2)		Cross-stream friction coefficient

Variables 2

1 Right-click **Definitions** and choose **Variables**.

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

3 Right-click and choose **Paste**.

4 In the table, enter the following settings:

Name	Expression	Unit	Description
utau_av	sqrt(aveop1(-spf.T_stressz/rho_w))	m/s	Average friction coefficient
utau_wall	sqrt(comp1.at1(x,eps,z,-spf.T_stressz/rho_w))	m/s	Friction coefficient at the base

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Measure**.

2 In the **Measure** window for **Measure**, locate the **Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 On the object **ext1**, select Boundary 7 only.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in > Water, liquid**.

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

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

TURBULENT FLOW, SSG-LRR (SPF)

1 In the **Settings** window for **Turbulent Flow, SSG-LRR**, locate the **Physical Model** section.

2 In the T_{ref} text field, type `T_w`.

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 **Model Input** section.

3 From the T list, choose **User defined**. In the associated text field, type `T_w`.

4 Locate the **Fluid Properties** section. From the ρ list, choose **User defined**. In the associated text field, type `rho_w`.

5 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	z

Periodic Flow Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Periodic Flow Condition**.

2 Select Boundaries 3 and 7 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 5 only.

Build very coarse, normal, and fine meshes. Notice that all three meshes resolve boundary layers but have only two cells in the streamwise direction.

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

Size

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

Free Triangular 1

1 In the **Model Builder** window, click **Free Triangular 1**.

2 Select Boundary 3 only.

Size 1

1 Right-click **Free Triangular 1** 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 **Extremely coarse**.

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 3 only.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 Select Edges 2, 3, and 10 only.

3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

4 In the **Number of layers** text field, type `10`.

5 In the **Stretching factor** text field, type `1.5`.

6 From the **Thickness specification** list, choose **First layer**.

7 In the **Thickness** text field, type 1E-2.

Swept 1

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

2 In the **Settings** window for **Swept**, click to expand the **Source Faces** section.

3 Select Boundary 3 only.

4 Click to expand the **Destination Faces** section. Select Boundary 7 only.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type 1.

4 In the **Model Builder** window, right-click **Mesh 1** and choose **Duplicate**.

MESH 2

In the **Model Builder** window, expand the **Component 1 (comp1)** > **Meshes** > **Mesh 2** node.

Size 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Meshes** > **Mesh 2** > **Free Triangular 1** node, then click **Size 1**.

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

3 From the **Predefined** list, choose **Coarse**.

Boundary Layer Properties

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Meshes** > **Mesh 2** > **Boundary Layers 1** node, then click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

3 In the **Number of layers** text field, type 15.

4 In the **Stretching factor** text field, type 1.2.

5 In the **Model Builder** window, right-click **Mesh 2** and choose **Duplicate**.

MESH 3

Size

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Meshes** > **Mesh 3** node, then click **Size**.

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

3 From the **Predefined** list, choose **Fine**.

Size 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Meshes** > **Mesh 3** > **Free Triangular 1** node, then click **Size 1**.

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

3 From the **Predefined** list, choose **Fine**.

Boundary Layers 1

In simulations of duct flows, many iterations are required to transport the high turbulence viscosity generated by high shear near the walls to the duct center. Thus, first obtain a solution on the coarse **Mesh 1**, and then continue by solving on **Mesh 2** and **Mesh 3**.

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Meshes** > **Mesh 3** > **Boundary Layers 1** node.

STUDY 1

Step 1: Wall Distance Initialization, Step 2: Stationary

1 In the **Model Builder** window, under **Study 1**, Ctrl-click to select **Step 1: Wall Distance Initialization** and **Step 2: Stationary**.

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

Step 3: Wall Distance Initialization 1

1 In the **Settings** window for **Wall Distance Initialization**, click to expand the **Mesh Selection** section.

2 In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 2

Step 4: Stationary 1

1 In the **Model Builder** window, click **Step 4: Stationary 1**.

2 In the **Settings** window for **Stationary**, click to expand the **Mesh Selection** section.

3 In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 2

Step 3: Wall Distance Initialization 1, Step 4: Stationary 1

1 In the **Model Builder** window, under **Study 1**, Ctrl-click to select **Step 3: Wall Distance Initialization 1** and **Step 4: Stationary 1**.

