Created in COMSOL Multiphysics 6.1

Electrochemical Machining of a Microbore

Introduction

Electrochemical Machining (ECM) is based on the anodic dissolution of a metallic work piece. ECM has the advantages of a relatively low operating temperature (<80 °C) in combination with a process basically free of mechanical forces.

Microbores are used in various precision applications, such as hydraulic systems and fuel injectors. For a direct-injection fuel system the shape of the injection hole is of paramount importance for the combustion process.

This example models a primary current distribution and the resulting shape evolution of a bore during electrochemical machining.

Model Definition

The model geometry is shown in Figure 1. The electrolyte is fed into the cell from the top boundary at high velocity and exits through the bore at the bottommost boundary. A potential difference of 20 V is applied between the anode work piece and cathode tool, causing gas evolution on the cathode and metal dissolution or removal on the anode. An insulating layer is placed on top of the anode.

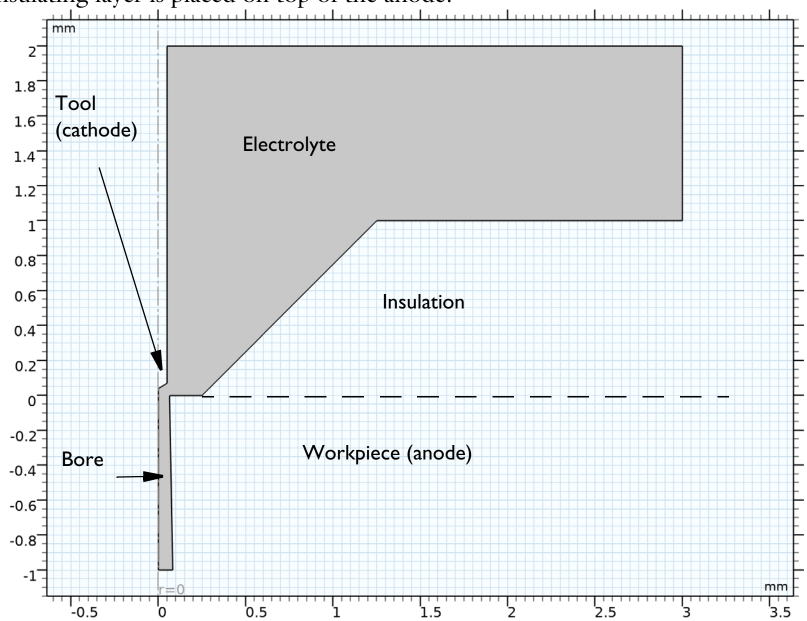


Figure 1: Model geometry. The geometry is symmetrical around the axis r=0.

Since only small potential gradients are expected in the electrodes due to the high metal conductivities, the electrode domains are not included in the model. The insulating layer is electrochemically inert and is hence not included either. The only modeled domain is the electrolyte.

Due to the high velocity of the flowing electrolyte, causing turbulent mixing, the electrolyte conductivity is assumed to be constant (7 S/m). In addition, the activation potentials of the electrode reactions are assumed to negligible so that a primary current distribution can be used to model the cell. The cathode is grounded and the anode is set to 20 V. The same equilibrium potential is used for both electrodes.

The anode electrode reaction gives rise to a boundary geometry deformation velocity v_n (m/s) in the normal direction defined as

$$v_n = \nu_{\mathrm{eff}} \frac{M}{\rho n F} i_{\mathrm{loc}}$$

where M (kg/mol) is the molar mass and ρ (kg/m^3) the density of the dissolving metal, n the number of electrons involved in the dissolution reaction, F (C/mol) Faraday's constant, and i_{loc} (A/m^2) is the local current density of the electrode reaction. For a primary current distribution i_{loc} can be calculated as

$$i_{\mathrm{loc}} = \mathbf{i}_l \cdot \mathbf{n}$$

where $\mathbf{i}_l$ (A/m^2) is the electrolyte current density vector and $\mathbf{n}$ the boundary normal.

v_{eff} is the effective stoichiometric coefficient for the metal in the anode electrode reaction. From experiments it has been seen that no metal dissolution occurs for current densities below 10 A/cm^2. v_{eff} is therefore defined as:

$$\nu_{\mathrm{eff}} = 1 \qquad \mathbf{i}_l \cdot \mathbf{n} \ge 10 \text{ A/cm}^2$$
$$\nu_{\mathrm{eff}} = 0 \qquad \mathbf{i}_l \cdot \mathbf{n} < 10 \text{ A/cm}^2$$

Note that a current efficiency lower than unity is an indication of multiple competing electrode reactions being present, for instance oxygen evolution. A more detailed model would consider a secondary current distribution including multiple, competing, electrode reactions.

