Model created in COMSOL Multiphysics 6.4

Spanwise Rotating Turbulent Channel Flow

Introduction

In this model, the Single-Phase Flow, Elliptic Blending R-ε interface is employed to investigate fully developed turbulent flow in a spanwise rotating planar channel, as well as the subsequent flow development in a suddenly expanded section. The dynamic characteristics of the resulting flows are analyzed in detail and visualized.

Model Definition

Correctly accounting for the impact of system rotation on turbulent flows is a challenge. Eddy-viscosity models are built so as to properly react to shear and strain but, commonly, not to vorticity. Activating **Include rotation-curvature correction** mends this deficiency to a certain extent. Nevertheless, the Reynolds stress tensor remains strictly proportional to the strain-rate tensor, which is too limiting in the general case. Differential Reynolds stress models of turbulence (RANS-RSM) are based on a deeper foundation, and appropriately respond both to the in-frame flow vorticity and to the frame rotation. However, even in the simplest case of a spanwise rotating turbulent channel, Wilcox R-ω and SSG-LRR underestimate the extent of the region with zero absolute vorticity, and predict laminarization at a too low rotation rate. Fortunately, the Elliptic Blending R-ε model from Ref. 1 properly blends turbulence behavior in the bulk and near-wall regions, which in turn leads to improved reaction to the system rotation.

In this example, spanwise rotating turbulent channel flow is extensively analyzed. The channel Reynolds number,

$$\mathrm{Re} = \frac{U_\mathrm{b}H}{\nu}$$

is $\mathrm{Re} = 14{,}000$. Above, U_b is the streamwise bulk velocity and H is the height of the channel. The range of rotation numbers investigated,

$$\mathrm{Ro} = \frac{\Omega H}{U_\mathrm{b}}$$

is between $\mathrm{Ro} = 0$ and $\mathrm{Ro} = 3$ (note that Rossby number has the same denotation, Ro, although they are inverses of each other). Ω is the angular rotation rate. A Periodic Flow Condition in a geometry that is short in the streamwise direction is employed to compute the fully developed channel flow (periodic channel study). Then, the channel suddenly and symmetrically expands with a 1.5 height ratio. The deflection of the expanding jet and the asymmetry between the separation bubbles are the most apparent outcomes of the system rotation. The case $\mathrm{Ro} = 0.1$ is presented below (expanded channel study).

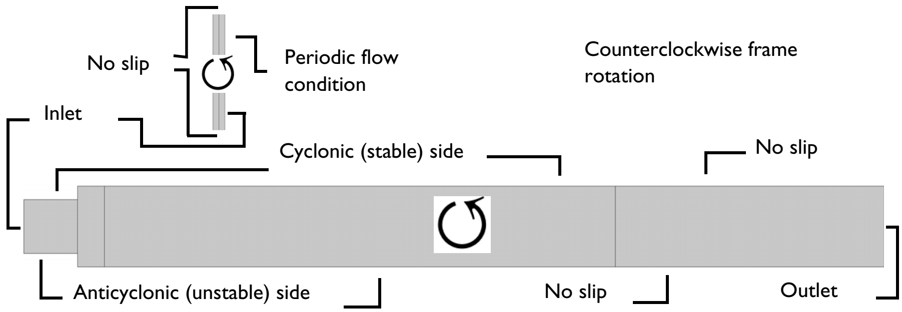

Figure 1: The model geometry of the rotating periodic channel (top) and expanded rotating channel (bottom). The converged fully developed state of the periodic channel study is used as an inlet condition for the expanded channel study.

Assuming positive (counterclockwise, cyclonic) frame rotation and rightward flow direction, the Coriolis force acts downward. The in-frame flow vorticity is cyclonic near the top wall of the channel and anticyclonic near the bottom wall. At the anticyclonic side of the channel, turbulence is enhanced, while it is strongly suppressed at the cyclonic side. Correspondingly, the anticyclonic side is referred to as the "unstable side" and the cyclonic side as the "stable side". In the central region of the turbulent channel flow (far away from the walls), the flow velocity possesses a constant cross-stream slope with the value of the anticyclonic in-frame vorticity, -2Ω, which exactly compensates the cyclonic vorticity, $+2\Omega$, inherent to the system rotation.

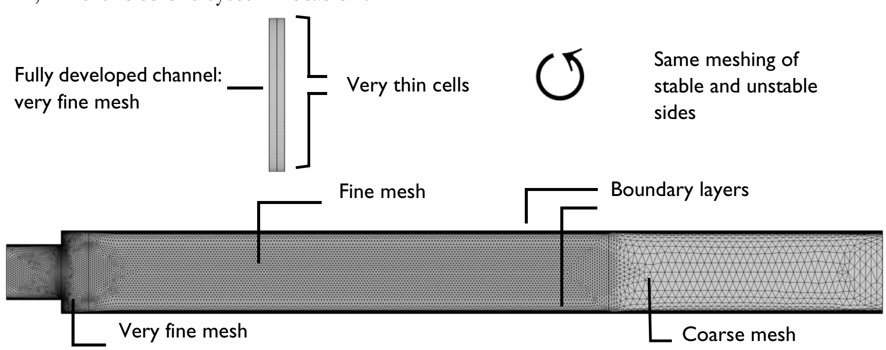

Figure 2: Meshes used for the studies of the rotating periodic channel (top) and expanded channel (bottom). Very thin cells at the walls of the rotating channel are used to ensure accurate evaluation of traction. The initial developing region of the expanding jet is meshed fine, while the remaining near-outlet region serves mostly to reduce the effect of the outlet boundary condition.

