Created in COMSOL Multiphysics 6.2

# Turbulent Parallel Flow in a Channel with Adjacent Fluid and Porous Regions

## Introduction

This verification example reproduces the characteristics of turbulent flow in a channel modified by the presence of an adjacent porous region. Asymmetric profiles of velocity, turbulence kinetic energy, and shear stress, as well as higher turbulence levels and higher friction coefficients both at the solid wall and the interface between the clear-flow and porous regions, are observed. High-permeability cases reveal excellent agreement between COMSOL Multiphysics simulations, DNS, and experimental results without the need for specifically calibrated empirical functions. The turbulent flow field in both the clear-flow and porous domains is computed using the Turbulent Flow, Low Re $k$-ε interface.

## Model Definition

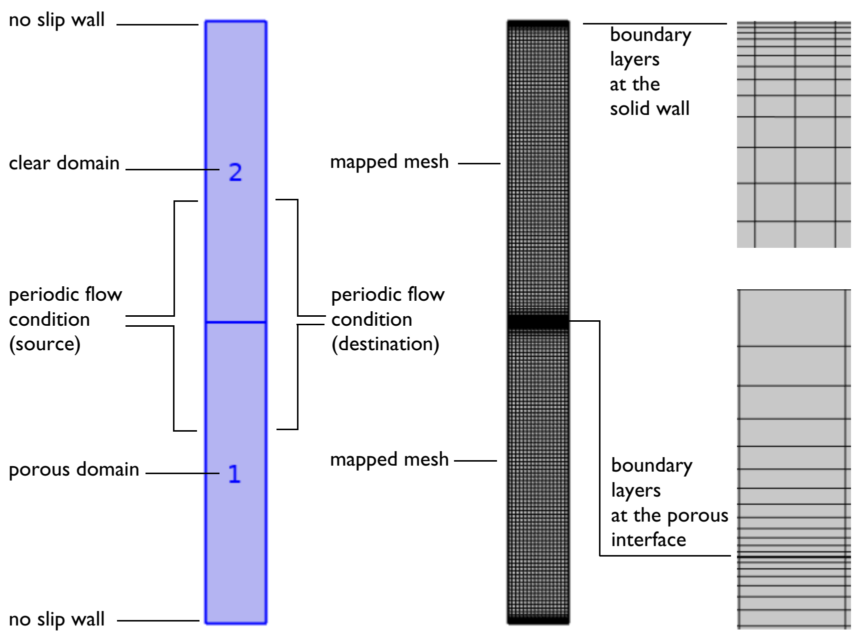


*Figure 1: Geometry with clear (2) and porous (1) domains, boundary conditions, mesh with boundary layer regions (at the solid wall and at the porous interface) zoomed in.*

Turbulent flows in porous media can occur in many cases of academic and practical interest: filters, catalytic converters in exhaust systems, chemical reactors, oil transport and recovery in wells, transport of contaminants in groundwater, porous river beds, plant canopies and dense urban areas, thermal power plants, heat exchangers with porous-metal inserts, and even propagation of forest fires. Here, a simple model of parallel flows in adjacent free and porous domains is considered.

Figure 1 presents the model's geometry, domain properties, boundary conditions, and mesh. A Periodic Flow Condition is employed to avoid simulating an upstream region with developing flow in the streamwise direction. A mapped mesh is used, and the region near porous interface (the boundary between the clear flow and porous domains) is meshed using a finer boundary layers mesh than that near the no-slip walls since the friction velocity at the porous interface, $u_\tau{}^{\mathrm{p}}$, is expected to be significantly higher than the friction velocity at the solid walls, $u_\tau{}^{\mathrm{s}}$.

The channel Reynolds number based on the average (bulk) velocity and the height of the clear channel, $\mathrm{Re} = U_\mathrm{b}H/\nu$, is higher than 5000 for the cases considered, which ensures that the flow is turbulent. The low Reynolds number $k$-ε turbulence model is chosen because it correctly predicts the friction coefficient in turbulent channel flow. The extension of the model to porous media is described in the theory section for the Turbulent Flow interfaces in the *CFD Module User's Guide*.

In the Direct Numerical Simulations (DNS), Ref. 1, and experiments, Ref. 2, the average (bulk) velocity through the clear part of the channel was imposed. To obtain the required driving force, the Pressure difference option of the Periodic Flow Condition is combined with a Global Equations feature, as described in the Modeling Instructions section.

Here, only the cases with high permeability from both DNS and experiments are simulated, because at lower permeabilities the effect of increasingly "solid wall" behavior requires special treatment of the porous interface to correctly model pressure and momentum losses as well as turbulence generation at the interface (especially for nearly impermeable cases).

Note that before solving the problem of interest, with the porous domain activated, the cases are solved with the clear channel region only (applying a no-slip condition at the porous interface). A Parametric Sweep is used to facilitate the process of parameter variation.

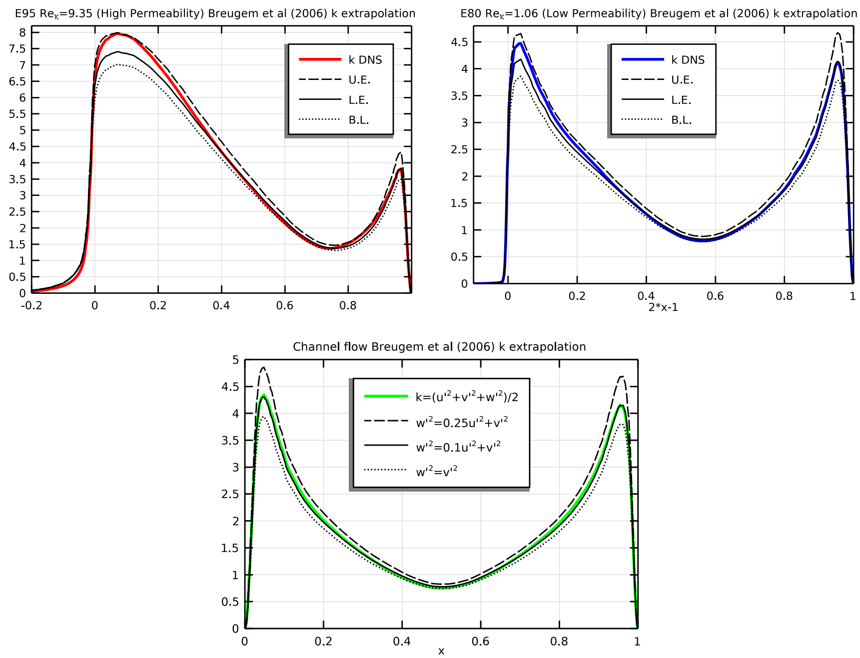


*Figure 2: Extrapolation of turbulence kinetic energy using only streamwise and wall-normal rms-velocities. The DNS results of* Ref. 1 *are used. U.E. is the "Upper Estimate", L.E. is the Lower Estimate and B.L. is the "Bottom Limit", as defined in the lower figure and in the main text.*

Flows in such "adjacent" configurations are referred to as flows with a "permeable wall" in Ref. 1 and Ref. 2. Indeed, a porous interface resembles a partial slip permeable wall since homogenization of the flow inside the porous region occurs within a few porous length scales ~$\kappa^{1/2}$, where $\kappa$ is the permeability. This is typically a quite small distance. Both Ref. 1 and Ref. 2 fitted the clear flow velocity near the porous interface by a log-law applicable to rough walls. However, as indicated in Ref. 1, the effect of permeability is intrinsically different from the effect of roughness and is of primary importance. To characterize it, the permeability Reynolds number was introduced:

$$\mathrm{Re}_\kappa = \frac{u_\tau^{\mathrm{p}}\sqrt{\kappa}}{\nu} \tag{1}$$

where $u_\tau{}^{\mathrm{p}}$ is the friction velocity on the clear (fluid) side of the porous interface.

The experimental results in Ref. 2 contain only measurements of streamwise and wall-normal rms-velocities, $u'$ and $v'$, respectively. The DNS data of Ref. 1 is analyzed to find a reasonable way to extrapolate the spanwise rms-velocity, $w'$, via the other two components, and to reconstruct the turbulence kinetic energy, $k = (u'^2 + v'^2 + w'^2)/2$, consequently.

