Model created in COMSOL Multiphysics 6.4

Air Filter

Introduction

This example demonstrates simulation of a turbulent flow in a geometry with an air filter. The porous material of the filter induces an abrupt pressure drop and a drastic increase of the turbulence level inside the filter. The turbulent flow field in both the clear-flow and porous domains is computed using the Turbulent Flow, k-ω interface.

Model Definition

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 three-dimensional model of a turbulent duct flow with an air filter represented by a porous domain is considered.

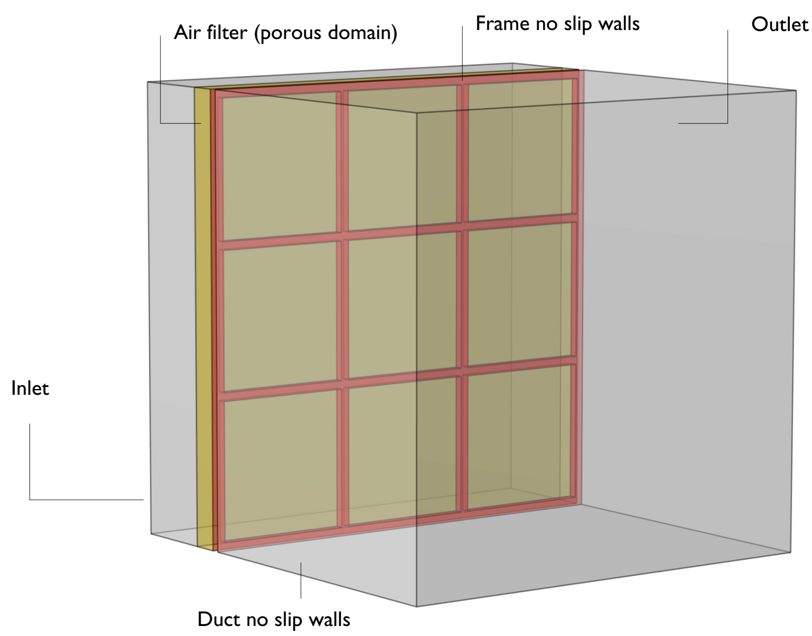

Figure 1: Duct geometry with the porous domain of the air filter (light green) and no-slip walls of the holding frame (red).

Figure 1 shows the model geometry, domain properties, and boundary conditions while Figure 2 and Figure 3 illustrate the mesh. At the Inlet, a Normal inflow velocity condition is applied. At the Outlet, a Static pressure condition is used. The air filter is modeled as a highly porous domain with cylindrical pores 0.1 mm in diameter occupying 90% of the material. The support of the air filter is represented by a frame with no-slip walls. The duct walls are also no slip. The upstream domain, relative to the air filter and its supporting frame, can be taken quite short since the porous domain homogenizes the flow very efficiently. The downstream domain must be long enough to ensure that the frame-generated wake has almost homogenized. Correspondingly, the frame walls and frame plane use a Finer mesh, while the air filter domain, the region upstream of it with thickness D_filter, and the wake region with thickness L_wake = 10 D_frame, use a Fine mesh. The remaining geometry uses the Coarser mesh level. The duct Reynolds number based on the average (bulk) velocity and the height of the duct, $\mathrm{Re} = U_\mathrm{b}H/\nu$, is equal to 5311, which ensures that the flow is turbulent in the clear part of the duct. The k-ω turbulence model is chosen because it consistently models situations with a lot of small no-slip walls (the frame surface). 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*.

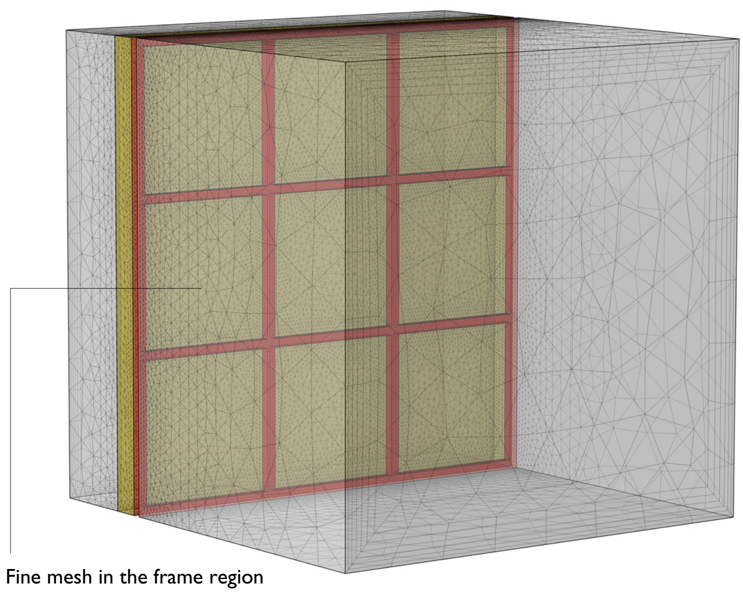

Figure 2: Model mesh. The focus is on the fine meshing of the frame region.

