Created in COMSOL Multiphysics 5.3

Separation Through Dialysis

This model is licensed under the COMSOL Software License Agreement 5.3.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Dialysis is a widely used separation method. An example is hemodialysis, where membranes are used as artificial kidneys for people with renal failure. Other applications include the recovery of caustic colloidal hemicellulose during viscose manufacturing as well as the removal of alcohol from beer (Ref. 1).

In the dialysis process specific components are preferentially transported through a membrane. The process is diffusion-driven, that is, components diffuse through a membrane due to concentration differences between the dialysate and the permeate sides of the membrane. Separation between solutes is achieved as a result of the different diffusion rates across the membrane, which arise from differences in molecular size and solubility.

This example examines a process aimed at lowering the concentration of a contaminant component in an aqueous product stream. The dialysis equipment is made of a hollow fiber module, where a large number of hollow fibers act as the membrane. It focuses on the transport of the contaminant in the hollow fiber and through its wall.

Figure 1 shows a diagram of the hollow fiber assembly within a dialysis module where the dialysate flows inside while the permeate flows on the outside in a co-current manner. The contaminant is transported through the fiber walls to the permeate side. Species with a higher molecular weight are retained in the dialysate side, due to their low solubility and diffusivity through the membrane.

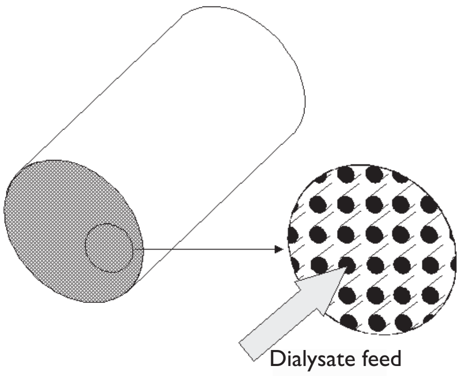

Figure 1: The hollow fiber assembly in a dialysis module.

Model Definition

This example models a piece of hollow fiber through which the dialysate flows with a fully developed laminar parabolic velocity profile. The fiber is surrounded by a permeate that flows laminarly in the same direction as the dialysate.The dialysate, the permeate, and the membrane are all examined in the results. The model domain is shown in Figure 2.Here, the angular gradients are considered negligible, so an axisymmetrical approximation can be used.

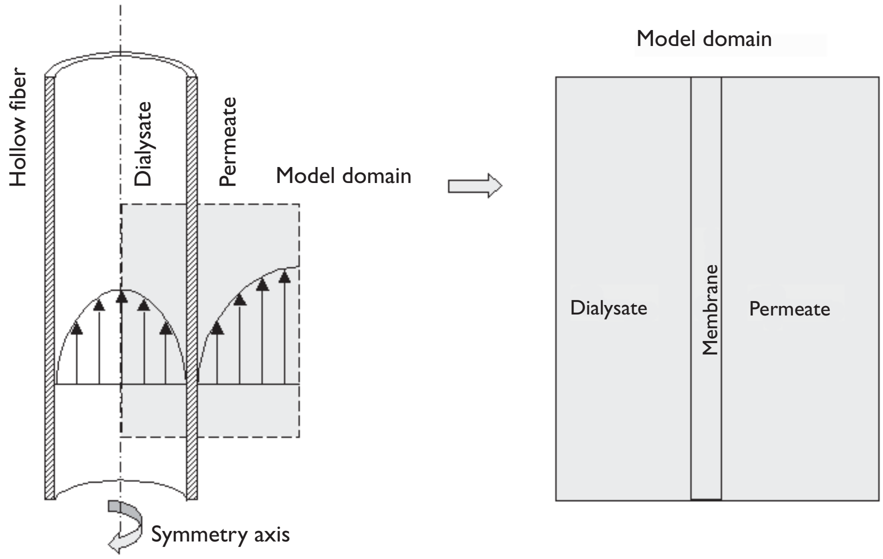

Figure 2: Illustrations of the hollow fiber setup with the dialysate and permeate, and of the model domain.

