Model created in COMSOL Multiphysics 6.4

# Reverse Osmosis Water Desalination

## *Introduction*

This example illustrates how to model the reverse osmosis process used to desalinate seawater. The modeled desalination unit consists of a spirally wound semi-permeable membrane through which the water is forced under high pressure. The membrane retains the salt, such that on the permeate side fresh water is produced and on the concentrate side (also called retentate side) a high salinity brine is obtained. This tutorial shows how to set up the flow and transport equations and assesses the high pressure needed to operate the process.

**Note:** This model requires the CFD Module.

## *Model Definition*

The desalination unit is built up around a sheet of porous spacer material with on both sides a semi-permeable membrane, which is rolled-up in a spiral manner. The sheet is not rolled-up tightly, but in such a way that there is space (with approximately the same width as the thickness of the sheet) between the windings for the feed flow of sea water. The sheet in the present model measures 1.3 m by 21 cm with a thickness of 2 mm. It is rolled-up along the long side resulting in a cylinder-like shape with a length of 21 cm. Sea water is fed through the spaces in between the rolled-up sheet along the axis of the cylinder at a high pressure such that fresh water is forced through the semi-permeable membranes. The fresh water flows through the porous sheet along the windings to the inside of the spiral where it is collected in a tube running along the center axis of the cylinder.

The sea water is fed into the unit at 27 l/h, of which 30% is forced through the semi-permeable membranes, resulting in a fresh water flow of 8.1 l/h (or 194.4 l/d). The present simulation is designed to determine the pressure needed to operate the process at these flow conditions and also how much salt is still in the permeate stream.

The geometry in the model is created by extruding along the $z$-axis a tightly wound 2D spiral with two layers drawn in an $xy$ work plane: one layer being the sheet of porous spacer material and the other being the flow domain for the feed flow. See Figure 1 below for a graphic representation of the geometry.

The semi-permeable membrane is modeled as a surface with no thickness and the volumetric water flux $J_{\mathrm{w}}$ (m/s) across the surface is given by

$$J_w = A_w(\Pi_r - \Pi_p - (p_r - p_p)) \tag{1}$$

. .

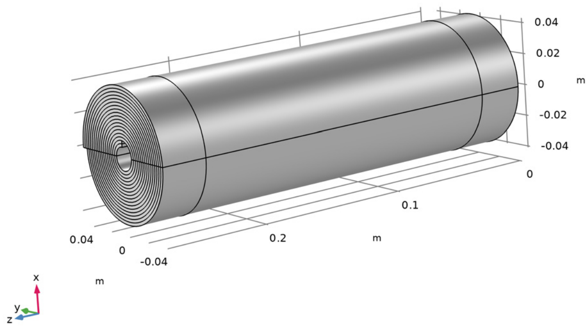


*Figure 1: Geometry of the model, created by extruding a 2D spiral along the x-axis.*

where $\Pi_r$ and $\Pi_p$ are the osmotic pressures and $p_r$ and $p_p$ the pressures on the retentate and permeate side, respectively. The parameter $A_w$ (m/Pa·s) is the resistance of the membrane. The osmotic pressure of a salt solution is given by

$$\Pi = 2cRT \tag{2}$$

where $c$ is the salt concentration (mol/m$^3$), $R$ the universal gas constant, and $T$ the temperature (K). The pressure jump across the membrane can thus be written as

$$(p_r - p_p) = 2RT(c_r - c_p) - \frac{J_w}{A_w} \tag{3}$$

where $c_r$ and $c_p$ are the salt concentrations on the retentate and permeate sides of the membrane. In the case of the high salt concentration in the feed sea water, the contribution to the pressure jump due to the osmotic pressure is expected to be much larger than the contribution due to the membrane flow resistance, so that the following approximation is made for the pressure jump

$$(p_r - p_p) \approx 2RT(c_r - c_p) \tag{4}$$

The salt flux $J_s$ (mol·m/s) across the membrane is given by

$$J_s = A_s(c_r - c_p) \quad (5)$$

where the parameter $A_s$ (m/s) is a property of the membrane. See Table 1 for the used value of this parameter and other model parameters.

