



Model created in COMSOL Multiphysics 6.4

Transient Arc Discharge in Guided Copper Rails

Introduction

Unwanted arcing can have serious adverse effects on electrical and electronic equipment and systems. To improve the understanding and prediction of arc dynamics, it is crucial to conduct multiphysics simulations of transient arc processes. This model features a 3D simulation of transient arc discharge movement in guided copper rails, utilizing the Arc Discharge multiphysics interface based on a magnetohydrodynamics formulation. The simulated arc voltage and arc velocity show good agreement with experimental results.

Model Definition

EXPERIMENTS

The experimental data are provided by associate professor Hengxin He from Huazhong University of Science and Technology (HUST). The experimental setup is shown in Figure 1, where an electric arc is generated and move along a copper rail. A thin wire connecting the upper and lower rail is installed to ignite the arc discharge. The measured voltage and current are recorded in the file `data0400.txt`. The arc root displacement is computed from the high-speed images and the data is stored in the file `data0400x.txt`.

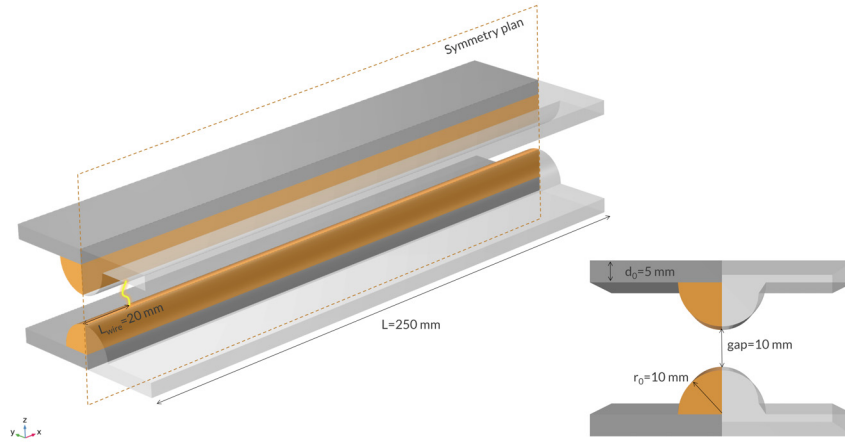


Figure 1: Experimental setup.

SIMULATION

The Arc Discharge multiphysics interface is used to simulate the transient arc. To save computation time, a symmetry plan is assumed such that only half of the domain is modeled. The magnetohydrodynamics (MHD) model solved is given as

$$\begin{aligned}
\nabla \times \mathbf{H} &= \mathbf{J} \\
\nabla \cdot \mathbf{J} &= 0 \\
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= 0 \\
\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) &= \nabla \cdot \left[-p \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I} \right] + \mathbf{F} \\
\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) - \nabla \cdot (k \nabla T) &= Q
\end{aligned}$$

where the first equation is Ampère's Law and the second is the current conservation equation. They are reformulated by the A–V formulation and solved by the Magnetic and Electric Fields interface. The third is the mass conservation equation and the fourth is the momentum conservation equation. They are solved with the Laminar Flow interface. The last equation is the energy conservation equation solved by the Heat Transfer in Fluid interface. The coupling between electromagnetics and fluid flow is defined by the Magnetohydrodynamics multiphysics coupling feature. The coupling between electromagnetics and heat transfer is defined by the Equilibrium Discharge Heat Source multiphysics coupling feature.

Results and Discussion

Figure 2 shows the temperature distribution of the electric arc developed at $t = 4$ ms, where the arc reaches a maximum temperature of 20 kK.

Figure 3 and Figure 4 compare the simulated arc voltage and displacement, respectively, with experimental results. Overall, the simulated results are in good agreement with those from experiments.