You can draw a hexagonal-shaped unit cell of the fiber assembly as in Figure 3:

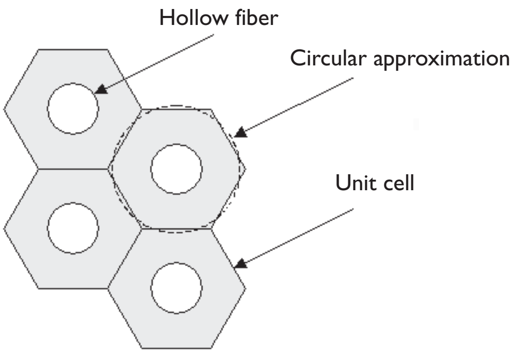

Figure 3: Hexagonal-shaped unit cell of the fiber assembly.

As a simplification, the hexagon is approximated as a circle in the model.

The contaminant is transported by diffusion and convection within the two liquids, whereas diffusion is the only transport mechanism through the membrane. The mass transport is modeled with the Transport of Diluted Species interface. To analyze the convective flux, the Laminar Flow interface is utilized, assuming that the flow is laminar.

The contaminant must dissolve into the membrane in order to be transported through it. The interface conditions between the liquids and the membrane are described by the dimensionless partition coefficient K:

$$K = \frac{c_2^{\mathrm{d}}}{c_1^{\mathrm{d}}} = \frac{c_2^{\mathrm{p}}}{c_3^{\mathrm{p}}} \tag{1}$$

where c_i denotes the concentration of the contaminant (SI unit: mol/m^3). The subscripts and superscripts describe the location in the dialysis fiber as displayed in Figure 4. This figure also shows a schematic concentration profile.

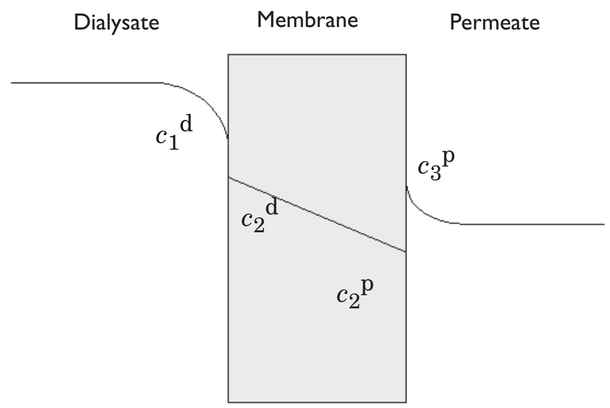

Figure 4: The concentration profile across the membrane (see Equation 1). Note that there are discontinuities in the concentration profile at the membrane boundaries.

To obtain a well-posed problem, an appropriate set of boundary conditions must be defined. Figure 5 displays the boundaries that need to be accounted for. Note that the boundaries are discontinuous and boundary conditions need to be set on both sides of the membrane at each liquid interface. Equation 1, set as Pointwise Constraints at the two boundaries, implements these boundary conditions in the model.

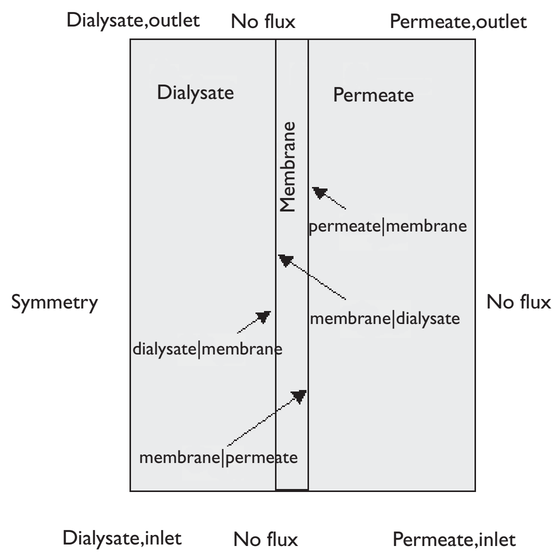

Figure 5: The boundaries accounted for in the model.

Danckwerts' inflow conditions are set at the inlet to the dialysate and permeate. At the outlet, the convective contribution to the mass transport is assumed to be much larger than the diffusive contribution and is modeled by setting outflow conditions. Symmetry applies at the leftmost boundary for this axisymmetrical model geometry and no flux is set at the membrane edges and the rightmost boundary, since no species pass these.