Figure 2 shows $k$ for three different cases: E95 (high permeability, $\mathrm{Re}_\kappa = 9.35$), E80 (low permeability, $\mathrm{Re}_\kappa = 1.06$), and clear channel flow (zero permeability) as presented in Ref. 1. Near its lowest point $k$ reaches the "bottom limit" (B.L.) based on $w'^2 = v'^2$ (it can be expected that unrestricted spanwise fluctuations are not smaller than restricted wall-normal fluctuations). Mostly, $k$ is situated between a proposed "upper estimate" (U.E.) based on $w'^2 = 0.25u'^2 + v'^2$ and a proposed "lower estimate" (L.E.) based on $w'^2 = 0.1u'^2 + v'^2$. The lower estimate represents $k$ very well not only for clear-channel flow with no-slip walls but also in more than half of the clear part of the channel (on the solid wall side) for both cases with a "permeable wall". Near the porous interface, $k$ approaches the upper estimate, the closer the higher the permeability Reynolds number $\mathrm{Re}_\kappa$. Note that these rules of thumb were formulated based on a limited number of cases and for the specific value $\mathrm{Re} = 5500$. Nevertheless, these tentatively proposed upper and lower estimates are employed to extrapolate experimental values of $k$.

## *Results and Discussion*

Table 1 contains parameters from DNS case E95 in Ref. 1, and from two experimental cases from series #06 in Ref. 2 (a bit corrected in Ref. 3). The Reynolds number is based on the average (bulk) velocity $U_\mathrm{b}$ and height of the clear channel $H$, $\mathrm{Re} = U_\mathrm{b}H/\nu$, while the Darcy number is defined as $\mathrm{Da} = \kappa/H^2$, where $\kappa$ is the permeability of the porous medium. $\varepsilon_\mathrm{p}$ is the porosity, and $c_\mathrm{F}$ is the Forchheimer parameter.

The numerical results of the COMSOL Multiphysics simulations are summarized in Table 2 for clear-channel flow and in Table 3 for the setup with adjacent clear and porous regions. The configurations are referred to as "clear" and "adjacent" (including the model plots). The following quantities are presented:

- Friction coefficients at the solid wall and at the fluid side of the porous interface:

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

- Friction velocities at the solid wall and at the fluid side of the porous interface:

$$u_\tau^s = \sqrt{\frac{\tau_w^s}{\rho}}, \qquad u_\tau^p = \sqrt{\frac{\tau_w^p}{\rho}}$$

- Friction Reynolds number based on $u_\tau^p$ and $H$ (clear channel height):

$$\mathrm{Re}_\tau = \frac{u_\tau^p H}{\nu}$$

where $\tau_w^s$ and $\tau_w^p$ are the shear stress at the solid wall and at the porous interface, respectively:

$$\tau_w^s = -\mu\frac{\partial u}{\partial y}, \qquad \tau_w^p = (\mu + \mu_T)\frac{\partial u}{\partial y}$$

TABLE 1: PARAMETERS OF THE DNS AND EXPERIMENTAL CASES.

| | Re | H | $\varepsilon_p$ | $c_F$ | Da |
|---|---|---|---|---|---|
| E95 | 5500 | 0.03 m | 0.95 | 0.14 | $1.9\cdot10^{-4}$ |
| #06_5400 | 5400 | 0.029 m | 0.8 | 0.095 | $1.04\cdot10^{-4}$ |
| #06_9500 | 9500 | 0.029 m | 0.8 | 0.095 | $1.04\cdot10^{-4}$ |

TABLE 2: CLEAR CHANNEL RESULTS.

| | $C_f^s\times10^3$ | $C_f^p\times10^3$ | $\mathrm{Re}_\tau$ | $u_\tau^p/u_\tau^s$ |
|---|---|---|---|---|
| E95 | 8.28 | 8.25 | 353 | 0.998 |
| #06_5400 | 8.32 | 8.28 | 348 | 0.998 |
| #06_9500 | 7.26 | 7.23 | 571 | 0.998 |

TABLE 3: ADJACENT CHANNEL RESULTS: SIMULATION VERSUS DNS/EXPERIMENT (BOLD).

| | $C_f^s\times10^3$ | $C_f^p\times10^3$ | $\mathrm{Re}_\tau$ | $u_\tau^p/u_\tau^s$ |
|---|---|---|---|---|
| E95 | 11.03 (**10.9**) | 31.2 (**30.4**) | 687 (**678**) | 1.68 (**1.67**) |
| #06_5400 | 10.59 (**10.65**) | 26.5 (**25.6**) | 622 (**611**) | 1.58 (**1.55**) |
| #06_9500 | 9.21 (**9.79**) | 22.3 (**26**) | 1003 (**1084**) | 1.56 (**1.63**) |

For the case of clear channel flow at $\mathrm{Re} = 5500$, the low Reynolds number $k$-$\varepsilon$ model gives friction coefficients equal to $8.28\cdot10^{-3}$ and $8.25\cdot10^{-3}$ at the solid wall and at the porous interface, respectively (the difference is due to a finer resolution at the latter), while the DNS prediction is $8.18\cdot10^{-3}$, so the accuracy of the model is near 1%. $u_\tau^p/u_\tau^s$ deviates from 1 by 0.2%, which indicates that for clear-channel flow further refinement is not needed.

[Figure 3](#) demonstrates the E95 case for which the computed profiles of velocity, turbulence kinetic energy, and shear stress are highly asymmetric. The velocity has a maximum magnitude close to the solid wall, a quarter of the clear-channel height from it. The minimum of the turbulence kinetic energy and the zero of the shear stress are situated approximately at the same location, while the maxima of $k$ and the shear stress near the porous interface are much larger than the maxima near the solid wall. Note that even at large $\mathrm{Re}_\kappa = 9.35$, the flow gradients do not penetrate deep into the porous region; the velocity and $k$ quickly attain small limiting values in the porous matrix while the shear stress effectively becomes zero.

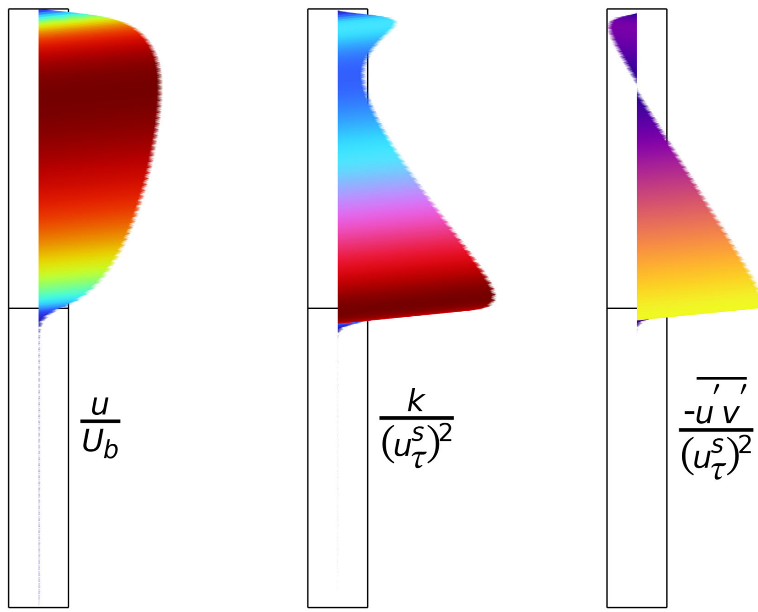


*Figure 3: Profiles of velocity, turbulence kinetic energy, and shear stress for the E95 case. The upper domain is a clear flow domain, whereas the lower domain is a porous domain. Note the quick damping of all the variables inside the latter.*

For E95, Figure 4 demonstrates very good correspondence of $u/U_b$, $k/(u_\tau^s)^2$, and $-u'v'/(u_\tau^s)^2$ obtained in COMSOL Multiphysics simulations with the DNS results from Ref. 1. In the clear-channel flow, the maximum of $k/(u_\tau^s)^2$ is underestimated by approximately 15%, and in the adjacent configuration the same underestimation near the solid wall is present, which seems to be characteristic for the low Reynolds number $k$-ε turbulence model. At the same time, the maximum of $k/(u_\tau^s)^2$ near the porous interface is surprisingly correct, although both the maximum and minimum positions are somewhat shifted. The behavior of the quantities close to the porous interface is not captured flawlessly but even the absence of larger deviations is remarkable since we do not treat the porous interface specifically.