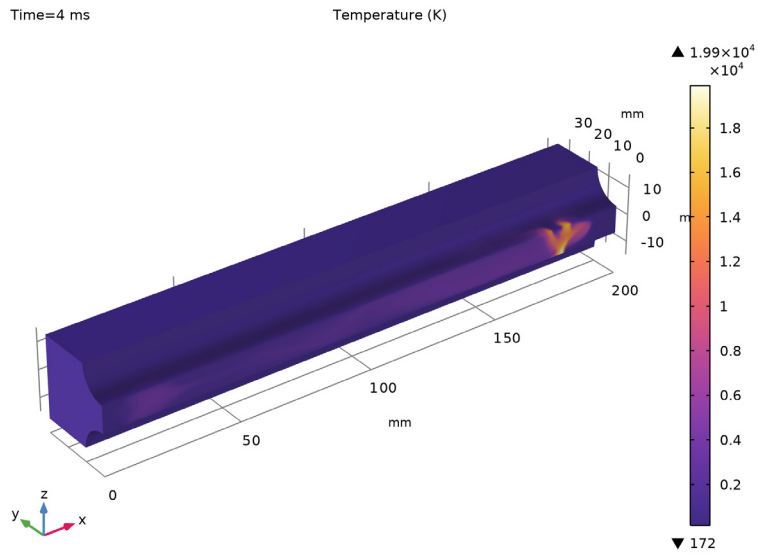


Figure 2: LTE arc temperature.

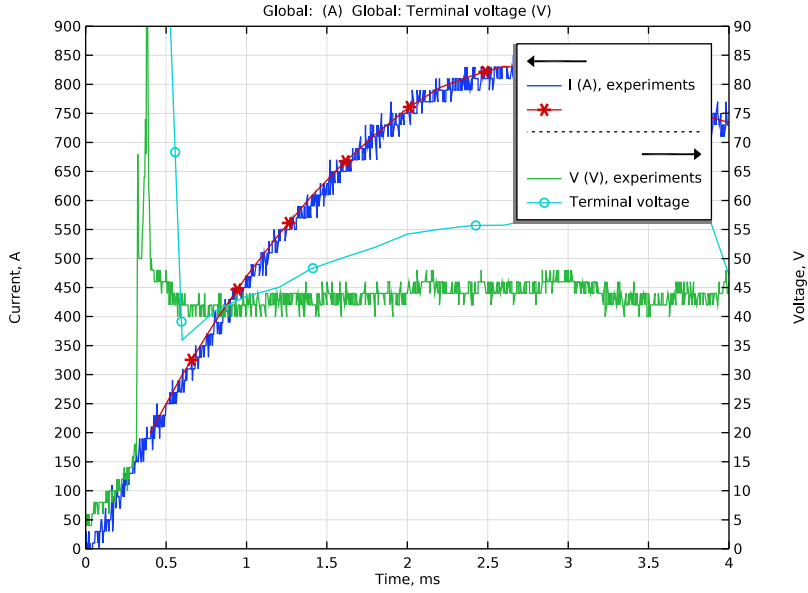


Figure 3: Comparison of the simulated and measured arc voltages.

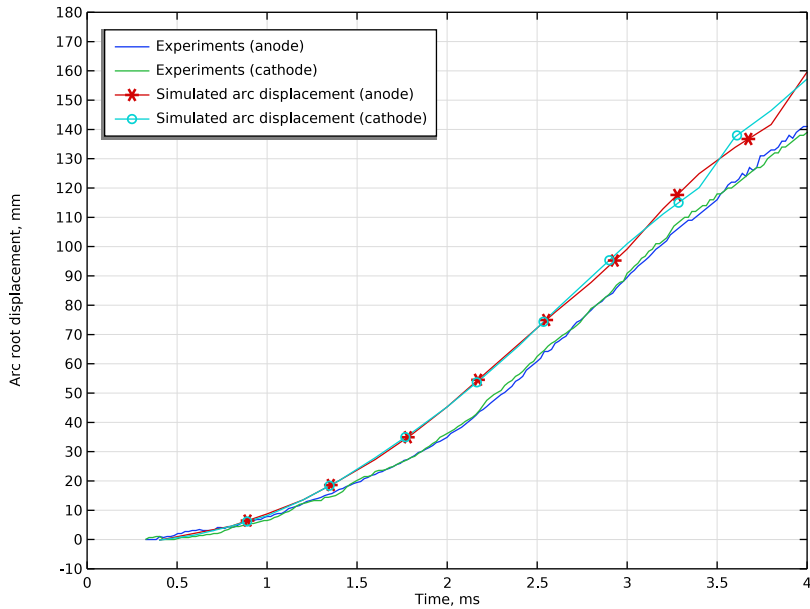



Figure 4: Comparison of the simulated and measured arc displacements.