SUMMARY OF INPUT DATA

The input data are listed in the table:

PROPERTY	VALUE	DESCRIPTION
D	10^{-9} m^2/s	Diffusion coefficient, liquids
D_{m}	10^{-9} m^2/s	Diffusion coefficient, membrane
R_{hf}	0.2 mm	Inner radius, hollow fiber
L_{m}	0.28 mm	Thickness, membrane
L_{pc}	0.7 mm	Width, concentric permeate channel
H	21 mm	Length, fiber
$U_{\mathrm{ave\_dia}}$	0.5 mm/s	Average velocity, dialysate
$U_{\mathrm{ave\_per}}$	0.8 mm/s	Average velocity, permeate
K	0.7	Partition coefficient
c_0	1 M	Inlet concentration, dialysate

Results

The surface plot in Figure 6 visualizes the concentration distribution throughout the three model domains in 3D: the dialysate liquid inside the hollow fiber (nearest the center), the membrane, and the permeate liquid outside the hollow fiber. As the plot shows, the concentration inside the hollow fiber decreases markedly over the first 10 mm from the inlet. The maximum concentration in the permeate occurs close to the outlet. If there is a risk of deposition on the fiber's outer surface due to a high concentration of filtrated species, it is largest at the location of this maximum. The figure also shows the developing diffusion layers on both sides of the fiber wall.

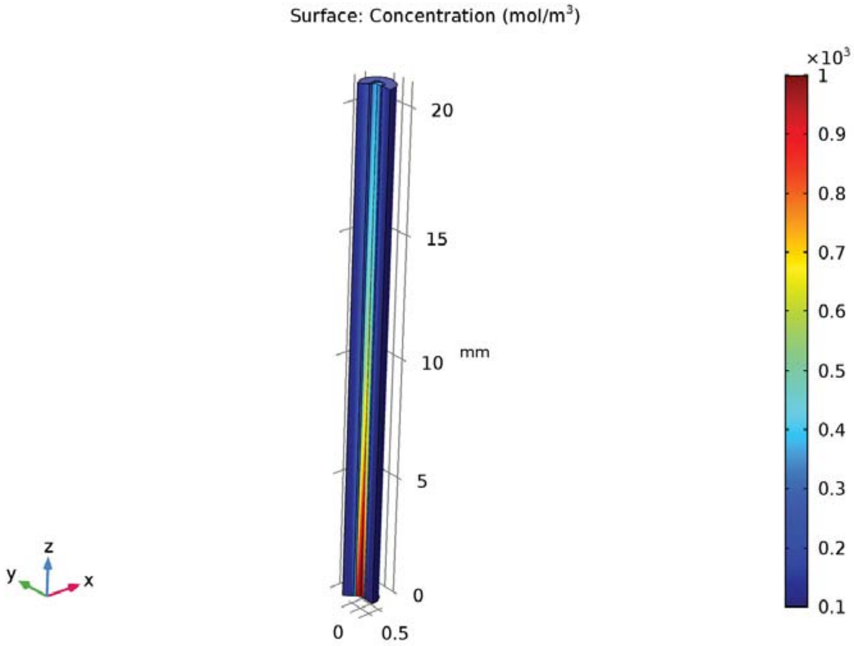

Figure 6: Concentration in the three domains as seen from the inlet of the fiber.

Concentration jumps arise at the boundaries between the domains. This is shown in Figure 7 where the concentration profile at the middle of the fiber length is plotted along the radius of the model geometry.