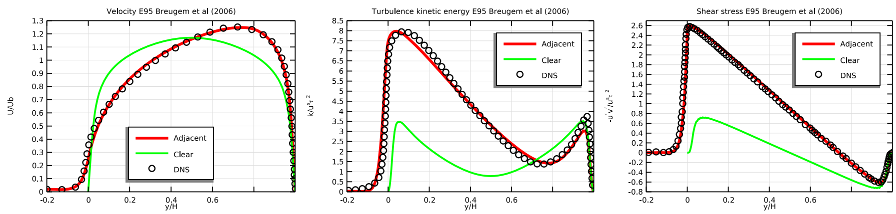


*Figure 4: Velocity, turbulence kinetic energy and shear stress for E95.*

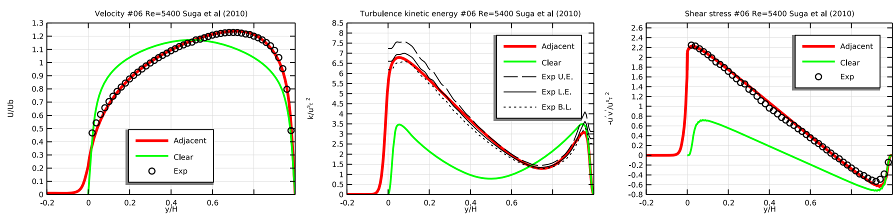


*Figure 5: Velocity, turbulence kinetic energy and shear stress for #06* Re = *5400.U.E. is the "Upper Estimate", L.E. is the Lower Estimate and B.L. is the "Bottom Limit", as defined in the main text.*

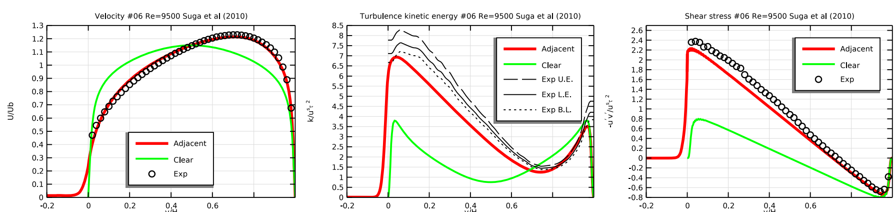


*Figure 6: Velocity, turbulence kinetic energy and shear stress for #06* Re = *9500. U.E. is the "Upper Estimate", L.E. is the Lower Estimate and B.L. is the "Bottom Limit", as defined in the main text.*

A comparison of simulations and experimental results from Ref. 2 for the case #06, $\mathrm{Re} = 5400$ and $\mathrm{Re}_K = 6.23$, is shown in Figure 5. The velocity profile and normalized shear stress are very well captured. The maximum of $k/(u_\tau^s)^2$ is slightly shifted toward the "permeable wall" and seems to be underestimated by approximately 10%, while both the minimum value and its position are captured correctly.

Figure 6 presents a simulations versus experiment comparison for the case #06, $\mathrm{Re} = 9500$ and $\mathrm{Re}_\kappa = 11.05$. The qualitative change from the “clear” profiles to the “adjacent” profiles is still very well captured, but now the computed velocity profile is a bit off the experimental one, and the profiles of turbulence kinetic energy and shear stress even more so: the maximum of $-u'v'/(u_\tau^{s})^2$ is 8% lower and the maximum of $k/(u_\tau^{s})^2$ is 13% lower, while its minimum is shifted and under predicted, too. Indeed, it is easy to see that the increase in $\mathrm{Re}$ from 5400 to 9500 does not result in a sufficiently significant shift in the computed profiles.These facts seem to be explained by the absence of additional treatment of the porous interface. Allegedly, a quantitative change of the influence of roughness occurs when increasing the velocity.

Finally, Figure 7 and Figure 8 graphically show the results presented in Table 2 and Table 3, the solution order 1-3 corresponds to Table 1. The remarkable accuracy of the calculations and excellent qualitative predictions, even in the last case when quantitative deviations become apparent, are visually emphasized.

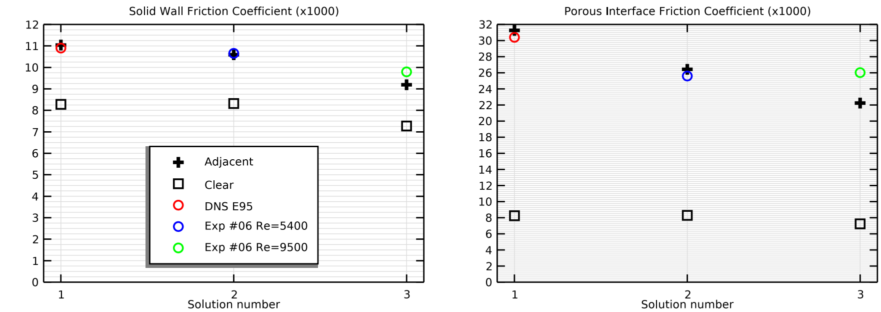


*Figure 7:* Friction coefficients (×1000*) at the solid wall (left) and at the porous interface (right).*

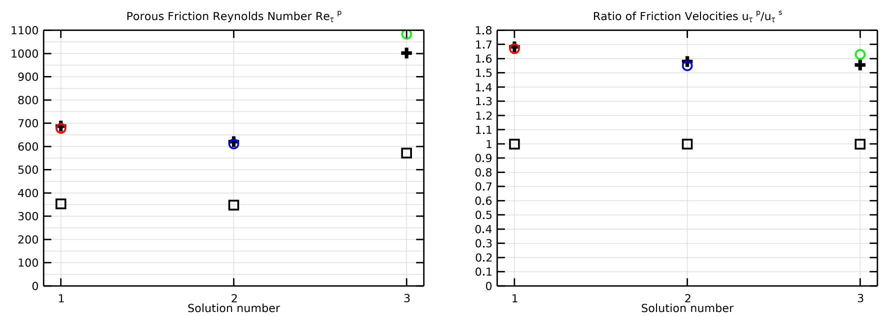


*Figure 8: Turbulence friction Reynolds number based on $u_\tau{}^p$ and H (left panel). Ratio of friction velocities at the solid wall and the porous interface $u_\tau{}^p/u_\tau{}^s$ (right panel).*

## *References*

1. W.P. Breugem, B.J. Boersma, and R.E. Uittenbogaard, “The Influence of Wall Permeability on Turbulent Channel Flow,” *J. Fluid Mech.*, vol. 562, pp. 35–72, 2006.

2. K. Suga, Y. Matsumura, Y.Ashitaka, S.Tominaga, and M. Kaneda, “Effects of Wall Permeability on Turbulence,” *Int. J. Heat Fluid Flow*, vol. 31, pp. 974–984, 2010.

3. Y. Kuwata, K. Suga, and Y. Sakurai, “Development and Application of a Multi-Scale k-ε Model for Turbulent Porous Medium Flows,” *Int. J. Heat Fluid Flow*, vol. 49, pp. 135–150, 2014.

**Application Library path:** `CFD_Module/Verification_Examples/turbulent_free_porous`

## 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, Low Re k-ε (spf)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

### GLOBAL DEFINITIONS

*Parameters 1*

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

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

3 Click **Load from File**.

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

### DEFINITIONS

*View 1*

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **View 1**.

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

**3** Select the **Show geometry labels** check box.

## GEOMETRY 1

Create a geometry that is short in the x (streamwise) direction because effectively fully developed flow will be modeled.

