Created in COMSOL Multiphysics 6.2

Electrodeposition of a Microconnector Bump with Deforming Geometry in 3D

Introduction

This model simulates the shape evolution of a microconnector bump over time as copper deposits on an electrode surface. Transport of cupric ions in the electrolyte occurs by convection and diffusion. The electrode kinetics are described by a concentration dependent Butler-Volmer expression.

The model is an extension to 3D of the Electrodeposition of a Microconnector Bump in 2D example.

Model Definition

The basics of the electrochemical cell, geometry and model problem are described in Electrodeposition of a Microconnector Bump in 2D. For this 3D model the Péclet number of the cell is 41.6.

Figure 1 shows the 3D model geometry. Due to symmetry the unit cell has been cut in half along the x-axis.

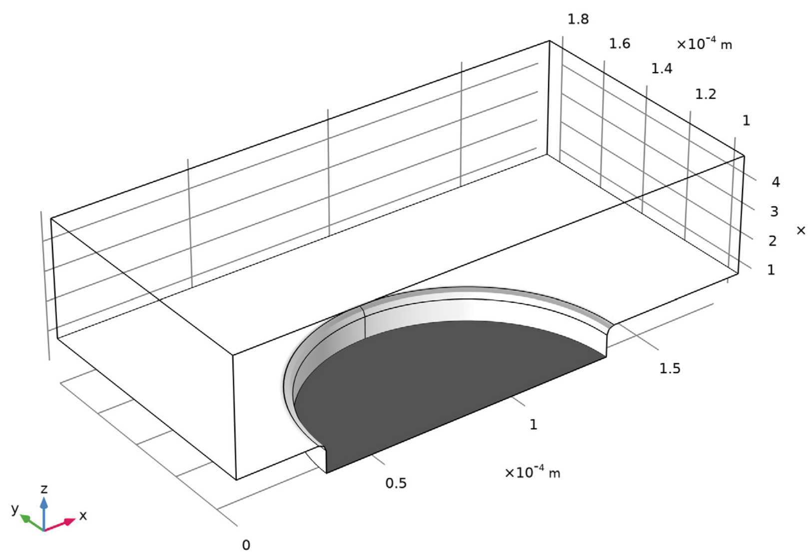


Figure 1: Model geometry. The electrolyte, flows from left to right in the x direction, over the circular hole in the photoresist mask, and exits on the right. The cathode (gray) is placed at the bottom of the circular hole. The top boundary is in contact with the electrolyte bulk.

In the 2D model, the cupric ion concentration is set to zero on the electrode surface, and the electrode current density is calculated from the electrolyte flux variable. In this 3D model however, in order to improve the stability of the deforming boundary, a concentration dependent Butler-Volmer expression is used to describe the current density at the cathode.

On the bottom boundary, the cathode, the electrode reaction

$$\mathrm{Cu}^{2+}(\mathrm{l}) + 2e^{-} = \mathrm{Cu}(\mathrm{s})$$

follows the following kinetics expression for the charge transfer current i_{ct}:

$$i_{\mathrm{ct}} = i_0\left(\exp\left(\frac{1.5F\eta}{RT}\right) - \frac{c_{\mathrm{Cu}^{2+}}}{c_{\mathrm{Cu}^{2+},\mathrm{ref}}}\exp\left(-\frac{0.5F\eta}{RT}\right)\right)$$

where i_0 is the exchange current density (10 A/m^2), η the overpotential, F Faraday's constant (96,485 C/mol), R the molar gas constant (8.13 J/(mol·K)), T the temperature, $c_{\mathrm{Cu}^{2+}}$ the electrolyte cupric ion concentration (mol/m^3), and $c_{\mathrm{Cu}^{2+},\mathrm{ref}}$, the reference cupric concentration in the bulk electrolyte (600 mol/m^3).

