



Model created in COMSOL Multiphysics 6.4

Two-Dimensional Transient Arc Discharge Simulation

Introduction

This model presents a 2D simulation of transient arc discharge movement along guided copper rails. While accurately modeling transient arcs typically requires a 3D simulation, the 2D approach offers greater efficiency and remains valuable for initial investigations and demonstration purposes.

Model Definition

For more information, see the 3D version model *Transient Arc Discharge in Guided Copper Rails*.

Results and Discussion

[Figure 1](#) shows the temperature distribution of the electric arc developed at $t = 4.5$ ms, where the arc reaches a maximum temperature of 20 kK. Although it is a 2D model, the arc roots at the anode and cathode are clearly distinguishable and closely resemble those observed in a 3D model.

[Figure 2](#) and [Figure 3](#) compare the simulated arc voltage and displacement with experimental results, respectively. While the simulated values show some deviation from the experimental data, they can still be useful for quick preliminary investigations.

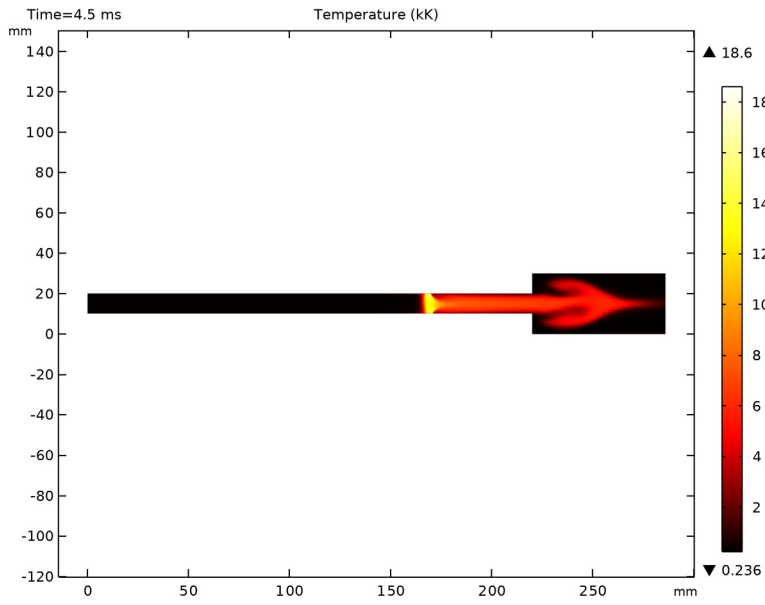


Figure 1: LTE arc temperature.

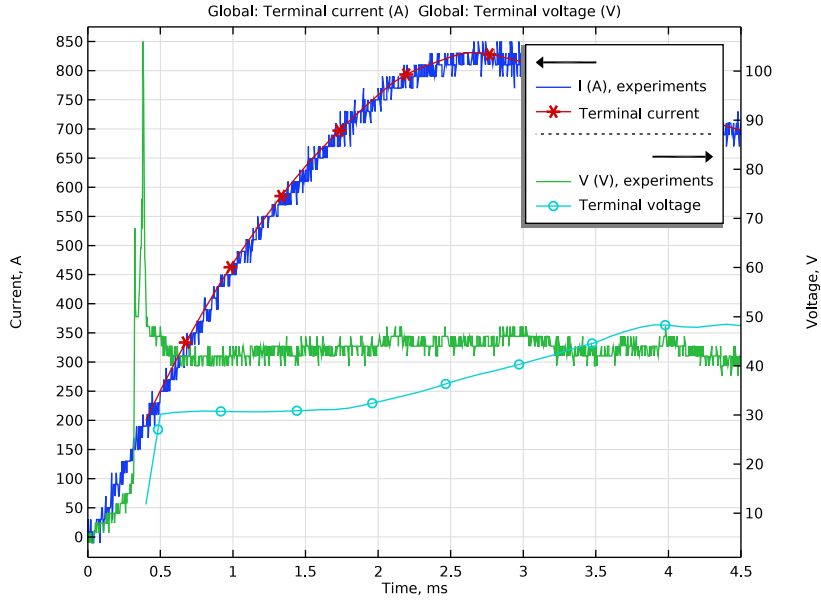


Figure 2: Comparison of the simulated and measured arc voltage.