### *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 `L`.

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

**5** Right-click **Rectangle 1 (r1)** and choose **Duplicate**.

### *Rectangle 2 (r2)*

**1** In the **Model Builder** window, click **Rectangle 2 (r2)**.

**2** In the **Settings** window for **Rectangle**, locate the **Position** section.

**3** In the **y** text field, type `-H`.

### *Point 1 (pt1)*

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

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

**3** In the **x** text field, type `CL`.

**4** Right-click **Point 1 (pt1)** and choose **Duplicate**.

### *Point 2 (pt2)*

**1** In the **Model Builder** window, click **Point 2 (pt2)**.

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

**3** In the **y** text field, type `H`.

**4** Click **Build All Objects**.

## DEFINITIONS

### *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** Select Boundary 9 only.

### *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 `CL`.

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

5 Select the **Snap to closest boundary** check box.

### *Friction at the solid wall*

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 `Friction at the solid wall` in the **Label** text field.

3 In the **Variable name** text field, type `friction_s`.

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

### *Domain Point Probe 1*

In the **Model Builder** window, right-click **Domain Point Probe 1** and choose **Duplicate**.

### *Domain Point Probe 2*

1 In the **Model Builder** window, click **Domain Point Probe 2**.

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

3 In row **Coordinates**, set **y** to `0`.

### *Friction at the porous interface*

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

2 In the **Settings** window for **Point Probe Expression**, type `Friction at the porous interface` in the **Label** text field.

3 In the **Variable name** text field, type `friction_p`.

4 Locate the **Expression** section. In the **Expression** text field, type `side(2,(spf.nuT+spf.nu)*pprint(uy))`.

Here `pprint()` is employed to guarantee that near the porous interface the smoothing is made using only data from the clear flow domain.

### *Variables 1*

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

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

3 Click **Load from File**.

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

### ADD MATERIAL

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

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

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

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

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

6 Click the **Show More Options** button in the **Model Builder** toolbar.

7 In the **Show More Options** dialog box, click **Select All**.

8 Click **OK**.

### TURBULENT FLOW, LOW RE K-ε (SPF)

1 In the **Settings** window for **Turbulent Flow, Low Re k-ε**, locate the **Physical Model** section.

2 Select the **Enable porous media domains** check box.

3 Locate the **Turbulence** section. From the **Wall treatment** list, choose **Low Re**.

*Fluid Properties 1*

1 In the **Model Builder** window, under **Component 1 (comp1)>Turbulent Flow, Low Re k-ε (spf)** click **Fluid Properties 1**.

2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.

3 From the $\rho$ list, choose **User defined**. In the associated text field, type `rho_i`.

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

5 Locate the **Distance Equation** section. From the $l_{\mathrm{ref}}$ list, choose **Manual**.

6 In the text field, type `H`.

The low Reynolds number k-\u03b5 model admits a solution with vanishing turbulence levels. To ensure that the solver does not converge to this spurious solution during the first iterations, a variable initial velocity profile is chosen.

*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*cos(2*pi*y/H) | x |

*Porous Medium 1*

1 In the **Physics** toolbar, click **Domains** and choose **Porous Medium**.

2 Select Domain 1 only.

*Fluid 1*

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

2 In the **Settings** window for **Fluid**, 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`.

*Porous Matrix 1*

1 In the **Model Builder** window, click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the $\varepsilon_p$ list, choose **User defined**. In the associated text field, type `epsilon_p_i`.

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

5 In the $c_F$ text field, type `cF_i`.

*Periodic Flow Condition 1*

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

2 Select Boundaries 1, 3, 8, and 9 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 **Mass flow** text field, type `rho_i*Ub*H*1[m]`.

*Pressure Point Constraint 1*

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

2 Select Point 4 only.

Delete the porous domain from the Interface selection to start with the computation of the clear channel flow.

3 In the **Model Builder** window, click **Turbulent Flow, Low Re k-ε (spf)**.

4 In the **Domain Selection** window, clear Domain **1**.

## MESH 1

Use **Mapped** to guarantee the mesh's consistency with the **Periodic Flow Condition**.

### *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 **Entire geometry**.

### *Distribution 1*

1. Right-click **Mapped 1** and choose **Distribution**.
2. Select Boundaries 4–7 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `8`.

### *Distribution 2*

1. In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2. Select Boundaries 3 and 9 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `Nmesh/2`.

### *Distribution 3*

1. Right-click **Mapped 1** and choose **Distribution**.
2. Select Boundaries 1 and 8 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `Nmesh/2`.

### *Boundary Layers 1*

1. In the **Mesh** toolbar, click **Boundary Layers**.
2. In the **Settings** window for **Boundary Layers**, click to expand the **Transition** section.
3. Clear the **Smooth transition to interior mesh** check box.

### *Boundary Layer Properties*

1. In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties**.
2. Select Boundaries 2, 5, and 7 only.
3. In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.
4. In the **Number of layers** text field, type `8`.

5 In the **Stretching factor** text field, type `1.2`.

6 From the **Thickness specification** list, choose **All layers**.

7 In the **Total thickness** text field, type `3*H/Nmesh`.

*Boundary Layers 1*

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

*Boundary Layer Properties*

1 In the **Model Builder** window, expand the **Boundary Layers 2** node, then click **Boundary Layer Properties**.

2 Select Boundaries 4 and 6 only.

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

4 In the **Number of layers** text field, type `12`.

5 From the **Thickness specification** list, choose **All layers**.

6 In the **Total thickness** text field, type `4*H/Nmesh`.

Note that the boundary layers mesh is finer near the porous interface where the turbulence level and the friction velocity are higher.

7 Click **Build All**.

## STUDY 1

*Parametric Sweep*

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| ReH (Bulk Reynolds number in clear channel) | 5500 5400 9500 | 1 |

5 Click **Add**.

6 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| H (Clear channel height) | H_E95 H_06 H_06 | m |

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

*Parametric Solutions 1 (sol3)*

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

2 Right-click **Parametric Solutions 1 (sol3)** and choose **Solution>Copy**.

*Clear channel flow*

1 In the **Model Builder** window, under **Study 1>Solver Configurations** click **Parametric Solutions 1 - Copy 1 (sol7)**.

2 In the **Settings** window for **Solution**, type `Clear channel flow` in the **Label** text field.

## RESULTS

Create a centerline for plotting the results to avoid distortion effects near the source and destination boundaries.

*CL clear*

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

2 In the **Settings** window for **Cut Line 2D**, type `CL clear` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Clear channel flow (sol7)**.

4 Locate the **Line Data** section. In row **Point 1**, set **x** to `CL`.

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

6 In row **Point 1**, set **y** to `-H`.

7 In row **Point 2**, set **y** to `H`.

## TURBULENT FLOW, LOW RE K-ε (SPF)

Include the porous domain back to the Interface selection.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, Low Re k-ε (spf)**.

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

*Global Equations 1 (ODE1)*

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

**Global equations** is needed to compute the pressure gradient, P_driving, which imposes the prescribed average (bulk) velocity in the *clear part* of the channel, as was done in the DNS and the experiments.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

**3** In the table, enter the following settings:

| Name | f(u,ut,utt,t) (1) | Initial value (u_0) (1) | Initial value (u_t0) (1/s) | Description |
|---|---|---|---|---|
| `P_driving` | `aveop1(u)-Ub` | `0` | `0` | |

**4** Locate the **Units** section. Click **Define Dependent Variable Unit**.

**5** In the **Dependent variable quantity** table, enter the following settings:

| Dependent variable quantity | Unit |
|---|---|
| Custom unit | `Pa` |

**6** Click **Define Source Term Unit**.

**7** In the **Source term quantity** table, enter the following settings:

| Source term quantity | Unit |
|---|---|
| Custom unit | `m/s` |

The **Mass Flow** option of the **Periodic Flow Condition** should be replaced by the **Pressure difference** with the pressure difference equal to P_driving.

### *Periodic Flow Condition 1*

**1** In the **Model Builder** window, click **Periodic Flow Condition 1**.

**2** In the **Settings** window for **Periodic Flow Condition**, locate the **Flow Condition** section.

**3** From the **Flow condition** list, choose **Pressure difference**.

**4** In the $\Delta p$ text field, type `P_driving`.

## STUDY 1

### *Parametric Sweep*

**1** In the **Model Builder** window, under **Study 1** click **Parametric Sweep**.

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

**3** Click **Load from File**.

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

### *Solution 1 (sol1)*

**1** In the **Model Builder** window, right-click **Solver Configurations** and choose **Show Default Solver**.

**2** In the **Home** toolbar, click **Compute**.

*Adjacent free-porous flow*

In the **Settings** window for **Solution**, type `Adjacent free-porous flow` in the **Label** text field.