Application Library path: Electric_Discharge_Module/Arc_Discharges/transient_arc_3d



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  **3D**.
- 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	200[mm]	0.2 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
d0	5[mm]	0.005 m	Insulator 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**.


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 **yz-plane**.

Work Plane 1 (wp1) > Plane Geometry


In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Rectangle 1 (r1)


- 1 In the **Work Plane** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type $r0*3$.
- 4 In the **Height** text field, type $(r0+d0+gap/2)$.
- 5 Locate the **Position** section. In the **yw** text field, type $-(r0+d0+gap/2)$.
- 6 Click to expand the **Layers** section. In the table, enter the following settings:


Layer name	Thickness (mm)
Layer 1	d0

Work Plane 1 (wp1) > Circle 1 (c1)

- 1 In the **Work Plane** toolbar, click  **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type $r0$.
- 4 In the **Sector angle** text field, type 90 .
- 5 Locate the **Position** section. In the **yw** text field, type $-(r0+gap/2)$.

Work Plane 1 (wp1) > Mirror 1 (mir1)



- 1 In the **Work Plane** toolbar, click  **Transforms** and choose **Mirror**.
- 2 Click in the **Graphics** window and then press **Ctrl+A** to select both objects.
- 3 In the **Settings** window for **Mirror**, locate the **Normal Vector to Line of Reflection** section.
- 4 In the **xw** text field, type 0 .
- 5 In the **yw** text field, type 1 .

- 6 In the **Work Plane** toolbar, click  **Build All**.
- 7 In the **Model Builder** window, click **Mirror 1 (mir1)**.
- 8 Locate the **Input** section. Select the **Keep input objects** checkbox.
- 9 In the **Work Plane** toolbar, click  **Build All**.
- 10 Click the  **Zoom Extents** button in the **Graphics** toolbar.


Extrude 1 (ext1)

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.
- 2 In the **Settings** window for **Extrude**, locate the **Distances** section.
- 3 In the table, enter the following settings:

Distances (mm)
L

- 4 Click  **Build All Objects**.
- 5 Click the  **Go to Default View** button in the **Graphics** toolbar.


Work Plane 2 (wp2)

- 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 **zx-plane**.

Work Plane 2 (wp2) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.


Work Plane 2 (wp2) > Circle 1 (c1)

- 1 In the **Work Plane** toolbar, click  **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 1 [m].
- 4 In the **Sector angle** text field, type 180.
- 5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	0.2 [m]

Revolve 1 (rev1)

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Revolve**.
- 2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

- 3 Click the **Angles** button.
- 4 In the **End angle** text field, type 90.
- 5 Locate the **Revolution Axis** section. Find the **Direction of revolution axis** subsection. In the **xw** text field, type 1.
- 6 In the **yw** text field, type 0.
- 7 Click  **Build All Objects**.

Form Union (fin)

- 1 In the **Geometry** toolbar, click  **Build All**.
- 2 Click the  **Go to Default View** button in the **Graphics** toolbar.

DEFINITIONS

Infinite Element Domain 1 (ie1)

- 1 In the **Model Builder** window, expand the **Component 1 (comp1) > Definitions** node.
- 2 Right-click **Definitions** and choose **Infinite Element Domain**.
- 3 Select Domains 1 and 9 only.
- 4 In the **Settings** window for **Infinite Element Domain**, locate the **Geometry** section.
- 5 From the **Type** list, choose **Spherical**.

Hide for Physics 1

- 1 In the **Model Builder** window, right-click **View 1** and choose **Hide for Physics**.
- 2 Select Domains 1, 2, and 9 only.

Define a terminal boundary for the input current. Note that the input current should be half of the total current because the model only solves for half of the geometry, assuming symmetry.

