Created in COMSOL Multiphysics 5.6

Electrodeposition of a Microconnector Bump in 2D

Introduction

This model demonstrates the impact of convection and diffusion on the transport-limited electrodeposition of a copper microconnector bump (metal post). Microconnector bumps are used in various types of electronic applications for interconnecting components, for instance liquid crystal displays (LCDs) and driver chips.

The location of the bumps on the electrode surface is controlled by the use of a photoresist mask. Control of the current distribution in terms of uniformity and shape is important for ensuring the shape and resulting reliability of the interconnector bumps.

The cell is running at a high overpotential so the deposition rate is governed by the transport rate of the depositing ion in the electrolyte. A result of this operating condition is that the electric potentials in the electrolyte and electrode need not be modeled to determine the current distribution on the bump.

The model is based on a paper by Kondo and others (Ref. 1).

For an extension of this model to 3D, including the microconnector shape evolution over time, see Electrodeposition of a Microconnector Bump with Deforming Geometry in 3D.

Model Definition

The model geometry is shown in Figure 1. The rectangular hole in the photoresist film is significantly longer in one direction so the model can be represented in 2D.

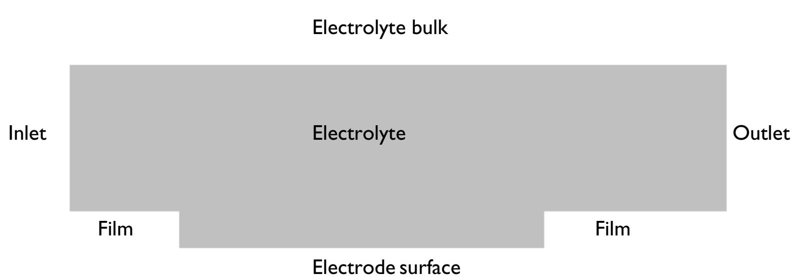

Figure 1: Model geometry.

FLOW MODEL

Assume laminar incompressible flow conditions (described by the Navier-Stokes equations), and use the Laminar Flow interface to model the electrolyte flow.

Apply a linear velocity profile (Couette flow) on the left vertical inlet boundary, ranging from zero at the photoresist surface up to a specified bulk flow velocity, depending on the Peclet number (see Equation 1). Set the velocity at the top boundary equal to the bulk velocity using a moving wall condition. Apply a pressure conditions on the right vertical outlet boundary.

The default no slip condition applies to all other boundaries.

MASS TRANSPORT MODEL

Assume the electrolyte to be diluted so that the transport of copper ions can be described using Fick's law. Use the Transport of Diluted Species interface coupled to the flow velocity in the Laminar Flow interface to model the mass transport.

Apply a fixed bulk concentration on the left inlet and top boundaries. Use outflow conditions on the right outlet boundary.

The cell is operating at high overpotential and is limited by transport of copper ions to the electrode surface. This is described by applying a zero concentration of ions at the electrode surface.

The default no-flux condition applies to all other boundaries.

THE PECLET NUMBER

The current density distribution on the electrode surface for this cell is governed by the Peclet number, which is a dimensionless quantity that relates the convective transport to the diffusion. For this cell, use the following definition of the Peclet number:

$$\mathrm{Pe} = \frac{h u_{\mathrm{Pe}}}{D} \tag{1}$$

where h is the thickness of the photoresist film, u_{Pe} is the velocity in the x direction at a height of 10 µm at the inlet boundary, and D is the diffusion coefficient of the copper ions.

STUDY SETTINGS

Use a Parametric Sweep of a Stationary Study to solve the problem for two different Peclet numbers (1.31 and 41.6).

Results and Discussion

Figure 2 and Figure 3 show the streamlines and velocity magnitudes for the two Peclet numbers. The shape of the streamlines are quite similar, although the velocity magnitudes differ. Vortices that are formed close the corners both on the upstream and downstream sides.