TABLE 1: MODEL PARAMETERS.

| Name | Value | Description |
|---|---|---|
| D_s | $1.64 \cdot 10^{-9}$ m$^2$/s | Salt diffusion coefficient |
| T_w | 270 K | Water temperature |
| c_in | 600 mol/m$^3$ | Feed water salt concentration |
| rho_w | 1000 kg/m$^3$ | (Sea) water density |
| mu_w | 0.001 Pa·s | (Sea) water viscosity |
| A_s | $10^{-7}$ m/s | Membrane parameter |
| kappa_m | $10^{-9}$ m$^2$ | Permeability porous sheet |
| poro_m | 0.6 | Porosity porous sheet |
| F_fr | $7.5 \cdot 10^{-3}$ kg/s | Feed mass flow rate |
| R_r | 0.3 | Production fraction |

## *Results and Discussion*

In Figure 2 the pressure of the concentrate flow is plotted. It can be seen that the pressure for the reverse osmosis process under the assumed operating conditions is around 76 bar. Also note that the pressure drop along the desalination unit is small compared to the applied pressure.

In Figure 3 the pressure of the permeate flow through the spirally wound porous material in between the semi-permeable membranes is plotted. The pressure is high at the outside windings of the spiral and lower at the inside windings, indicating that the permeate flows from the outside to the inside of the spiral, as is intended by the design of the unit.

Figure 4 shows the salt concentration in the concentrate flow. At the inlet the concentration is equal to 600 mol/m$^3$, and the concentration increases as the sea water flows in between the spiral sheet, with boundary layers of high salt concentration near the membranes.

In Figure 5 the salt concentration in the permeate flow is plotted. The concentration here is much lower than in the feed stream, indicating that the membrane does form a barrier for the salt. The average salt concentration at the outlet of the permeate flow is equal to 37.33 mol/m$^3$, which is about 16 times lower than the salt concentration of the sea water that is fed into the unit. In the Modeling Instructions section it will be shown how to obtain this value from the simulation.

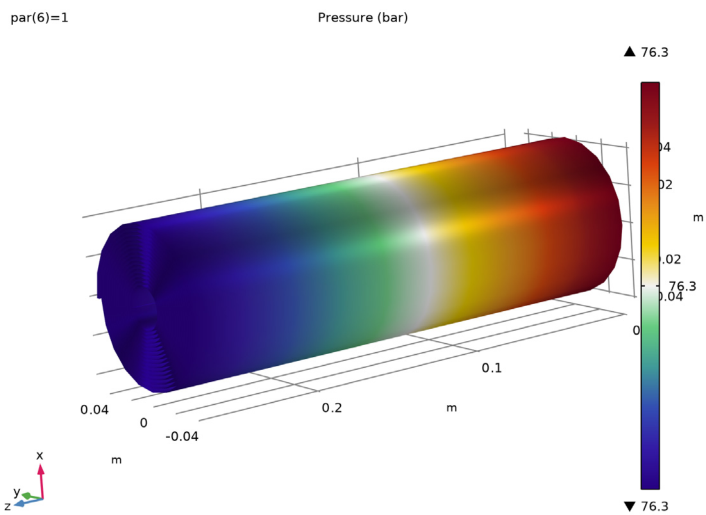


*Figure 2: The pressure of the concentrate flow.*

.