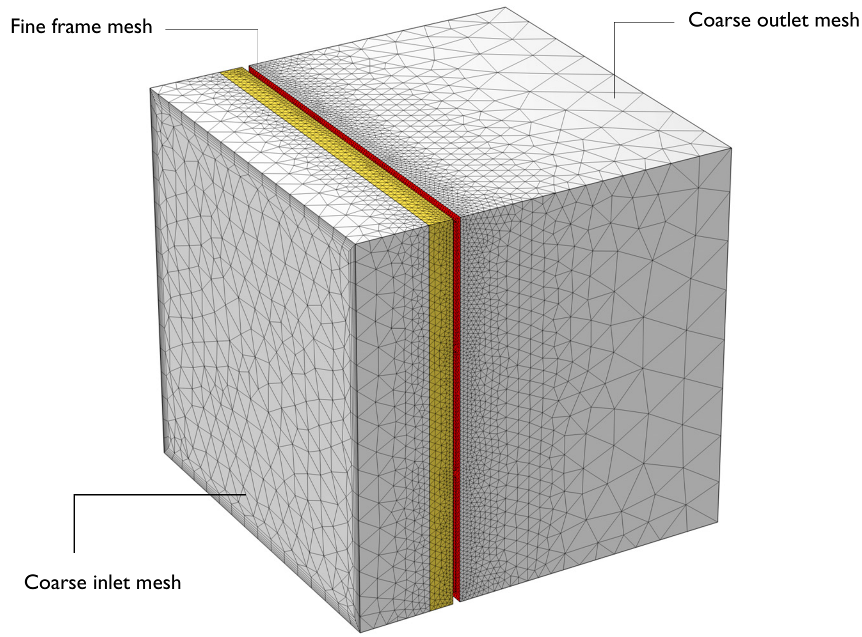

Figure 3: Model mesh.

Results and Discussion

The presence of the porous domain in the turbulent flow does not negatively influence the simulation stability of this particular model. On the contrary, the solution time is approximately twice as small compared to the case with a completely clear flow.

Figure 4 illustrates how the frame influences the velocity field at the inflow surface of the air filter. The velocity streamlines are shown and the abrupt pressure drop across the porous domain is plotted. Figure 5 is an analogous figure illustrating the drastic increase of the turbulence kinetic energy inside the air filter and its fast attenuation both upstream and downstream.

Figure 6 shows the velocity magnitude on the vertical and horizontal slices. The wakes produced by the frame and boundary layers on the duct walls are visible. Figure 7 uses the same slices and plots $\log10(k)$ to emphasize that the pore-scale turbulence production boosts the turbulence kinetic energy up to several orders of magnitude inside the porous domain, while k falls very rapidly outside.

Figure 8 demonstrates how the velocity is first influenced by the porous domain and the frame, and is subsequently homogenized through the wake region. Figure 9 shows the same slices with the turbulence kinetic energy. Notice that unlike the velocity, whose evolution is apparent in the wake region, the turbulence kinetic energy has a pronounced peak in the air filter region and is insignificant outside of it.