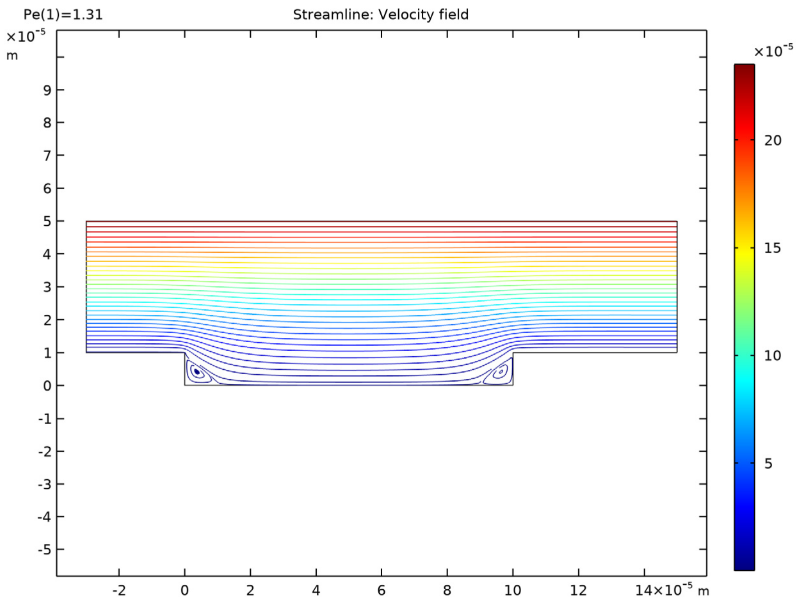

Figure 2: Velocity streamlines for Pe = 1.31

.

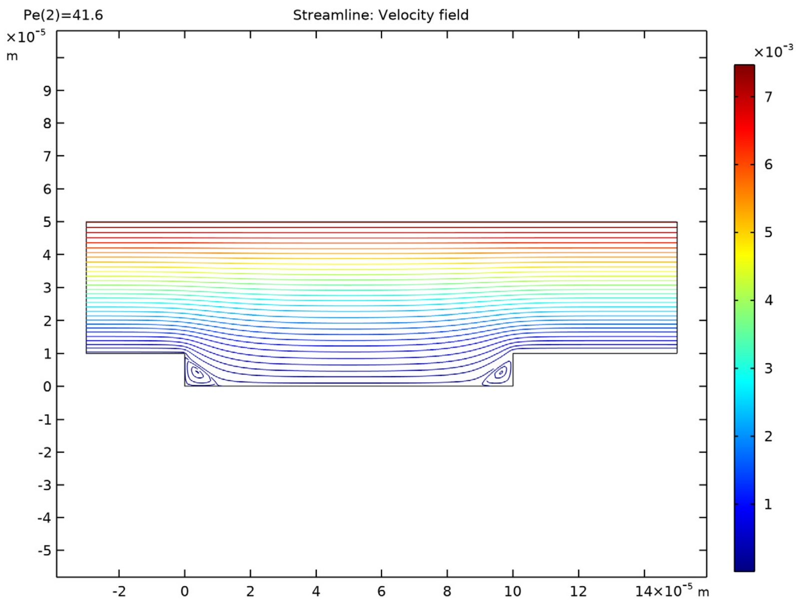

Figure 3: Velocity stream lines for Pe = 41.6.

Figure 4 and Figure 5 show the concentrations for the two different Peclet number cases. The concentration profiles are quite different, with the depletion zone of copper ions extending farther into the electrolyte for the lower Peclet number.

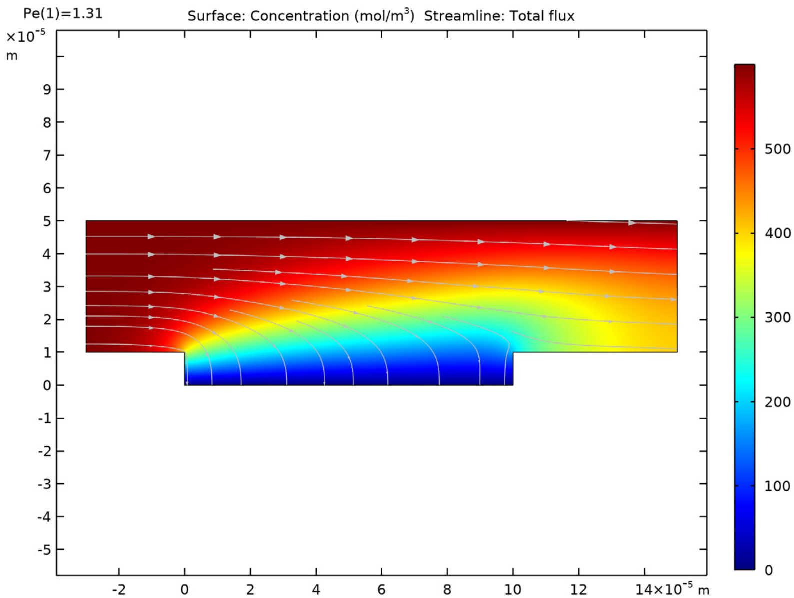

Figure 4: Copper ion concentration for Pe = 1.31.