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 Click the  **Go to Default View** button in the **Graphics** toolbar.
- 3 Select Boundary 19 only.

- 4 In the **Settings** window for **Boundary Terminal**, locate the **Terminal** section.
- 5 In the I_0 text field, type $ic(t)/2$.



Magnetic Insulation I

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

Electric Insulation I

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Electric Insulation**.
- 2 Select Boundaries 1, 3, 4, 7, 13, 16, 22, 26, and 29 only.



Perfect Magnetic Conductor I

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Perfect Magnetic Conductor**.
- 2 Select Boundaries 2, 5, 8, 11, 14, 17, 20, 23, and 27 only.
- 3 In the **Settings** window for **Perfect Magnetic Conductor**, locate the **Boundary Selection** section.
- 4 Click  **Create Selection**.
- 5 In the **Create Selection** dialog, type Symmetry for EM in the **Selection name** text field.
- 6 Click **OK**.



HEAT TRANSFER IN FLUIDS (HT)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.
- 2 Select Domains 5 and 6 only.

Outflow I

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Outflow**.
- 2 Select Boundaries 13, 16, 33, 34, 39, and 40 only.
- 3 In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.
- 4 Click  **Create Selection**.
- 5 In the **Create Selection** dialog, type Outflow in the **Selection name** text field.
- 6 Click **OK**.

Symmetry I

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 14 and 17 only.
- 3 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
- 4 Click  **Create Selection**.

5 In the **Create Selection** dialog, type Symmetry for Heat and Flow in the **Selection name** text field.

6 Click **OK**.

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	$15e3[K] * \exp(-((x - L_wire)/1.5[mm])^2 - ((y - 0[mm])/1[mm])^2) + 293.15[K]$	K	Initial temperature

HEAT TRANSFER IN FLUIDS (HT)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Fluids (ht)** click **Initial Values 1**.

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

3 In the T text field, type T0.

Heat Flux 1

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

2 Select Boundaries 15, 21, 30, and 31 only.

3 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

4 Click  **Create Selection**.

5 In the **Create Selection** dialog, type Wall in the **Selection name** text field.

6 Click **OK**.

7 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.


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

9 In the h text field, type 600.


LAMINAR FLOW (SPF)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
- 2 Select Domains 5 and 6 only.

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**.

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 for Heat and Flow**.

MATERIALS

Copper Rail

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
- 2 Select Domains 4 and 7 only.
- 3 In the **Settings** window for **Material**, type Copper Rail in the **Label** text field.
- 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 ; sigmair = sigma_iso, sigmairj = 0	1e7[S/m]	S/m	Basic
Relative permittivity	epsilon_r_iso ; epsilon_rir = epsilon_r_iso, epsilon_rirj = 0	1		Basic



Insulator

- 1 Right-click **Materials** and choose **Blank Material**.

- 2 Select Domains 3 and 8 only.
- 3 In the **Settings** window for **Material**, type Insulator in the **Label** text field.
- 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 ; sigmair = sigma_iso, sigmairj = 0	1e-3	S/m	Basic
Relative permittivity	epsilon_r_iso ; epsilon_rii = epsilon_r_iso, epsilon_rij = 0	1		Basic

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 (I[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 (I[atm]) (mat3)

Select Domains 1, 2, 5, 6, and 9 only.

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 **Geometric Analysis**, **Detail Size**, **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**.

Free Tetrahedral 1

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Mesh 1** click **Free Tetrahedral 1**.
- 2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Click  **Clear Selection**.
- 5 Click  **Paste Selection**.
- 6 In the **Paste Selection** dialog, type 2-8 in the **Selection** text field.
- 7 Click **OK**.

Size 1

- 1 Right-click **Free Tetrahedral 1** and choose **Size**.

A coarse mesh was used here to save computation time. It is always good to refine the mesh later to ensure that the results do not change significantly.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Edge**.
- 4 Select Edges 15 and 21 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 0.6.

Size 2



- 1 Right-click **Free Tetrahedral 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 14 and 17 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 0.9.