## RESULTS

*CL clear*

In the **Model Builder** window, under **Results>Datasets** right-click **CL clear** and choose **Duplicate**.

*CL adjacent free-porous flow*

**1** In the **Model Builder** window, under **Results>Datasets** click **CL clear 1**.

**2** In the **Settings** window for **Cut Line 2D**, type `CL adjacent free-porous flow` in the **Label** text field.

**3** Locate the **Data** section. From the **Dataset** list, choose **Study 1/Adjacent free-porous flow (sol3)**.

Import the DNS and the experimental results.

*U_E95*

**1** In the **Results** toolbar, click **Table**.

**2** In the **Model Builder** window, under **Results** click **Tables**.

**3** Analogously, create 7 more instances of **Table**.

**4** In the **Settings** window for **Table**, type `U_E95` in the **Label** text field.

**5** Locate the **Data** section. Click **Import**.

**6** Browse to the model's Application Libraries folder and double-click the file `turbulent_free_porous_U_E95.txt`.

*k_E95*

**1** In the **Model Builder** window, under **Results>Tables** click **Table 3**.

**2** In the **Settings** window for **Table**, type `k_E95` in the **Label** text field.

**3** Locate the **Data** section. Click **Import**.

**4** Browse to the model's Application Libraries folder and double-click the file `turbulent_free_porous_k_E95.txt`.

*uv_E95*

**1** In the **Model Builder** window, under **Results>Tables** click **Table 4**.

2 In the **Settings** window for **Table**, type `uv_E95` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `turbulent_free_porous_uv_E95.txt`.

*U_06_5400*

1 In the **Model Builder** window, under **Results>Tables** click **Table 5**.

2 In the **Settings** window for **Table**, type `U_06_5400` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `turbulent_free_porous_U_#06_5400.txt`.

*uv_06_5400*

1 In the **Model Builder** window, under **Results>Tables** click **Table 6**.

2 In the **Settings** window for **Table**, type `uv_06_5400` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `turbulent_free_porous_uv_#06_5400.txt`.

*U_06_9500*

1 In the **Model Builder** window, under **Results>Tables** click **Table 7**.

2 In the **Settings** window for **Table**, type `U_06_9500` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `turbulent_free_porous_U_#06_9500.txt`.

*uv_06_9500*

1 In the **Model Builder** window, under **Results>Tables** click **Table 8**.

2 In the **Settings** window for **Table**, type `uv_06_9500` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `turbulent_free_porous_uv_#06_9500.txt`.

*u-rms and v-rms #06*

1 In the **Model Builder** window, under **Results>Tables** click **Table 9**.

2 In the **Settings** window for **Table**, type `u-rms and v-rms #06` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `turbulent_free_porous_rms_#06.txt`.

### DEFINITIONS

Build Interpolation functions for the u-rms and v-rms velocities taken from the experimental #06-series. The aim is to reconstruct the w-rms velocity by extrapolation as explained in the main text.

*u_rms #06 5400*

1 In the **Home** toolbar, click f(x) **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, type `u_rms  #06 5400` in the **Label** text field.

3 Locate the **Definition** section. From the **Data source** list, choose **Result table**.

4 From the **Table from** list, choose **u-rms and v-rms #06**.

5 Find the **Functions** subsection. In the table, enter the following settings:

| Function name | Position in file |
|---|---|
| u1 | 1 |

6 Right-click **u_rms #06 5400** and choose **Duplicate**.

*v_rms #06 5400*

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **u_rms #06 5400.1 (u2)**.

2 In the **Settings** window for **Interpolation**, type `v_rms  #06 5400` in the **Label** text field.

3 Locate the **Definition** section. Find the **Functions** subsection. In the table, enter the following settings:

| Function name | Position in file |
|---|---|
| v1 | 2 |

4 Right-click **v_rms #06 5400** and choose **Duplicate**.

*u_rms #06 9500*

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **v_rms #06 5400.1 (v3)**.

2 In the **Settings** window for **Interpolation**, type `u_rms  #06 9500` in the **Label** text field.

3 Locate the **Definition** section. Find the **Functions** subsection. In the table, enter the following settings:

| Function name | Position in file |
|---|---|
| u2 | 3 |

4 Right-click **u_rms #06 9500** and choose **Duplicate**.

*v_rms #06 9500*

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **u_rms #06 9500.1 (u4)**.

2 In the **Settings** window for **Interpolation**, type `v_rms #06 9500` in the **Label** text field.

3 Locate the **Definition** section. Find the **Functions** subsection. In the table, enter the following settings:

| Function name | Position in file |
|---|---|
| v2 | 4 |

## RESULTS

*Grid Exp*

1 In the **Results** toolbar, click **More Datasets** and choose **Grid>Grid 1D**.

2 In the **Settings** window for **Grid 1D**, type `Grid Exp` in the **Label** text field.

3 Locate the **Data** section. From the **Source** list, choose **Function**.

4 From the **Function** list, choose **All**.

*U_E95*

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

2 In the **Settings** window for **1D Plot Group**, type `U_E95` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **CL adjacent free-porous flow**.

4 From the **Parameter selection (ReH, H, epsilon_p_i, cF_i, Da_i)** list, choose **From list**.

5 In the **Parameter values (ReH,H (m),epsilon_p_i,cF_i,Da_i)** list, select **1: ReH=5500, H=0.03 m, epsilon_p_i=0.95, cF_i=0.14, Da_i=1.9E-4**.

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

7 In the **Title** text area, type `Velocity E95 Breugem et al (2006)`.

8 Locate the **Plot Settings** section.

9 Select the **x-axis label** check box. In the associated text field, type `y/H`.

10 Select the **y-axis label** check box. In the associated text field, type `U/Ub`.

11 Locate the **Axis** section. Select the **Manual axis limits** check box.

12 In the **x minimum** text field, type `-0.2`.

13 In the **y minimum** text field, type `0`.

14 In the **y maximum** text field, type `1.3`.

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

*Adjacent*

1 Right-click **U_E95** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Adjacent` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the **Expression** text field, type `u/Ub`.

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

5 In the **Expression** text field, type `y_H`.

6 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Red**.

7 From the **Width** list, choose **3**.

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

9 Find the **Include** subsection. Clear the **Solution** check box.

10 Select the **Label** check box.

11 Right-click **Adjacent** and choose **Duplicate**.

*Clear*

1 In the **Model Builder** window, under **Results>U_E95** click **Adjacent 1**.

2 In the **Settings** window for **Line Graph**, type `Clear` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **CL clear**.

4 From the **Parameter selection (ReH, H)** list, choose **From list**.

5 In the **Parameter values (ReH,H (m))** list, select **1: ReH=5500, H=0.03 m**.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

7 From the **Width** list, choose **2**.

*DNS*

1 In the **Model Builder** window, right-click **U_E95** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, type `DNS` in the **Label** text field.

3 Locate the **Data** section. From the **Table** list, choose **U_E95**.

4. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
5. From the **Color** list, choose **Black**.
6. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
7. Click to expand the **Legends** section. Select the **Show legends** check box.
8. Find the **Include** subsection. Clear the **Headers** check box.
9. Select the **Label** check box.

### *U_E95*

Right-click **U_E95** and choose **Duplicate**.

### *k_E95*

1. In the **Model Builder** window, under **Results** click **U_E95.1**.
2. In the **Settings** window for **1D Plot Group**, type k_E95 in the **Label** text field.
3. Locate the **Title** section. In the **Title** text area, type Turbulence kinetic energy E95 Breugem et al (2006).
4. Locate the **Plot Settings** section. In the **y-axis label** text field, type k/u<sup>s</sup> <sub>\tau</sub> <sup>2</sup>.
5. Locate the **Axis** section. In the **y maximum** text field, type 8.5.
6. Locate the **Legend** section. From the **Position** list, choose **Upper right**.

### *Adjacent*

1. In the **Model Builder** window, expand the **k_E95** node, then click **Adjacent**.
2. In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type k/friction_s.