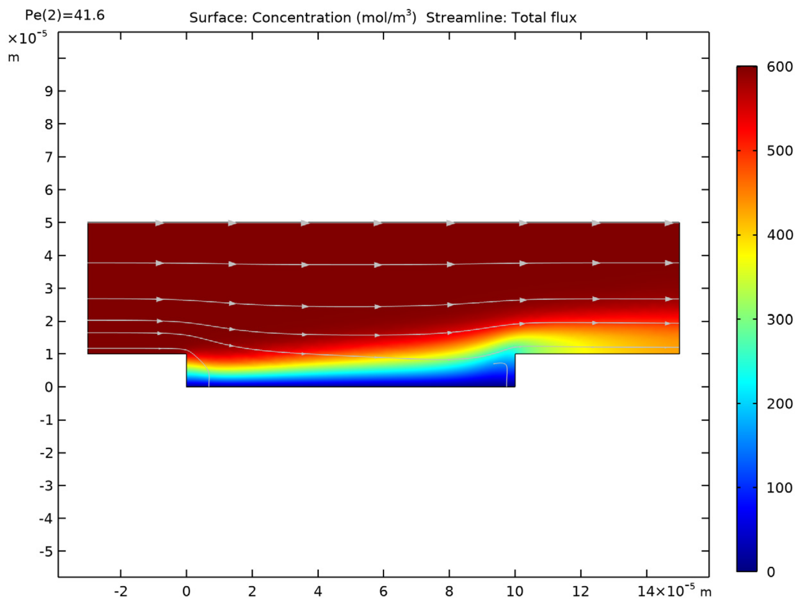

Figure 5: Copper ion concentration for Pe = 41.6.

The fluxes of the copper ions over the electrode surface in combination with Faraday's law can be used to calculate the local current density on the electrode surface:

$$i_{\mathrm{loc}} = nFN_{\mathrm{Cu}^{-}}$$

where n is the number of electrons in the electrode reactions, F is Faraday's constant, and $N_{\mathrm{Cu-}}$ is the normal flux of copper ions over the electrode surface.

Figure 6 shows the local current density for the two Peclet numbers. The higher flow velocity for the higher Peclet number increases the local current density significantly due to the increased transport velocity of copper ions.

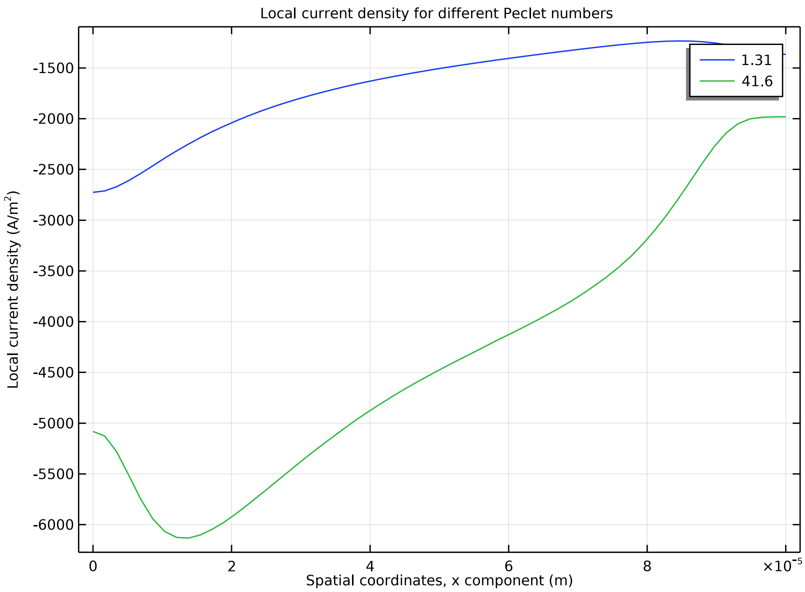

Figure 6: Local current densities at the electrode surface

In Figure 7 the local current densities have been normalized to their maximum values. The lower Peclet number results in a more uniform current density distribution on the electrode surface.

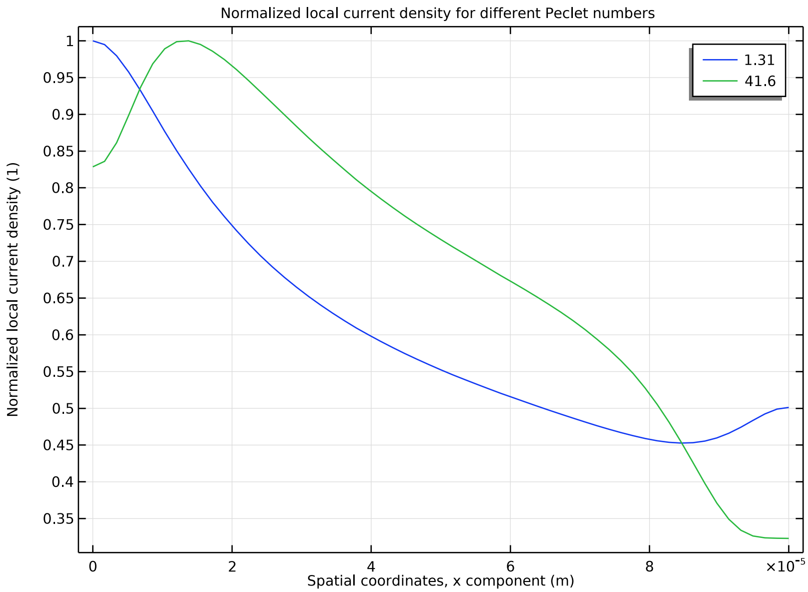

Figure 7: Local current density at the electrode surface normalized by the maximum value.