Implementation in COMSOL Multiphysics

Figure 1 contains geometries along with boundary conditions for both studies, while Figure 2 presents the corresponding meshes. A **Rotating Frame** feature is employed to impose the frame rotation. It is crucial for the efficient computation of the periodic channel, since the **Periodic Flow Condition** feature is not inherently compatible with a moving wall-based approach. An **Auxiliary sweep** with continuation is used for quicker computation through the bunch of rotation numbers. The expanded channel flow is computed subsequently; values of variables at its inlet are taken from the converged state of the periodic channel flow. Thus, the inlet represents a fully developed turbulent channel flow in the rotating frame. The near-wall cells of the channel should be meshed very well, and the regions with separation bubbles and the ensuing wake should have very fine or at least fine meshing. The remaining part mainly serves to relieve the influence of the outlet boundary condition and might have quite coarse mesh. The pressure at the outlet is taken as if a Coriolis force of a constant velocity flow would be balanced by the cross-stream pressure gradient. Note that **Use reduced pressure** is activated to filter out centrifugal pressure.

The details of the implementation of the Single-Phase Flow, Elliptic Blending R-ε interface can be found in the section *Theory for the Turbulent Flow Interfaces* of the *CFD Module User's Guide*, while specifics of the Rotating Frame feature are listed in *Rotating Frame* description of *Theory for the Single-Phase Flow Interfaces*.

Results and Discussion

Figure 3 shows velocity profiles in rotating turbulent channel flow. With increasing Ro, the profiles first become anticyclonically skewed with boosted slopes near the anticyclonic (unstable) wall and reduced slopes near the cyclonic (stable) wall. Consequently, the velocity maximum moves closer to the stable wall; for Ro in the interval $(0.3, 1.1)$ the position of the maximum is at $y > H/4$, which means that it is near the stable side within one-quarter of H. At higher rotation rate, say at Ro larger than 1.1, the profiles show a clear tendency to laminarization, taking on progressively more parabolic form. At $\mathrm{Ro} = 3$, laminarization is almost complete. Note that at small Ro the increase in the velocity slope near the unstable side is much less visible than the slope's decrease near the stable wall.

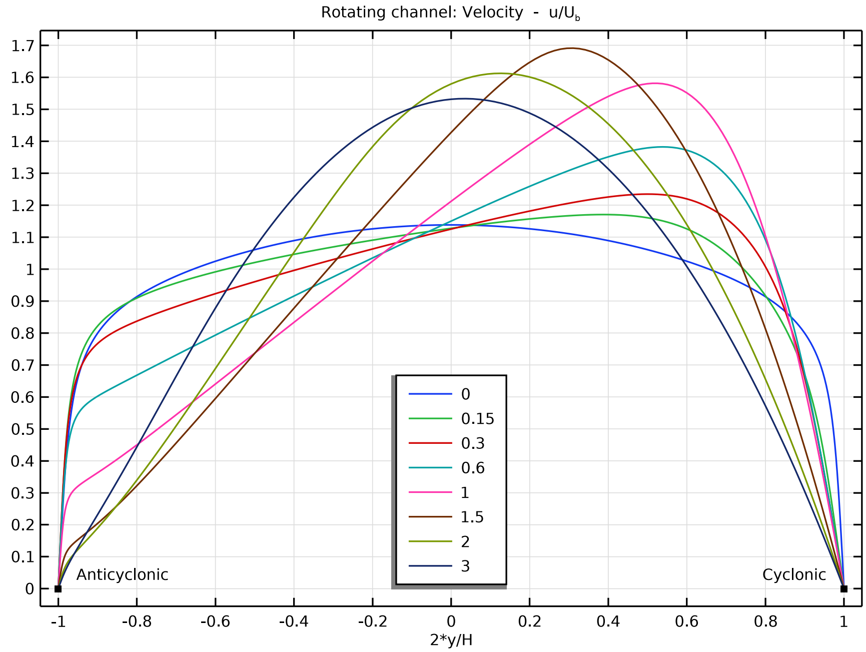

Figure 3: Rotating channel flow at various rotation numbers Ro*: velocity profiles. The channel is situated within* $[-H/2, H/2]$.

Indeed, Figure 4 presents the dependence of the friction velocity $u_\tau = \tau_w/\rho$ on Ro (normalized by the nonrotating value). Clearly, while u_τ at the unstable side grows only slightly with Ro (by 10% at $\mathrm{Ro} = 0.1$), u_τ at the stable side falls abruptly (by 42% at $\mathrm{Ro} = 0.2$). The picture is consistent with Ref. 1. The stable-side plunge in u_τ has correct magnitude, but happens too quickly ($\mathrm{Ro} = 0.5$ is an approximate rotation number when u_τ reduces by 42% according to Large-Eddy Simulation).