### *Clear*

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

### *DNS*

1. In the **Model Builder** window, click **DNS**.
2. In the **Settings** window for **Table Graph**, locate the **Data** section.
3. From the **Table** list, choose **k_E95**.
4. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Positioning** list, choose **Interpolated**.

5 In the **Number** text field, type 100.

### U_E95

In the **Model Builder** window, under **Results** right-click **U_E95** and choose **Duplicate**.

### uv_E95

1 In the **Model Builder** window, click **U_E95.1**.

2 In the **Settings** window for **1D Plot Group**, locate the **Title** section.

3 In the **Title** text area, type Shear stress E95 Breugem et al (2006).

4 In the **Label** text field, type uv_E95.

5 Locate the **Plot Settings** section. In the **y-axis label** text field, type -u<sup>\prime</sup>v<sup>\prime</sup>/u<sup>s</sup><sub>\tau</sub> <sup>2</sup>.

6 Locate the **Axis** section. In the **y minimum** text field, type -0.8.

7 In the **y maximum** text field, type 2.7.

8 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

### Adjacent

1 In the **Model Builder** window, expand the **uv_E95** node, then click **Adjacent**.

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

3 In the **Expression** text field, type shear_stress/friction_s.

### Clear

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

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

3 In the **Expression** text field, type shear_stress/friction_s.

### DNS

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

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

3 From the **Table** list, choose **uv_E95**.

### U_E95

In the **Model Builder** window, under **Results** right-click **U_E95** and choose **Duplicate**.

### U_#06_Re=5400

1 In the **Model Builder** window, under **Results** click **U_E95.1**.

2 In the **Settings** window for **1D Plot Group**, type U_#06_Re=5400 in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (ReH,H (m),epsilon_p_i,cF_i,Da_i)** list, select **2: ReH=5400, H=0.029 m, epsilon_p_i=0.8, cF_i=0.095, Da_i=1.04E-4**.

4 Locate the **Title** section. In the **Title** text area, type `Velocity #06 Re=5400 Suga et al (2010)`.

*Clear*

1 In the **Model Builder** window, expand the **U_#06_Re=5400** node, then click **Clear**.

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

3 In the **Parameter values (ReH,H (m))** list, select **2: ReH=5400, H=0.029 m**.

*Exp*

1 In the **Model Builder** window, under **Results>U_#06_Re=5400** click **DNS**.

2 In the **Settings** window for **Table Graph**, type `Exp` in the **Label** text field.

3 Locate the **Data** section. From the **Table** list, choose **U_06_5400**.

*k_E95*

In the **Model Builder** window, under **Results** right-click **k_E95** and choose **Duplicate**.

*k_#06_Re=5400*

1 In the **Model Builder** window, under **Results** click **k_E95.1**.

2 In the **Settings** window for **1D Plot Group**, type `k_#06_Re=5400` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (ReH,H (m),epsilon_p_i,cF_i,Da_i)** list, select **2: ReH=5400, H=0.029 m, epsilon_p_i=0.8, cF_i=0.095, Da_i=1.04E-4**.

4 Locate the **Title** section. In the **Title** text area, type `Turbulence kinetic energy #06 Re=5400 Suga et al (2010)`.

*Clear*

1 In the **Model Builder** window, expand the **k_#06_Re=5400** node, then click **Clear**.

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

3 In the **Parameter values (ReH,H (m))** list, select **2: ReH=5400, H=0.029 m**.

4 Right-click **Clear** and choose **Duplicate**.

*DNS*

In the **Model Builder** window, right-click **DNS** and choose **Delete**.

In the experiments #06 the spanwise rms-velocity was not measured. Here its "upper estimate" and "lower estimate", as well as "bottom limit", are reconstructed (extrapolated) using the streamwise and the wall-normal rms-velocities according to the main text.

*Exp Upper Estimate*

1 In the **Model Builder** window, under **Results>k_#06_Re=5400** click **Clear 1**.

2 In the **Settings** window for **Line Graph**, type `Exp Upper Estimate` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Grid Exp**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `((1+cUu)*u1(x)^2+(1+cUv)*v1(x)^2)/2`.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `x`.

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

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

8 From the **Width** list, choose **1**.

9 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

| Legends |
| --- |
| `Exp U.E.` |

11 Right-click **Results>k_#06_Re=5400>Exp Upper Estimate** and choose **Duplicate**.

*Exp Lower Estimate*

1 In the **Model Builder** window, under **Results>k_#06_Re=5400** click **Exp Upper Estimate 1**.

2 In the **Settings** window for **Line Graph**, type `Exp Lower Estimate` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the **Expression** text field, type `((1+cLu)*u1(x)^2+(1+cLv)*v1(x)^2)/2`.

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 |
| --- |
| `Exp L.E.` |

6 Right-click **Exp Lower Estimate** and choose **Duplicate**.

*Exp Bottom Limit*

1 In the **Model Builder** window, under **Results>k_#06_Re=5400** click **Exp Lower Estimate 1**.

2 In the **Settings** window for **Line Graph**, type `Exp Bottom Limit` in the **Label** text field.

**3** Locate the **y-Axis Data** section. In the **Expression** text field, type (u1(x)^2+2*v1(x)^2)/2.

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

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

| Legends |
| --- |
| Exp B.L. |

*uv_E95*

In the **Model Builder** window, under **Results** right-click **uv_E95** and choose **Duplicate**.

*uv_#06_Re=5400*

**1** In the **Model Builder** window, under **Results** click **uv_E95.1**.

**2** In the **Settings** window for **1D Plot Group**, type uv_#06_Re=5400 in the **Label** text field.

**3** Locate the **Data** section. In the **Parameter values (ReH,H (m),epsilon_p_i,cF_i,Da_i)** list, select **2: ReH=5400, H=0.029 m, epsilon_p_i=0.8, cF_i=0.095, Da_i=1.04E-4**.

**4** Locate the **Title** section. In the **Title** text area, type Shear stress #06 Re=5400 Suga et al (2010).

*Clear*

**1** In the **Model Builder** window, expand the **uv_#06_Re=5400** node, then click **Clear**.

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

**3** In the **Parameter values (ReH,H (m))** list, select **2: ReH=5400, H=0.029 m**.

*Exp*

**1** In the **Model Builder** window, under **Results>uv_#06_Re=5400** click **DNS**.

**2** In the **Settings** window for **Table Graph**, type Exp in the **Label** text field.

**3** Locate the **Data** section. From the **Table** list, choose **uv_06_5400**.

*U_#06_Re=5400*

In the **Model Builder** window, under **Results** right-click **U_#06_Re=5400** and choose **Duplicate**.

*U_#06_Re=9500*

**1** In the **Model Builder** window, under **Results** click **U_#06_Re=5400.1**.

**2** In the **Settings** window for **1D Plot Group**, type U_#06_Re=9500 in the **Label** text field.

**3** Locate the **Data** section. In the **Parameter values (ReH,H (m),epsilon_p_i,cF_i,Da_i)** list, select **3: ReH=9500, H=0.029 m, epsilon_p_i=0.8, cF_i=0.095, Da_i=1.04E-4**.

4 Locate the **Title** section. In the **Title** text area, type `Velocity #06 Re=9500 Suga et al (2010)`.

*Clear*

1 In the **Model Builder** window, expand the **U_#06_Re=9500** node, then click **Clear**.

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

3 In the **Parameter values (ReH,H (m))** list, select **3: ReH=9500, H=0.029 m**.

*Exp*

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

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

3 From the **Table** list, choose **U_06_9500**.

*k_#06_Re=5400*

In the **Model Builder** window, under **Results** right-click **k_#06_Re=5400** and choose **Duplicate**.

*k_#06_Re=9500*

1 In the **Model Builder** window, under **Results** click **k_#06_Re=5400.1**.

2 In the **Settings** window for **1D Plot Group**, type `k_#06_Re=9500` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (ReH,H (m),epsilon_p_i,cF_i,Da_i)** list, select **3: ReH=9500, H=0.029 m, epsilon_p_i=0.8, cF_i=0.095, Da_i=1.04E-4**.

4 Locate the **Title** section. In the **Title** text area, type `Turbulence kinetic energy #06 Re=9500 Suga et al (2010)`.