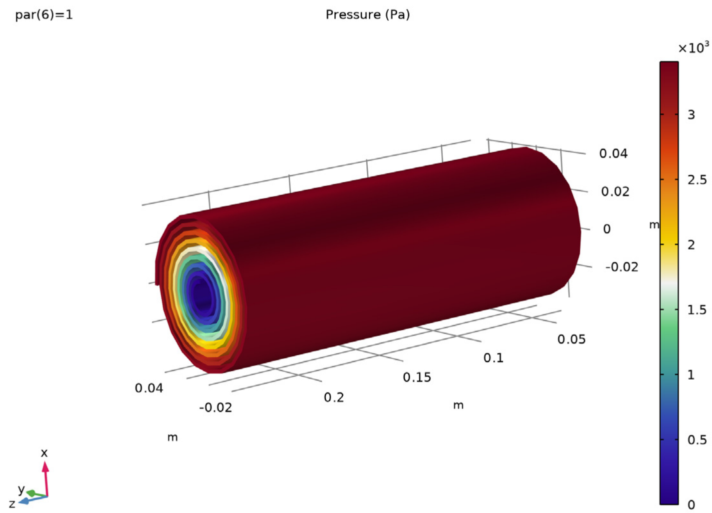


*Figure 3: The pressure of the permeate flow through the spirally wound porous material in between the semi-permeable membranes.*

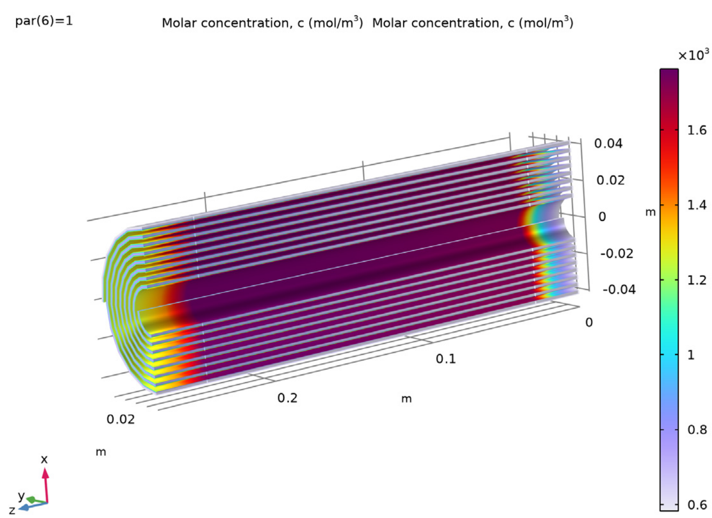


*Figure 4: The salt concentration in the feed flow.*

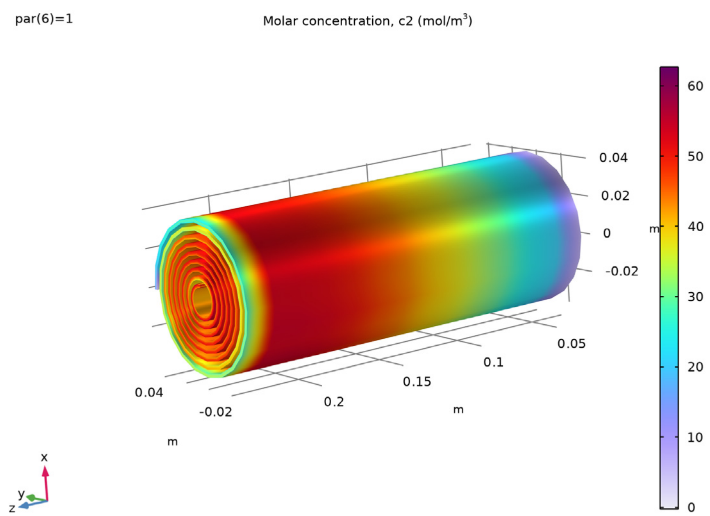


*Figure 5: The salt concentration in the permeate flow.*

## *Notes About the COMSOL Implementation*

In this model it is assumed that the density of the water does not depend on the salt concentration. The feed flow in between the spirally wound sheet is modeled using the Laminar Flow interface and the permeate flow in the porous sheet is modeled using the Darcy's Law interface. These two interface are coupled through the Free and Porous Media Flow Coupling, which allows to include the pressure jump across the semi-permeable membrane due to the osmotic pressure.