Figure 5 illustrates flow pattern in a suddenly expanded channel section at $\mathrm{Ro} = 0.1$ (rotation number is based on the parameters of the periodic channel). The jet becomes quickly deflected to the unstable side by the Coriolis force, and then slowly approaches strictly straight direction in the expanded section. Apparently, two major separation bubbles are formed. The one at the unstable wall is quite compact, shorter than in nonrotating case, with easily distinguishable boundary. Meantime, that on the stable wall is very extended in the streamwise direction and has long fuzzy streaks, which make it hard to determine reconnection point visually. Indeed, according to Ref. 1, the Elliptic

Blending R-ε model accurately reproduces behavior on the unstable side, but is less reliable at the stable side of the expanded channel setup. Allegedly, imperfect sensitivity of the model to the laminarization tendency is responsible for this. It might be suggested that true longitudinal length of the stable-side bubble is approximately half the length predicted by the current version of the Elliptic Blending R-ε. The major bubbles are accompanied by small counterrotating secondary "corner" bubbles (pushed to the corners of the expanded channel).

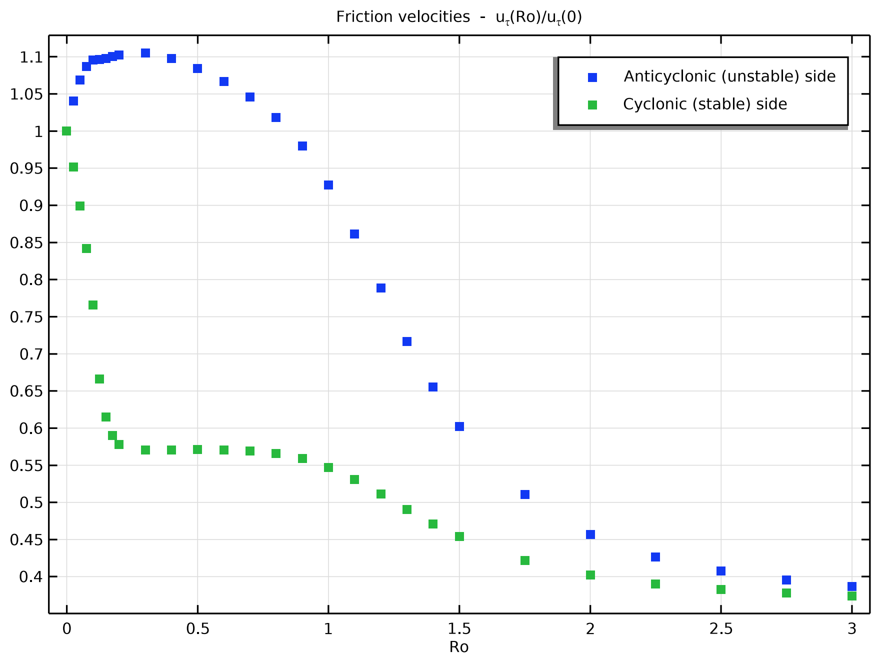

Figure 4: Rotating channel flow at various rotation numbers Ro*:* $u_\tau(\mathrm{Ro})/u_\tau(0)$ *is presented; y is within* $[-H/2, H/2]$.

Figure 6 shows friction coefficients along the stable and unstable walls of the expanded section,

$$C_\mathrm{f} = \frac{\tau_\mathrm{w}}{\frac{1}{2}\rho U_{\mathrm{exp,b}}^2}, \qquad U_{\mathrm{exp,b}} = \frac{H\,U_\mathrm{b}}{H_\mathrm{exp}}, \qquad h_\mathrm{step} = \frac{H_\mathrm{exp} - H}{2}$$

with streamwise length normalized by the step height h_step. Notice that even on unstable side it takes quite a distance before friction coefficient magnitude settles to a constant value. On the stable side C_f is negative for very long stretch, which indicates that the turbulence model overestimates rotation-induced tendency to laminarization. Thus, flow recovery to the new fully developed state is significantly delayed. The friction coefficient variations inside the secondary bubbles are also revealed.

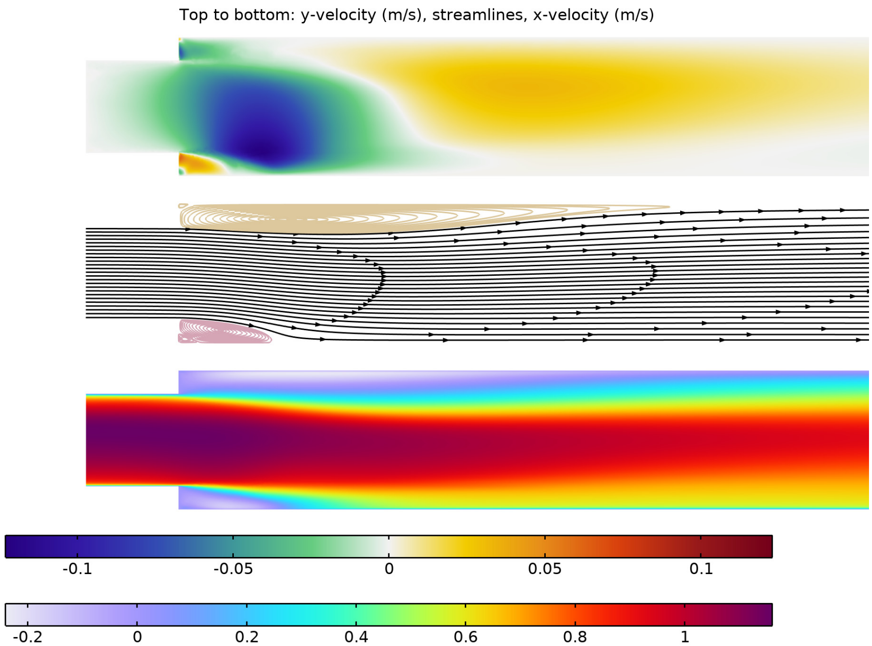

Figure 5: Suddenly expanded rotating channel. The jet is deflected down by the Coriolis force, the negative maximum of the y-velocity is twice higher than its positive maximum, and two different separation regions are formed. The separation regions are in color, and each consists of a major bubble and a counterrotating secondary "corner bubble". $\mathrm{Re} = 14{,}000$ *and* $\mathrm{Ro} = 0.1$ *(parameters for the periodic channel).*