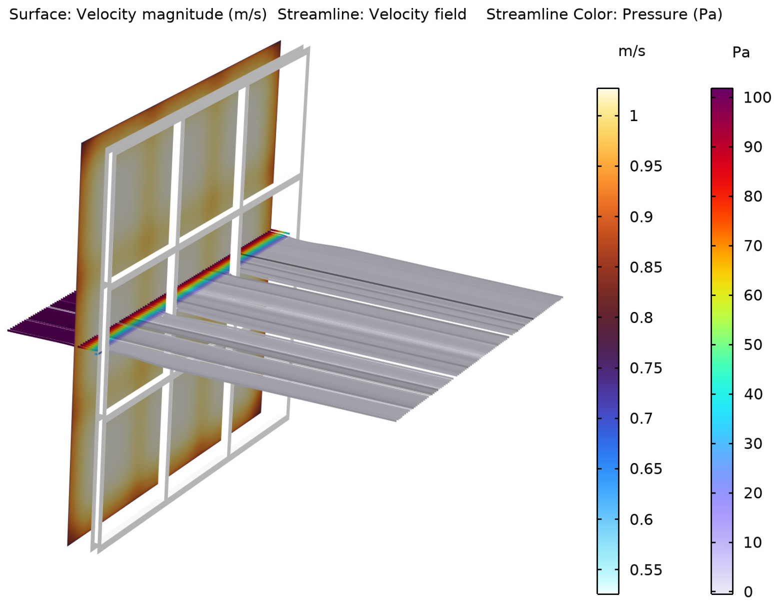

Figure 4: Velocity magnitude at the inflow surface of the air filter; velocity streamlines are colored with the pressure magnitude.

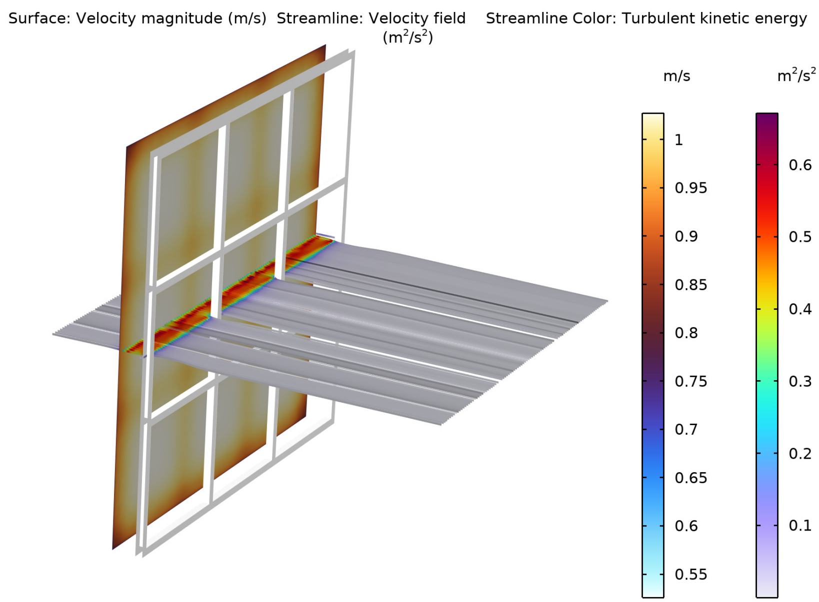

Figure 5: Velocity magnitude at the inflow surface of the air filter; velocity streamlines are colored with the turbulence kinetic energy magnitude.

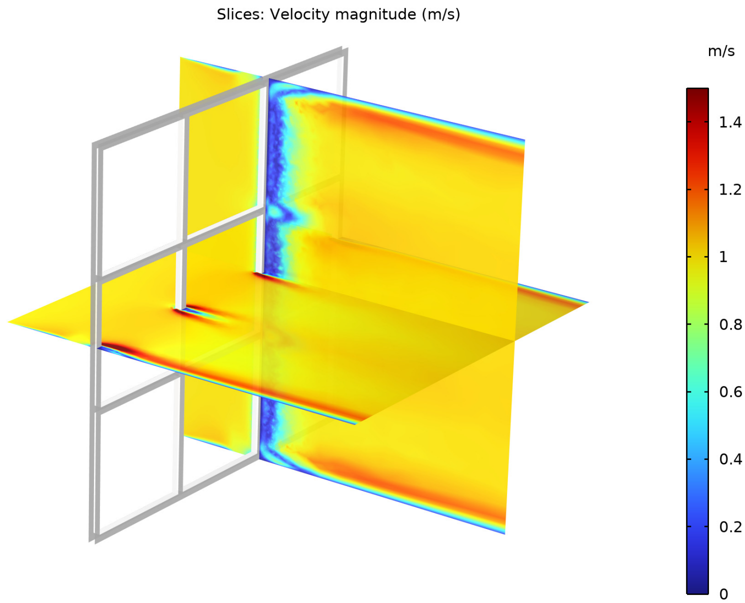

Figure 6: Horizontal and vertical slices of the velocity magnitude showing frame-produced wakes. Boundary layers on the duct walls are visible, too.