The electrode reaction causes the electrode boundary to move in the normal direction with a velocity v_{dep} (m/s) according to

$$v_{\mathrm{dep}} = -\frac{M_{\mathrm{Cu}}}{\rho_{\mathrm{Cu}}}\frac{i_{\mathrm{ct}}}{2F}$$

where M_{Cu} is the molar mass (0.06355 kg/mol) and ρ_{Cu} the density (8,960 kg/m^3) of copper, respectively.

The electrode potential is set to –0.45 V. The electrolyte conductivity is set to 1 S/m, and the top bulk electrolyte boundary potential is set 0 V. All boundaries except the cathode and the top bulk electrolyte boundary are isolated.

The problem is solved in a time-dependent simulation to simulate the electrode deformation during 120 s.

Results and Discussion

Figure 2 shows the concentration in the cell at t = 0 s. The concentration along the zx-plane is qualitatively similar to Figure 5 of the 2D model, with the main differences being the cupric concentration at the electrode surface. This difference is due to the changed boundary condition, with a limiting current condition in the 2D model and a mixed concentration/activation condition in the 3D model.

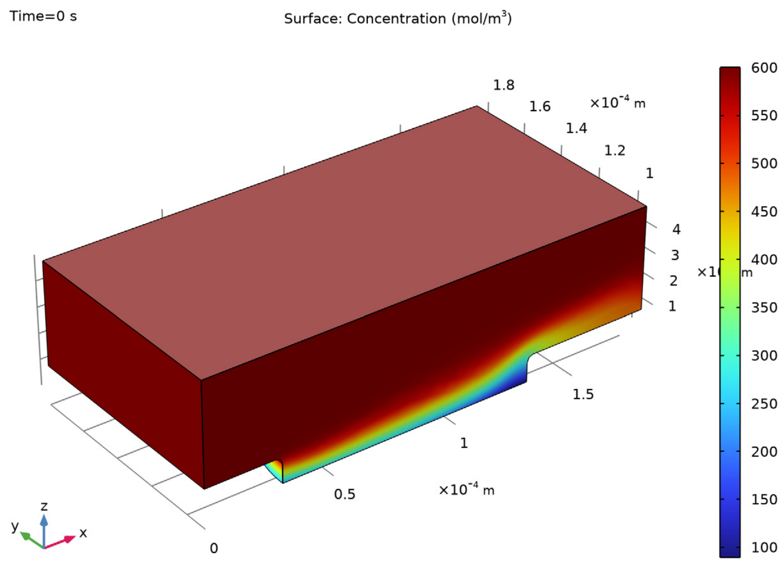


Figure 2: Cu^{2+} concentration in the cell at t = 0.

Figure 3 shows the concentration at $t = 120$ s. The minimum concentration is now higher compared to $t = 0$ s due to a shorter transport length of ions toward the electrode surface.

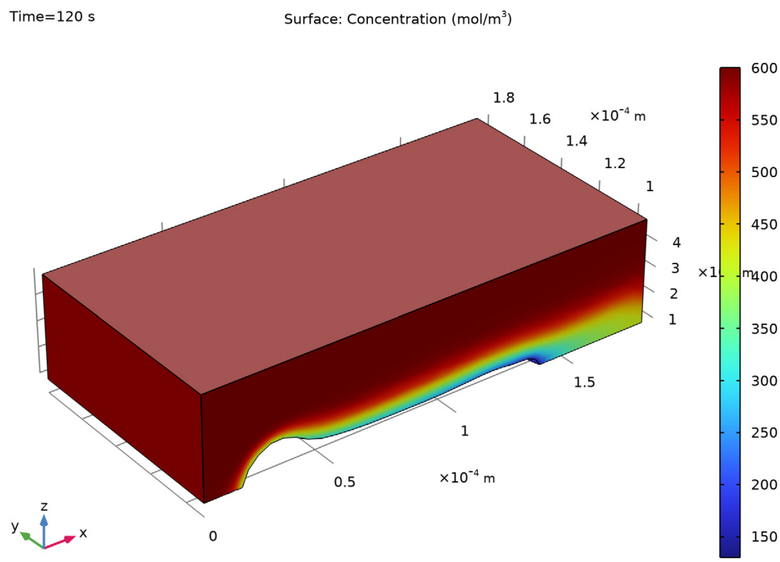


Figure 3: Cu^{2+} concentration in the cell at t = 120.

