Created in COMSOL Multiphysics 5.6

# Electrochemical Polishing

## *Introduction*

This example illustrates the principle of electrochemical polishing. The simplified 2D model geometry consists of two electrodes and an intermediate electrolyte domain. The positive electrode has a protrusion, representing a surface defect. The purpose of the application is to examine how this protrusion and the surrounding electrode material are depleted over a period of time.

## *Model Definition*

The potential drop over the electrodes is 30 V, and the electrolyte has a conductivity of 10 S/m.

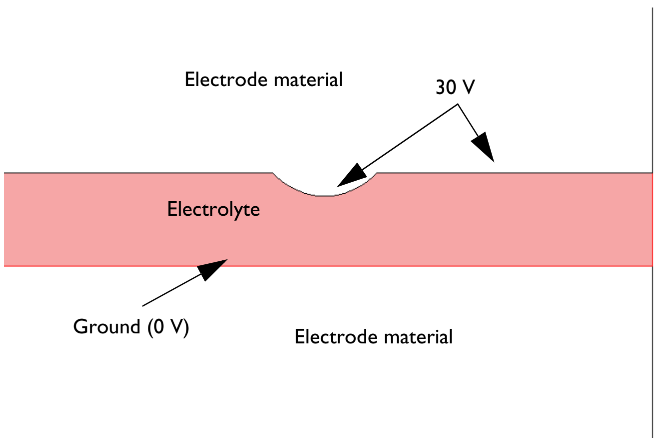



*Figure 1: Model geometry.*

Modeling the depletion of the positive electrode requires a moving boundary because the geometry changes and the current density distribution with it. A simple model for the depletion is based on the assumption that the depletion rate is proportional to the normal current density at the electrode surface. The velocity, $U$, normal to the mesh at the electrode surface then becomes

$$U = -KJ_n \qquad (1)$$

where $K$ is the coefficient of proportionality, and $J_n$ is the normal current density. In this example, $K = 10^{-11}\ \mathrm{m^3/As}$.

The part of the electrode and electrolyte that the model includes is about 3 mm wide and the distance between the electrodes is 0.4 mm.

## *Results and Discussion*

After a period of 10 s, the protrusion is somewhat smoothed out, and a significant portion of the positive electrode has been depleted.

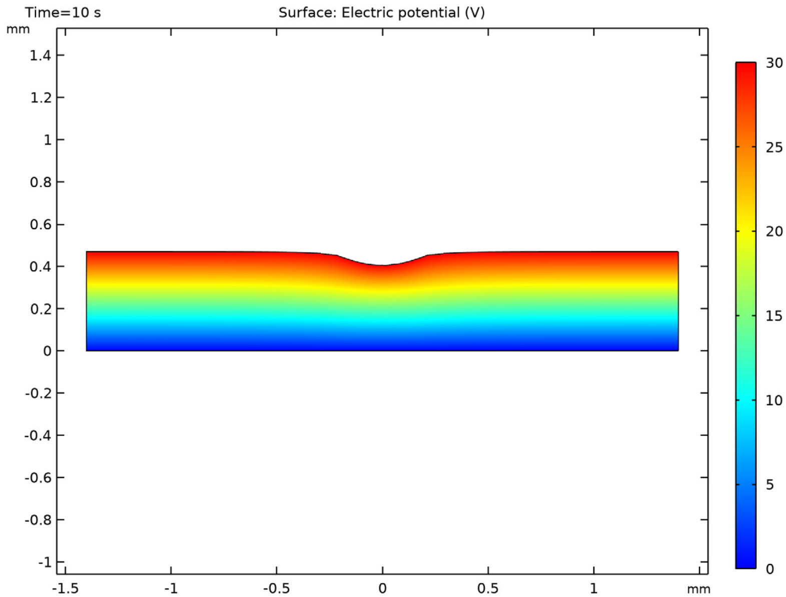



*Figure 2: Potential distribution and electrode depletion after 10 s.*

Using Equation 1, the expected total depletion increment, $d(\Delta t = 10\text{ s})$, over the simulated time interval can be estimated as

$$d(\Delta t) = |U|\Delta t = K|J_n|\Delta t = \left(10^{-11}\frac{\text{m}^3}{\text{As}}\right)\cdot\left(10^6\frac{\text{A}}{\text{m}}\right)\cdot(10^1\text{s}) = 10^{-4}\text{ m} \qquad (2)$$

This estimate agrees with the maximum value for the $y$-displacement obtained for the model, showing that the approximate formula (which does not take effects from the curved boundary into account) is in fact very accurate.

## *Notes About the COMSOL Implementation*

This application uses the Electric Currents and Deformed Geometry interfaces. The variable for the normal current density defines the mesh velocity. The dynamics in this example is quasi static in nature, and the time dependence only enters in the depletion (removal of material) of the electrode.

**Application Library path:** `COMSOL_Multiphysics/Electromagnetics/electrochemical_polishing`

## *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 **Mathematics>Deformed Mesh>Deformed Geometry (dg)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **AC/DC>Electric Fields and Currents>Electric Currents (ec)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Time Dependent**.
8 Click **Done**.