All other boundaries but the anode are set to zero geometrical deformation in the normal direction.

The electrochemical machining of the bore is simulated in a time-dependent study for 1 s.

Figure 2 shows the electrolyte potential distribution in the cell and the corresponding current density streamlines at $t = 1$ s. The current is concentrated around the top of the workpiece.

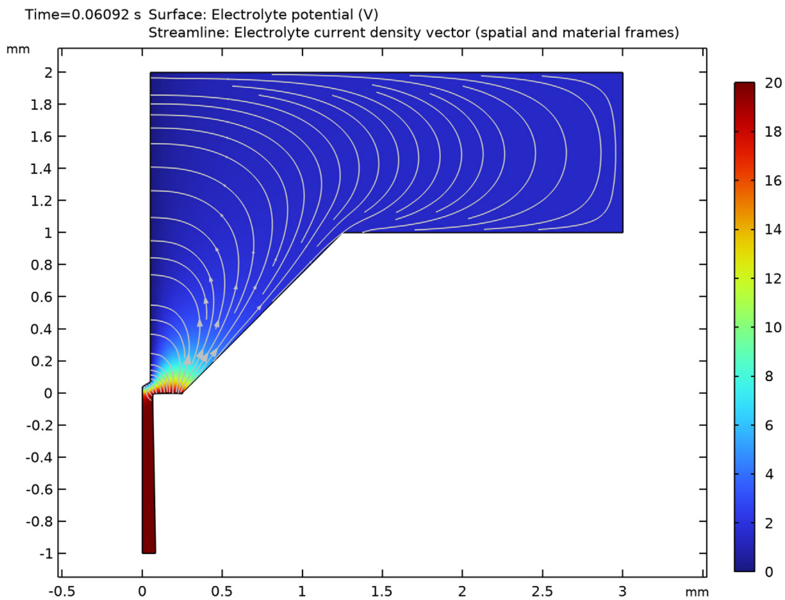


Figure 2: Electrolyte potential and electrolyte current density streamlines at $t = 1$ s.

Figure 3 shows the profile of the workpiece at $t = 0$, 0.5, and 1 s. Most of the material removal occurs in the region located closest to the cathode tool, which is located outside the upper-left corner of the figure, but the material removal also reaches a less pronounced local maximum toward the right. This latter local maximum is an effect of the insulating layer.

Figure 4 shows a revolved plot of the current density at the workpiece surface at $t = 1$ s. For the lower parts of the bore the current density does not exceed the threshold current density for material removal (10 A/cm^2).