Figure 4 shows the electrode surface at t = 120 s. The deposit is thicker toward the left in the figure, and the shape follows the same trend as was seen in Figure 7 of the 2D model.

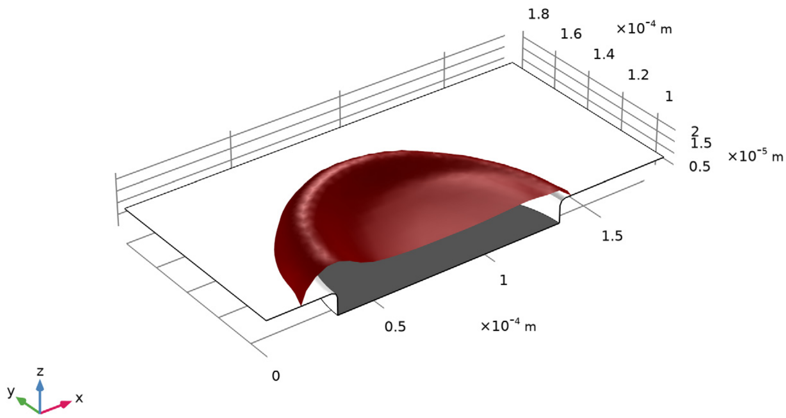


Figure 4: Microconnector bump surface (red) at t = 120 s.

Notes About the COMSOL Implementation

The model is solved using two stationary steps followed by a time-dependent step. The first stationary step solves for the laminar flow only. The second stationary step solves for the concentration and initial current distribution in the cell. The results from the two initial steps are used as initial values for the third time-dependent study step.

Reference

K. Kondo, K. Fukui, K. Uno, and K. Shonohara, "Shape Evolution of Electrodeposited Copper Bumps," *J. Electrochemical Society*, vol. 143, pp 1880–1886, 1996.

Application Library path: `Electrodeposition_Module/Tutorials/microconnector_bump_3d`

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 **Electrochemistry>Electrodeposition, Deformed Geometry>Electrodeposition, Tertiary with Supporting Electrolyte**.

3 Click **Add**.

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

5 In the **Concentrations (mol/m³)** table, enter the following settings:

c

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

7 Click **Add**.

8 Click **Done**.

GLOBAL DEFINITIONS

Load the model parameters from a text file.

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 `microconnector_bump_parameters.txt`.

DEFINITIONS

Load also some variables from a text file.

Variables 1

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `microconnector_bump_3d_variables.txt`.

GEOMETRY 1

Draw the geometry as a block and a cylinder (for the hole in the photoresist film). Round off the sharp corners of the film by using an additional cylinder and a torus. Finally, use the symmetry of the problem by cutting the model geometry in half.

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 `Ltot`.

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

5 In the **Height** text field, type `h2`.

6 Locate the **Position** section. In the **z** text field, type `h1`.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `L1/2`.

4 In the **Height** text field, type `h1`.

5 Locate the **Position** section. In the **x** text field, type `L3+L1/2`.

6 In the **y** text field, type `Ltot/2`.

7 Click **Build Selected**.

8 Click the **Transparency** button in the **Graphics** toolbar.

9 Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

Cylinder 2 (cyl2)

1 In the **Model Builder** window, click **Cylinder 2 (cyl2)**.

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

3 In the **Radius** text field, type `L1/2+r_edge`.

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

5 Locate the **Position** section. In the **z** text field, type `h1-r_edge`.

6 Click **Build Selected**.

Torus 1 (tor1)

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

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

3 In the **Major radius** text field, type `L1/2+r_edge`.

4 In the **Minor radius** text field, type `r_edge`.

5 Locate the **Position** section. In the **x** text field, type `L3+L1/2`.

6 In the **y** text field, type `Ltot/2`.

7 In the **z** text field, type `h1-r_edge`.

8 Click **Build Selected**.

Difference 1 (dif1)

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

2 Select the object **cyl2** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **tor1** only.

6 Click **Build Selected**.

Union 1 (uni1)

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

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

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

4 Clear the **Keep interior boundaries** check box.

5 Click **Build Selected**.

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 **xz-plane**.