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

Step 5: Wall Distance Initialization 2

1 In the **Settings** window for **Wall Distance Initialization**, locate the **Mesh Selection** section.

2 In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 3

Step 6: Stationary 2

1 In the **Model Builder** window, click **Step 6: Stationary 2**.

2 In the **Settings** window for **Stationary**, locate the **Mesh Selection** section.

3 In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 3

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** node, then click **Segregated 1**.

4 In the **Settings** window for **Segregated**, locate the **General** section.

5 From the **Termination technique** list, choose **Iterations or tolerance**.

6 In the **Number of iterations** text field, type 100.

7 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 4** node, then click **Segregated 1**.

8 In the **Settings** window for **Segregated**, locate the **General** section.

9 From the **Termination technique** list, choose **Iterations or tolerance**.

10 In the **Number of iterations** text field, type 30.

11 In the **Initial CFL number** text field, type 5.

12 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 6** node, then click **Segregated 1**.

13 In the **Settings** window for **Segregated**, locate the **General** section.

14 From the **Termination technique** list, choose **Iterations or tolerance**.

15 In the **Number of iterations** text field, type 30.

16 In the **Initial CFL number** text field, type 20.

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

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

19 Clear the **Generate default plots** checkbox.

20 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.

Create datasets to be used in plots.

Surface 1

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

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

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

4 From the **x- and y-axes** list, choose **xy-plane**.

5 Locate the **Selection** section. From the **Selection** list, choose **Central Plane**.

Cut Line 2D 1

1 In the **Results** toolbar, click **Cut Line 2D**.

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

3 In row **Point 2**, set **x** to 0.

4 In row **Point 2**, set **y** to h_duct.

5 Select the **Additional parallel lines** checkbox.

6 Click **Range**.

7 In the **Range** dialog, type 6 in the **Start** text field.

8 In the **Step** text field, type 6.

9 In the **Stop** text field, type 42.

10 Click **Add**.

Cut Line 2D 2

1 In the **Results** toolbar, click **Cut Line 2D**.

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

3 In row **Point 1**, set **x** to 5.

4 In row **Point 2**, set **x** to 5.

5 In row **Point 2**, set **y** to `h_duct`.

Cut Line 2D 3

1 Right-click **Cut Line 2D 2** and choose **Duplicate**.

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

3 In row **Point 1**, set **x** to 15.

4 In row **Point 2**, set **x** to 15.

Cut Line 2D 4

1 Right-click **Cut Line 2D 3** and choose **Duplicate**.

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

3 In row **Point 1**, set **x** to 25.

4 In row **Point 2**, set **x** to 25.

Cut Line 2D 5

1 Right-click **Cut Line 2D 4** and choose **Duplicate**.

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

3 In row **Point 1**, set **x** to 35.

4 In row **Point 2**, set **x** to 35.

Evaluate control quantities on all the meshes.

Evaluation Group 1

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

Streamwise Friction Coefficient

1 Right-click **Evaluation Group 1** and choose **Average** > **Surface Average**.

2 In the **Settings** window for **Surface Average**, type `Streamwise Friction Coefficient` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Walls**.

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

Expression	Unit	Description
withsol('sol3',C_z)	1	
withsol('sol5',C_z)	1	
C_z	1	Streamwise friction coefficient

Cross-Stream Friction Coefficient

1 Right-click **Streamwise Friction Coefficient** and choose **Duplicate**.

2 In the **Settings** window for **Surface Average**, type `Cross-Stream Friction Coefficient` in the **Label** text field.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`withsol('sol3',C_xy)`	1	
`withsol('sol5',C_xy)`	1	
`C_xy`	1	`Cross-stream friction coefficient`

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

The results presented in the plots are from the solution obtained on **Mesh 3**.

Surface Streamlines

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Surface Streamlines` 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 `Streamline surface and Arrow surface of the Secondary velocity field`.

5 Click to expand the **Plot Array** section. Select the **Enable** checkbox.

6 From the **Array axis** list, choose **y**.

7 In the **Relative padding** text field, type `-2.25`.

Streamline 1

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

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

3 From the **Positioning** list, choose **Uniform density**.

4 In the **Density level** text field, type `10.8`.

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(u^2+v^2)/Ub`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Ranitomeya**.