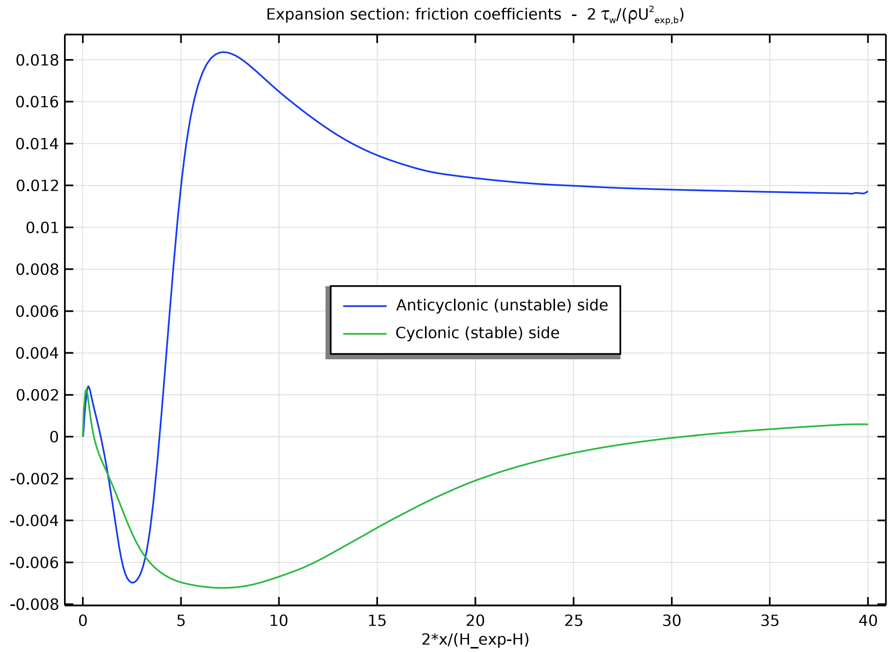

Figure 6: Friction coefficients along the walls of the expanded channel at $\mathrm{Re} = 14{,}000$ *and* $\mathrm{Ro} = 0.1$ *(parameters for the narrower section). The length is normalized by the step height* $h_{\mathrm{step}} = (H_{\mathrm{exp}} - H)/2$.

Figure 7 illustrates contours of reduced pressure, which are almost perpendicular to the Coriolis force and are strongly distorted by the minima associated with pressure losses in the separation regions. The trough on the stable side is especially strong. Velocity magnitude is included too. The recirculation streamline line at the edge of the bottom bubble is well distinguishable, while the recirculation streamline at the edge of the top bubble is sticking to the wall almost in parallel fashion.

Figure 8 demonstrates Reynolds stresses. All the components possess much higher maxima near the unstable side. In particular, the region on the stable side where vv is suppressed extends downstream a lot. This might be one of the reasons for the observed behavior of Elliptic Blending R-ε on the stable side, since turbulent diffusion is proportional to vv in the Daly–Harlow turbulent diffusion model. The shear layers of uu and ww are more elongated at the stable side than at the unstable side, while the opposite applies to the shear layers of uv and vv. The boundary layer of uu (with peak values) is well pronounced near the unstable wall, while it does not develop near the stable wall at the distances shown.