4 In the **y-coordinate** text field, type `Ltot/2`.

Partition Objects 1 (par1)

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

2 Select the object **uni1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

5 Click **Build Selected**.

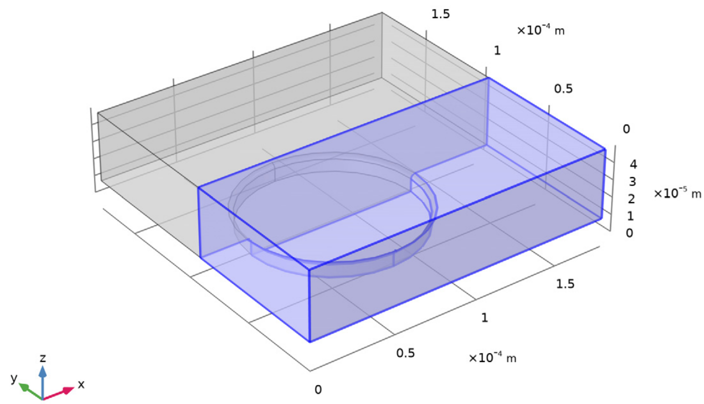
Delete Entities 1 (del1)

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

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **par1**, select Domain 1 only.



5 Click **Build Selected**.

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

DEFINITIONS

Add a number of selections to facilitate choosing various parts of the geometry when setting up the physics later.

Inlet

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

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

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

4 Select Boundary 1 only.

5 In the **Label** text field, type `Inlet`.

Outlet

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

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

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

4 Select Boundary 11 only.

5 In the **Label** text field, type `Outlet`.

Symmetry Walls

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

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

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

4 Select Boundaries 2 and 5 only.

5 In the **Label** text field, type `Symmetry Walls`.

Cathode

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

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

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

4 Select Boundary 8 only.

5 In the **Label** text field, type `Cathode`.

Bulk Electrolyte

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

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

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

4 Select Boundary 4 only.

5 In the **Label** text field, type `Bulk Electrolyte`.

Inlet + Bulk Electrolyte

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

2 In the **Settings** window for **Union**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

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

5 In the **Add** dialog box, in the **Selections to add** list, choose **Inlet** and **Bulk Electrolyte**.

6 Click **OK**.

7 In the **Settings** window for **Union**, type `Inlet + Bulk Electrolyte` in the **Label** text field.

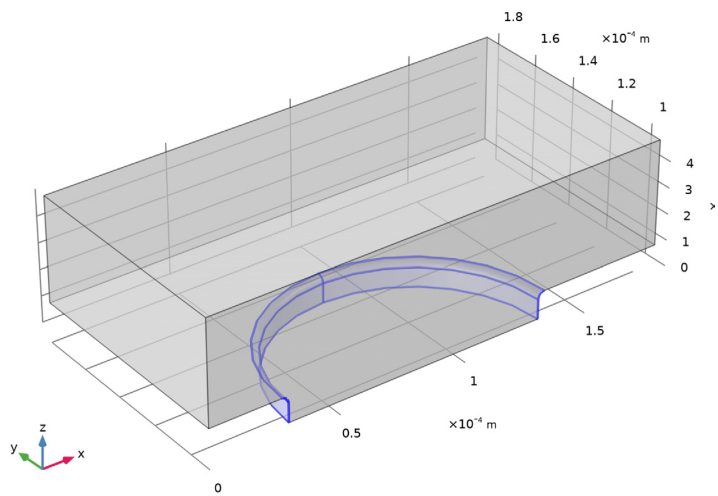
Insulator Hole Walls

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

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

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

4 Select Boundaries 6, 7, 9, and 10 only.



5 In the **Label** text field, type `Insulator Hole Walls`.

TERTIARY CURRENT DISTRIBUTION, NERNST-PLANCK (TCD)

Set up the current distribution and deforming geometry in the Electrodeposition, Tertiary with Supporting Electrolyte interface.