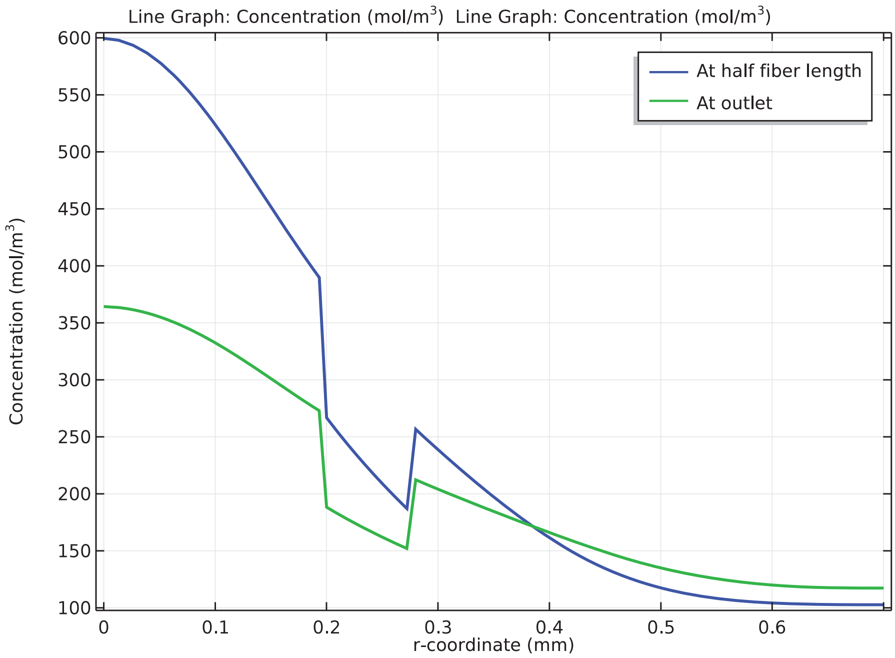

Figure 7: Concentration across the three domains at the middle of the fiber length and at the outlet.

Modeling in COMSOL Multiphysics

Manually defined point-wise constraints are used to model the discontinuities in the concentration profile at the boundaries between the liquid and membrane phases. The constraint expressions are defined according to Equation 1.

When defining these constraints the constraint forces need to be set to ensure a continuous flux over the phase boundaries. This is done by using `test()` operators, operating on the dependent variables that are solved for in each domain, for example `test(c1)`. What the `test()` terms in the force expressions represent depends on the partial differential equation solved for in each domain. In this model, the `test()` terms in the force expression on the liquid-membrane boundaries represent the diffusive flux.

References

1. M. Mulder, *Basic Principles of Membrane Technology*, 2nd ed., Kluwer Academic Publishers, 1998.

2. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, *Transport Phenomena*, John Wiley & Sons, 1960.

Application Library path: `Chemical_Reaction_Engineering_Module/Mixing_and_Separation/dialysis`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Chemical Species Transport>Transport of Diluted Species (tds)**.

3 Click **Add**.

4 In the **Concentrations** table, enter the following settings:

c1

5 In the **Select Physics** tree, select **Chemical Species Transport>Transport of Diluted Species (tds)**.

6 Click **Add**.

7 In the **Concentrations** table, enter the following settings:

c2

8 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

9 Click **Add**.

10 Click **Study**.

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

12 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

Load parameters from a text-file.

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 `dialysis_parameters.txt`.

Draw the geometry and make selections.

GEOMETRY 1

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

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

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

Rectangle 1 (r1)

1 ~~On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.~~ Right-click **Geometry 1** in Model Builder and choose **Rectangle.**

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

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

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

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

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

Rectangle 2 (r2)

1 ~~On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.~~ Right-click **Geometry 1** in Model Builder and choose **Rectangle.**

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

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

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

5 Locate the **Position** section. In the **r** text field, type `Rhf`.

6 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

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

Rectangle 3 (r3)

1 ~~On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.~~ Right-click **Geometry 1** in Model Builder and choose **Rectangle.**

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

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

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

5 Locate the **Position** section. In the **r** text field, type `Rhf+Lm`.

6 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

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

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

Explicit Selection 1 (sel1)

1 ~~On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.~~ Right-click **Geometry 1** in Model Builder and choose **Selection > Explicit Selection.**

2 In the **Settings** window for **Explicit Selection**, type `Dialysate and Permeate` in the **Label** text field.

3 ~~On the object **fin**, select Domains 1 and 3 only.~~ Select Domains 1 and 3 in the Graphics window.

Explicit Selection 2 (sel2)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 ~~On the object **fin**, select Domain 2 only.~~ Select Domain 2 in the Graphics window.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

Add a common variable `c_all` for all concentrations.

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

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

4 From the **Selection** list, choose **Dialysate and Permeate**.

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

Name	Expression	Unit	Description
c_all	c1	mol/m³	Concentration

Variables 2

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

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

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

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

Name	Expression	Unit	Description
`c_all`	`c2`	mol/m³	`Concentration`

Model transport by convection and diffusion in the dialysate and permeate, and diffusion in the membrane.

TRANSPORT OF DILUTED SPECIES (TDS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species (tds)**.

2 In the **Settings** window for **Transport of Diluted Species**, type `Transport of Diluted Species - Dialysate and Permeate` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Dialysate and Permeate**.

TRANSPORT OF DILUTED SPECIES - DIALYSATE AND PERMEATE (TDS)

On the **Physics** toolbar, click **Transport of Diluted Species (tds)** and choose **Transport of Diluted Species - Dialysate and Permeate (tds)**.