The pressure jump condition depending on the salt concentrations makes the model quite nonlinear. To get the simulation to converge under the chosen operating conditions, the diffusion coefficient of the dissolved salt is ramped down to the physical value, starting with a value that is 100,000 times larger.

**Application Library path:** `Chemical_Reaction_Engineering_Module/Mixing_and_Separation/reverse_osmosis_desalination`

## *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** > **Porous Media and Subsurface Flow** > **Free and Porous Media Flow, Darcy**.

3 Click **Add**.

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

5 Click **Add**.

6 In the **Select Physics** tree, select **Chemical Species Transport** > **Transport of Diluted Species in Porous Media (tds)**.

7 Click **Add**.

8 Click **Study**.

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

10 Click **Done**.

### GEOMETRY 1

*Work Plane 1 (wp1)*

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

*Work Plane 1 (wp1) > Plane Geometry*

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

*Work Plane 1 (wp1) > Parametric Curve 1 (pc1)*

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.

3 In the **Minimum** text field, type `4*pi`.

4 In the **Maximum** text field, type `20.5*pi`.

5 Locate the **Expressions** section. In the **xw** text field, type `0.00065*s*cos(s)`.

6 In the **yw** text field, type `0.00065*s*sin(s)`.

### *Work Plane 1 (wp1) > Thicken 1 (thi1)*

1 In the **Work Plane** toolbar, click **Conversions** and choose **Thicken**.
2 In the **Settings** window for **Thicken**, locate the **Input** section.
3 Select the **Keep input objects** checkbox.
4 Select the object **pc1** only.
5 Locate the **Options** section. From the **Offset** list, choose **Asymmetric**.
6 In the **Upside thickness** text field, type `0.00205`.

### *Work Plane 1 (wp1) > Thicken 2 (thi2)*

1 Right-click **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** > **Thicken 1 (thi1)** and choose **Duplicate**.
2 In the **Settings** window for **Thicken**, locate the **Options** section.
3 In the **Upside thickness** text field, type `0`.
4 In the **Downside thickness** text field, type `0.00205`.

### *Extrude 1 (ext1)*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.
2 In the **Settings** window for **Extrude**, locate the **Distances** section.
3 In the table, enter the following settings:

| Distances (m) |
|---|
| 0.03 |
| 0.24 |
| 0.27 |

### *Work Plane 2 (wp2)*

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**.

### *Partition Objects 1 (par1)*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.
2 Select the object **ext1** only.
3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.
4 From the **Partition with** list, choose **Work plane**.

### *Remove Details 1 (rmd1)*

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Remove Details**.

2 Click **Build All**.

## 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 `reverse_osmosis_desalination_parameters.txt`.

## MATERIALS

### *Material 1 (mat1)*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

3 In the table, enter the following settings:

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Density | rho | `rho_w` | kg/m³ | Basic |
| Dynamic viscosity | mu | `mu_w` | Pa·s | Basic |
| Porosity | epsilon | `poro_m` | 1 | Basic |
| Permeability | kappa_iso ; kappaii = kappa_iso, kappaij = 0 | `kappa_m` | m² | Basic |

## LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 Select Domains 1–3, 7–9, 13–18, 25–30, 37–42, 49–54, 61–66, 73–78, and 85–90 only.

3 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog, type `Laminar flow domain` in the **Selection name** text field.

6 Click **OK**.

*Inlet 1*

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

2 Select Boundaries 3, 13, 41, 42, 75, 76, 109, 110, 143, 144, 177, 178, 211, 212, 245, and 246 only.

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

4 Click **Create Selection**.

5 In the **Create Selection** dialog, type `Inlet` in the **Selection name** text field.

6 Click **OK**.

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

8 From the list, choose **Mass flow**.

9 Locate the **Mass Flow** section. In the $m$ text field, type `F_fr`.

*Outlet 1*

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

2 Select Boundaries 10, 20, 53, 54, 87, 88, 121, 122, 155, 156, 189, 190, 223, 224, 257, and 258 only.

3 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog, type `Outlet` in the **Selection name** text field.