### 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 In the table, enter the following settings:

| Name | Expression | Description |
|---|---|---|
| K | 1e-11[m^3/(A*s)] | Coefficient of proportionality |

### 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 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 `2.8`.

4 In the **Height** text field, type `0.4`.

5 Locate the **Position** section. In the **x** text field, type `-1.4`.

6 Click **Build Selected**.

*Circle 1 (c1)*

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

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

3 In the **Radius** text field, type `0.3`.

4 Locate the **Position** section. In the **y** text field, type `0.6`.

*Difference 1 (dif1)*

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

2 Select the object **r1** only to add it to the **Objects to add** list.

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

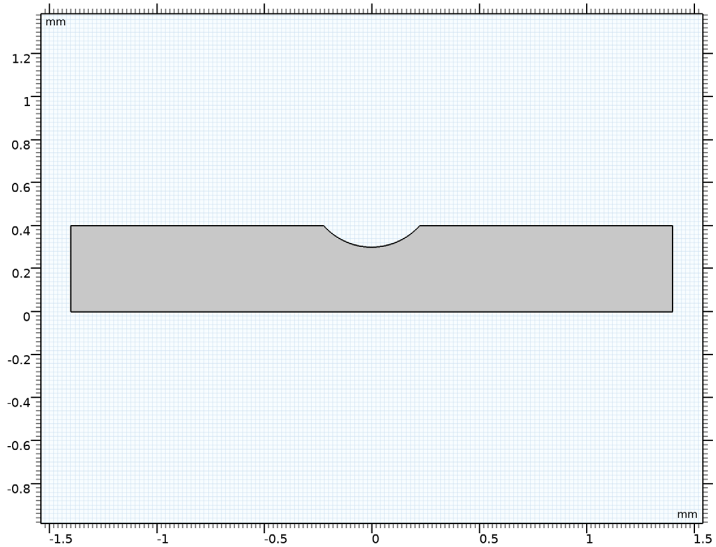
4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the object **c1** only.

6 Click **Build Selected**.

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

The model geometry is now complete.




Before turning to the **Deformed Geometry** interface settings, define variables for the local displacement components.

## DEFINITIONS

### *Variables 1*

**1** In the **Home** toolbar, click a= **Variables** and choose **Local Variables**.

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

**3** In the table, enter the following settings:

| Name | Expression | Unit | Description |
|---|---|---|---|
| dx | x-Xg | m | x-displacement |
| dy | y-Yg | m | y-displacement |

Here, Xg and Yg are geometry-frame coordinates corresponding to $x$ and $y$.

## DEFORMED GEOMETRY (DG)

### *Free Deformation 1*

**1** In the **Model Builder** window, under **Component 1 (comp1)** right-click **Deformed Geometry (dg)** and choose **Free Deformation**.

**2** Select Domain 1 only.

*Prescribed Mesh Velocity 1*

**1** In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Velocity**.

**2** Select Boundaries 1 and 5 only.

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

**4** Clear the **Prescribed Y velocity** check box.

*Prescribed Normal Mesh Velocity 1*

**1** In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Normal Mesh Velocity**.

**2** Select Boundaries 3, 4, 6, and 7 only.

**3** In the **Settings** window for **Prescribed Normal Mesh Velocity**, locate the **Normal Mesh Velocity** section.

**4** In the $v_n$ text field, type `-K*(-ec.nJ)`.

### ELECTRIC CURRENTS (EC)

**1** In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

**2** In the **Settings** window for **Electric Currents**, click to expand the **Equation** section.

**3** From the **Equation form** list, choose **Stationary**.

With this setting you specify that the current distribution can be regarded as stationary on the time scale determined by the depletion rate.

*Current Conservation 1*

**1** In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents (ec)** click **Current Conservation 1**.

**2** In the **Settings** window for **Current Conservation**, locate the **Constitutive Relation Jc-E** section.

**3** From the σ list, choose **User defined**. In the associated text field, type `10`.

*Electric Potential 1*

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

**2** Select Boundaries 3, 4, 6, and 7 only.

**3** In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

**4** In the $V_0$ text field, type `30`.

*Ground 1*

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

2 Select Boundary 2 only.

*Electric Insulation 1*

For the left and right boundaries, the default boundary condition is a good approximation if you want to simulate that the electrodes are extended indefinitely in both directions.