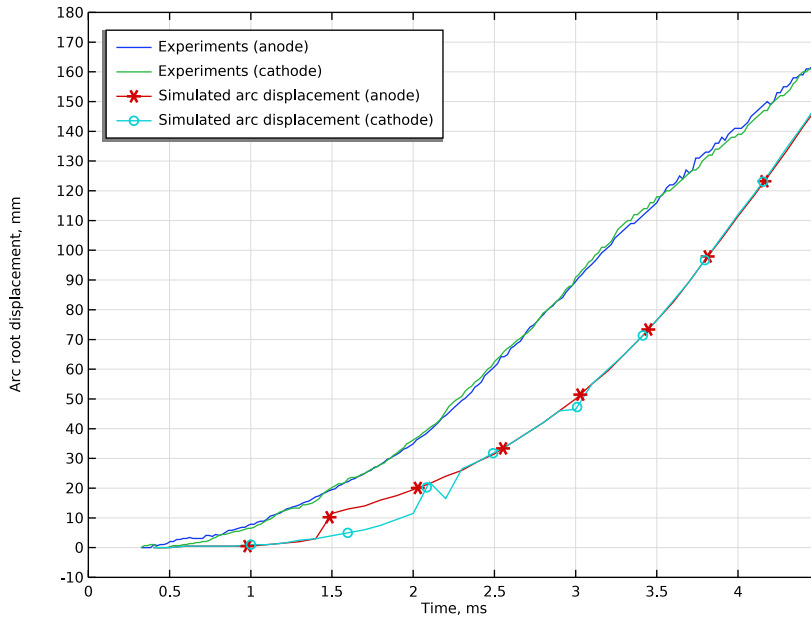



Figure 3: Comparison of the simulated and measured arc displacement.

Application Library path: Electric_Discharge_Module/Arc_Discharges/transient_arc_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 .
- 2 In the **Select Physics** tree, select **Electric Discharge > Arc Discharge**.
- 3 Click **Add**.
- 4 Click  **Study**.

- 5 In the **Select Study** tree, select **General Studies > Stationary**.
- 6 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	Value	Description
L	220[mm]	0.22 m	Copper rail length
L_wire	20[mm]	0.02 m	Ignition wire position
r0	10[mm]	0.01 m	Electrode radius
gap	10[mm]	0.01 m	Rail gap
d	6[mm]	0.006 m	Out-of-plane thickness
ts_ms	0.4	0.4	Simulation start time in ms
t	ts_ms[ms]	4E-4 s	Time

Interpolation 1 (int1)

- 1 In the **Home** toolbar, click  **Functions** and choose **Global > Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type **ic**.
- 4 Click  **Load from File**.
- 5 Browse to the model's Application Libraries folder and double-click the file **ic400.txt**.
- 6 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.
- 7 From the **Extrapolation** list, choose **Linear**.
- 8 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
ic	A


- 9 In the **Argument** table, enter the following settings:

Argument	Unit
t	ms

GEOMETRY I

- 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 L.
- 4 In the **Height** text field, type $r0+gap+r0$.
- 5 Click to expand the **Layers** section. In the table, enter the following settings:




Layer name	Thickness (mm)
Layer 1	r0
Layer 2	gap

- 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 $L*0.3$.
- 4 In the **Height** text field, type $r0+gap+r0$.
- 5 Locate the **Position** section. In the **x** text field, type L.

Rectangle 3 (r3)

- 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 $L*2$.
- 4 In the **Height** text field, type $(r0+gap+r0)*11$.
- 5 Locate the **Position** section. In the **y** text field, type $-(r0+gap+r0)*5$.
- 6 In the **Geometry** toolbar, click  **Build All**.
- 7 Click the  **Zoom Extents** button in the **Graphics** toolbar.

MAGNETIC AND ELECTRIC FIELDS (MEF)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic and Electric Fields (mef)**.