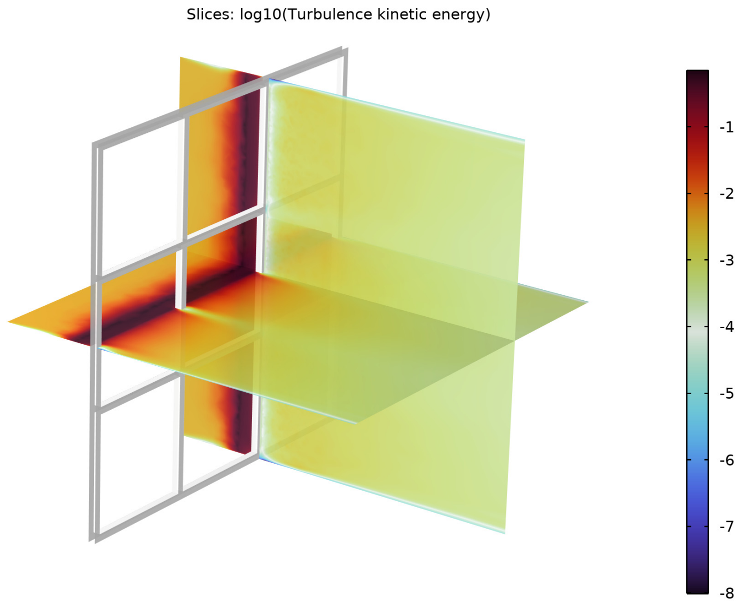

Figure 7: Horizontal and vertical slices of the decimal logarithm of the turbulence kinetic energy, illustrating that the particular porous domain produces k values several orders of magnitude higher than the values upstream and downstream.

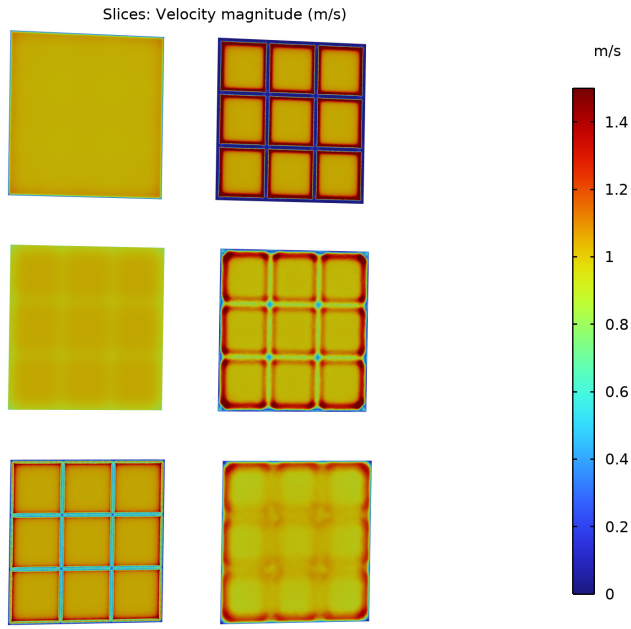

Figure 8: Slices of the velocity magnitude. Left column, top to bottom: D_filter *distance upstream of the air-filter plane, inflow surface of the air filter, outflow surface of the air filter. It is visible that the frame placed on the outflow surface makes a clear imprint on the inflow surface of the air filter. Right column, top to bottom: frame plane, plane at* L_wake/2 *distance from the frame, plane at* L_wake *distance from the frame. Here, homogenization in the frame-generated wake is apparent.*

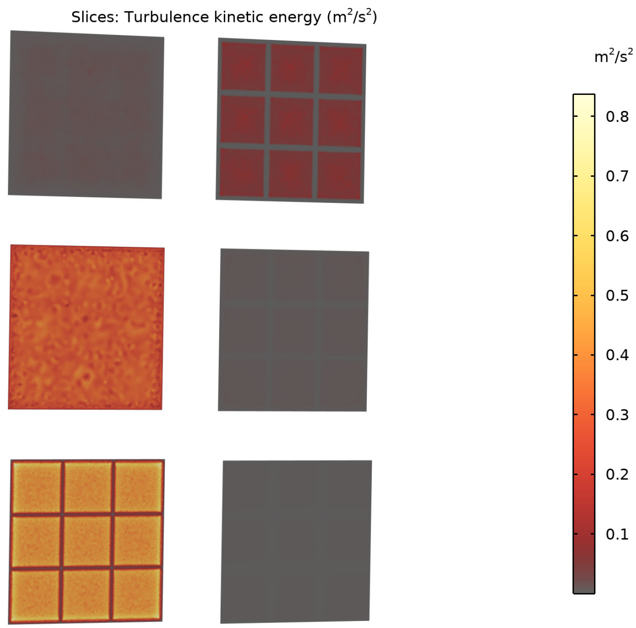

Figure 9: Slices of the turbulence kinetic energy. Left column, top to bottom: D_filter *distance upstream of the air-filter plane, inflow surface of the air filter, outflow surface of the air filter. Right column, top to bottom: frame plane, plane at* L_wake/2 *distance from the frame, plane at* L_wake *distance from the frame. Apparently, the turbulence kinetic energy is high on the surfaces of the air filter only and rapidly falls in both directions.*

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

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Fluid Flow** > **Single-Phase Flow** > **Turbulent Flow** > **Turbulent Flow, k-ω** (spf).