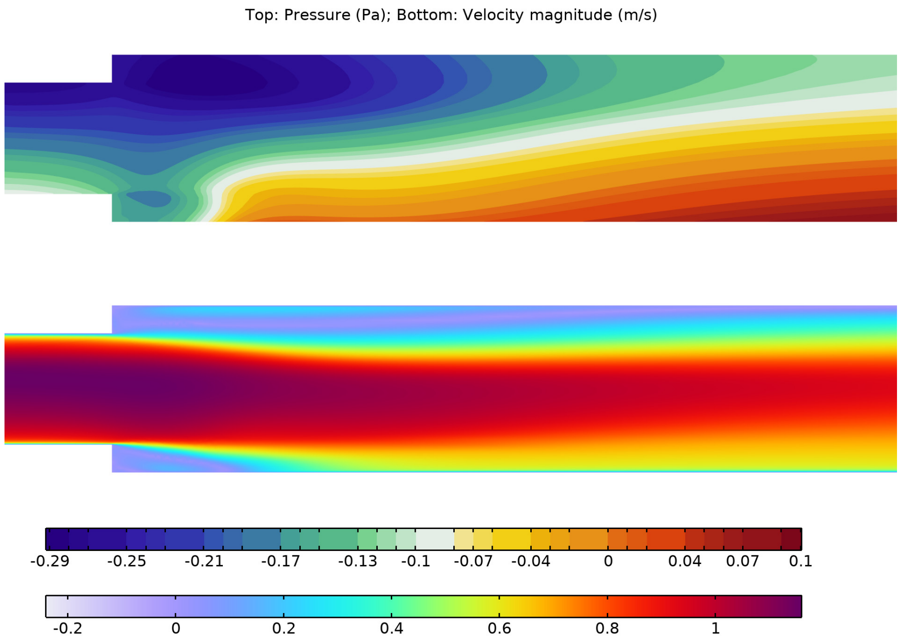

Figure 7: Levels of the reduced pressure with pronounced troughs (especially in the top bubble), and velocity magnitude in the expanded channel. Recirculation lines have purple color. Notice that the unstable side recirculation line terminates abruptly, while the stable side recirculation line leans to the wall smoothly.

Summary and Outlook

The Elliptic Blending R-ε Reynolds stress model implemented in COMSOL Multiphysics can predict most features of rotating turbulent channel flow by properly accounting for the influence of rotation and near-wall effects. Both periodic rotating channels and suddenly expanded rotating channels can be analyzed with the approach.

In rotating periodic channel flow, velocity profiles are captured correctly at various rotation numbers. Friction velocities at the anticyclonic (unstable) side are quite reliable. Due to oversensitivity to rotation, friction velocities at the cyclonic (stable) side abruptly fall with slightly nonzero rotation number.

In rotating expanded channel flow, the major separation bubble and the friction coefficient are predicted well at the unstable side. At the stable side, laminarization tendencies are too strong, which results in an overpredicted bubble size with very high recirculation length.

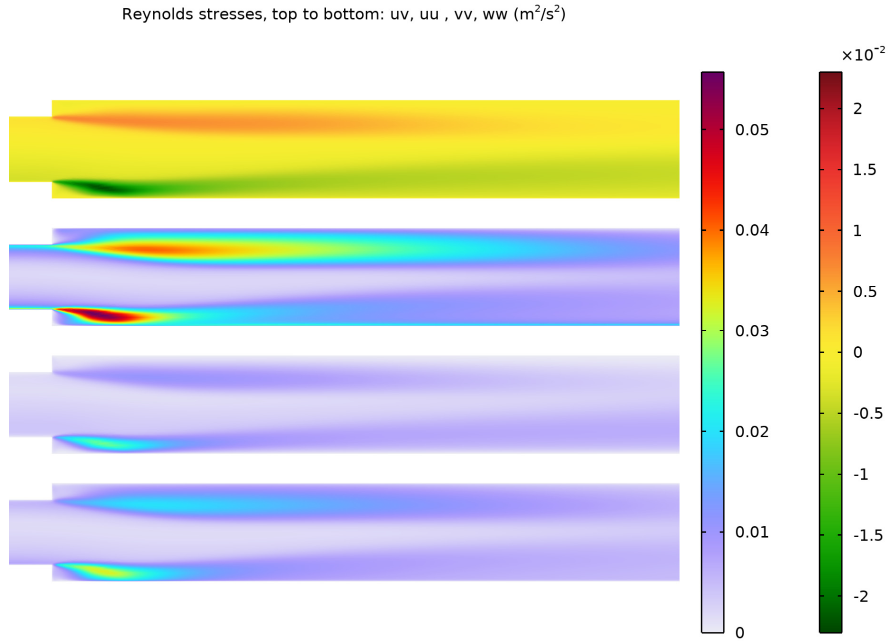

Figure 8: Reynolds stresses in the expanded channel section. Top to bottom: shear stress uv, diagonal components (common color legend) - streamwise uu, cross-stream vv, spanwise ww. There are clear bands of increased stresses in the shear layers, which are stronger at the unstable side. uu has very pronounced region of high values in the boundary layer along the unstable wall, but along the stable wall the boundary layer is not visible at the distances shown.

Subsequently, the friction coefficient recovers, approaching its fully developed value slowly, too.

To summarize, Elliptic Blending R-ε is superior to other RSMs in prediction of near-wall effects, such as friction and Reynolds stress profiles. For example, it is able to correctly reproduce near-wall peaks of the diagonal streamwise component of R_{ij}. The model is a good tool for capturing the effect of system rotation on turbulence development, although its predictions at the stable side (lying opposite to the direction of the Coriolis force) should be taken with caution.

Reference

1. R. Manceau, “Recent progress in the development of the Elliptic Blending Reynolds-stress model,” *Int. J. Heat Fluid Flow*, vol. 51, pp. 195–220, 2015; doi.org/10.1016/j.ijheatfluidflow.2014.09.002.

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

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**.

2 In the **Select Physics** tree, select **Fluid Flow** > **Single-Phase Flow** > **Turbulent Flow** > **Turbulent Flow, Elliptic Blending R-ε** (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 **Load from File**.

4 Browse to the model’s Application Libraries folder and double-click the file `rotating_turbulent_channel_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 **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `rotating_turbulent_channel_parameters_2.txt`.

GEOMETRY 1

Periodic channel

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

2 In the **Settings** window for **Rectangle**, type `Periodic channel` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `2*l_delta`.

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

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **x** text field, type `-L_i-l_delta`.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

8 In the **New Cumulative Selection** dialog, type `Channel` in the **Name** text field.

9 Click **OK**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type `-L_i-l_delta`.

5 In the **y** text field, type `-H/2`.

6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **x** text field, type `-L_i-l_delta`.