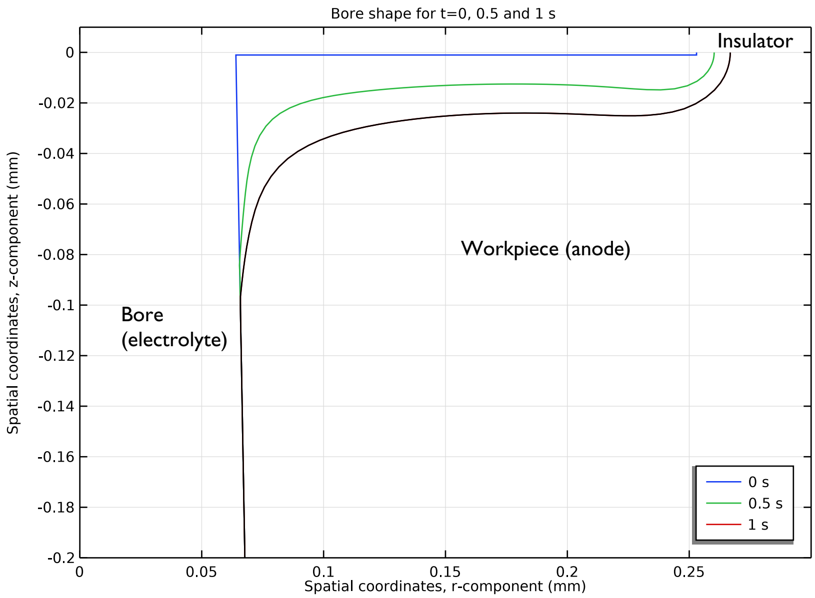


Figure 3: Geometry deformation of the bore mouth at t = 0, 0.5 and 1 s.

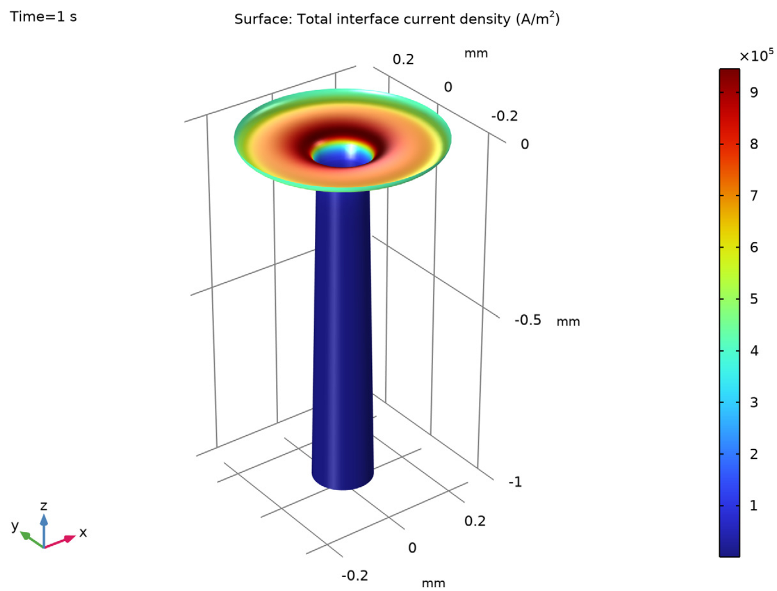


Figure 4: Anode current density and geometry shape at t = 1 s.

Notes About the COMSOL Implementation

A small rectangular seed deformation along the anode at z=0, extending into/below the insulating layer is introduced in the initial geometry (see Figure 3, t = 0 graph). The reason for this is to avoid singularities in the contact point between the electrolyte and the insulating boundary, and to enforce a zero vertical deformation along the insulator/anode contact line (the dotted line in Figure 1).

The default Nondeforming Boundary node is modified to use a Zero normal displacement type of boundary condition for the geometry deformation. This ensures that all corners are fixed in the geometry.

Reference

1. M. Hackert-Oschätzchen, M. Kowalick, G. Meichsner, and A. Schubert, "Multiphysics Simulation of the Electrochemical Finishing of Micro Bores," *Proceedings of the 2012 COMSOL Conference in Milan*, 2012.

Application Library path: `Electrodeposition_Module/Tutorials/ecm_microbore`

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

2 In the **Select Physics** tree, select **Electrochemistry>Electrodeposition, Deformed Geometry>Electrodeposition, Primary**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

6 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 `ecm_microbore_parameters.txt`.

GEOMETRY 1

Set the default length unit of the model to millimeters.

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)

Now draw the geometry as a set of rectangles and polygons.

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 3.

4 In the **Height** text field, type 3.

5 Locate the **Position** section. In the **z** text field, type -1.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

r (mm)	z (mm)
0	2
0.05	2
0.05	0.07
0	0.04

Polygon 2 (pol2)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

r (mm)	z (mm)
a1	-1
a2	0
3	0
3	-1

Polygon 3 (pol3)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

r (mm)	z (mm)
0.25	0
1.25	1
3	1
3	0

Difference 1 (dif1)

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

2 Select the object **r1** only.

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

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

5 Select the objects **pol1**, **pol2**, and **pol3** only.

6 Click **Build Selected**.

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 `0.25+s_seed*3-a2`.