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Start with loading predefined parameters and expressions to be used in the model.

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 `air_filter_parameters_1.txt`.

Parameters 2

1 In the **Home** toolbar, click Pi **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 `air_filter_parameters_2.txt`.

GEOMETRY 1

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

Work Plane 1 (wp1) > Plane Geometry

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

Work Plane 1 (wp1) > If 1 (if1)

1 In the **Work Plane** toolbar, click **Programming** and choose **If + End If**.

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

3 In the **Condition** text field, type `cell_type==1`.

Work Plane 1 (wp1) > Rectangle 1 (r1)

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

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

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

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

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

6 In the **xw** text field, type `Th_h+cL_h/2`.

7 In the **yw** text field, type `Th_v+cL_v/2`.

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

9 In the **New Cumulative Selection** dialog, type `Inner Grill` in the **Name** text field.

10 Click **OK**.

Add alternative option for openings in the air filter frame (ellipses).

Work Plane 1 (wp1) > Else If 1 (elseif1)

1 Right-click **Plane Geometry** and choose **Programming** > **Else If**.

2 In the **Settings** window for **Else If**, locate the **Else If** section.

3 In the **Condition** text field, type `cell_type==2`.

Work Plane 1 (wp1) > Ellipse 1 (e1)

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

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

3 In the **a-semiaxis** text field, type `cL_h/2`.

4 In the **b-semiaxis** text field, type `cL_v/2`.

5 Locate the **Position** section. In the **xw** text field, type `Th_h+cL_h/2`.

6 In the **yw** text field, type `Th_v+cL_v/2`.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Inner Grill**.

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

Work Plane 1 (wp1) > Array 1 (arr1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 In the **Settings** window for **Array**, locate the **Input** section.

3 From the **Input objects** list, choose **Inner Grill**.

4 Locate the **Size** section. In the **xw size** text field, type `N_h`.

5 In the **yw size** text field, type `N_v`.

6 Locate the **Displacement** section. In the **xw** text field, type `cL_h+Th_h`.

7 In the **yw** text field, type `cL_v+Th_v`.

Work Plane 1 (wp1) > Rectangle 2 (r2)

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

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

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

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

Work Plane 1 (wp1) > Difference 1 (dif1)

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

2 Select the object **r2** only.

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

4 From the **Objects to subtract** list, choose **Inner Grill**.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Inner Grill**.

Extrude 1 (ext1)

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

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

3 In the table, enter the following settings:

Distances (m)
`D_frame`

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

5 From the **Show in physics** list, choose **Boundary selection**.

6 From the **Color** list, choose **None** or — if you are running the cross-platform desktop — **Custom**. On the cross-platform desktop, click the **Color** button.

7 Click **Define custom colors**.

8 Set the RGB values to 255, 0, and 0, respectively.

9 Click **Add to custom colors**.

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

11 Find the **Cumulative selection** subsection. Click **New**.

12 In the **New Cumulative Selection** dialog, type `Frame` in the **Name** text field.

13 Click **OK**.

Block 1 (blk1)

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

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

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

4 In the **Depth** text field, type `L_h`.

5 In the **Height** text field, type `L_v`.

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

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

8 From the **Color** list, choose **Color 4**.

9 Find the **Cumulative selection** subsection. Click **New**.

10 In the **New Cumulative Selection** dialog, type `Filter` in the **Name** text field.

11 Click **OK**.

DEFINITIONS

View 3

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

2 Right-click **Definitions** and choose **View**.

3 In the **Settings** window for **View**, click to expand the **Transparency** section.

4 Select the **Transparency** checkbox.

GEOMETRY 1

Block 1 (blk1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Block 1 (blk1)**.