Size 3



- 1 Right-click **Free Tetrahedral 1** and choose **Size**.
- 2 Select Domains 5 and 6 only.
- 3 In the **Settings** window for **Size**, locate the **Element Size** section.
- 4 Click the **Custom** button.

- 5 Locate the **Element Size Parameters** section.
- 6 Select the **Maximum element size** checkbox. In the associated text field, type 10.
- 7 Select the **Maximum element growth rate** checkbox. In the associated text field, type 1.15.

Swept 1

- 1 In the **Mesh** toolbar, click  **Swept**.
- 2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Click  **Paste Selection**.
- 5 In the **Paste Selection** dialog, type 1,9 in the **Selection** text field.
- 6 Click **OK**.

Distribution 1


- 1 Right-click **Swept 1** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 In the **Number of elements** text field, type 3.
- 4 Click  **Build All**.
- 5 Click the  **Go to Default View** button in the **Graphics** toolbar.

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)**.
- 4 In the **Solve for** column of the table, under **Component 1 (comp1) > Multiphysics**, clear the checkboxes for **Equilibrium Discharge Heat Source 1 (phs1)** and **Magnetohydrodynamics 1 (mhd1)**.

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,0.2,4).

Solution 1 (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
In order to save computational memory, an iterative solver is used here.
- 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) > Stationary Solver 1 > Iterative 1** node.
- 4 In the **Model Builder** window, expand the **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Segregated 1** node, then click **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1**.
- 5 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 6 Select the **Initial step** checkbox. In the associated text field, type 0.1 [us].
- 7 From the **Maximum step constraint** list, choose **Constant**.
- 8 In the **Maximum step** text field, type 2[us].
- 9 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1** click **Segregated 1**.
- 10 In the **Settings** window for **Segregated**, locate the **General** section.
- 11 In the **Tolerance factor** text field, type 1.
- 12 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Segregated 1** right-click **Velocity u, Pressure p** and choose **Delete**.
- 13 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Segregated 1** click **Temperature**.
- 14 In the **Settings** window for **Segregated Step**, type Fluid & Heat in the **Label** text field.
- 15 Locate the **General** section. Under **Variables**, click **+ Add**.
- 16 In the **Add** dialog, in the **Variables** list, choose **Pressure (comp1.p)** and **Velocity Field (comp1.u)**.
- 17 Click **OK**.
- 18 In the **Settings** window for **Segregated Step**, locate the **General** section.
- 19 From the **Linear solver** list, choose **AMG, fluid flow variables (spf)**.
- 20 Click to expand the **Method and Termination** section. In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Segregated 1** click **Magnetic and Electric Fields**.



- 21 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.
- 22 From the **Jacobian update** list, choose **Once per time step**.
- 23 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1)** right-click **Time-Dependent Solver 1** and choose **Iterative**.
- 24 Expand the **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Iterative 2** node.
- 25 Right-click **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Iterative 2** and choose **Multigrid**.
- 26 In the **Settings** window for **Multigrid**, locate the **General** section.
- 27 From the **Solver** list, choose **Algebraic multigrid**.
- 28 Click to expand the **Hybridization** section. From the **Use as** list, choose **Multi preconditioner**.
- 29 In the **Preconditioner variables** list, choose **Pressure (comp1.p)**, **Temperature (comp1.T)**, **Velocity Field (comp1.u)**, **Electric Potential (comp1.V)**, and **Terminal Voltage (comp1.mef.mil.term1.V0_ode)**.
- 30 Under **Preconditioner variables**, click  **Delete**.
- 31 Right-click **Iterative 2** and choose **Direct Preconditioner**.
- 32 In the **Settings** window for **Direct Preconditioner**, locate the **General** section.
- 33 From the **Solver** list, choose **PARDISO**.
- 34 Click to expand the **Hybridization** section. In the **Preconditioner variables** list, choose **Magnetic Vector Potential (comp1.A)**, **Pressure (comp1.p)**, **Temperature (comp1.T)**, and **Velocity Field (comp1.u)**.
- 35 Under **Preconditioner variables**, click  **Delete**.
- 36 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Segregated 1** click **Magnetic and Electric Fields**.
- 37 In the **Settings** window for **Segregated Step**, locate the **General** section.
- 38 From the **Linear solver** list, choose **Iterative 2**.
- 39 In the **Study** toolbar, click  **Compute**.