8 In the **y** text field, type `H/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 `L_i`.

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

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **x** text field, type `-L_i/2`.

Rectangle 3 (r3)

1 Right-click **Rectangle 2 (r2)** and choose **Duplicate**.

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

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

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

5 Locate the **Position** section. In the **x** text field, type `H/4`.

Rectangle 4 (r4)

1 Right-click **Rectangle 3 (r3)** and choose **Duplicate**.

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

3 In the **Width** text field, type `L_1`.

4 Locate the **Position** section. In the **x** text field, type `L_1/2`.

Rectangle 5 (r5)

1 Right-click **Rectangle 4 (r4)** and choose **Duplicate**.

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

3 In the **Width** text field, type `L_2`.

4 Locate the **Position** section. In the **x** text field, type `L_1+L_2/2`.

DEFINITIONS

Expanded channel

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

2 In the **Settings** window for **Complement**, type `Expanded channel` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog, select **Channel** in the **Selections to invert** list.

5 Click **OK**.

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 > Air**.

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, ELLIPTIC BLENDING R-ε (SPF)

1 In the **Settings** window for **Turbulent Flow, Elliptic Blending R-ε**, locate the **Domain Selection** section.

2 From the **Selection** list, choose **Channel**.

3 Locate the **Physical Model** section. Select the **Rotating frame** checkbox.

4 Select the **Use reduced pressure** checkbox.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Turbulent Flow, Elliptic Blending R-ε** (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 `rho_i`.

4 From the μ list, choose **User defined**. In the associated text field, type `mu_i`.

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

U_i	x

Periodic Flow Condition 1

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

2 Select Boundaries 1 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 `rho_i*U_i*H*1[m]`.

Pressure Point Constraint 1

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

2 Select Point 3 only.

Rotating Frame 1

1 In the **Model Builder** window, click **Rotating Frame 1**.

2 In the **Settings** window for **Rotating Frame**, locate the **Rotating Frame** section.

3 In the Ω_f text field, type `Ro`.

DEFINITIONS

Domain Point Probe 1

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.

2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.

3 In row **Coordinates**, set **x** to `-L_i-l_delta`.

4 In row **Coordinates**, set **y** to `H/2`.

Point Probe Expression 1 (ppb1)

1 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, type `utau_c` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `sqrt(-spf.nu*ppr(uy))`.

Domain Point Probe 2

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Definitions** right-click **Domain Point Probe 1** and choose **Duplicate**.

2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.

3 In row **Coordinates**, set **y** to `-H/2`.

Point Probe Expression 1 (ppb2)

1 In the **Model Builder** window, expand the **Domain Point Probe 2** node, then click **Point Probe Expression 1 (ppb2)**.

2 In the **Settings** window for **Point Probe Expression**, type `utau_ac` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `sqrt(spf.nu*ppr(uy))`.

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 From the **Selection** list, choose **Channel**.

Distribution 1

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

2 Select Boundaries 2, 3, 5, and 6 only.

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

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

Distribution 2

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

2 Select Boundaries 1, 4, and 7 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 15.

7 Select the **Symmetric distribution** checkbox.

8 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, Elliptic Blending R-ε 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, Elliptic Blending R-ε 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
Ro (Rotation number)	`range(0,0.025,0.2) range(0.3,0.1,1.4) range(1.5,0.25,3)`	

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, 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 **Channel (Domain)** in the **Selections** list.
7 Click **OK**.
8 In the **Study** toolbar, click **Compute**.

TURBULENT FLOW, ELLIPTIC BLENDING R-ε 2 (SPF2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, Elliptic Blending R-ε 2 (spf2)**.
2 In the **Settings** window for **Turbulent Flow, Elliptic Blending R-ε**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Expanded channel**.
4 Locate the **Physical Model** section. Select the **Rotating frame** checkbox.
5 Select the **Use reduced pressure** checkbox.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Turbulent Flow, Elliptic Blending R-ε** 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_i`.
4 From the μ list, choose **User defined**. In the associated text field, type `mu_i`.

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

U_i*H/H_exp	x

Rotating Frame 1

1 In the **Model Builder** window, click **Rotating Frame 1**.

2 In the **Settings** window for **Rotating Frame**, locate the **Rotating Frame** section.

3 In the Ω_f text field, type `Ro_2`.

Inlet 1

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

2 Select Boundary 7 only.

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

4 Click the **Velocity field** button.

5 Specify the $\mathbf{u}_0$ vector as

withsol('sol1',u,setval(Ro,Ro_2))	x
withsol('sol1',v,setval(Ro,Ro_2))	y

6 Locate the **Turbulence Conditions** section. Click the **Specify turbulence variables** button.

7 Specify the $\mathbf{R}_0$ matrix as

withsol('sol1',uu, setval(Ro,Ro_2))	withsol('sol1',uv, setval(Ro,Ro_2))	0
withsol('sol1',uv,setval(Ro, Ro_2))	withsol('sol1',vv, setval(Ro,Ro_2))	0
0	0	withsol('sol1',ww, setval(Ro,Ro_2))

8 In the ε_0 text field, type `withsol('sol1',spf.ep_global,setval(Ro,Ro_2))`.

9 In the α_0 text field, type `withsol('sol1',alpha,setval(Ro,Ro_2))`.

Outlet 1

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

2 Select Boundary 21 only.

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

4 In the p_0 text field, type `-2*rho_i*spf2.Omegaz*U_i*(H/H_exp)*y`.

5 Select the **Normal flow** 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**.

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

Distribution 1

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

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

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type `200`.