Transport Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)> Transport of Diluted Species - Dialysate and Permeate (tds)** node, then click **Transport Properties 1**.

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

3 In the D_{c1} text field, type `D`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Transport of Diluted Species - Dialysate and Permeate (tds)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the $c1$ text field, type `c0_dia`.

Initial Values 2

1 Right-click **Component 1 (comp1)>Transport of Diluted Species - Dialysate and Permeate (tds)>Initial Values 1** and choose **Duplicate**.

2 ~~Select Domain 3 only.~~ Change Selection to Manual and leave only Domain 3 in the Selection box.

3 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

4 In the $c1$ text field, type `c0_per`.

Inflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Inflow**, locate the **Concentration** section.

4 In the $c_{0,c1}$ text field, type `c0_dia`.

~~**5** Locate the **Boundary Condition Type** section. From the list, choose **Flux (Danckwerts)**.~~

Inflow 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 Select Boundary 8 only.

3 In the **Settings** window for **Inflow**, locate the **Concentration** section.

4 In the $c_{0,c1}$ text field, type `c0_per`.

~~**5** Locate the **Boundary Condition Type** section. From the list, choose **Flux (Danckwerts)**.~~

Outflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 Select Boundaries 3 and 9 only.

3 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Pointwise Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Pointwise Constraint**.

Select a pointwise boundary constraint to account for the transport between the domains.

2 Select Boundaries 4 and 7 only.

3 In the **Settings** window for **Pointwise Constraint**, locate the **Pointwise Constraint** section.

4 From the **Apply reaction terms on** list, choose **User defined**.

5 In the **Constraint expression** text field, type `c2-K*c1`.

6 In the **Constraint force expression** text field, type `test(c2)-test(c1)`.

TRANSPORT OF DILUTED SPECIES 2 (TDS2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species 2 (tds2)**.
2 In the **Settings** window for **Transport of Diluted Species**, type `Transport of Diluted Species - Membrane` in the **Label** text field.
3 Locate the **Domain Selection** section. From the **Selection** list, choose **Membrane**.
4 Locate the **Transport Mechanisms** section. Clear the **Convection** check box.

TRANSPORT OF DILUTED SPECIES - MEMBRANE (TDS2)

On the **Physics** toolbar, click **Transport of Diluted Species 2 (tds2)** and choose **Transport of Diluted Species - Membrane (tds2)**.

Transport Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Transport of Diluted Species - Membrane (tds2)** click **Transport Properties 1**.
2 In the **Settings** window for **Transport Properties**, locate the **Diffusion** section.
3 In the D_{c2} text field, type `Dm`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Transport of Diluted Species - Membrane (tds2)** click **Initial Values 1**.
2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3 In the $c2$ text field, type `c0_mem`.

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Liquids and Gases>Liquids>Water**.
4 Click **Add to Component** in the window toolbar.
5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Dialysate and Permeate**.
4 In the **Model Builder** window, click **Laminar Flow (spf)**.

Inlet 1

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

2 Select Boundary 2 only.

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

4 From the list, choose **Laminar inflow**.

5 Locate the **Laminar Inflow** section. In the U_{av} text field, type `Uave_dia`.

Inlet 2

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

2 Select Boundary 8 only.

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

4 From the list, choose **Laminar inflow**.

5 Locate the **Laminar Inflow** section. In the U_{av} text field, type `Uave_per`.

Outlet 1

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

2 Select Boundaries 3 and 9 only.

MULTIPHYSICS

Flow Coupling 1 (fc1)

On the **Physics** toolbar, click **Multiphysics** and choose **Global>Flow Coupling**.

MESH 1

Distribution 1

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

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

3 Select Boundaries 1, 4, 7, and 10 only.

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

5 From the **Distribution properties** list, choose **Predefined distribution type**.

6 In the **Number of elements** text field, type `250`.

7 In the **Element ratio** text field, type `25`.

Distribution 2

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

2 Select Boundaries 5 and 6 only.

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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 7.

6 In the **Element ratio** text field, type 2.

7 Select the **Symmetric distribution** check box.

Distribution 3

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

2 Select Boundaries 2 and 3 only.

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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 20.