RESULTS

Streamline Multislice 1

- 1 In the **Model Builder** window, expand the **Results > Magnetic Flux Density (mef)** node.
- 2 Right-click **Streamline Multislice 1** and choose **Delete**.

Temperature (ht)

- 1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.
- 2 In the **Settings** window for **3D 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.

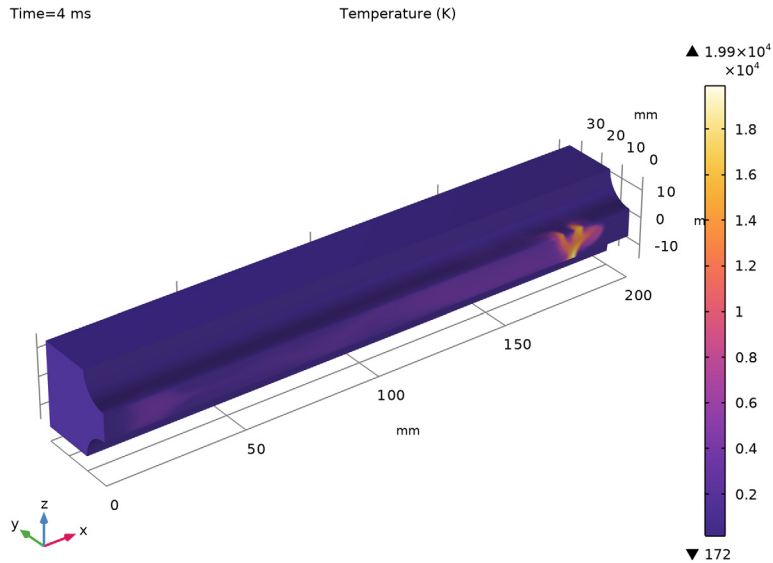


Table 1





- 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*2	A	

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.VO_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**.

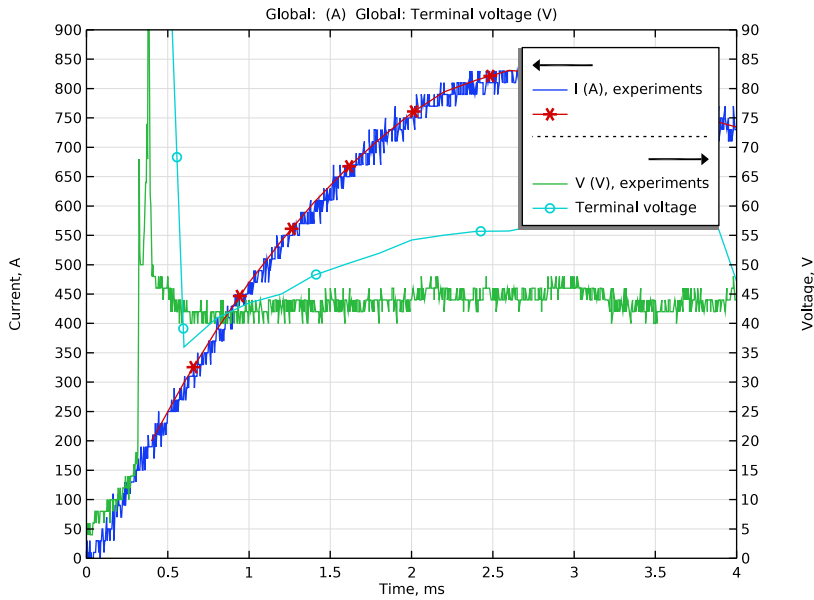
9 In the **y minimum** text field, type **0**.

10 In the **y maximum** text field, type **900**.

11 In the **Secondary y minimum** text field, type **0**.


12 In the **Secondary y maximum** text field, type **90**.

13 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 **Edge**.
- 4** Select Edge 21 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 **Edge**.
- 4** Select Edge 15 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**.

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**.