6 Click **OK**.

7 In the **Settings** window for **Outlet**, locate the **Boundary Condition** section.

8 From the list, choose **Mass flow**.

9 Locate the **Mass Flow** section. In the $m$ text field, type `F_fr*(1-R_r)`.

*Wall 2*

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

2 Select Boundaries 347–349 only.

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

4 From the **Wall condition** list, choose **Slip**.

5 In the **Model Builder** window, click **Wall 2**.

6 Select Boundaries 292, 295, 298, and 347–349 only.

### DARCY'S LAW (DL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Darcy's Law (dl)**.

2 Select Domains 5, 11, 21, 22, 33, 34, 45, 46, 57, 58, 69, 70, 81, 82, 93, 94, and 98 only.

3 In the **Settings** window for **Darcy's Law**, locate the **Domain Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog, type `Porous permeate flow` in the **Selection name** text field.

6 Click **OK**.

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

8 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

9 Click **OK**.

10 In the **Model Builder** window, click **Darcy's Law (dl)**.

11 In the **Settings** window for **Darcy's Law**, click to expand the **Discretization** section.

12 From the **Pressure** list, choose **Linear**.

13 Select the **Compute boundary fluxes** checkbox.

*Pressure 1*

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

2 Select Boundaries 350–352 only.

3 In the **Settings** window for **Pressure**, locate the **Boundary Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog, type `Permeate outlet` in the **Selection name** text field.

6 Click **OK**.

### MULTIPHYSICS

*Free and Porous Media Flow Coupling 1 (nsd1)*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Free and Porous Media Flow Coupling 1 (nsd1)**.

2 In the **Settings** window for **Free and Porous Media Flow Coupling**, locate the **Boundary Selection** section.

3 Click **Create Selection**.

4 In the **Create Selection** dialog, type `Membrane` in the **Selection name** text field.

5. Click **OK**.
6. In the **Settings** window for **Free and Porous Media Flow Coupling**, locate the **Coupling Settings** section.
7. Select the **Include pressure jump across free-porous boundary** checkbox.
8. In the $p_j$ text field, type `2*R_const*T_w*(c-c2)`.

## 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**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Laminar flow domain**.
4. In the **Model Builder** window, click **Transport of Diluted Species (tds)**.
5. Click to expand the **Advanced Settings** section. From the **Material balance form** list, choose **Conservative**.

### *Fluid 1*

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Transport of Diluted Species (tds)** click **Fluid 1**.
2. In the **Settings** window for **Fluid**, locate the **Convection** section.
3. From the $\mathbf{u}$ list, choose **Velocity field (spf)**.
4. Locate the **Diffusion** section. In the $D_c$ text field, type `D_s*par`.

### *Inflow 1*

1. In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.
2. In the **Settings** window for **Inflow**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Inlet**.
4. Locate the **Concentration** section. In the $c_{0,c}$ text field, type `c_in`.

### *No Flux 1*

1. In the **Model Builder** window, click **No Flux 1**.
2. In the **Settings** window for **No Flux**, locate the **Convection** section.
3. Select the **Include** checkbox.

### *Outflow 1*

1. In the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
2. In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.

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

### *Flux 1*

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

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

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

**4** Locate the **Convection** section. Select the **Include** checkbox.

**5** Locate the **Inward Flux** section. Select the **Species c** checkbox.

**6** In the $J_{0,c}$ text field, type `-A_s*(c-c2)`.

## TRANSPORT OF DILUTED SPECIES IN POROUS MEDIA 2 (TDS2)

**1** In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species in Porous Media 2 (tds2)**.

**2** In the **Settings** window for **Transport of Diluted Species in Porous Media**, locate the **Domain Selection** section.

**3** From the **Selection** list, choose **Porous permeate flow**.

**4** Click to expand the **Advanced Settings** section. From the **Material balance form** list, choose **Conservative**.