*Clear*

1 In the **Model Builder** window, expand the **k_#06_Re=9500** node, then click **Clear**.

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

3 In the **Parameter values (ReH,H (m))** list, select **3: ReH=9500, H=0.029 m**.

*Exp Upper Estimate*

1 In the **Model Builder** window, click **Exp Upper Estimate**.

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

3 In the **Expression** text field, type `((1+cUu)*u2(x)^2+(1+cUv)*v2(x)^2)/2`.

*Exp Lower Estimate*

1 In the **Model Builder** window, click **Exp Lower Estimate**.

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

3 In the **Expression** text field, type `((1+cLu)*u2(x)^2+(1+cLv)*v2(x)^2)/2`.

### *Exp Bottom Limit*

1 In the **Model Builder** window, click **Exp Bottom Limit**.

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

3 In the **Expression** text field, type `(u2(x)^2+2*v2(x)^2)/2`.

### *uv_#06_Re=5400*

In the **Model Builder** window, under **Results** right-click **uv_#06_Re=5400** and choose **Duplicate**.

### *uv_#06_Re=9500*

1 In the **Model Builder** window, under **Results** click **uv_#06_Re=5400.1**.

2 In the **Settings** window for **1D Plot Group**, type `uv_#06_Re=9500` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (ReH,H (m),epsilon_p_i,cF_i,Da_i)** list, select **3: ReH=9500, H=0.029 m, epsilon_p_i=0.8, cF_i=0.095, Da_i=1.04E-4**.

4 Locate the **Title** section. In the **Title** text area, type `Shear stress #06 Re=9500 Suga et al (2010)`.

### *Clear*

1 In the **Model Builder** window, expand the **uv_#06_Re=9500** node, then click **Clear**.

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

3 In the **Parameter values (ReH,H (m))** list, select **3: ReH=9500, H=0.029 m**.

### *Exp*

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

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

3 From the **Table** list, choose **uv_06_9500**.

### *Solid Wall Friction Coefficient (x1000)*

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

2 In the **Settings** window for **1D Plot Group**, type `Solid Wall Friction Coefficient (x1000)` 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 `Solid Wall Friction Coefficient (x1000)`.

5 Locate the **Axis** section. Select the **Manual axis limits** check box.

6 In the **x minimum** text field, type `0.9`.

7 In the **x maximum** text field, type `manual_x_max`.

8 In the **y minimum** text field, type `0`.

9 In the **y maximum** text field, type `12`.

10 Locate the **Grid** section. Select the **Manual spacing** check box.

11 In the **y spacing** text field, type `0.25`.

*Adjacent*

1 Right-click **Solid Wall Friction Coefficient (x1000)** and choose **Global**.

2 In the **Settings** window for **Global**, type `Adjacent` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Adjacent free-porous flow (sol3)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| Cs1000 | 1 | Friction coefficient (x1000) at the solid wall |

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

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

7 Find the **Line markers** subsection. From the **Marker** list, choose **Plus sign**.

8 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

9 Clear the **Solution** check box.

10 Select the **Label** check box.

11 Right-click **Adjacent** and choose **Duplicate**.

*Clear*

1 In the **Model Builder** window, under **Results>Solid Wall Friction Coefficient (x1000)** click **Adjacent 1**.

2 In the **Settings** window for **Global**, type `Clear` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Clear channel flow (sol7)**.

4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

*Adjacent*

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

*DNS E95*

1 In the **Model Builder** window, under **Results>Solid Wall Friction Coefficient (x1000)** click **Adjacent 1**.

2 In the **Settings** window for **Global**, type `DNS E95` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (ReH, H, epsilon_p_i, cF_i, Da_i)** list, choose **From list**.

4 In the **Parameter values (ReH,H (m),epsilon_p_i,cF_i,Da_i)** list, select **1: ReH=5500, H=0.03 m, epsilon_p_i=0.95, cF_i=0.14, Da_i=1.9E-4**.

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

| Expression | Unit | Description |
| --- | --- | --- |
| 10.9 | 1 | |

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

8 Right-click **DNS E95** and choose **Duplicate**.

*Exp #06 Re=5400*

1 In the **Model Builder** window, under **Results>Solid Wall Friction Coefficient (x1000)** click **DNS E95.1**.

2 In the **Settings** window for **Global**, type `Exp #06 Re=5400` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (ReH,H (m),epsilon_p_i,cF_i,Da_i)** list, select **2: ReH=5400, H=0.029 m, epsilon_p_i=0.8, cF_i=0.095, Da_i=1.04E-4**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

| Expression | Unit | Description |
| --- | --- | --- |
| C_help_var*(6.23/1.55)^2 | 1 | |

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

6 Right-click **Exp #06 Re=5400** and choose **Duplicate**.

*Exp #06 Re=9500*

1 In the **Model Builder** window, under **Results>Solid Wall Friction Coefficient (x1000)** click **Exp #06 Re=5400.1**.

2 In the **Settings** window for **Global**, type `Exp #06 Re=9500` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (ReH,H (m),epsilon_p_i,cF_i,Da_i)** list, select **3: ReH=9500, H=0.029 m, epsilon_p_i=0.8, cF_i=0.095, Da_i=1.04E-4**.

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

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| `C_help_var*(11.05/1.63)^2` | 1 | |

### *Annotation 1*

1 In the **Model Builder** window, right-click **Solid Wall Friction Coefficient (x1000)** and choose **Annotation**.

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

3 In the **Text** text field, type `Sol 1: E95 \\ Re=5500 \\ $\epsilon_p=0.95$ \\ Da=1.9e-4 \\ $c_F=0.14$ \\ ~ \\ Sol 2: #06 Re=5400 \\ Sol 3: #06 Re=9500 \\ $\epsilon_p$=0.8 \\ Da=1.04e-4 \\ $c_F=0.095$`.

4 Select the **LaTeX markup** check box.

5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Upper right**.

6 Locate the **Position** section. In the **x** text field, type `3.9`.

7 In the **y** text field, type `5`.

8 Locate the **Coloring and Style** section. Clear the **Show point** check box.

9 Select the **Show frame** check box.

10 From the **Background color** list, choose **White**.

### *Solid Wall Friction Coefficient (x1000)*

Right-click **Solid Wall Friction Coefficient (x1000)** and choose **Duplicate**.

### *Porous Interface Friction Coefficient (x1000)*

1 In the **Model Builder** window, under **Results** click **Solid Wall Friction Coefficient (x1000) 1**.

2 In the **Settings** window for **1D Plot Group**, type `Porous Interface Friction Coefficient (x1000)` in the **Label** text field.

3 Click to expand the **Title** section. In the **Title** text area, type `Porous Interface Friction Coefficient (x1000)`.

4 Locate the **Axis** section. In the **y maximum** text field, type `32`.

### *Adjacent*

1 In the **Model Builder** window, expand the **Porous Interface Friction Coefficient (x1000)** node, then click **Adjacent**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| Cp1000 | 1 | Friction coefficient (x1000) at the porous interface |

*Clear*

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

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| Cp1000 | 1 | Friction coefficient (x1000) at the porous interface |

*DNS E95*

1 In the **Model Builder** window, click **DNS E95**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| 30.4 | 1 | |

*Exp #06 Re=5400*

1 In the **Model Builder** window, click **Exp #06 Re=5400**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| C_help_var*(6.23)^2 | 1 | |

*Exp #06 Re=9500*

1 In the **Model Builder** window, click **Exp #06 Re=9500**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| C_help_var*(11.05)^2 | 1 | |

### *Annotation 1*

1. In the **Model Builder** window, click **Annotation 1**.
2. In the **Settings** window for **Annotation**, locate the **Position** section.
3. In the **y** text field, type 14.

### *Porous Interface Friction Coefficient (x1000)*

In the **Model Builder** window, right-click **Porous Interface Friction Coefficient (x1000)** and choose **Duplicate**.

### *Porous Friction Reynolds Number*

1. In the **Model Builder** window, under **Results** click **Porous Interface Friction Coefficient (x1000) 1**.
2. In the **Settings** window for **1D Plot Group**, type Porous Friction Reynolds Number in the **Label** text field.
3. Click to expand the **Title** section. In the **Title** text area, type Porous Friction Reynolds Number Re<sub>\tau</sub> <sup>p</sup>.
4. Locate the **Axis** section. In the **y maximum** text field, type 1100.
5. Locate the **Grid** section. In the **y spacing** text field, type 50.