2 In the **Settings** window for **Block**, click **Build Selected**.

Block 2 (blk2)

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

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

3 In the **Width** text field, type `L_pre+L_post`.

4 In the **Depth** text field, type `L_h`.

5 In the **Height** text field, type `L_v`.

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

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

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

9 Click **OK**.

Block 3 (blk3)

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

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

3 In the **x** text field, type `-2*D_filter`.

4 Locate the **Size and Shape** section. In the **Width** text field, type `D_filter`.

5 In the **Depth** text field, type `L_h`.

6 In the **Height** text field, type `L_v`.

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

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

9 Click **OK**.

Block 4 (blk4)

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

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

3 In the **Depth** text field, type `L_h`.

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

5 In the **Width** text field, type `D_frame+L_wake`.

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

7 In the **New Cumulative Selection** dialog, type `MD_post` in the **Name** text field.

8 Click **OK**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **blk2** and **blk4** only.

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

4 From the **Objects to subtract** list, choose **Frame**.

Add **Mesh Control Domains** to impose coarser mesh outside the regions of air filter, frame and wake.

Mesh Control Domains 1 (mcd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Domains**.

2 On the object **fin**, select Domains 1 and 5 only.

3 In the **Geometry** toolbar, click **Build All**.

4 Adjust camera to reproduce **Figure 1**.

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

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

Complement air properties with porosity and permeability to be used in the porous domain of air filter.

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, click to expand the **Material Properties** section.

2 In the **Material properties** tree, select **Basic Properties** > **Permeability**.

3 Click **Add to Material**.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	kappa_i	m^2	Basic

5 Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties** > **Porosity**.

6 Click **Add to Material**.

7 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Porosity	epsilon	`epsilon_p_i`	1	Basic

Define selections to be used below.

DEFINITIONS

Filter front

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

2 In the **Settings** window for **Intersection**, type `Filter front` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

5 In the **Add** dialog, in the **Selections to intersect** list, choose **Filter** and **MD_pre**.

6 Click **OK**.

Filter-frame

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

2 In the **Settings** window for **Intersection**, type `Filter-frame` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

5 In the **Add** dialog, in the **Selections to intersect** list, choose **Filter** and **MD_post**.

6 Click **OK**.

TURBULENT FLOW, K-ω (SPF)

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

2 In the **Settings** window for **Turbulent Flow, k-ω**, locate the **Physical Model** section.

3 Select the **Enable porous media domains** checkbox.

Porous Medium 1

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

2 In the **Settings** window for **Porous Medium**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Filter**.

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 In the c_F text field, type `cF_i`.

Inlet 1

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

2 Select Boundary 1 only.

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

4 In the U_0 text field, type `U_in`.

Outlet 1

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

2 Select Boundary 62 only.

Mesh should be fine in the porous filter, frame region and in the wake region, which is fulfilled by varying settings in **Size** features of the **Mesh**node.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Coarser**.

4 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** click **Size 1**.

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

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

4 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Frame**.

5 Click **Build Selected**.

Size 2

1 Right-click **Component 1 (comp1)** > **Mesh 1** > **Size 1** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Filter-frame**.

4 Click **Build Selected**.

Size 3

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

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

3 From the **Selection** list, choose **Manual**.

4 Click **Clear Selection**.

5 Select Boundaries 6, 71, and 72 only.

6 Locate the **Element Size** section. From the **Predefined** list, choose **Fine**.

7 Click **Build Selected**.

Size 4

1 Right-click **Size 3** and choose **Duplicate**.

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

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

4 Select Domains 2–4 only.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

6 Reproduce **Figure 2**.

DEFINITIONS

View 4

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

MESH 1

Adjust camera to reproduce **Figure 3**.

STUDY 1

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

RESULTS

Cut Line is needed as a seed of points for the **Streamline**.

Cut Line 3D 1

1 In the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Line Data** section.

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

4 In row **Point 2**, set **y** to `L_h`.

5 In row **Point 1**, set **z** to `Cut_v_cell`.

6 In row **Point 2**, set **z** to `Cut_v_cell`.

Velocity and pressure

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

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

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

4 In the **Label** text field, type `Velocity and pressure`.

5 Locate the **Color Legend** section. Select the **Show units** checkbox.

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

Surface 1

1 Right-click **Velocity and pressure** and choose **Surface**.

2 In the **Settings** window for **Surface**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

Selection 1

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

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

3 From the **Selection** list, choose **Frame**.

Material Appearance 1

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

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Plastic (shiny)**.