### *Fluid 1*

**1** In the **Model Builder** window, under **Component 1 (comp1)** > **Transport of Diluted Species in Porous Media 2 (tds2)** > **Porous Medium 1** click **Fluid 1**.

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

**3** From the $\mathbf{u}$ list, choose **Total Darcy velocity field (dl/porous1)**.

**4** Locate the **Diffusion** section. In the $D_{F,c2}$ text field, type `D_s*par`.

### *No Flux 1*

**1** In the **Model Builder** window, under **Component 1 (comp1)** > **Transport of Diluted Species in Porous Media 2 (tds2)** click **No Flux 1**.

**2** In the **Settings** window for **No Flux**, locate the **Convection** section.

**3** Select the **Include** checkbox.

### *Outflow 1*

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

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

**3** From the **Selection** list, choose **Permeate outlet**.

*Flux 1*

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

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

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

4 Locate the **Convection** section. Select the **Include** checkbox.

5 Locate the **Inward Flux** section. Select the **Species c2** checkbox.

6 In the $J_{0,c2}$ text field, type `A_s*(c-c2)`.

## MESH 1

*Mapped 1*

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

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

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

*Distribution 1*

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

2 Select Edges 2, 21, 47, 73, 103, 133, 163, 193, 223, 275, 297, 311, 325, 339, 353, 367, 384, and 401 only.

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

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

5 In the **Number of elements** text field, type `10`.

6 In the **Element ratio** text field, type `10`.

7 Select the **Symmetric distribution** checkbox.

*Distribution 2*

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

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

3 In the **Number of elements** text field, type `10`.

4 Select Edges 1, 16, 22, 33, 34, 48, 59, 60, 74, 75, 89, 90, 104, 105, 119, 120, 134, 135, 149, 150, 164, 165, 179, 180, 194, 195, 209, 210, 224, 225, 239, and 240 only.

*Distribution 3*

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

2 Select Edges 276 and 286 only.

*Swept 1*

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

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

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

**4** Select Domains 1, 7, 13, 14, 25, 26, 37, 38, 49, 50, 61, 62, 73, 74, 85, and 86 only.

*Distribution 1*

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

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

**3** In the **Number of elements** text field, type 2.

*Mapped 2*

**1** In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

**2** Select Boundaries 29, 30, 63, 64, 97, 98, 131, 132, 165, 166, 199, 200, 233, 234, 267, 268, and 287 only.

*Distribution 1*

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

**2** Select Edges 8, 36, 62, 92, 122, 152, 182, 212, 267, 293, 307, 321, 335, 349, 363, and 377 only.

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

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

*Distribution 2*

**1** In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.

**2** Select Edges 268 and 279 only.

*Swept 2*

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

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

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

**4** Select Domains 2, 5, 8, 11, 15, 16, 21, 22, 27, 28, 33, 34, 39, 40, 45, 46, 51, 52, 57, 58, 63, 64, 69, 70, 75, 76, 81, 82, 87, 88, 93, 94, and 98 only.

*Distribution 1*

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

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

**3** In the **Number of elements** text field, type 13.

### *Swept 3*

1 In the **Mesh** toolbar, click **Swept**.
2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domains 3, 9, 17, 18, 29, 30, 41, 42, 53, 54, 65, 66, 77, 78, 89, and 90 only.

### *Distribution 1*

1 Right-click **Swept 3** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type 2.
4 Click **Build All**.

## 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**, click to expand the **Study Extensions** section.
3 Select the **Auxiliary sweep** checkbox.
4 Click **Add**.
5 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| par (Ramping parameter) | `10^range(5,-1,0)` | |

6 From the **Run continuation for** list, choose **No parameter**.
7 From the **Reuse solution from previous step** list, choose **Yes**.

### *Solution 1 (sol1)*

1 In the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node.
4 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** and choose **Fully Coupled**.
5 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** node, then click **Concentration (comp1.c)**.
6 In the **Settings** window for **Field**, locate the **Scaling** section.