5 In the **Element ratio** text field, type `15`.

6 Select the **Symmetric distribution** checkbox.

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 From the **Selection** list, choose **Expanded channel**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 Select Domains 3 and 4 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 **Finer**.

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 **Point**.

4 Select Points 7–10 only.

5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

6 From the **Predefined** list, choose **Extremely fine**.

7 Click the **Custom** button.

8 Locate the **Element Size Parameters** section.

9 Select the **Maximum element size** checkbox. In the associated text field, type `0.0101/1`.

10 Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.03`.

Size 3

1 Right-click **Free Triangular 1** 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 **Fine**.

Size 4

1 Right-click **Free Triangular 1** and choose **Size**.

2 Select Domain 6 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 **Coarser**.

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 From the **Selection** list, choose **Expanded channel**.

Boundary Layer Properties

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

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

3 Click **Clear Selection**.

4 Select Boundaries 8–11, 13, 14, 16, and 17 only.

5 Locate the **Layers** section. In the **Number of layers** text field, type `18`.

6 In the **Stretching factor** text field, type `1.1`.

7 In the **Thickness adjustment factor** text field, type `0.5`.

Boundary Layer Properties 1

1 Right-click **Boundary Layer Properties** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundaries 19 and 20 only.

5 Locate the **Layers** section. In the **Thickness adjustment factor** text field, type `0.25`.

6 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, Elliptic Blending R-ε (spf)**.

Step 2: Stationary

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

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

3 From the **Tolerance** list, choose **User controlled**.

4 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Turbulent Flow, Elliptic Blending R-ε (spf)**.

5 Click to expand the **Results While Solving** section. From the **Probes** list, choose **None**.

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 **Expanded channel** in the **Selections** list.

7 Click **OK**.

8 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 2** node, then click **Segregated 1**.

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

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

11 In the **Number of iterations** text field, type `400`.

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

RESULTS

Rotating channel: velocity profiles

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

2 In the **Settings** window for **1D Plot Group**, type `Rotating channel: velocity profiles` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (Ro)** list, choose **From list**.

4 In the **Parameter values (Ro)** list, choose **0**, **0.15**, **0.3**, **0.6**, **1**, **1.5**, **2**, and **3**.

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

6 In the **Title** text area, type `Rotating channel: Velocity - u/U<sub>b</sub>`.

7 Locate the **Plot Settings** section.

8 Select the **x-axis label** checkbox. In the associated text field, type `2*y/H`.

9 Select the **y-axis label** checkbox.

10. Locate the **Legend** section. From the **Position** list, choose **Lower middle**.

Line Graph 1

1. Right-click **Rotating channel: velocity profiles** 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 `u/U_i`.
5. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6. In the **Expression** text field, type `2*y/H`.
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.

Annotation 1

1. In the **Model Builder** window, right-click **Rotating channel: velocity profiles** and choose **Annotation**.
2. In the **Settings** window for **Annotation**, locate the **Coloring and Style** section.
3. From the **Anchor point** list, choose **Lower left**.
4. Locate the **Position** section. In the **x** text field, type `-1`.
5. Locate the **Annotation** section. In the **Text** text field, type `Anticyclonic`.

Annotation 2

1. Right-click **Annotation 1** and choose **Duplicate**.
2. In the **Settings** window for **Annotation**, locate the **Coloring and Style** section.
3. From the **Anchor point** list, choose **Lower right**.
4. Locate the **Position** section. In the **x** text field, type `1`.
5. Locate the **Annotation** section. In the **Text** text field, type `Cyclonic`.

Rotating channel: friction velocities

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Rotating channel: friction velocities` in the **Label** text field.
3. Locate the **Title** section. From the **Title type** list, choose **Manual**.
4. In the **Title** text area, type `Friction velocities - u<sub>\tau</sub>(Ro)/u<sub>\tau</sub>(0)`.
5. Locate the **Plot Settings** section.
6. Select the **x-axis label** checkbox. In the associated text field, type `Ro`.

7 Select the **y-axis label** checkbox.

Global 1

1 Right-click **Rotating channel: friction velocities** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
utau_ac/withsol('sol1',utau_ac,setind(Ro,1))	1	
utau_c/withsol('sol1',utau_c,setind(Ro,1))	1	

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

6 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
Anticyclonic (unstable) side
Cyclonic (stable) side

Expanded section: friction coefficients

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

2 In the **Settings** window for **1D Plot Group**, type Expanded section: friction coefficients in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.

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

5 In the **Title** text area, type Expansion section: friction coefficients - 2 \tau_w/(\rho U²_{exp,b}).

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** checkbox. In the associated text field, type 2*x/(H_exp-H).

8 Select the **y-axis label** checkbox.

9 Locate the **Legend** section. From the **Position** list, choose **Center**.

Line Graph 1

1 Right-click **Expanded section: friction coefficients** and choose **Line Graph**.

2 Select Boundaries 11 and 16 only.

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

4 In the **Expression** text field, type `2*spf2.nu*ppr(u2y)/(U_i*H/H_exp)^2`.

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

6 In the **Expression** text field, type `4*x/H`.

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
Anticyclonic (unstable) side

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 `-2*spf2.nu*ppr(u2y)/(U_i*H/H_exp)^2`.

