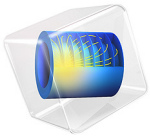


A Geoelectrical Forward Problem

Introduction

Electrical resistivity tomography (ERT) is a geophysical method used for imaging the resistivity of soil and rocks in the subsurface, based on measurement of electrical quantities by means of electrodes at the surface or inside boreholes. The method has its roots in the vertical electrical sounding techniques (VES) (Ref. 1) and exploits the fact that resistivity varies over several orders of magnitude in different types of rock, in particular depending on the conductivity and quantity of fluids and minerals contained in them.

ERT is today routinely applied to investigate targets from diverse applications such as mining, oil and gas exploration, groundwater, waste disposal, landslides, dams and embankments, geological faults, and volcanoes and archaeology, to name but a few. The method is closely related to electrical impedance tomography (EIT) as it is used in medical imaging or nondestructive material testing. One peculiarity of geophysical ERT is, however, that the boundaries of the target region are typically far away from the measurement system, virtually at infinity.

Typically, the physical domain in which the geoelectrical survey takes place is so large that it can be considered unbounded to the sides and the bottom. This property must be properly taken into account during modeling.

A single geoelectrical measurement involves the injection of a current I through two point-like electrodes, typically called C_1 and C_2, and the measurements of the potential difference V_{12} between two other electrodes (P_1 and P_2) resulting in a measured resistance $R = V_{12}/I$. This application describes the 3D ERT forward problem for 25 electrodes in a homogeneous ground of resistivity 100 Ω·m, and compares the result with the analytical solution.

Although technically a low-frequency alternating current is used, inductive and capacitive effects are typically ignored and the governing equation is

$$\nabla \cdot (-\sigma \nabla V) = I \cdot (\delta(\mathbf{r} - \mathbf{r}_{C1}) - \delta(\mathbf{r} - \mathbf{r}_{C2})) \tag{1}$$

Here, V represents the electric potential, σ is the subsurface electrical conductivity, and $I \cdot (\delta(\mathbf{r} - \mathbf{r}_{C1}) - \delta(\mathbf{r} - \mathbf{r}_{C2}))$ are point current sources from the electrodes located at positions $\mathbf{r}_{C1}$ and $\mathbf{r}_{C2}$ (actually, one electrode acts as current source, and the other as current sink).

The goal of ERT is to determine the unknown subsurface resistivity from measurements with a multitude of electrode configurations, $R(C_1,C_2,P_1,P_2)_i$, $i = 1,\ldots, N$. The procedure typically involves two steps:

- The prediction of measurement responses on a given model (forward step)
- The correction of the model to achieve a better fit (inverse step)

The procedure is repeated iteratively until an acceptable fit between predicted and measured data, and possibly an agreement with additional constraints (for example, smoothness or *a priori* information), are found.

While the forward step is well defined by Equation 1, many algorithms have been proposed for the inverse step. Most commonly, gradient-based inverse methods are used (Ref. 2), which require the calculation of a parameter sensitivity matrix S that describes how the measured resistance would be affected by the change of a parameter (typically the resistivity in a certain subsurface region).

It can be shown (Ref. 3) that the measured resistance can also be expressed as

$$R_i = \int_\Omega S_i(\mathbf{r}, \rho)\rho(\mathbf{r})dV$$

with the sensitivity function S_i defined as

$$S_i(\mathbf{r}, \rho) = J(\mathbf{r}, \mathbf{r}_{C1,i}, \mathbf{r}_{C2,i}) \cdot J'(\mathbf{r}, \mathbf{r}_{P1,i}, \mathbf{r}_{P2,i}) \tag{2}$$

which is given by the reciprocity theorem (Ref. 4). Here, $\mathbf{r}_{C1}$ and $\mathbf{r}_{C2}$ designate the positions of the electrodes injecting the electric current into the ground, and $\mathbf{r}_{P1}$ and $\mathbf{r}_{P2}$ are the position for the electrodes measuring the difference in electric potential V_{12}, for a given configuration "i".

Model Definition

The model illustrates the forward step and the sensitivity calculation for a typical situation of near-surface ERT with 25 point electrodes spaced one meter from each other. The model is geometrically parameterized to be easily adaptable to various sizes.