- 2 In the **Settings** window for **Magnetic and Electric Fields**, locate the **Thickness** section.
- 3 In the d text field, type d .

HEAT TRANSFER IN FLUIDS (HT)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.
- 2 In the **Settings** window for **Heat Transfer in Fluids**, locate the **Physical Model** section.
- 3 In the d_z text field, type d .
- 4 Select Domains 1, 3, and 5 only.

LAMINAR FLOW (SPF)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
- 2 Select Domains 1, 3, and 5 only.

DEFINITIONS

Outflow


- 1 In the **Model Builder** window, expand the **Component 1 (comp1) > Definitions** node.
- 2 Right-click **Definitions** and choose **Selections > Explicit**.
- 3 In the **Settings** window for **Explicit**, locate the **Input Entities** section.
- 4 From the **Geometric entity level** list, choose **Boundary**.
- 5 In the **Label** text field, type **Outflow**.
- 6 Select Boundaries 1, 2, 5, 9, 11, and 18 only.

MAGNETIC AND ELECTRIC FIELDS (MEF)

Magnetic Insulation 1

In the **Model Builder** window, under **Component 1 (comp1) > Magnetic and Electric Fields (mef)** click **Magnetic Insulation 1**.


Boundary Terminal 1

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Boundary Terminal**.
- 2 Select Boundary 7 only.
- 3 In the **Settings** window for **Boundary Terminal**, locate the **Terminal** section.
- 4 In the I_0 text field, type $ic(t)$.

Magnetic Insulation 1

In the **Model Builder** window, click **Magnetic Insulation 1**.

Electric Insulation 1

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Electric Insulation**.
- 2 In the **Settings** window for **Electric Insulation**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Outflow**.

Gauge Fixing for A-Field 1

- In the **Physics** toolbar, click  **Domains** and choose **Gauge Fixing for A-Field**.

DEFINITIONS


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 In the table, enter the following settings:


Name	Expression	Unit	Description
T0	$12000[\text{K}] * \exp(-((x - L_{\text{wire}}) / d)^2) + 293.15[\text{K}]$	K	Initial temperature

HEAT TRANSFER IN FLUIDS (HT)



Initial Values 2

- 1 In the **Physics** toolbar, click  **Domains** and choose **Initial Values**.
- 2 Select Domain 3 only.
- 3 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 4 In the T text field, type T0.

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 **Outflow**.

Heat Flux 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Heat Flux**.
- 2 Click the  **Select Box** button in the **Graphics** toolbar.
- 3 Select Boundaries 4, 6, 8, 10, 12, and 15 only.
- 4 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

5 From the **Flux type** list, choose **Convective heat flux**.

6 In the h text field, type 600.

Surface-to-Ambient Radiation 1

1 In the **Physics** toolbar, click  **Boundaries** and choose **Surface-to-Ambient Radiation**.

2 Select Boundaries 6 and 8 only.

3 In the **Settings** window for **Surface-to-Ambient Radiation**, locate the **Surface-to-Ambient Radiation** section.

4 From the ϵ list, choose **User defined**. In the associated text field, type 0.5.

LAMINAR FLOW (SPF)

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 **Outflow**.

MULTIPHYSICS

Equilibrium Discharge Heat Source 1 (phs1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Equilibrium Discharge Heat Source 1 (phs1)**.

2 Select Domains 1, 3, and 5 only.

ADD MATERIAL

1 In the **Materials** toolbar, click  **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Equilibrium Discharge** > **Air (1[atm])**.

4 Right-click and choose **Add to Component 1 (comp1)**.

5 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

MATERIALS

Air (1[atm]) (mat1)

Select Domains 1, 3, and 5 only.


Cu

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

- 2 In the **Settings** window for **Material**, type Cu in the **Label** text field.
- 3 Select Domains 2 and 4 only.
- 4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1		Basic
Electric conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	1e7	S/m	Basic
Relative permittivity	epsilonIso ; epsilonrii = epsilonIso, epsilonrij = 0	1		Basic

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 In the table, clear the **Use** checkboxes for **Magnetic and Electric Fields (mef)**, **Heat Transfer in Fluids (ht)**, **Laminar Flow (spf)**, **Equilibrium Discharge Heat Source 1 (phs1)**, and **Magnetohydrodynamics 1 (mhd1)**.
- 4 Click  **Build All**.
- 5 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

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 Select Boundaries 6 and 8 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section.
- 7 Select the **Maximum element size** checkbox. In the associated text field, type 0.5.