6 In the **Element ratio** text field, type 2.

Distribution 4

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

~~2 Select Boundary 9 only.~~

~~3 Click the **Zoom Extents** button on the **Graphics** toolbar.~~

~~4 Click the **Zoom Box** button on the **Graphics** toolbar.~~

5 Select Boundaries 8 and 9 only.

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

7 From the **Distribution properties** list, choose **Predefined distribution type**.

8 In the **Number of elements** text field, type 30.

9 In the **Element ratio** text field, type 3.

10 Select the **Reverse direction** check box.

Solve the model in two steps. First, the **Laminar Flow** interface and thereafter the **Transport of Diluted Species** interfaces.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Transport of Diluted Species - Dialysate and Permeate (tds)** and **Transport of Diluted Species - Membrane (tds2)**.

Step 2: Stationary 2

1 On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Laminar Flow (spf)**.

4 On the **Study** toolbar, click **Compute**.

Plot the concentration distribution in Figure 6. For this axisymmetric geometry, the **Revolution 2D** data set is used.

RESULTS

Concentration (tds2) 1

1 In the **Model Builder** window, under **Results** click **Concentration (tds2) 1**.

2 In the **Settings** window for **3D Plot Group**, type `Concentration 2D Revolution - All` in the **Label** text field.

Revolution 2D 2

1 On the **Results** toolbar, click **More Data Sets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution layers** section.

3 Locate the **Revolution Layers** section. In the **Start angle** text field, type `-90`.

4 In the **Revolution angle** text field, type `225`.

Concentration 2D Revolution - All

1 In the **Model Builder** window, expand the **Results>Concentration 2D Revolution - All** node, then click **Concentration 2D Revolution - All**.

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

3 From the **Data set** list, choose **Revolution 2D 2**.

Surface

1 In the **Model Builder** window, under **Results>Concentration 2D Revolution - All** click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Definitions>Variables>c_all - Concentration**.

If necessary, the view angle of the plot can be adjusted with the mouse.

Concentration 2D Revolution - All

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

2 On the **Concentration 2D Revolution - All** toolbar, click **Plot**.

Velocity (spf) 1

1 In the **Model Builder** window, expand the **Concentration 2D Revolution - All** node, then click **Results>Velocity (spf) 1**.

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

3 From the **Data set** list, choose **Revolution 2D 2**.

4 On the **Velocity (spf) 1** toolbar, click **Plot**.

Create cut lines at two locations along the fiber length to illustrate the concentration jump between the domains in Figure 7.

Cut Line 2D 1

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

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

3 In row **Point 1**, set **Z** to `H/2`.

4 In row **Point 2**, set **R** to `Rhf+Lm+Lpc` and **z** to `H/2`.

Cut Line 2D 2

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

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

3 In row **Point 1**, set **Z** to `H`.

4 In row **Point 2**, set **R** to `Rhf+Lm+Lpc` and **z** to `H`.

1D Plot Group 8

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

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

Line Graph 1

1 Right-click **Concentration Jump** and choose **Line Graph**.

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

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `c_all`.

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

6 In the **Expression** text field, type `r`.

7 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

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

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

10 In the table, enter the following settings:

Legends
`At half fiber length`

Line Graph 2

1 Right-click **Results>Concentration Jump~~>At H/2~~** and choose **Line Graph**.

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

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 2**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `c_all`.

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

6 In the **Expression** text field, type `r`.

7 Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

8 Locate the **Legends** section. Select the **Show legends** check box.

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

10 In the table, enter the following settings:

Legends
`At outlet`

11 On the **Concentration Jump** toolbar, click **Plot**.

Fix the name of all plot groups.

Concentration (tds)

1 In the **Model Builder** window, under **Results** click **Concentration (tds)**.

2 In the **Settings** window for **2D Plot Group**, type `Concentration Surface - Dialysate and Permeate` in the **Label** text field.

Concentration (tds) 1

1 In the **Model Builder** window, under **Results** click **Concentration (tds) 1**.

2 In the **Settings** window for **3D Plot Group**, type `Concentration 2D revolution - Dialysate and Permeate` in the **Label** text field.

Concentration (tds2)

1 In the **Model Builder** window, under **Results** click **Concentration (tds2)**.

2 In the **Settings** window for **2D Plot Group**, type `Concentration Surface - Membrane` in the **Label** text field.

Velocity (spf)

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

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

Velocity (spf) 1

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

2 In the **Settings** window for **3D Plot Group**, type `Velocity 2D Revolution` in the **Label** text field.