At the sides and at the bottom of the computational region, the infinite element domains account for the fact that the domain is not bounded.

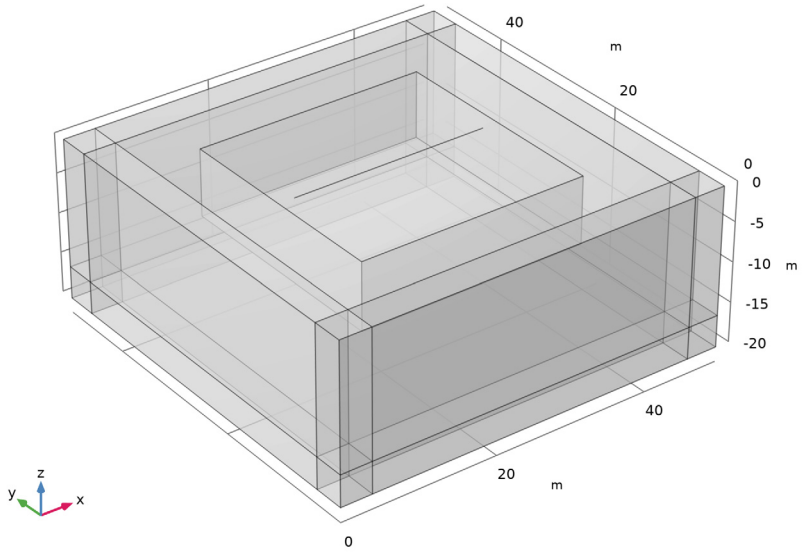


Figure 1: Computational domain. A line of 25 electrodes is placed at the top of a 50-by-50-by-20 meters box bounded by infinite element regions.

As a material, a homogeneous half-space that often serves as a reference model for forward algorithms is used. The electric conductivity is $\sigma = 0.01$ S/m, which corresponds to a resistivity of 100 Ωm.

The solution is compared to the analytical solution

$$V = \frac{I}{2\pi\sigma}\left(\frac{1}{|\mathbf{r}-\mathbf{r}_{C1}|} - \frac{1}{|\mathbf{r}-\mathbf{r}_{C2}|}\right) \tag{3}$$

To achieve sufficient accuracy, the mesh density is refined directly at the electrodes and in the region of particular interest underneath them. A swept mesh is used in infinite element regions; see Figure 2.

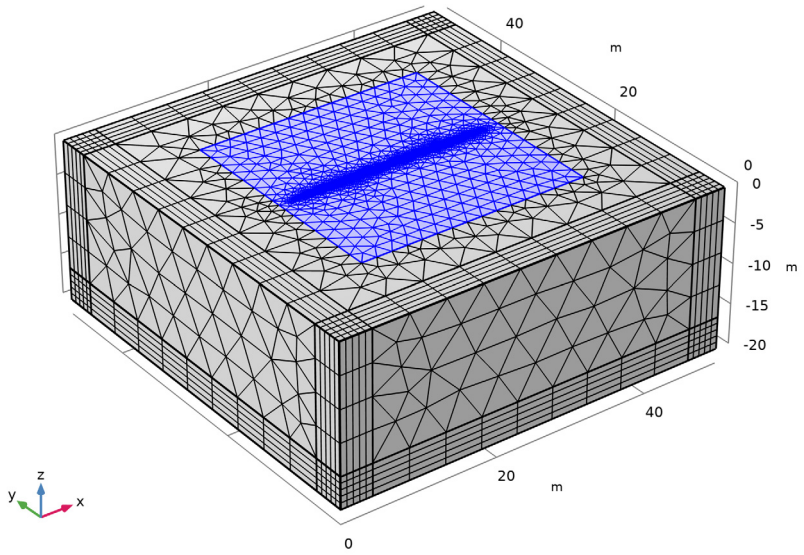


Figure 2: Refined mesh around the electrodes and swept mesh in infinite element regions.

Use the continuation solver to switch between different electrode configurations.

Results and Discussion

Figure 3 shows the potential distribution around the active pair for the second of the two electrode configurations that the model solves for.

Figure 4 compares the simulation results for the potential at the electrode positions for both electrode configurations to the analytical solution given by Equation 3.