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 **Domain**.
- 4 Select Domain 3 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section.
- 7 Select the **Maximum element size** checkbox. In the associated text field, type 0.8.

Size 3

- 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 **Domain**.
- 4 Select Domain 5 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section.
- 7 Select the **Maximum element size** checkbox. In the associated text field, type 2.
- 8 Click  **Build All**.


The simulation will start from the moment the arc is formed. First, a stationary study step is added to compute the initial electromagnetic fields. This is necessary in case the electromagnetic fields are non-zero before the transient analysis.

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**, locate the **Physics and Variables Selection** section.
- 3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Heat Transfer in Fluids (ht)** and **Laminar Flow (spf)**.


Step 2: Time Dependent

- 1 In the **Study** toolbar, click  **Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 3 From the **Time unit** list, choose **ms**.
- 4 In the **Output times** text field, type range (ts_ms, 0.1, 4.5).

Solution 1 (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.

- 2 In the **Model Builder** window, expand the **Solution I (sol1)** node.
- 3 In the **Model Builder** window, under **Study I > Solver Configurations > Solution I (sol1)** click **Time-Dependent Solver I**.
- 4 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 5 From the **Steps taken by solver** list, choose **Free**.
- 6 From the **Maximum step constraint** list, choose **Constant**.
- 7 In the **Maximum step** text field, type 2[us].
- 8 Right-click **Study I > Solver Configurations > Solution I (sol1) > Time-Dependent Solver I** and choose **Segregated**.
- 9 In the **Model Builder** window, expand the **Study I > Solver Configurations > Solution I (sol1) > Time-Dependent Solver I > Segregated I** node.
- 10 Right-click **Study I > Solver Configurations > Solution I (sol1) > Time-Dependent Solver I > Segregated I** and choose **Segregated Step**.
- 11 In the **Settings** window for **Segregated Step**, locate the **General** section.
- 12 In the **Variables** list, choose **Pressure (comp1.p)**, **Temperature (comp1.T)**, and **Velocity Field (comp1.u)**.
- 13 Under **Variables**, click  **Delete**.
- 14 Click to expand the **Method and Termination** section. From the **Jacobian update** list, choose **Once per time step**.
- 15 In the **Label** text field, type Electromagnetics.
- 16 In the **Model Builder** window, under **Study I > Solver Configurations > Solution I (sol1) > Time-Dependent Solver I > Segregated I** click **Segregated Step I**.
- 17 In the **Settings** window for **Segregated Step**, locate the **General** section.
- 18 Under **Variables**, click  **Add**.
- 19 In the **Add** dialog, in the **Variables** list, choose **Pressure (comp1.p)**, **Temperature (comp1.T)**, and **Velocity Field (comp1.u)**.
- 20 Click **OK**.
- 21 In the **Settings** window for **Segregated Step**, type Fluid & Heat in the **Label** text field.
- 22 Locate the **Method and Termination** section. In the **Damping factor** text field, type 0.8.
- 23 From the **Jacobian update** list, choose **Once per time step**.
- 24 In the **Model Builder** window, under **Study I > Solver Configurations > Solution I (sol1) > Time-Dependent Solver I** click **Direct (Merged)**.

- 25 In the **Settings** window for **Direct**, locate the **General** section.
- 26 From the **Solver** list, choose **PARDISO**.
- 27 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1** click **Segregated 1**.
- 28 In the **Settings** window for **Segregated**, locate the **General** section.
- 29 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.
- 30 In the **Study** toolbar, click  **Compute**.

RESULTS


Selection 1


- 1 In the **Model Builder** window, expand the **Temperature (ht)** node.
- 2 Right-click **Surface 1** and choose **Selection**.
- 3 Select Domains 3 and 5 only.

