Created in COMSOL Multiphysics 6.1

Lightning-Induced Voltage of an Overhead Line Over Lossy Ground

Introduction

Lightning strikes can induce high voltages on power transmission lines and cause insulation failure and power outage. Therefore, it is of great importance to compute lightning-induced voltage during the design, operation, and maintenance stages. Apart from the height of the overhead line and the striking point, other parameters like soil conductivity and lightning channel inclination, also play a big role.

This model shows how to calculate the lightning induced voltage of an overhead line over a lossy ground. Lightning channels with different inclinations are considered. The effect of soil conductivity is also studied. The configuration of the model is shown in Figure 1, where the voltage at both ends of an overhead line (A and B) are assumed to be measured with an oscilloscope that has a capacitance of 50 pF and a resistance of 600 Ω.

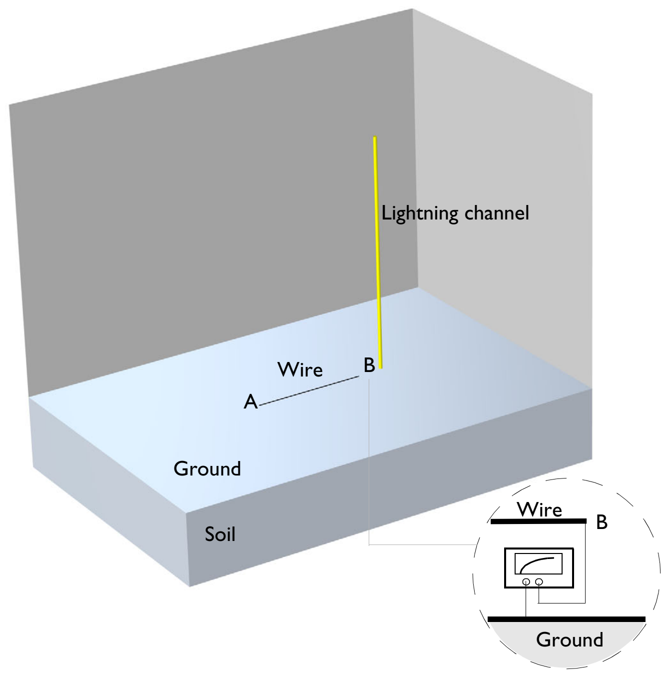


Figure 1: Configuration of the model.

Model Definition

Lightning current is varying in time and a transient 3D study is required. The **RF Module** provides a physics interface named as **Electromagnetic Waves, Transient** that is dedicated for this purpose. The **Electromagnetic Waves, Transient** interface provides predefined lightning current. The current can be easily visualized by clicking the **Plot Pulse Shape** button in the **Settings** window, as shown in Figure 2.

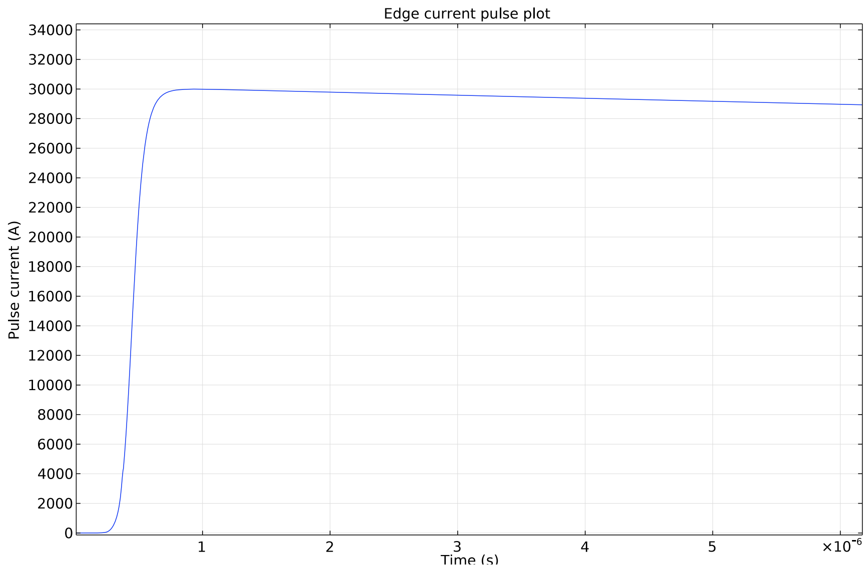


Figure 2: The lightning current used in the model.