Figure 5 shows the relative error of the computed solutions with respect to the analytical solutions given by Equation 3 for the two electrode configurations. The plot is taken along the line joining the point sources.

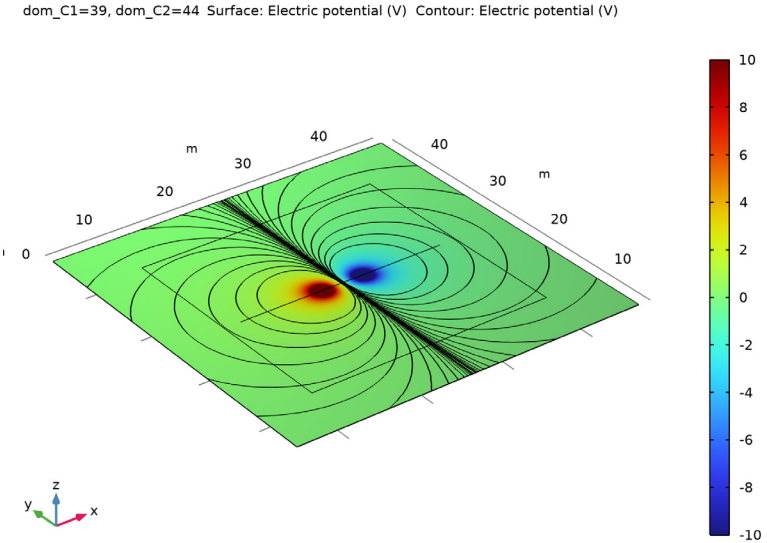


Figure 3: Electric potential around the active electrode pair.

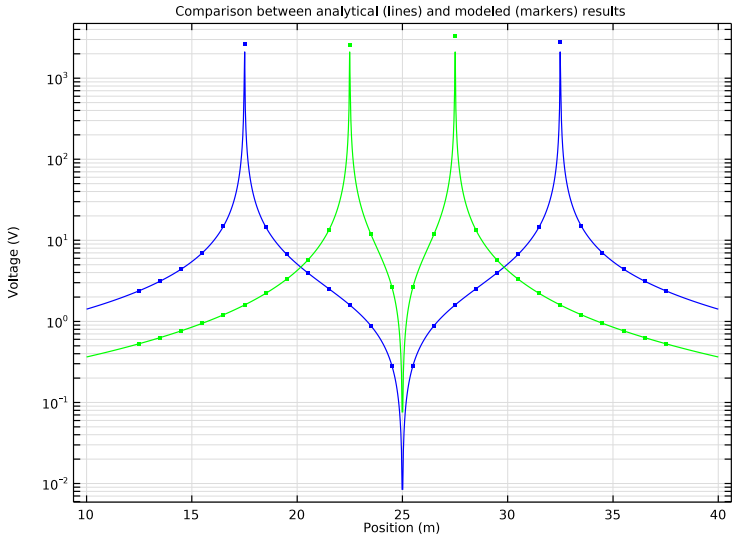


Figure 4: Comparison between analytical (lines) and modeled (markers) results.

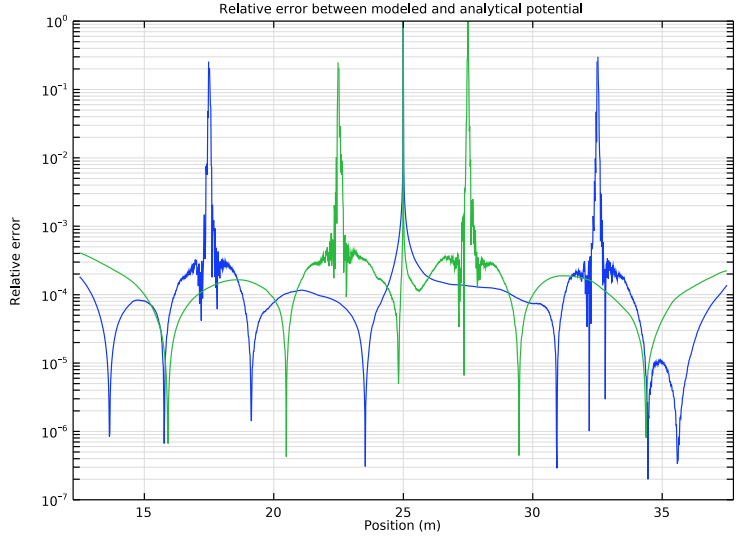


Figure 5: The relative error of the computed solutions with respect to the analytical solutions for the two electrode configurations. The plot is taken along the line joining the point sources.