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

5 Locate the **Position** section. In the **r** text field, type `a2`.

6 In the **z** text field, type `-s_seed`.

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.

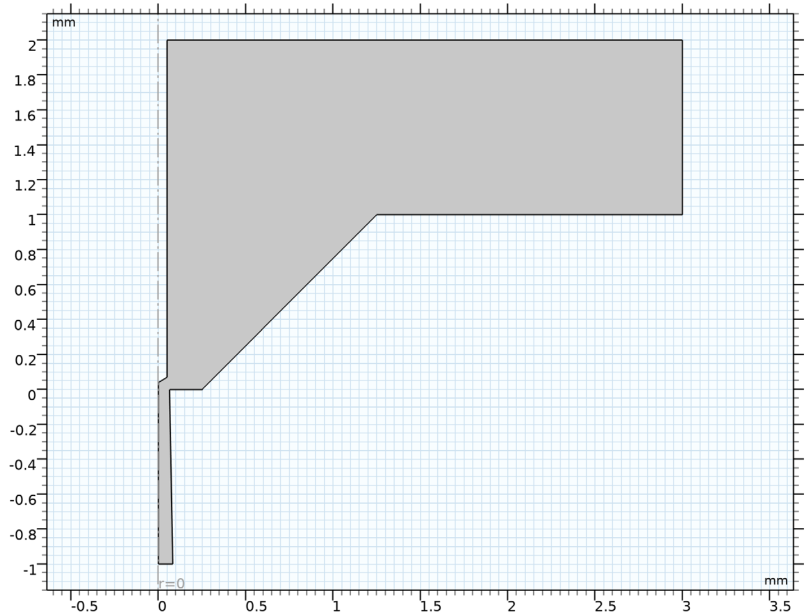
3 In the **Settings** window for **Union**, locate the **Union** section.

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

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

6 Click **Build All Objects**.

Your finished geometry should now look like this:



DEFINITIONS

Now define some selections of the geometry. They will be used later when setting up the physics.

Anode

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 Click **Paste Selection**.

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

6 Click **OK**.

7 In the **Settings** window for **Explicit**, type `Anode` in the **Label** text field.

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 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `3, 4` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Explicit**, type `Cathode` in the **Label** text field.

Cathode tip

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 3 only.

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

Add a step function that will be used later when defining the 10 A/cm^2 threshold current density for material removal.

Step 1 (step1)

In the **Definitions** toolbar, click f(X) **More Functions** and choose **Step**.

PRIMARY CURRENT DISTRIBUTION (CD)

Electrolyte 1

Now set up the physics. Start with the electrolyte.

1 In the **Model Builder** window, under **Component 1 (comp1)>Primary Current Distribution (cd)** click **Electrolyte 1**.

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

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

Electrode Surface 1

Use an Electrode Surface to describe the anode. Add a Dissolving-Depositing species to define the deformation velocity.

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

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

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

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

5 Click to expand the **Dissolving-Depositing Species** section. Click **Add**.

6 In the table, enter the following settings:

Species	Density (kg/m^3)	Molar mass (kg/mol)
s1	rho_Me	M_Me

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

Electrode Reaction 1

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 `n`.

The stoichiometric coefficient will determine the direction and velocity of the geometry deformation. Use the step function you defined before to set the velocity to zero below a certain threshold current density.

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

Species	Stoichiometric coefficient (1)
s1	1*step1(cd.itot/i_threshold-1)

Electrode Surface 2

Use a second Electrode Surface feature to define the cathode.

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

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

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

MULTIPHYSICS

Nondeforming Boundary 1 (ndbdg1)

The following applies a stronger constraint (than the default condition) for the planar nondepositing walls in order to enforce a zero boundary movement in the normal direction.

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Nondeforming Boundary 1 (ndbdg1)**.

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

3 From the **Boundary condition** list, choose **Zero normal displacement**.

Deforming Electrode Surface 1 (desdg1)

Two of the electrode boundaries are not deforming. Deselect the corresponding boundaries from the Deforming Electrode Surface node.

1 In the **Model Builder** window, click **Deforming Electrode Surface 1 (desdg1)**.
2 In the **Settings** window for **Deforming Electrode Surface**, locate the **Boundary Selection** section.
3 In the list, choose **3** and **4**.
4 Click **Remove from Selection**.
5 Select Boundaries 1, 2, and 5–12 only.