The radius of a lightning channel is usually a few millimeters and it is here modeled with the **Edge Current** feature. Lightning current typically travels at one-third the speed of light due to corona discharges around the channel. The **Edge Current** feature defines the waveform of the current as well as how it is propagating along the edge (lightning path).

The oscilloscope is simulated with a real capacitor in parallel with a resistor modeled with the **Lumped Element** feature. The modeling of thin wire is critical in computing the induced voltage. The section below introduces a simplified method of modeling thin wire using the **Electromagnetic Waves, Transient** interface.

THEORY

Thin wires are widely presented in electrical and electronic systems. The aspect ratio of the wire can be more than six orders of magnitude such that volumetric meshing of the wire is impractical. There are numerous methods to model thin wires based on either Boundary Element Method (BEM) or *ad hoc* assumptions as in Finite Difference Time Domain (FDTD) method. The main idea of modeling thin wire in *ad hoc* assumptions is to make the wire lay in an edge of the structured mesh and manually modify material properties in the cell containing the wire, as illustrated in Figure 3.

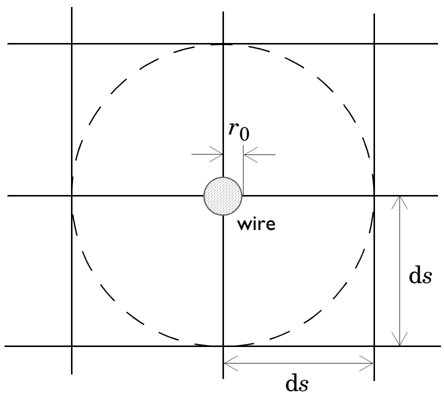


Figure 3: Thin wire numerical representation

Let $\mathrm{d}s$ be the side cell length of the structured mesh. The thin wire with a radius of r_0 in air has an equivalent radius of $0.23\mathrm{d}s$ if the electric field along the axis of the wire (the out-of-plane direction in Figure 3) is set to zero. It is further noticed that a thin wire can be equivalently represented by placing a zero-radius wire in a rectangular prism, coaxial with the thin wire, with a cross-sectional area of $2\mathrm{d}s$ by $2\mathrm{d}s$ and the modified relative permittivity and permeability given by

$$\varepsilon_r^* = \frac{\ln\left(\frac{1}{0.23}\right)}{\ln\left(\frac{\mathrm{d}s}{r_0}\right)}, \mu_r^* = \frac{1}{\varepsilon_r^*}$$

ADVANTAGES AND DISADVANTAGES

The advantages of the above approach are:

- It reduces the total DoFs significantly as the volumetric meshing is avoided
- The results are acceptable in engineering applications

The disadvantages are:

- It relies on a specifically structured mesh, for example, in the domain containing the wire structure
- The this wire cannot be too close to other thin wires and objects of interest

IMPLEMENTATIONS IN FEM

It is possible to model the thin wire in the Finite Element framework that the **Electromagnetic Waves, Transient** interface is built on. The modeling requires several settings as follows:

- In the **Geometry**, add a block that is coaxial to the thin wire
- In the **Electromagnetic Waves, Transient** interface settings, set the **Discretization** to linear
- Add a **Pointwise Constraint** on the wire to configure it effectively as a perfect electric conductor by constraining the tangential magnetic vector potential to zero
- In the **Mesh**, add a mapped mesh on one side of the block and then use a **Swept** feature

Note that the method described above is based on *ad hoc* assumptions and the abovementioned disadvantages should always be kept in mind. The method therefore only provide a first approximation and it requires further benchmarking and validation with other numerical methods and experiments.

Results and Discussion

Figure 4 shows the computed induced voltage at both ends of the overhead line with different soil conductivity. It clearly shows that increasing soil conductivity can significantly decrease the induced voltage.

The position and inclination of the lightning channel is parameterized and therefore it is straightforward to investigate their effects on induced voltage. The model can also be extended to model multiple overhead lines and to include other objects like buildings.