Surface 2

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

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

3 From the **Color table** list, choose **ThermalWave**.

Selection 1

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

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

3 From the **Selection** list, choose **Filter front**.

Streamline 1

1 In the **Model Builder** window, right-click **Velocity and pressure** and choose **Streamline**.
2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
3 From the **Positioning** list, choose **Starting-point controlled**.
4 In the **Points** text field, type `100`.
5 From the **Along curve or surface** list, choose **Cut Line 3D 1**.
6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Ribbon**.
7 Select the **Width scale factor** checkbox. In the associated text field, type `2E-4`.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.
2 In the **Settings** window for **Color Expression**, locate the **Expression** section.
3 In the **Expression** text field, type `p`.
4 Click to expand the **Title** section. From the **Title type** list, choose **Automatic**.
5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.
6 Adjust camera to reproduce **Figure 4**.

Velocity and turbulence kinetic energy

1 In the **Model Builder** window, right-click **Velocity and pressure** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `Velocity and turbulence kinetic energy` in the **Label** text field.
3 In the **Model Builder** window, expand the **Velocity and turbulence kinetic energy** node.

Color Expression 1

1 In the **Model Builder** window, expand the **Results** > **Velocity and turbulence kinetic energy** > **Streamline 1** node, then click **Color Expression 1**.
2 In the **Settings** window for **Color Expression**, locate the **Expression** section.
3 In the **Expression** text field, type `k`.
4 Reproduce **Figure 5**.

Velocity slices xy and xz

1 In the **Model Builder** window, right-click **Velocity and turbulence kinetic energy** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `Velocity slices xy and xz` in the **Label** text field.

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

4 In the **Velocity slices xy and xz** toolbar, click **Plot**.

View 3D 6

1 In the **Model Builder** window, expand the **Results** > **Views** node, then click **View 3D 6**.

2 In the **Settings** window for **View 3D**, click to expand the **Transparency** section.

3 Select the **Transparency** checkbox.

4 In the **Transparency** text field, type `0.1`.

Velocity slices xy and xz

In the **Model Builder** window, expand the **Results** > **Velocity slices xy and xz** node.

Streamline 1, Surface 2

1 In the **Model Builder** window, under **Results** > **Velocity slices xy and xz**, Ctrl-click to select **Surface 2** and **Streamline 1**.

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

Slice 1

1 In the **Model Builder** window, right-click **Velocity slices xy and xz** and choose **Slice**.

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

3 From the **Plane** list, choose **xy-planes**.

4 From the **Entry method** list, choose **Coordinates**.

5 In the **z-coordinates** text field, type `Cut_v_cell`.

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

7 In the **Maximum** text field, type `1.5`.

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

9 In the **Title** text area, type `Slices: Velocity magnitude (m/s)`.

Slice 2

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

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

3 From the **Plane** list, choose **zx-planes**.

4 From the **Entry method** list, choose **Coordinates**.

5 In the **y-coordinates** text field, type `Cut_h_edge`.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

7 Locate the **Title** section. From the **Title type** list, choose **None**.

8 Adjust camera to reproduce **Figure 6**.

log10(Turbulence kinetic energy) slices xy and xz

1 In the **Model Builder** window, right-click **Velocity slices xy and xz** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `log10(Turbulence kinetic energy) slices xy and xz` in the **Label** text field.

Slice 1

1 In the **Model Builder** window, expand the **log10(Turbulence kinetic energy) slices xy and xz** node, then click **Slice 1**.
2 In the **Settings** window for **Slice**, locate the **Range** section.
3 Clear the **Manual color range** checkbox.
4 Locate the **Title** section. In the **Title** text area, type `Slices: log10(Turbulence kinetic energy)`.
5 Locate the **Expression** section. In the **Expression** text field, type `log10(k)`.
6 Locate the **Coloring and Style** section. From the **Color table** list, choose **DipoleDark**.

Slice 2

1 In the **Model Builder** window, click **Slice 2**.
2 In the **Settings** window for **Slice**, locate the **Expression** section.
3 In the **Expression** text field, type `log10(k)`.
4 Reproduce **Figure 7**.

Velocity slices yz planes

1 In the **Model Builder** window, right-click **Velocity slices xy and xz** and choose **Duplicate**.

   The following **Plot group** serves to illustrate how velocity field is influenced by the porous domain and, predominantly, by the frame, and then becomes gradually homogenized in the wake region.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.
3 From the **View** list, choose **New view**.
4 In the **Label** text field, type `Velocity slices yz planes`.
5 Click to expand the **Plot Array** section. From the **Array type** list, choose **Square**.
6 From the **Array plane** list, choose **yz**.
7 In the **Model Builder** window, expand the **Velocity slices yz planes** node.