Surface 1

- 1 In the **Model Builder** window, click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **kK**.
- 4 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalClassic**.

Temperature (ht)

- 1 In the **Model Builder** window, click **Temperature (ht)**.
- 2 In the **Settings** window for **2D Plot Group**, locate the **Color Legend** section.
- 3 Select the **Show maximum and minimum values** checkbox.
- 4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.
- 5 In the **Temperature (ht)** toolbar, click  **Plot**.

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

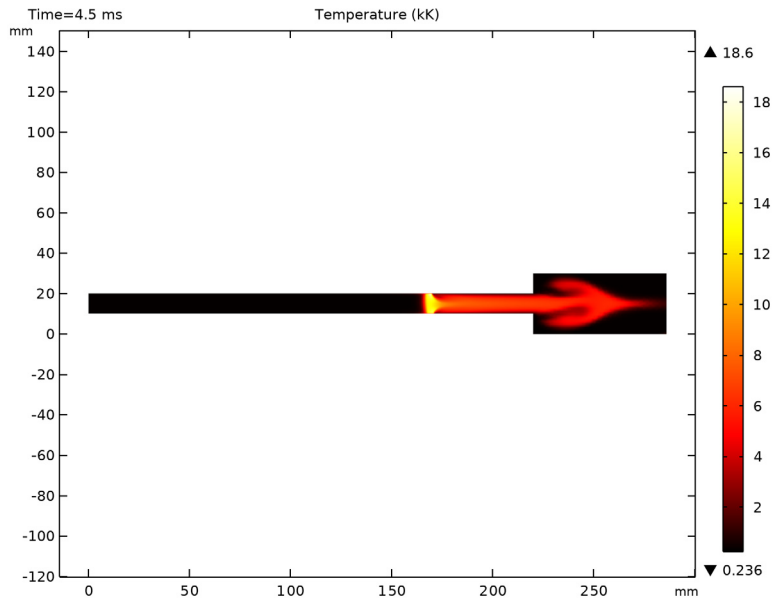


Table 1






- 1 In the **Results** toolbar, click  **Table**.
- 2 In the **Settings** window for **Table**, locate the **Data** section.
- 3 Click  **Import**.
- 4 Browse to the model's Application Libraries folder and double-click the file data0400.txt.

Table 2


- 1 In the **Results** toolbar, click  **Table**.
- 2 In the **Settings** window for **Table**, locate the **Data** section.
- 3 Click  **Import**.
- 4 Browse to the model's Application Libraries folder and double-click the file data0400x.txt.

Current and Voltage

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Current** and **Voltage** in the **Label** text field.

- 3 Locate the **Plot Settings** section. Select the **Two y-axes** checkbox.

Table Graph 1

- 1 Right-click **Current and Voltage** and choose **Table Graph**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 From the **Plot columns** list, choose **Manual**.
- 4 In the **Columns** list box, select **I (A)**.
- 5 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 6 In the **Current and Voltage** toolbar, click  **Plot**.
- 7 From the **Legends** list, choose **Manual**.
- 8 In the table, enter the following settings:

Legends
I (A), experiments

Table Graph 2

- 1 Right-click **Table Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 In the **Columns** list box, select **V (V)**.
- 4 Locate the **y-Axis** section. Select the **Plot on secondary y-axis** checkbox.
- 5 Locate the **Legends** section. In the table, enter the following settings:

Legends
V (V), experiments

Global 1

- 1 In the **Model Builder** window, right-click **Current and Voltage** and choose **Global**.
- 2 In the **Settings** window for **Global**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
- 4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
comp1.mef.IO_1	A	Terminal current

- 5 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 6 From the **Positioning** list, choose **Interpolated**.