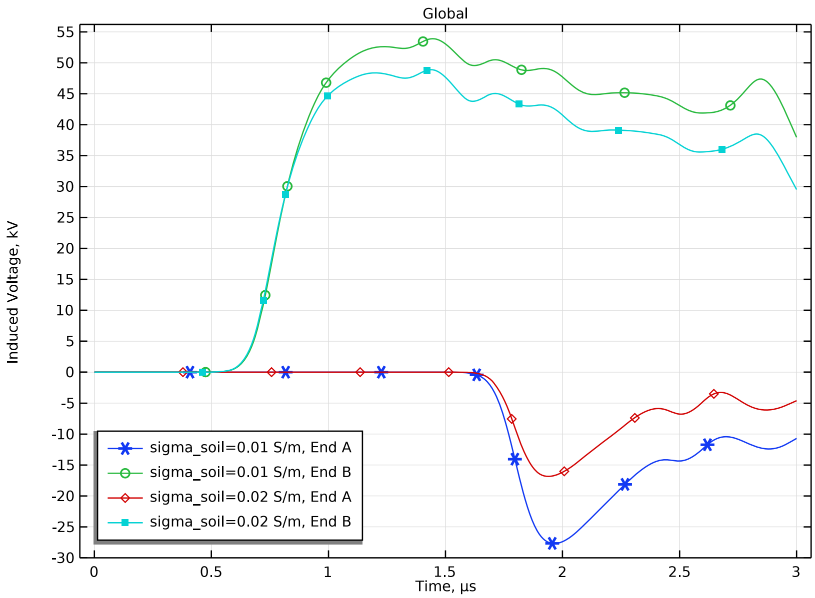


Figure 4: Induced voltage at End A and End B as a function of time under two different soil conductivities.

Application Library path: `RF_Module/ESD_and_Lightning_Surge/lightning_induced_voltage_overhead_lines`

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 **Radio Frequency>Electromagnetic Waves, Transient (temw)**.

3 Click **Add**.

4 Click **Study**.

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

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 Click **Clear Table**.

4 In the table, enter the following settings:

Name	Expression	Value	Description
W_air	1200[m]	1200 m	Width of the air domain
D_air	800[m]	800 m	Depth of the air domain
H_air	1000[m]	1000 m	Height of the air domain
H_soil	200[m]	200 m	Soil layer thickness
L_wire	300[m]	300 m	Length of the wire
H_wire	6[m]	6 m	Height of the wire
r_wire	1[cm]	0.01 m	Radius of the wire

Name	Expression	Value	Description
LDx	80[m]	80 m	X distance between wire and striking point
LDy	20[m]	20 m	Y distance between wire and striking point
ds	1[m]	1 m	Thin wire domain mesh size
vr	150[m/us]	1.5E8 m/s	Current velocity
epsilonr	log(1/0.23)/ log(ds/r_wire)	0.31914	Modified relative permittivity
mur	1/epsilonr	3.1335	Modified relative permeability
C_term	50[pF]	5E-11 F	Capacitance used for termination
R_term	600[ohm]	600 Ω	Resistance used for termination
S_capacitor	0.5[m]	0.5 m	Separation of the parallel plate capacitor
W_capacitor	3[m]	3 m	Width of the capacitor
L_capacitor	C_term* S_capacitor/ epsilon0_const/ W_capacitor	0.94117 m	Length of the capacitor
angle	90[degree]	1.5708 rad	Angle of inclination of the wire
sigma_soil	0.01[S/m]	0.01 S/m	Soil conductivity

GEOMETRY 1

Block 1 (blk1)

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

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

3 In the **Width** text field, type W_air.

4 In the **Depth** text field, type D_air.

5 In the **Height** text field, type H_air.

6 Locate the **Position** section. In the **z** text field, type -H_soil.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	H_soil

8 Click **Build Selected**.

DEFINITIONS

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

Hide for Geometry 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>View 1** node.

2 Right-click **View 1** and choose **Hide for Geometry**.

3 In the **Settings** window for **Hide for Geometry**, locate the **Selection** section.

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

5 On the object **blk1**, select Boundaries 4, 5, and 7 only.

GEOMETRY 1

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type W_air/2-L_wire/2.