Electrolyte 1

Now set up the convection and diffusion part of the problem using Tertiary Current Distribution Nernst-Planck.

1 In the **Model Builder** window, under **Component 1 (comp1)>Tertiary Current Distribution, Nernst-Planck (tcd)** click **Electrolyte 1**.

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

5 Locate the **Solvent** section. From the σ_l list, choose **User defined**. In the associated text field, type `1`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

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

3 In the c text field, type `c_bulk`.

Concentration - Bulk

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

2 In the **Settings** window for **Concentration**, type `Concentration - Bulk` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Inlet + Bulk Electrolyte**.

4 Locate the **Concentration** section. Select the **Species c** check box.

5 In the $c_{0,c}$ text field, type `c_bulk`.

Electrolyte Potential - Bulk

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

2 In the **Settings** window for **Electrolyte Potential**, type `Electrolyte Potential - Bulk` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Bulk Electrolyte**.

Electrode Surface 1

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

2 In the **Settings** window for **Electrode Surface**, click to expand the **Dissolving-Depositing Species** section.

3 Click **Add**.

4 Clear the **Solve for surface concentration variables** check box.

5 Locate the **Boundary Selection** section. From the **Selection** list, choose **Cathode**.

6 Locate the **Electrode Phase Potential Condition** section. In the $\phi_{s,ext}$ text field, type `-0.45`.

Electrode Reaction 1

Now define the concentration-dependent kinetics for cupric ions on the cathode.

1 In the **Model Builder** window, click **Electrode Reaction 1**.

2 In the **Settings** window for **Electrode Reaction**, locate the **Stoichiometric Coefficients** section.

3 In the n text field, type `2`.

4 In the ν_c text field, type `-1`.

5 In the **Stoichiometric coefficients for dissolving-depositing species:** table, enter the following settings:

Species	Stoichiometric coefficient (1)
s1	1

6 Click to expand the **Reference Concentrations** section. In the table, enter the following settings:

Electrolyte species	Reference concentrations (mol/m^3)
c	c_bulk

7 Locate the **Electrode Kinetics** section. In the $i_{0,\mathrm{ref}}(T)$ text field, type `10[A/m^2]`.

8 In the α_a text field, type `1.5`.

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**.

Linear shape functions for concentration, electrolyte potential and electric potential are sufficient for this model setup. They also result in reduced computation time and memory requirements when compared to the default quadratic shape functions.

4 In the **Model Builder** window, click **Tertiary Current Distribution, Nernst-Planck (tcd)**.

5 In the **Settings** window for **Tertiary Current Distribution, Nernst-Planck**, click to expand the **Discretization** section.

6 From the **Concentration** list, choose **Linear**.

7 From the **Electrolyte potential** list, choose **Linear**.

8 From the **Electric potential** list, choose **Linear**.

LAMINAR FLOW (SPF)

Set up the flow.