Reference

1. 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/Verification_Examples/microconnector_bump_2d`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.
2 In the **Select Physics** tree, select **Chemical Species Transport>Transport of Diluted Species (tds)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Stationary**.
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`.

GEOMETRY 1

Create the geometry by drawing two rectangles. The union of the two defines the final geometry.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `L1+L2+L3`.
4 In the **Height** text field, type `h2`.
5 Locate the **Position** section. In the **x** text field, type `-L3`.
6 In the **y** text field, type `h1`.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `L1`.

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

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 both objects.

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

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

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

DEFINITIONS

Add some variable expressions from a text file.

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local 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_2d_variables.txt`.

Note that the `i_loc_norm` variable (that you will use for results processing) is colored orange. It needs a maximum operator that you have yet to define.

Maximum 1 (maxop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.

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

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

4 Select Boundary 5 only.

Variables 1

If you return to the variables again, the `i_loc_norm` variable should now have turned black.

LAMINAR FLOW (SPF)

Now start setting up the physics, beginning with the fluid flow model.

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`.

Inlet 1

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

2 Select Boundary 1 only.

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

4 Click the **Velocity field** button.

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

u_profile	x
0	y

Wall 2

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.

4 From the **Translational velocity** list, choose **Manual**.

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

u_bulk	x
0	y

Outlet 1

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

2 Select Boundary 8 only.

3 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.

4 Select the **Normal flow** check box.

Initial Values 1

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

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

3 Specify the **u** vector as

u_bulk	x
0	y

TRANSPORT OF DILUTED SPECIES (TDS)

Transport Properties 1

Now set up the mass transport model of the ions. Start by coupling the mass transport to the fluid velocity in the flow model, and specify the diffusion coefficient.

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

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

3 From the **u** list, choose **Velocity field (spf)**.

4 Locate the **Diffusion** section. In the D_c text field, type `D`.

Concentration 1

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

2 Select Boundaries 1 and 3 only.

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

4 Select the **Species c** check box.

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

Outflow 1

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

2 Select Boundary 8 only.

Concentration 2

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

2 Select Boundary 5 only.

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

4 Select the **Species c** check box.

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`.

STUDY 1

The model is now ready for solving. Create a parametric sweep to solve for two different Peclet numbers.

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Pe (Peclet number)	`1.31 41.6`	

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

RESULTS

Concentration (tds)

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

One concentration plot is created by default (Figure 5). To reproduce the concentration plot for the lower Peclet number (Figure 4), do the following.

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

3 From the **Parameter value (Pe)** list, choose **1.31**.

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

2D Plot Group 4

Create a new plot group with a streamline plot to reproduce Figure 2 and Figure 3.

In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

Streamline 1

1 Right-click **2D Plot Group 4** 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)>Laminar Flow>Velocity and pressure>u,v - Velocity field**.

3 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.

4 In the **Separating distance** text field, type `0.01`.

Color Expression 1

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

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Laminar Flow>Velocity and pressure>spf.U - Velocity magnitude - m/s**.

3 In the **2D Plot Group 4** toolbar, click **Plot**.

2D Plot Group 4

1 In the **Model Builder** window, click **2D Plot Group 4**.

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

3 From the **Parameter value (Pe)** list, choose **1.31**.

4 In the **2D Plot Group 4** toolbar, click **Plot**.

1D Plot Group 5

Now plot the current density on the electrode surface using the defined i_loc variable (Figure 6).

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Local current density for different Peclet numbers`.

Line Graph 1

1 Right-click **1D Plot Group 5** and choose **Line Graph**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>i_loc - Local current density - A/m^2**.

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

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

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

7 In the **1D Plot Group 5** toolbar, click **Plot**.

1D Plot Group 6

Proceed in a similar way as above to reproduce the normalized current density plot (Figure 7).

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

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

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Normalized local current density for different Peclet numbers`.

Line Graph 1

1 Right-click **1D Plot Group 6** and choose **Line Graph**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>i_loc_norm - Normalized local current density**.

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

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

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

7 In the **1D Plot Group 6** toolbar, click **Plot**.