### *Adjacent*

1. In the **Model Builder** window, expand the **Porous Friction Reynolds Number** node, then click **Adjacent**.
2. In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| Retp | 1 | |

### *Clear*

1. In the **Model Builder** window, click **Clear**.
2. In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| Retp | 1 | |

### *DNS E95*

1. In the **Model Builder** window, click **DNS E95**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| 678 | 1 | |

*Exp #06 Re=5400*

1 In the **Model Builder** window, click **Exp #06 Re=5400**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| 6.23/sqrt(Da_i) | | |

*Exp #06 Re=9500*

1 In the **Model Builder** window, click **Exp #06 Re=9500**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| 11.05/sqrt(Da_i) | | |

*Annotation 1*

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

2 In the **Settings** window for **Annotation**, locate the **Position** section.

3 In the **y** text field, type `600`.

*Porous Friction Reynolds Number*

In the **Model Builder** window, right-click **Porous Friction Reynolds Number** and choose **Duplicate**.

*Ratio of Friction Velocities*

1 In the **Model Builder** window, under **Results** click **Porous Friction Reynolds Number 1**.

2 In the **Settings** window for **1D Plot Group**, type `Ratio of Friction Velocities` in the **Label** text field.

3 Click to expand the **Title** section. In the **Title** text area, type `Ratio of Friction Velocities u<sub>\tau</sub> <sup>p</sup>/u<sub>\tau</sub> <sup>s</sup>`.

4 Locate the **Axis** section. In the **y maximum** text field, type `1.8`.

5 Locate the **Grid** section. In the **y spacing** text field, type `0.1`.

*Adjacent*

1 In the **Model Builder** window, expand the **Ratio of Friction Velocities** node, then click **Adjacent**.

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_p/utau_s | 1 | |

*Clear*

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

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_p/utau_s | 1 | |

*DNS E95*

1 In the **Model Builder** window, click **DNS E95**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| sqrt(30.4/10.9) | 1 | |

*Exp #06 Re=5400*

1 In the **Model Builder** window, click **Exp #06 Re=5400**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| 1.55 | 1 | |

*Exp #06 Re=9500*

1 In the **Model Builder** window, click **Exp #06 Re=9500**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| 1.63 | 1 | |

### *Annotation 1*

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

2 In the **Settings** window for **Annotation**, locate the **Position** section.

3 In the **y** text field, type `1`.

### *View 2D 2*

In the **Model Builder** window, under **Results** right-click **Views** and choose **View 2D**.

### *Axis*

1 In the **Model Builder** window, expand the **View 2D 2** node, then click **Axis**.

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

3 In the **x minimum** text field, type `-0.002`.

4 In the **x maximum** text field, type `0.08`.

5 In the **y minimum** text field, type `-0.032`.

6 In the **y maximum** text field, type `0.032`.

### *Arrow Surfaces E95*

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

2 In the **Settings** window for **2D Plot Group**, type `Arrow Surfaces E95` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Adjacent free-porous flow (sol3)**.

4 From the **Parameter value (ReH,H (m),epsilon_p_i,cF_i,Da_i)** list, choose **1: ReH=5500, H=0.03 m, epsilon_p_i=0.95, cF_i=0.14, Da_i=1.9E-4**.

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

6 In the **Title** text area, type `Velocity, turbulence kinetic energy and shear stress`.

7 Locate the **Color Legend** section. Clear the **Show legends** check box.

8 Click to expand the **Plot Array** section. Select the **Enable** check box.

9 In the **Relative padding** text field, type `4`.

*Velocity*

1 Right-click **Arrow Surfaces E95** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, type `Velocity` in the **Label** text field.

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

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

5 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `1`.

6 Find the **y grid points** subsection. In the **Points** text field, type `500`.

7 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.

8 Select the **Scale factor** check box. In the associated text field, type `1e-2`.

*Color Expression 1*

1 Right-click **Velocity** and choose **Color Expression**.

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

3 From the **Color data** list, choose **Arrow length**.

*Velocity*

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

*Turbulence kinetic energy*

1 In the **Model Builder** window, under **Results>Arrow Surfaces E95** click **Velocity 1**.

2 In the **Settings** window for **Arrow Surface**, type `Turbulence kinetic energy` in the **Label** text field.

3 Locate the **Expression** section. In the **x-component** text field, type `k/utau_s^2`.

4 Locate the **Coloring and Style** section. In the **Scale factor** text field, type `2e-3`.

*Color Expression 1*

1 In the **Model Builder** window, expand the **Turbulence kinetic energy** node, then click **Color Expression 1**.

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Wave>Disco** in the tree.

5 Click **OK**.

*Turbulence kinetic energy*

In the **Model Builder** window, right-click **Turbulence kinetic energy** and choose **Duplicate**.

*Shear stress*

1 In the **Model Builder** window, under **Results>Arrow Surfaces E95** click **Turbulence kinetic energy 1**.
2 In the **Settings** window for **Arrow Surface**, type `Shear stress` in the **Label** text field.
3 Locate the **Expression** section. In the **x-component** text field, type `shear_stress/utau_s^2`.
4 Locate the **Coloring and Style** section. In the **Scale factor** text field, type `6e-3`.

*Color Expression 1*

1 In the **Model Builder** window, expand the **Shear stress** node, then click **Color Expression 1**.
2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.
3 Click **Change Color Table**.
4 In the **Color Table** dialog box, select **Thermal>Plasma** in the tree.
5 Click **OK**.

*Annotation 1*

1 In the **Model Builder** window, right-click **Arrow Surfaces E95** and choose **Annotation**.
2 In the **Settings** window for **Annotation**, locate the **Annotation** section.
3 Select the **LaTeX markup** check box.
4 In the **Text** text field, type `\Huge$\frac{u}{U_b}$`.
5 Locate the **Position** section. In the **x** text field, type `L`.
6 In the **y** text field, type `-2*L`.
7 Locate the **Coloring and Style** section. Clear the **Show point** check box.
8 From the **Anchor point** list, choose **Middle left**.
9 Click to expand the **Plot Array** section. Clear the **Belongs to array** check box.
10 Right-click **Annotation 1** and choose **Duplicate**.

*Annotation 2*

1 In the **Model Builder** window, click **Annotation 2**.
2 In the **Settings** window for **Annotation**, locate the **Annotation** section.
3 In the **Text** text field, type `\Huge$\frac{k}{{(u_\tau^s})^2}$`.
4 Locate the **Position** section. In the **x** text field, type `6*L`.
5 Right-click **Annotation 2** and choose **Duplicate**.

*Annotation 3*

1 In the **Model Builder** window, click **Annotation 3**.

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

3 In the **Text** text field, type `\Huge$\frac{-\overline{u^\prime\,v^\prime}}{{(u_\tau^s})^2}$`.

4 Locate the **Position** section. In the **x** text field, type `11*L`.

*Clear channel*

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Clear channel` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Clear channel flow (sol7)**.

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

| Expression | Unit | Description |
|---|---|---|
| `Cs1000` | 1 | `Friction coefficient (x1000) at the solid wall` |
| `Cp1000` | 1 | `Friction coefficient (x1000) at the porous interface` |
| `utau_p*H/nu_i` | 1 | `Reynolds number based on the porous interface friction velocity` |
| `utau_p/utau_s` | 1 | `Ratio of friction velocities (porous interface over solid wall)` |

5 Click **Evaluate**.

6 Right-click **Clear channel** and choose **Duplicate**.

*Adjacent*

1 In the **Model Builder** window, under **Results>Derived Values** click **Clear channel 1**.

2 In the **Settings** window for **Global Evaluation**, type `Adjacent` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Adjacent free-porous flow (sol3)**.

4 Click **Evaluate**.

## GLOBAL DEFINITIONS

*Parameters 1*

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

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

3 In the table, enter the following settings:

| Name | Expression | Value | Description |
| --- | --- | --- | --- |
| manual_x_max | 3.95 | 3.95 | Manual x max limit |