Fluid Properties 1

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

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

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

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

Symmetry 1

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

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

3 From the **Selection** list, choose **Symmetry Walls**.

Inlet 1

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

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

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

4 Locate the **Velocity** section. Click the **Velocity field** button.

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

u_profile	x
0	y
0	z

Wall 2

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

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

3 From the **Selection** list, choose **Bulk Electrolyte**.

4 Click to expand the **Wall Movement** section. From the **Translational velocity** list, choose **Manual**.

5 Specify the $\mathbf{u}_{tr}$ vector as

u_bulk	x
0	y
0	z

Outlet 1

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

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

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

MULTIPHYSICS

Nondeforming Boundary 2 (ndbdg2)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary> Nondeforming Boundary**.
2 Select Boundaries 1, 2, 4, 5, and 11 only.
3 In the **Settings** window for **Nondeforming Boundary**, locate the **Nondeforming Boundary** section.
4 From the **Boundary condition** list, choose **Zero normal displacement**.

MESH 1

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Insulator Hole Walls**.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section.
7 Select the **Maximum element size** check box. In the associated text field, type `3E-6`.

Size

1 In the **Model Builder** window, click **Size**.
2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.
3 In the **Maximum element size** text field, type `8E-6`.
4 In the **Maximum element growth rate** text field, type `1.1`.
5 In the **Curvature factor** text field, type `0.7`.

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 **Insulator Hole Walls**.
4 Click to expand the **Advanced Settings** section. From the **Interpolation method** list, choose **Transfinite in 3D**.

Convert 1

1 In the **Mesh** toolbar, click **Modify** and choose **Convert**.
2 In the **Settings** window for **Convert**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Insulator Hole Walls**.
5 Click **Build Selected**.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.
2 In the **Settings** window for **Free Tetrahedral**, click **Build Selected**.

Boundary Layers 1

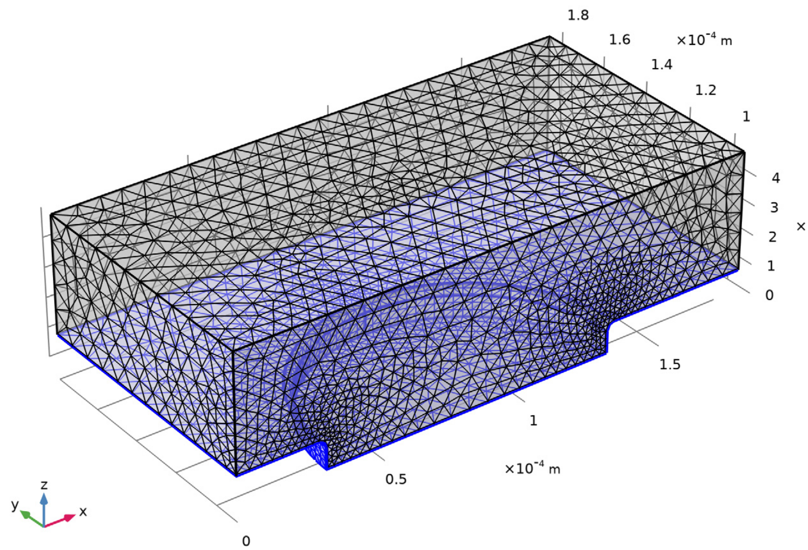
1 In the **Mesh** toolbar, click **Boundary Layers**.
2 In the **Settings** window for **Boundary Layers**, click to expand the **Corner Settings** section.
3 From the **Handling of sharp edges** list, choose **Trimming**.
4 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.
2 Select Boundaries 3 and 6–10 only.
3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.
4 In the **Number of layers** text field, type `1`.
5 From the **Thickness specification** list, choose **First layer**.
6 In the **Thickness** text field, type `0.5E-6`.
7 Click **Build All**.

Your finished mesh should now look like this:

8 Right-click **Boundary Layer Properties** and choose **Plot**.



ROOT

Solve the problem using three different steps. The first study step solves for the flow profile at $t = 0$.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

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

STUDY 1

Step 1: Stationary

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

2 Select the **Modify model configuration for study step** check box.

3 In the tree, select **Component 1 (comp1)>Tertiary Current Distribution, Nernst-Planck (tcd)**.

4 Click **Disable in Solvers**.

5 In the tree, select **Component 1 (comp1)>Multiphysics>Nondeforming Boundary 1 (ndbdg1)**, **Component 1 (comp1)>Multiphysics>Deforming Electrode Surface 1 (desdg1)**, and **Component 1 (comp1)>Multiphysics>Nondeforming Boundary 2 (ndbdg2)**.