Check that the results on **Mesh 1** and even on **Mesh 2** provide streamlines that are not as smooth as those obtained using **Mesh 3**.

Arrow Surface 1

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

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

3 Find the **x grid points** subsection. In the **Points** text field, type `30`.

4 Find the **y grid points** subsection. In the **Points** text field, type `30`.

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

6 Select the **Scale factor** checkbox. In the associated text field, type `250`.

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

8 In the **Surface Streamlines** toolbar, click **Plot**.

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

Streamwise Velocity: Average Friction Scaling

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

2 In the **Settings** window for **1D Plot Group**, type `Streamwise Velocity: Average Friction Scaling` 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 `Axial velocity: average friction scaling`.

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

6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **Streamwise Velocity: Average Friction Scaling** 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 `w/utau_av`.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `utau_av*y*rho_w/mu_w`.

6 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

7 Click to expand the **Coloring and Style** section. From the **Color cycle** list, choose **Long**.

8 Click to expand the **Legends** section. Select the **Show legends** checkbox.

Streamwise Velocity: Wall Friction Scaling

1 In the **Model Builder** window, right-click **Streamwise Velocity: Average Friction Scaling** and choose **Duplicate**.
2 In the **Settings** window for **1D Plot Group**, type `Streamwise Velocity: Wall Friction Scaling` in the **Label** text field.
3 Locate the **Title** section. In the **Title** text area, type `Axial velocity: wall friction scaling`.

Line Graph 1

1 In the **Model Builder** window, expand the **Streamwise Velocity: Wall Friction Scaling** 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 `w/utau_wall`.
4 Locate the **x-Axis Data** section. In the **Expression** text field, type `utau_wall*y*rho_w/mu_w`.

Secondary Flow: u-Component

1 In the **Model Builder** window, right-click **Streamwise Velocity: Average Friction Scaling** and choose **Duplicate**.
2 In the **Settings** window for **1D Plot Group**, type `Secondary Flow: u-Component` in the **Label** text field.
3 Locate the **Title** section. In the **Title** text area, type `Velocity x-component`.
4 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

Line Graph 1

1 In the **Model Builder** window, expand the **Secondary Flow: u-Component** 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 `u`.
4 Locate the **x-Axis Data** section. In the **Expression** text field, type `y`.
5 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

Secondary Flow: v-Component

1 In the **Model Builder** window, right-click **Secondary Flow: u-Component** and choose **Duplicate**.
2 In the **Settings** window for **1D Plot Group**, type `Secondary Flow: v-Component` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Velocity y-component`.

Line Graph 1

1 In the **Model Builder** window, expand the **Secondary Flow: v-Component** 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`.

Streamwise Vorticity Contours and Plots

1 In the **Results** toolbar, click **2D Plot Group**.

Illustrate the streamwise vorticity due to the secondary flow.

2 In the **Settings** window for **2D Plot Group**, type `Streamwise Vorticity Contours and Plots` in the **Label** text field.

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

4 In the **Title** text area, type `Streamwise vorticity. Top: contours [-15,15] Middle: surface [-10,10] Bottom: decimal logarithm`.

5 Locate the **Plot Array** section. Select the **Enable** checkbox.

6 From the **Array axis** list, choose **y**.

7 In the **Relative padding** text field, type `-2.25`.

Contour 1

1 Right-click **Streamwise Vorticity Contours and Plots** and choose **Contour**.

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

3 In the **Expression** text field, type `max(min(15,spf.vorticityz),-15)`.

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

5 In the **Levels** text field, type `range(-15,1,15)`.

Surface 1

1 In the **Model Builder** window, right-click **Streamwise Vorticity Contours and Plots** and choose **Surface**.

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

3 In the **Expression** text field, type `max(min(10,spf.vorticityz),-10)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

5 Clear the **Color legend** checkbox.

Surface 2

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

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

3 In the **Expression** text field, type `log10(spf.vorticityz)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

5 In the **Streamwise Vorticity Contours and Plots** toolbar, click **Plot**.

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

Plot the streamlines in a 3D view. Significant compression in the streamwise direction is needed for good visualization of the secondary flow.