Global 2

- 1 Right-click **Global 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

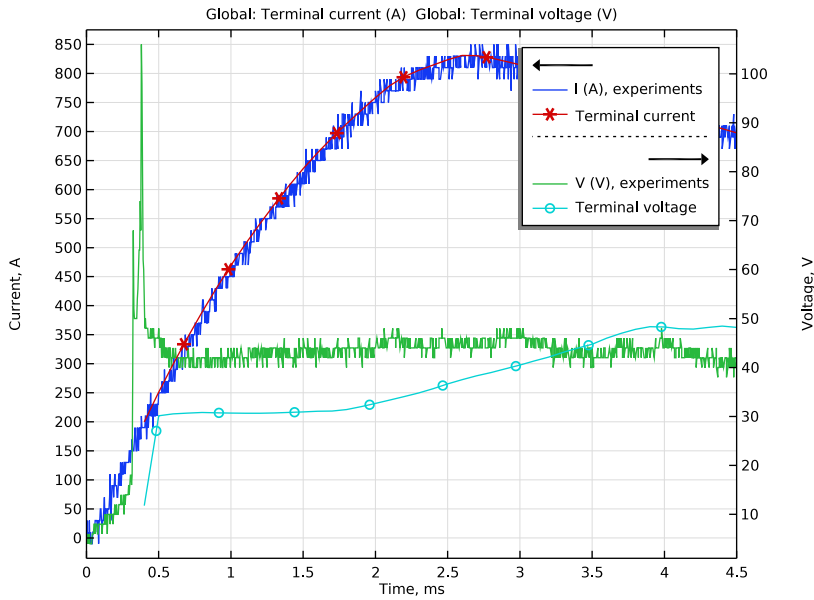
Expression	Unit	Description
comp1.mef.V0_1	V	Terminal voltage

- 4 Locate the **y-Axis** section. Select the **Plot on secondary y-axis** checkbox.

Current and Voltage


- 1 In the **Model Builder** window, click **Current and Voltage**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Plot Settings** section.
- 3 Select the **x-axis label** checkbox. In the associated text field, type **Time, ms**.
- 4 Select the **y-axis label** checkbox. In the associated text field, type **Current, A**.
- 5 Select the **Secondary y-axis label** checkbox. In the associated text field, type **Voltage, V**.
- 6 Locate the **Axis** section. Select the **Manual axis limits** checkbox.
- 7 In the **x minimum** text field, type **0**.
- 8 In the **x maximum** text field, type **4.5**.

9 In the **Current and Voltage** toolbar, click  **Plot**.




DEFINITIONS

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 8 only.

Maximum 2 (maxop2)

- 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 6 only.


Variables 1

- 1 In the **Model Builder** window, click **Variables 1**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
disp_a	$\maxop1(x*(T==\maxop1(T)))-L_wire$	m	Simulated arc displacement (anode)
disp_c	$\maxop2(x*(T==\maxop2(T)))-L_wire$	m	Simulated arc displacement (cathode)

STUDY 1

In the **Study** toolbar, click  **Update Solution**.

RESULTS

Arc Root Displacement



- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Arc Root Displacement in the **Label** text field.

Table Graph 1

- 1 Right-click **Arc Root Displacement** and choose **Table Graph**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 From the **Table** list, choose **Table 2**.
- 4 Locate the **Legends** section. Select the **Show legends** checkbox.
- 5 In the **Arc Root Displacement** toolbar, click  **Plot**.
- 6 From the **Legends** list, choose **Manual**.
- 7 In the table, enter the following settings:

Legends
Experiments (anode)
Experiments (cathode)

Global 1

- 1 In the **Model Builder** window, right-click **Arc Root Displacement** and choose **Global**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
comp1.disp_a	mm	Simulated arc displacement (anode)
comp1.disp_c	mm	Simulated arc displacement (cathode)

4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 From the **Positioning** list, choose **Interpolated**.

Arc Root Displacement

1 In the **Model Builder** window, click **Arc Root Displacement**.

2 In the **Settings** window for **ID Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower right**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type **Time, ms**.

6 Select the **y-axis label** checkbox. In the associated text field, type **Arc root displacement, mm**.

7 Locate the **Axis** section. Select the **Manual axis limits** checkbox.


8 In the **x minimum** text field, type **0**.

9 In the **x maximum** text field, type **4.5**.

10 In the **y minimum** text field, type **-10**.

11 In the **y maximum** text field, type **180**.

12 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

B3 In the **Arc Root Displacement** toolbar, click  **Plot**.