MESH 1

Create a user-defined mesh with higher resolution on the electrodes.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.
3 From the list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Predefined** list, choose **Finer**.
4 Click to expand the **Element Size Parameters** section. In the **Resolution of narrow regions** text field, type `3`.

Size 1

1 In the **Model Builder** window, right-click **Free Triangular 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 **Anode**.
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 `0.005`.

Size 2

1 Right-click **Free Triangular 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 **Cathode tip**.

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 `0.005`.

8 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 1: Time Dependent

Enable automatic remeshing. Then solve the model.

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

2 In the **Settings** window for **Time Dependent**, click to expand the **Study Extensions** section.

3 Select the **Automatic remeshing** check box.

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

RESULTS

Electrolyte Potential (cd)

The following will reproduce the plots from the Results and Discussion section. Start by disabling the arrow plot and instead adding streamlines to the default electrolyte potential plot.

Streamline 1

1 In the **Model Builder** window, expand the **Electrolyte Potential (cd)** node, then click **Streamline 1**.

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

3 From the **Positioning** list, choose **Magnitude controlled**.

4 In the **Minimum distance** text field, type `0.0010`.

5 In the **Maximum distance** text field, type `0.030`.

6 In the **Electrolyte Potential (cd)** toolbar, click **Plot**.

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

Bore Shape

Create line plots of the shape of the bore as follows:

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

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

Line Graph 1

1 Right-click **Bore Shape** and choose **Line Graph**.

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

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

4 From the **Time selection** list, choose **First**.

5 Locate the **Selection** section. From the **Selection** list, choose **Anode**.

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

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

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

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

10 In the **Bore Shape** toolbar, click **Plot**.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

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

3 From the **Time selection** list, choose **Interpolated**.

4 In the **Times (s)** text field, type `0.5`.

5 In the **Bore Shape** toolbar, click **Plot**.

Line Graph 3

1 Right-click **Line Graph 2** and choose **Duplicate**.

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

3 In the **Times (s)** text field, type `1`.

Line Graph 4

1 Right-click **Line Graph 3** and choose **Duplicate**.

2 Select Boundaries 8 and 9 only.

3 In the **Settings** window for **Line Graph**, click to expand the **Coloring and Style** section.

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

5 Locate the **Legends** section. Clear the **Show legends** check box.

Bore Shape

1 In the **Model Builder** window, click **Bore Shape**.

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 `Bore shape for t=0, 0.5 and 1 s`.

5 Locate the **Axis** section. Select the **Manual axis limits** check box.

6 In the **x minimum** text field, type `0`.

7 In the **x maximum** text field, type `0.3`.

8 In the **y minimum** text field, type `-0.2`.

9 In the **y maximum** text field, type `0.01`.

10 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

11 In the **Bore Shape** toolbar, click **Plot**.

Follow these steps to create a revolved dataset for the anode surface. Then plot the current density on this surface in 3D.

Study 1/Remeshed Solution 1 (3) (sol2)

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets>Study 1/Remeshed Solution 1 (sol2)** and choose **Duplicate**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

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

Revolution 2D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

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

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

Current Density on the Bore Surface

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

2 In the **Settings** window for **3D Plot Group**, type `Current Density on the Bore Surface` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D 2**.

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

Surface 1

1 Right-click **Current Density on the Bore Surface** 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)> Primary Current Distribution>Electrode kinetics>cd.itot - Total interface current density - A/m²**.

3 In the **Current Density on the Bore Surface** toolbar, click **Plot**.

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

Finally, create an animation of the electromachining process in the following way.

Animation 1

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

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

3 From the **Target** list, choose **Player**.

4 Locate the **Scene** section. From the **Subject** list, choose **Current Density on the Bore Surface**.

5 Locate the **Animation Editing** section. From the **Time selection** list, choose **Interpolated**.

6 Click **Range**.

7 In the **Range** dialog box, type `0` in the **Start** text field.

8 In the **Step** text field, type `0.1`.

9 In the **Stop** text field, type `1`.

10 Click **Replace**.

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