Secondary Flow in 3D

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

2 In the **Settings** window for **3D Plot Group**, type `Secondary Flow in 3D` 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 `Streamlines (compressed by a factor of 40). Color - streamwise vorticity.`.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

6 From the **View** list, choose **New view**.

7 In the **Secondary Flow in 3D** toolbar, click **Plot**.

Streamline 1

1 Right-click **Secondary Flow in 3D** and choose **Streamline**.

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

3 In the **z-component** text field, type `w/1000`.

4 Locate the **Streamline Positioning** section. In the **Number** text field, type `40`.

5 Select Boundary 3 only.

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

7 Select the **Radius scale factor** checkbox. In the associated text field, type `0.5`.

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 `spf.vorticityz`.

Transformation 1

1 In the **Model Builder** window, right-click **Streamline 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 `25`.

5 In the **Secondary Flow in 3D** toolbar, click **Plot**.

Axial and Secondary (u) Velocity, muT/mu

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Axial and Secondary (u) Velocity, muT/mu` in the **Label** text field.

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

4 In the **Title** text area, type `Top: axial velocity, middle: secondary velocity (u), bottom: muT/mu`.

5 Locate the **Plot Array** section. Select the **Enable** checkbox.

6 From the **Array axis** list, choose **y**.

7 In the **Relative padding** text field, type `0.25`.

Surface 1

1 Right-click **Axial and Secondary (u) Velocity, muT/mu** and choose **Surface**.

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

3 In the **Expression** text field, type `spf.muT/spf.mu`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

5 Clear the **Color legend** checkbox.

Surface 2

1 In the **Model Builder** window, right-click **Axial and Secondary (u) Velocity, muT/mu** and choose **Surface**.

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

3 In the **Expression** text field, type `sqrt(u^2+v^2)/Ub`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Ranitomeya**.

Surface 3

1 Right-click **Axial and Secondary (u) Velocity, muT/mu** and choose **Surface**.

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

3 In the **Expression** text field, type `w/Ub`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Viridis**.

5 Clear the **Color legend** checkbox.

6 In the **Axial and Secondary (u) Velocity, muT/mu** toolbar, click **Plot**.

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

uu, vv, uu-vv

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `uu, vv, uu-vv` in the **Label** text field.

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

4 In the **Title** text area, type `Top: uu, Middle: vv, Bottom: uu-vv`.

5 Locate the **Plot Array** section. Select the **Enable** checkbox.

6 From the **Array axis** list, choose **y**.

7 In the **Relative padding** text field, type `0.25`.

Surface 1

1 Right-click **uu, vv, uu-vv** and choose **Surface**.

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

3 In the **Expression** text field, type `uu-vv`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Ranitomeya**.

Surface 2

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

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

3 In the **Expression** text field, type `vv`.

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

Surface 3

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

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

3 In the **Expression** text field, type `uu`.

4 In the **uu, vv, uu-vv** toolbar, click **Plot**.

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

uv, uw, vw

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `uv, uw, vw` in the **Label** text field.

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

4 In the **Title** text area, type `Top: uv, Middle: uw, Bottom: vw`.

5 Locate the **Plot Array** section. Select the **Enable** checkbox.

6 From the **Array axis** list, choose **y**.

7 In the **Relative padding** text field, type `0.25`.

Surface 1

1 Right-click **uv, uw, vw** and choose **Surface**.

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

3 In the **Expression** text field, type `vw`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Ranitomeya**.

Surface 2

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

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

3 In the **Expression** text field, type `uw`.

4 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Surface 3

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

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

3 In the **Expression** text field, type `uv`.

4 In the **uv, uw, vw** toolbar, click **Plot**.

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

ww and Turbulence Kinetic Energy

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `ww and Turbulence Kinetic Energy` in the **Label** text field.

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

4 In the **Title** text area, type `Top: ww, Bottom: TKE`.

5 Locate the **Plot Array** section. Select the **Enable** checkbox.

6 From the **Array axis** list, choose **y**.

7 In the **Relative padding** text field, type `0.25`.

Surface 1

1 Right-click **ww and Turbulence Kinetic Energy** and choose **Surface**.

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

3 In the **Expression** text field, type `spf.tke`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Ranitomeya**.

Surface 2

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

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

3 In the **Expression** text field, type `ww`.

4 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

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