7 From the **Method** list, choose **Manual**.

8 In the **Scale** text field, type 600.

9 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** click **Concentration (comp1.c2)**.

10 In the **Settings** window for **Field**, locate the **Scaling** section.

11 From the **Method** list, choose **Manual**.

12 In the **Scale** text field, type 6.

13 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** click **Pressure Lagrange Multiplier (comp1.dl.pb_lm)**.

14 In the **Settings** window for **Field**, locate the **Scaling** section.

15 From the **Method** list, choose **Manual**.

16 In the **Scale** text field, type 100.

17 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** click **Pressure (comp1.p)**.

18 In the **Settings** window for **Field**, locate the **Scaling** section.

19 From the **Method** list, choose **Manual**.

20 In the **Scale** text field, type 1e7.

21 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** click **Pressure (comp1.p2)**.

22 In the **Settings** window for **Field**, locate the **Scaling** section.

23 From the **Method** list, choose **Manual**.

24 In the **Scale** text field, type 1000.

25 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** click **Velocity Field (comp1.u)**.

26 In the **Settings** window for **Field**, locate the **Scaling** section.

27 From the **Method** list, choose **Manual**.

28 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** click **Pressure from mass flow rate (comp1.spf.inl1.Pmf)**.

29 In the **Settings** window for **State**, locate the **Scaling** section.

30 From the **Method** list, choose **Manual**.

31 In the **Scale** text field, type 1e7.

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

To create Figure 2, follow the steps below.

## RESULTS

### *Pressure (spf)*

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

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

3 Clear the **Plot dataset edges** checkbox.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

### *Surface*

1 In the **Model Builder** window, expand the **Pressure (spf)** node, then click **Surface**.

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

3 From the **Unit** list, choose **bar**.

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

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

6 In the **Pressure (spf)** toolbar, click **Plot**.

The following steps create Figure 3.

### *Pressure (dl)*

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

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

3 Clear the **Plot dataset edges** checkbox.

### *Surface*

1 In the **Model Builder** window, expand the **Pressure (dl)** node, then click **Surface**.

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

3 From the **Color table type** list, choose **Continuous**.

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

5 In the **Pressure (dl)** toolbar, click **Plot**.

For Figure 4 and Figure 5, follow the instructions below.

### *Concentration, Surface (tds)*

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

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

**3** Clear the **Plot dataset edges** checkbox.

*Filter 1*

**1** In the **Model Builder** window, expand the **Concentration, Surface (tds)** node.

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

**3** In the **Settings** window for **Filter**, locate the **Element Selection** section.

**4** In the **Logical expression for inclusion** text field, type `y>0`.

*Slice 1*

**1** In the **Model Builder** window, right-click **Concentration, Surface (tds)** and choose **Slice**.

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

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

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

**5** Locate the **Expression** section. In the **Expression** text field, type `c`.

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

**7** In the **Concentration, Surface (tds)** toolbar, click **Plot**.

*Concentration, Surface (tds2)*

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

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

**3** Clear the **Plot dataset edges** checkbox.

**4** In the **Concentration, Surface (tds2)** toolbar, click **Plot**.

*Velocity, Streamline*

The following steps create the plot that is used as the model thumbnail.

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

**2** Right-click **3D Plot Group 9** and choose **Rename**.

**3** In the **Rename 3D Plot Group** dialog, type `Velocity, Streamline` in the **New label** text field.

**4** Click **OK**.

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

**6** Clear the **Plot dataset edges** checkbox.

*Streamline 1*

**1** Right-click **Velocity, Streamline** and choose **Streamline**.

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

3 In the **Number** text field, type `80`.

4 Locate the **Selection** section. From the **Selection** list, choose **Inlet**.

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

### *Color Expression 1*

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

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

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

### *Streamline 2*

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

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Darcy's Law** > **Velocity and pressure** > **dl.u,dl.v,dl.w - Total Darcy velocity field**.