The model solves for two current dipole situations with a common midpoint. Together they allow the calculation of the sensitivity of a Wenner-α configuration according to Equation 2.

```
S = with(1,ec.Jx)*with(2,ec.Jx) + with(1,ec.Jy)*with(2,ec.Jy) +
    with(1,ec.Jz)*with(2,ec.Jz))
```

The `with` operator makes reference to the modeled configuration (`1` or `2` in the first argument of the operator). Figure 3 shows a plot of this expression.

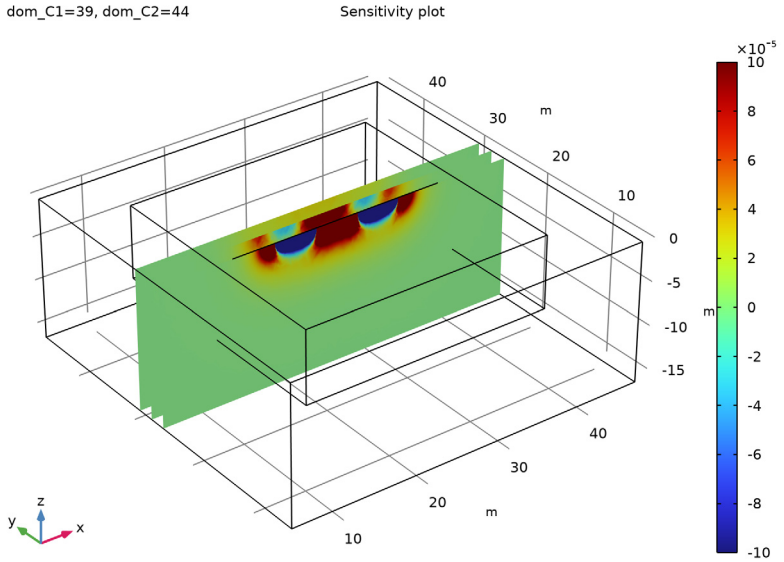


Figure 6: Sensitivity plot.

References

1. F. Wenner, "A Method of Measuring Earth Resistivity," *National Bureau of Standards, Bull.* 12(4) 258, pp. 478–496, 1915.

2. M.H. Loke and R.D. Barker, "Practical techniques for 3D resistivity surveys and data inversion," *Geophysical Prospecting*, vol. 44, pp. 499–523, 1996.

3. S. Friedel, "Resolution, Stability and Efficiency of Resistivity Tomography Estimated from a Generalized Inverse Approach," *Geophys. J. Int.*, vol. 153, pp. 305–316, 2003.

4. D.B. Geselowitz, "An application of electrocardiographic lead theory to impedance plethysmography," *IEEE Trans. Biomed. Eng.* BME-18, pp. 38–41, 1971.

Application Library path: `ACDC_Module/Devices,_Resistive/geoelectrics`

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 **AC/DC>Electric Fields and Currents>Electric Currents (ec)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Add parameters that are useful for defining physical and geometric quantities.

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	50[m]	50 m	Domain length
W	50[m]	50 m	Domain width
H	20[m]	20 m	Domain height
WI	4[m]	4 m	Infinite layer thickness
N	25	25	Number of electrodes
a	1[m]	1 m	Electrode separation
x0	L/2-N/2*a	12.5 m	First electrode x-position
y0	W/2	25 m	First electrode y-position
rho0	100[m/S]	100 Ω·m	Resistivity

Add two extra parameters which are going to be useful for sweeping among excitations applied to different entities. For further details, look at coming comments in these instructions, where the parameters are used.

4 In the table, enter the following settings:

Name	Expression	Value	Description
dom_C1	39	39	Positive electrode domain number
dom_C2	44	44	Negative electrode domain number

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 L.