5 In the **y** text field, type D_air/2.

6 In the **z** text field, type H_wire.

7 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

8 In the **x** text field, type W_air/2+L_wire/2.

9 In the **y** text field, type D_air/2.

10 In the **z** text field, type H_wire.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type W_air/2+L_wire/2+LDx.

5 In the **y** text field, type D_air/2+LDy.

6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **x** text field, type W_air/2+L_wire/2+LDx+(H_air-H_soil)*0.8*cos(angle).

8 In the **y** text field, type D_air/2+LDy.

9 In the **z** text field, type (H_air-H_soil)*0.8*sin(angle).

10 Click **Build Selected**.

Block 2 (blk2)

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

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

3 In the **Width** text field, type L_wire.

4 In the **Depth** text field, type 2*ds.

5 In the **Height** text field, type 2*ds.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **x** text field, type W_air/2.

8 In the **y** text field, type D_air/2.

9 In the **z** text field, type H_wire.

10 Locate the **Layers** section. Find the **Layer position** subsection. Select the **Front** check box.

11 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	ds

12 Click **Build Selected**.

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work 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 L_wire.

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

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **xw** text field, type `W_air/2`.

7 In the **yw** text field, type `D_air/2`.

8 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	L_capacitor

9 Clear the **Layers on bottom** check box.

10 Select the **Layers to the left** check box.

11 Select the **Layers to the right** check box.

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 (m)
S_capacitor
H_wire-ds

Delete Entities 1 (del1)

1 Right-click **Geometry 1** and choose **Delete Entities**.

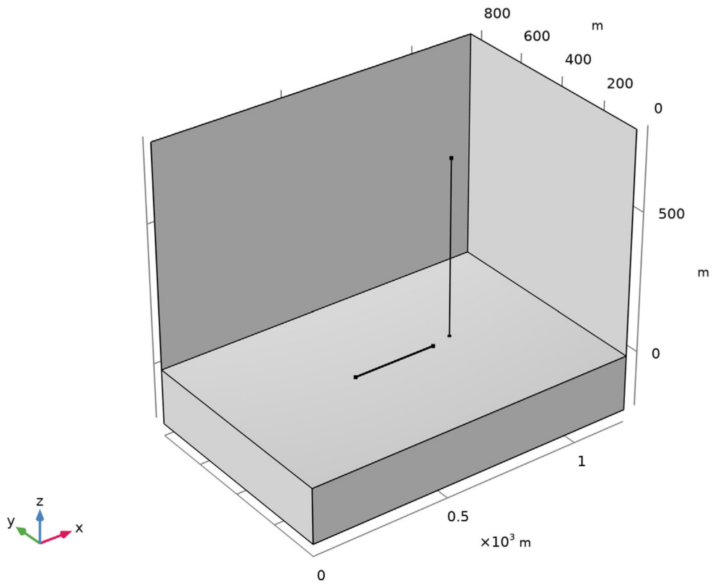
2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **ext1**, select Domains 3 and 4 only.

5 Click **Build All Objects**.

6 Click the **Go to Default View** button in the **Graphics** toolbar.



MATERIALS

Air

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Air` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

4 Locate the **Geometric Entity Selection** section. In the list, choose **1**, **5**, **6**, **7**, and **8**.

5 Click **Remove from Selection**.

6 Select Domains 2–4, 9, and 10 only.

Soil

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

2 In the **Settings** window for **Material**, type `Soil` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`sigma_soil`	S/m	Basic

Modified Material

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

2 In the **Settings** window for **Material**, type `Modified Material` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `5-8` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Material**, locate the **Material Contents** section.

7 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	`epsilonr`	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	`mur`	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Transient (temw)**.

2 In the **Settings** window for **Electromagnetic Waves, Transient**, click to expand the **Discretization** section.

3 From the **Magnetic vector potential** list, choose **Linear**.

Scattering Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Scattering Boundary Condition**.

2 In the **Settings** window for **Scattering Boundary Condition**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1-5, 7-9, 58, 59` in the **Selection** text field.