4 Locate the **Selection** section. Click **Clear Selection**.

5 Select Boundaries 14 and 17 only.

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

Legends
Cyclonic (stable) side

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 1**, set **y** to `-H_exp/2`.

5 In row **Point 2**, set **x** to `0.46`.

6 In row **Point 2**, set **y** to `-0.62`.

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 row **Point 1**, set **y** to `H_exp/2`.

4 In row **Point 2**, set **x** to `1.27`.

5 In row **Point 2**, set **y** to `0.58`.

Velocity and streamlines (spf2)

1 In the **Model Builder** window, under **Results** click **Velocity (spf2)**.

2 In the **Settings** window for **2D Plot Group**, type `Velocity and streamlines (spf2)` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

4 In the **Velocity and streamlines (spf2)** toolbar, click **Plot**.

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

6 In the **Title** text area, type `Top to bottom: y-velocity (m/s), streamlines, x-velocity (m/s)`.

7 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.

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

9 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

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

11 In the **Relative padding** text field, type `0.2`.

Surface

1 In the **Model Builder** window, expand the **Velocity and streamlines (spf2)** node, then click **Surface**.

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

3 In the **Expression** text field, type `v2`.

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

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

6 In the **Minimum** text field, type `-0.123`.

7 In the **Maximum** text field, type `0.123`.

8 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

9 In the **Index** text field, type `2`.

Streamline 1

1 In the **Model Builder** window, right-click **Velocity and streamlines (spf2)** and choose **Streamline**.

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

3 In the **x-component** text field, type `u2`.

4 In the **y-component** text field, type `v2`.

5 Locate the **Streamline Positioning** section. In the **Number** text field, type 25.

6 Select Boundary 12 only.

7 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

8 From the **Arrow distribution** list, choose **Equal time**.

9 Click to expand the **Advanced** section. In the **Loop tolerance** text field, type 0.003.

10 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

11 In the **Index** text field, type 1.

Streamline 2

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

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

3 From the **Positioning** list, choose **Starting-point controlled**.

4 From the **Along curve** list, choose **Cut Line 2D 1**.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **None**.

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

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

8 Click **Define custom colors**.

9 Set the RGB values to 213, 164, and 181, respectively.

10 Click **Add to custom colors**.

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

Streamline 3

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

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

3 From the **Along curve** list, choose **Cut Line 2D 2**.

4 Locate the **Coloring and Style** section. Click **Define custom colors**.

5 Set the RGB values to 220, 200, and 156, respectively.

6 Click **Add to custom colors**.

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

Surface 2

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

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

3 In the **Expression** text field, type `u2`.

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

5 Locate the **Range** section. In the **Minimum** text field, type `-0.24`.

6 In the **Maximum** text field, type `1.16`.

7 Locate the **Plot Array** section. In the **Index** text field, type `0`.

8 In the **Velocity and streamlines (spf2)** toolbar, click **Plot**.

Pressure and velocity (spf2)

1 In the **Model Builder** window, under **Results** click **Pressure (spf2)**.

2 In the **Settings** window for **2D Plot Group**, type `Pressure and velocity (spf2)` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

4 In the **Pressure and velocity (spf2)** toolbar, click **Plot**.

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

6 In the **Title** text area, type `Top: Pressure (Pa); Bottom: Velocity magnitude (m/s)`.

7 Locate the **Plot Array** section. From the **Array type** list, choose **Linear**.

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

9 In the **Relative padding** text field, type `0.5`.

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

11 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.

Surface 2

Right-click **Pressure and velocity (spf2)** and choose **Surface**.

Surface 1

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

2 In the **Number of bands** text field, type `30`.

3 Locate the **Plot Array** section. Select the **Manual indexing** checkbox.

4 In the **Index** text field, type `1`.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

3 In the **Expression** text field, type spf2.U.

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

5 Locate the **Range** section. Select the **Manual color range** checkbox.

6 In the **Minimum** text field, type -0.24.

7 In the **Maximum** text field, type 1.16.

8 Locate the **Plot Array** section. Select the **Manual indexing** checkbox.

9 In the **Pressure and velocity (spf2)** toolbar, click **Plot**.

Reynolds stresses

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

2 In the **Settings** window for **2D Plot Group**, type Reynolds stresses in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.

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

5 In the **Title** text area, type Reynolds stresses, top to bottom: uv, uu , vv, ww (m²/s²).

6 Clear the **Parameter indicator** text field.

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

8 Locate the **Plot Array** section. From the **Array type** list, choose **Linear**.

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

Surface 1

1 Right-click **Reynolds stresses** and choose **Surface**.

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

3 In the **Expression** text field, type ww2.

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

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 vv2.

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 `uu2`.

Surface 4

1 In the **Model Builder** window, under **Results** > **Reynolds stresses** 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 `uv2`.

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

5 Locate the **Range** section. Select the **Manual color range** checkbox.

6 In the **Minimum** text field, type `-0.023`.

7 In the **Maximum** text field, type `0.023`.

8 In the **Reynolds stresses** toolbar, click **Plot**.