Slice 2, Surface 1

1 In the **Model Builder** window, under **Results** > **Velocity slices yz planes**, Ctrl-click to select **Surface 1** and **Slice 2**.

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

Slice 1

1 In the **Settings** window for **Slice**, locate the **Plane Data** section.

2 From the **Plane** list, choose **yz-planes**.

3 From the **Entry method** list, choose **Coordinates**.

4 In the **x-coordinates** text field, type `-2*D_filter`.

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

Slice 2

1 Right-click **Results** > **Velocity slices yz planes** > **Slice 1** and choose **Duplicate**.

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

3 In the **x-coordinates** text field, type `-D_filter`.

4 Locate the **Plot Array** section. In the **Row index** text field, type `-1`.

5 Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

6 Locate the **Title** section. From the **Title type** list, choose **None**.

Slice 3

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

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

3 In the **x-coordinates** text field, type `0`.

4 Locate the **Plot Array** section. In the **Row index** text field, type `-2`.

Slice 4

1 Right-click **Slice 3** and choose **Duplicate**.

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

3 In the **x-coordinates** text field, type `D_frame`.

4 Locate the **Plot Array** section. In the **Row index** text field, type `0`.

5 In the **Column index** text field, type `1`.

Slice 5

1 Right-click **Slice 4** and choose **Duplicate**.

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

3. In the **x-coordinates** text field, type D_frame+L_wake/2.
4. Locate the **Plot Array** section. In the **Row index** text field, type -1.

Slice 6

1. Right-click **Slice 5** and choose **Duplicate**.
2. In the **Settings** window for **Slice**, locate the **Plane Data** section.
3. In the **x-coordinates** text field, type D_frame+L_wake.
4. Locate the **Plot Array** section. In the **Row index** text field, type -2.
5. Adjust camera to reproduce **Figure 8**.

Turbulence kinetic energy yz planes

1. In the **Model Builder** window, right-click **Velocity slices yz planes** and choose **Duplicate**.

   The following **Plot group** serves to illustrate how turbulence kinetic energy is drastically amplified in the porous domain and quickly becomes attenuated in both upstream and downstream directions.
2. In the **Settings** window for **3D Plot Group**, type Turbulence kinetic energy yz planes in the **Label** text field.

Slice 1

1. In the **Model Builder** window, expand the **Turbulence kinetic energy yz planes** node, then click **Slice 1**.
2. In the **Settings** window for **Slice**, locate the **Title** section.
3. In the **Title** text area, type Slices: Turbulence kinetic energy (m²/s²).
4. Locate the **Range** section. Clear the **Manual color range** checkbox.
5. Locate the **Expression** section. In the **Expression** text field, type k.
6. Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayBodyLight**.

Slice 2

1. In the **Model Builder** window, click **Slice 2**.
2. In the **Settings** window for **Slice**, locate the **Expression** section.
3. In the **Expression** text field, type k.

Slice 3

1. In the **Model Builder** window, click **Slice 3**.
2. In the **Settings** window for **Slice**, locate the **Expression** section.
3. In the **Expression** text field, type k.

Slice 4

1 In the **Model Builder** window, click **Slice 4**.

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

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

Slice 5

1 In the **Model Builder** window, click **Slice 5**.

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

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

Slice 6

1 In the **Model Builder** window, click **Slice 6**.

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

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

4 Reproduce **Figure 9**.

Produce high-resolution mesh figures by exporting the images.

Image 1

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

2 In the **Settings** window for **Image**, choose **Presentation and document** from the **Preset** list.

3 Click to expand the **Layout** section. From the **Background** list, choose **Transparent**.

4 Click to expand the **Image** section. In the **Width** text field, type `3165`.

5 In the **Height** text field, type `2374`.

6 In the **Resolution** text field, type `90`.

7 Locate the **Scene** section. In the tree, select **Model (root)** > **Component 1 (comp1)** > **Mesh 1**.

8 Click **Use as Source**.

9 Click to expand the **Output** section. From the **Target** list, choose **File**.

10 Locate the **Scene** section. From the **View** list, choose **View 3**.

11 Locate the **Output** section. In the **Filename** text field, type `air_filter_mesh_1_HiRes.png`.

12 Click **Export**.

Image 2

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

2 In the **Settings** window for **Image**, locate the **Scene** section.

3 From the **View** list, choose **View 4**.

4 Click to expand the **Output** section. In the **Filename** text field, type `air_filter_mesh_2_HiRes.png`.

5 Click **Export**.