5 Click **OK**.

Perfect Electric Conductor 2

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

2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `12, 13, 15, 17, 24, 42, 45, 53, 54, 56` in the **Selection** text field.

5 Click **OK**.

Lumped Element A

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

2 In the **Settings** window for **Lumped Element**, type `Lumped Element A` in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `10` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

7 In the Z_{element} text field, type `R_term`.

Lumped Element B

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

2 In the **Settings** window for **Lumped Element**, type `Lumped Element B` in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `52` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

7 In the Z_{element} text field, type `R_term`.

8 Click the **Show More Options** button in the **Model Builder** toolbar.

9 In the **Show More Options** dialog box, select **Physics>Equation-Based Contributions** in the tree.

10 In the tree, select the check box for the node **Physics>Equation-Based Contributions**.

11 Click **OK**.

Pointwise Constraint 1

1 In the **Physics** toolbar, click **Edges** and choose **Pointwise Constraint**.

2 In the **Settings** window for **Pointwise Constraint**, locate the **Edge Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `35` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Pointwise Constraint**, locate the **Pointwise Constraint** section.

7 In the **Constraint expression** text field, type `0-tAx`.

Pointwise Constraint 2

1 Right-click **Pointwise Constraint 1** and choose **Duplicate**.

2 In the **Settings** window for **Pointwise Constraint**, locate the **Pointwise Constraint** section.

3 In the **Constraint expression** text field, type `0-tAy`.

Pointwise Constraint 3

1 Right-click **Pointwise Constraint 2** and choose **Duplicate**.

2 In the **Settings** window for **Pointwise Constraint**, locate the **Pointwise Constraint** section.

3 In the **Constraint expression** text field, type `0-tAz`.

Edge Current 1

1 In the **Physics** toolbar, click **Edges** and choose **Edge Current**.

2 Select Edge 101 only.

3 In the **Settings** window for **Edge Current**, locate the **Edge Current** section.

4 From the **Edge current type** list, choose **Lightning**.

5 In the v_p text field, type `vr`.

MESH 1

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**.

Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

2 Drag and drop below **Size**.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `17, 20, 24, 27` in the **Selection** text field.

6 Click **OK**.

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 box, type 5-8 in the **Selection** text field.

6 Click **OK**.

Size 1

1 Right-click **Swept 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** check box. In the associated text field, type 3*ds.

6 Select the **Minimum element size** check box. In the associated text field, type ds.

7 Click **Build All**.

Size 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Mesh 1>Free Tetrahedral 1** node.

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

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

4 From the **Geometric entity level** list, choose **Edge**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type 101 in the **Selection** text field.

7 Click **OK**.

8 In the **Settings** window for **Size**, locate the **Element Size** section.

9 Click the **Custom** button.

10 Locate the **Element Size Parameters** section.

11 Select the **Maximum element size** check box. In the associated text field, type 5[m].

Size 2

1 In the **Model Builder** window, 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 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type 10, 52 in the **Selection** text field.

6 Click **OK**.

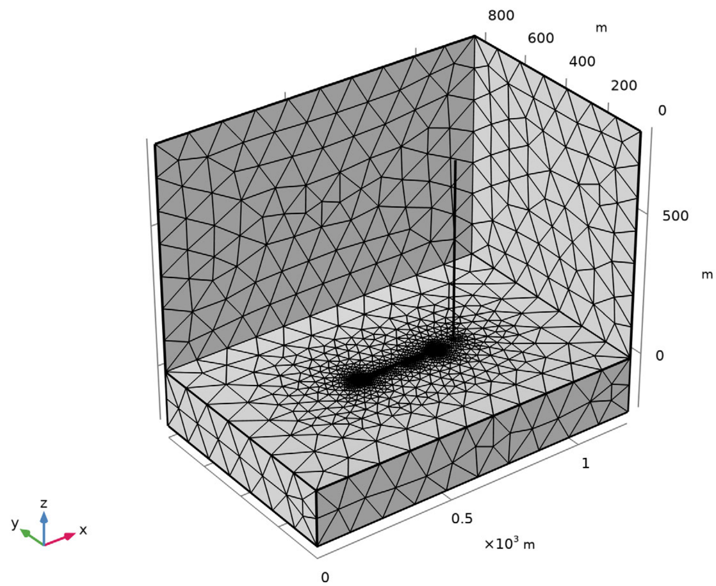
7 In the **Settings** window for **Size**, locate the **Element Size** section.