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

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

### *Color Expression 1*

1 Right-click **Streamline 2** and choose **Color Expression**.

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

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

4 Locate the **Expression** section. In the **Expression** text field, type `dl.U`.

### *Surface 1*

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

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

3 From the **Coloring** list, choose **Uniform**.

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

### *Filter 1*

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

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

3 In the **Logical expression for inclusion** text field, type `(y>0)*(z<0.265)`.

4 In the **Velocity, Streamline** toolbar, click **Plot**.

5 Click the **Show Legends** button in the **Graphics** toolbar.

6 Click the **Show Grid** button in the **Graphics** toolbar.

7 In the **Graphics** window toolbar, click next to **Scene Light**, then choose **Ambient Occlusion**.

8 In the **Graphics** window toolbar, click next to **Scene Light**, then choose **Direct Shadows**.

*Global Evaluation 1*

The following instructions check the mass conservation by comparing the mass flow rates at the inlet and outlets.

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

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

3 From the **Parameter selection (par)** list, choose **Last**.

4 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Laminar Flow** > **Auxiliary variables** > **spf.inl1.massFlowRate - Outward mass flow rate across feature selection - kg/s**.

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

| Expression | Unit | Description |
|---|---|---|
| spf.inl1.massFlowRate | kg/h | Outward mass flow rate across feature selection |

6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Laminar Flow** > **Auxiliary variables** > **spf.out1.massFlowRate - Outward mass flow rate across feature selection - kg/s**.

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

| Expression | Unit | Description |
|---|---|---|
| spf.out1.massFlowRate | kg/h | Outward mass flow rate across feature selection |

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Darcy's Law** > **Mass flow** > **dl.pr1.Mflow - Mass flow - kg/s**.

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

| Expression | Unit | Description |
|---|---|---|
| dl.pr1.Mflow | kg/h | Mass flow |

**10** Click **Evaluate**.

Note that the sum of the mass flow rates at the outlets matches the mass flow rate at the inlet, indicating the exact mass conservation of the flow.

*Global Evaluation 2*

The following instructions check the mass conservation of the salt.

1. In the **Results** toolbar, click **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, locate the **Data** section.
3. From the **Parameter selection (par)** list, choose **Last**.
4. Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Transport of Diluted Species** > **Inflow 1** > **tds.in1.nmflow_c - Normal molar flow rate - mol/s**.
5. Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| tds.in1.nmflow_c | mol/h | Normal molar flow rate |

6. Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Transport of Diluted Species** > **Outflow 1** > **tds.out1.nmflow_c - Normal molar flow rate - mol/s**.
7. Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| tds.out1.nmflow_c | mol/h | Normal molar flow rate |

8. Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Transport of Diluted Species in Porous Media 2** > **Outflow 1** > **tds2.out1.nmflow_c2 - Normal molar flow rate - mol/s**.
9. Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| tds2.out1.nmflow_c2 | mol/h | Normal molar flow rate |
| tds.out1.nmflow_c+ tds2.out1.nmflow_c2 | mol/h | |

**10** Click next to **Evaluate**, then choose **New Table**.

Note that again the sum of the salt mass flow rates at the outlets matches the salt mass flow rate at the inlet, indicating the exact mass conservation of the salt. The following last instructions evaluate the salt concentrations in the inlet and outlet streams.

*Global Evaluation 3*

**1** In the **Results** toolbar, click 8.5 **Global Evaluation**.

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

**3** From the **Parameter selection (par)** list, choose **Last**.

**4** Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Transport of Diluted Species** > **Inflow 1** > **tds.in1.c0_avg_c - Average concentration - mol/m³**.

**5** Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Transport of Diluted Species** > **Outflow 1** > **tds.out1.c0_avg_c - Average concentration - mol/m³**.

**6** Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Transport of Diluted Species in Porous Media 2** > **Outflow 1** > **tds2.out1.c0_avg_c2 - Average concentration - mol/m³**.

**7** Click next to **Evaluate**, then choose **New Table**.

Observe that the average salt concentration of the permeate outlet stream equals 37.33 mol/$m^3$ as reported in the Results section.