## MESH 1

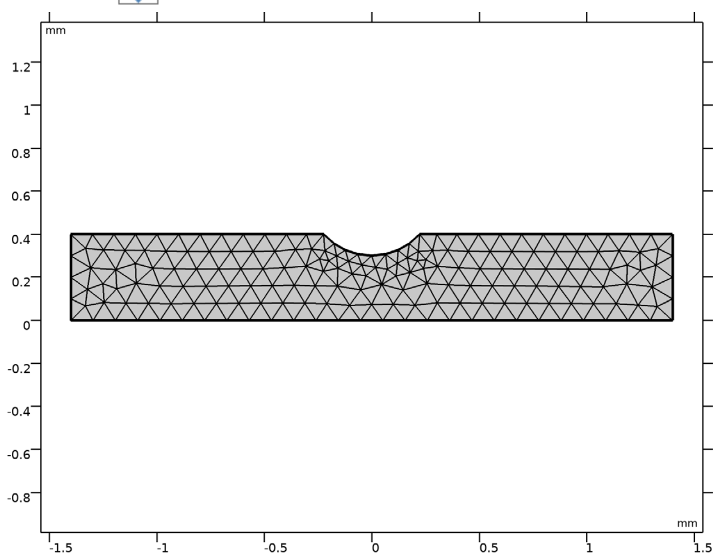
1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

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

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

4 Click **Build All**.

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




## STUDY 1

*Step 1: Time Dependent*

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

4 In the **Home** toolbar, click **Compute**.

## RESULTS

### *Electric Potential (ec)*

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

The default plot shows the potential field at the end of the simulation interval; compare with Figure 2.

### *2D Plot Group 2*

Next, plot the current distribution.

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

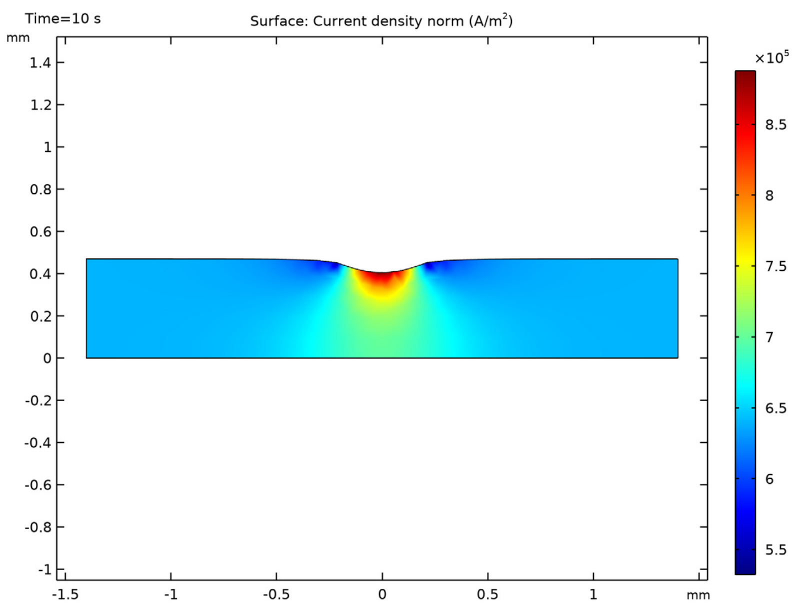
### *Surface 1*

**1** Right-click **2D Plot Group 2** and choose **Surface**.

**2** In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Electric Currents> Currents and charge>ec.normJ - Current density norm - A/m²**.

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

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




The maximum current density appears to be of the order of $10^6$ A/m$^2$.

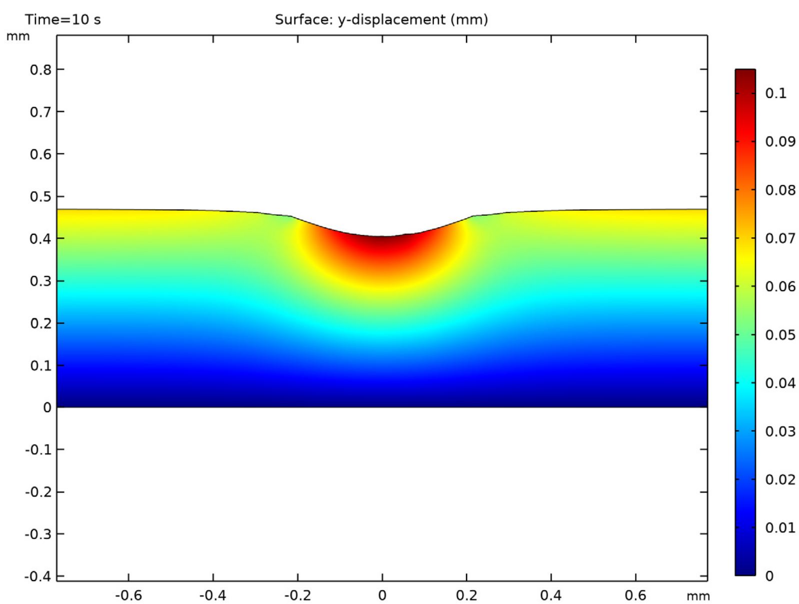
To see the magnitude of the depletion in the $y$ direction more easily, plot the $y$-component of the mesh displacement.

### *2D Plot Group 3*

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

### *Surface 1*

1 Right-click **2D Plot Group 3** and choose **Surface**.
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>dy - y-displacement - m**.
3 In the **2D Plot Group 3** toolbar, click **Plot**.
4 Click the **Zoom In** button in the **Graphics** toolbar.




The maximum value for the $y$-displacement is approximately 0.1 mm, which agrees with the value calculated in Equation 2.