8 Click the **Custom** button.

9 Locate the **Element Size Parameters** section.

10 Select the **Maximum element size** check box. In the associated text field, type `S_capacitor*0.6`.

11 Click **Build All**.



STUDY 1

1 In the **Model Builder** window, click **Study 1**.

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

3 Clear the **Generate default plots** check box.

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 From the **Time unit** list, choose **μs**.

4 In the **Output times** text field, type `range(0,0.01,3)`.

5 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sigma_soil (Soil conductivity)	0.01 0.02	S/m

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Manual**.

5 In the **Time step** text field, type `0.01[us]`.

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

RESULTS

Electric Field

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Electric Field` in the **Label** text field.

Surface 1

1 Right-click **Electric Field** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1-3, 6, 8, 9, 12, 42, 58, 59` in the **Selection** text field.

5 Click **OK**.

Electric Field

1 In the **Model Builder** window, under **Results** click **Electric Field**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** check box.

Line 1

1 Right-click **Electric Field** and choose **Line**.

2 In the **Settings** window for **Line**, locate the **Coloring and Style** section.

3 From the **Line type** list, choose **Tube**.

Selection 1

1 Right-click **Line 1** and choose **Selection**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 101 in the **Selection** text field.

5 Click **OK**.

Line 2

In the **Model Builder** window, right-click **Electric Field** and choose **Line**.

Selection 1

1 In the **Model Builder** window, right-click **Line 2** and choose **Selection**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 35 in the **Selection** text field.

5 Click **OK**.

Electric Field

1 Click the **Show Grid** button in the **Graphics** toolbar.

2 Click the **Show Legends** button in the **Graphics** toolbar.

3 Click the **Show Grid** button in the **Graphics** toolbar.

4 Click the **Show Grid** button in the **Graphics** toolbar.

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

6 In the **Model Builder** window, under **Results** click **Electric Field**.

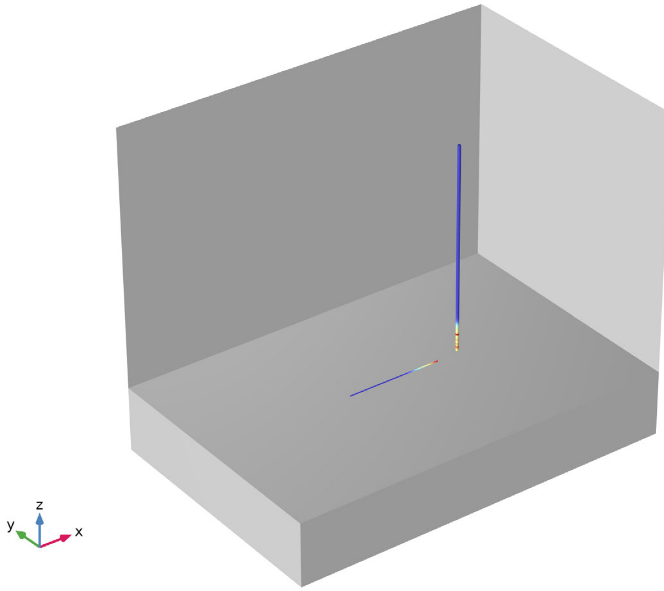
7 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

8 From the **Title type** list, choose **None**.

9 Locate the **Data** section. From the **Time (µs)** list, choose **1**.

10 In the **Electric Field** toolbar, click **Plot**.

11 Click the **Go to Default View** button in the **Graphics** toolbar.



Induced Voltage

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

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

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** check box. In the associated text field, type `Time, µs`.

5 Select the **y-axis label** check box. In the associated text field, type `Induced Voltage, kV`.

6 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 Right-click **Induced Voltage** 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
temw.Velement_1	kV	End A
temw.Velement_2	kV	End B

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

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

6 In the **Induced Voltage** toolbar, click **Plot**.