6 Click **Disable in Solvers**.

Step 2: Stationary 2

Add a second step to solve for the concentration and current distribution at $t = 0$.

1 In 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 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Laminar Flow (spf)**.

5 Right-click and choose **Disable in Solvers**.

6 In the tree, select **Component 1 (comp1)>Multiphysics>Nondeforming Boundary 1 (ndbdg1)**, **Component 1 (comp1)>Multiphysics>Deforming Electrode Surface 1 (desdg1)**, and **Component 1 (comp1)>Multiphysics>Nondeforming Boundary 2 (ndbdg2)**.

7 Right-click and choose **Disable in Solvers**.

Step 3: Time Dependent

Add a third and final time dependent step to solve for the problem during 120 s. The results from the first two steps will be used as initial values automatically.

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

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

3 In the **Output times** text field, type `range(0,10,120)`.

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)>Time-Dependent Solver 1** node, then click **Segregated 1**.

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

5 In the **Maximum number of iterations** text field, type `15`.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1>Segregated 1** node, then click **Velocity u, Pressure p**.

7 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

8 From the **Jacobian update** list, choose **On first iteration**.

9 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Time-Dependent Solver 1>Segregated 1** click **Merged Variables**.

10 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

11 From the **Termination technique** list, choose **Tolerance**.

12 In the **Model Builder** window, click **Study 1**.

13 In the **Settings** window for **Study**, locate the **Study Settings** section.

14 Clear the **Generate default plots** check box.

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

RESULTS

Concentration

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

Surface 1

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

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

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

4 In the **Concentration** toolbar, click **Plot**.

5 Click the **Transparency** button in the **Graphics** toolbar.

Concentration

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

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

3 From the **Time (s)** list, choose **0**.

4 In the **Concentration** toolbar, click **Plot**.

5 Right-click **Concentration** and choose **Duplicate**.

Microconnector Bump Surface

1 In the **Model Builder** window, expand the **Results>Concentration 1** node, then click **Concentration 1**.

2 In the **Settings** window for **3D Plot Group**, type `Microconnector Bump Surface` in the **Label** text field.

3 Locate the **Data** section. From the **Time (s)** list, choose **120**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

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

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

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

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

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

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

7 Click **Define custom colors**.

8 Set the RGB values to 128, 0, and 0, respectively.

9 Click **Add to custom colors**.

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

11 Click to expand the **Quality** section. From the **Resolution** list, choose **Fine**.

Selection 1

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

2 Select Boundary 8 only.

Surface 2

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

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

3 From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

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

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

7 Locate the **Quality** section. From the **Resolution** list, choose **Fine**.

Selection 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `3, 6-7, 9-10` in the **Selection** text field.

5 Click **OK**.

RESULTS

Surface 2

1 In the **Model Builder** window, collapse the **Results>Microconnector Bump Surface>Surface 2** node.

2 Right-click **Surface 2** and choose **Duplicate**.

Surface 3

1 In the **Model Builder** window, expand the **Results>Microconnector Bump Surface>Surface 3** node, then click **Surface 3**.

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

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

Selection 1

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

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `8` in the **Selection** text field.

6 Click **OK**.

Line 1

1 In the **Model Builder** window, right-click **Microconnector Bump Surface** and choose **Line**.

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

3 From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

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

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

Selection 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2-3, 7, 9, 11-13, 16, 20-22, 24` in the **Selection** text field.

5 Click **OK**.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **Player**.

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

3 From the **Subject** list, choose **Microconnector Bump Surface**.

4 Locate the **Frames** section. In the **Number of frames** text field, type `13`.

5 In the **Frame number** text field, type `13`.

6 Click the **Play** button in the **Graphics** toolbar.