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

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

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

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

Layer name	Thickness (m)
Layer 1	WI

8 Find the **Layer position** subsection. Select the **Left** check box.

9 Select the **Right** check box.

10 Select the **Front** check box.

11 Select the **Back** check box.

12 Click **Build Selected**.

Polygon 1 (pol1)

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

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

3 From the **Data source** list, choose **Vectors**.

4 In the **x** text field, type `range(x0,a,x0+N*a)`.

5 In the **y** text field, type `y0+0*range(0,1,N)`.

6 In the **z** text field, type `0*range(0,1,N)`.

7 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 `(N+4)*a`.

4 In the **Depth** text field, type `(N+4)*a`.

5 In the **Height** text field, type `N*a/3`.

6 Locate the **Position** section. In the **x** text field, type `x0-2*a`.

7 In the **y** text field, type `y0-(N+4)*a/2`.

8 In the **z** text field, type `-N*a/3`.

9 Click **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

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

4 Click the **Transparency** button in the **Graphics** toolbar to see the interior of the geometry.

The completed geometry is shown in Figure 1.

5 Click the **Transparency** button in the **Graphics** toolbar to return to the default state.

DEFINITIONS

Analytic 1 (an1)

1. In the **Home** toolbar, click f(x) **Functions** and choose **Local>Analytic**.
2. In the **Settings** window for **Analytic**, type `V_ref` in the **Function name** text field.
3. Locate the **Definition** section. In the **Expression** text field, type `1[A]*rho0/(2*pi)*(1/abs(x-x1)-1/abs(x-x2))`.
4. In the **Arguments** text field, type `x, x1, x2`.

   The analytical solution given by `V_ref` will be used to validate the computed solution.

Ground Boundaries

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 all the exterior boundaries located in the ground. This operation can be performed quickly by selecting the **Group by continuous tangent** check box, then clicking on one boundary from each of the sides and the bottom of the box.
5. In the **Label** text field, type `Ground Boundaries`.

Infinite Element Domains

1. In the **Definitions** toolbar, click **Explicit**.
2. Select Domains 1–9, 11, 12, and 14–19 only.
3. In the **Settings** window for **Explicit**, type `Infinite Element Domains` in the **Label** text field.

Infinite Elements Boundaries

1. In the **Definitions** toolbar, click **Explicit**.
2. Select Domains 1–8, 11, 12, and 14–19 only.
3. In the **Settings** window for **Explicit**, locate the **Output Entities** section.
4. From the **Output entities** list, choose **Adjacent boundaries**.
5. Select the **Interior boundaries** check box.
6. In the **Label** text field, type `Infinite Elements Boundaries`.

Infinite Element Domain 1 (ie1)

1. In the **Definitions** toolbar, click **Infinite Element Domain**.
2. In the **Settings** window for **Infinite Element Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Infinite Element Domains**.

Set the physical size of the infinite elements domain to be 100 times the size of the geometry. Larger sizes can give even more accurate results but will require a more refined mesh to be resolved.

4 Locate the **Scaling** section. In the **Physical width** text field, type `1e2*dGeomChar`.

MATERIALS

100 Ohmm Homogeneous

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

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`1/rho0`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

4 In the **Label** text field, type `100 Ohmm Homogeneous`.

ELECTRIC CURRENTS (EC)

Point Current Source 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electric Currents (ec)** and choose **Points>Point Current Source**.

2 Select Points 29–54 only.

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

4 In the $Q_{j,p}$ text field, type `1*(dom==dom_C1)-1*(dom==dom_C2)`.

The `dom` variable is a built-in variable that represents the domain number. It identifies uniquely each entity among the specified dimensionality. In this model, it is used to select different points, as on point 1 it has a value of 1; in point 2, it has a value of 2; and so on. The == operator, together with a sweep over a parameter, allows for scanning

over different sources, identifying them by their domain number. Point number `dom_C1` injects a unit charge. Point number `dom_C2` extracts a unit charge. See the *COMSOL Multiphysics Reference Manual* for more information about built-in variables and operators.

5 Locate the **Point Selection** section. Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Point sources` in the **Selection name** text field.

7 Click **OK**.

Ground 1

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

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

3 From the **Selection** list, choose **Ground Boundaries**.

MESH 1

Size

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size 1

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 **Edge**.

4 Select Edges 67–91 only.

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 `0.05`.

Size 2

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

2 Select Domain 13 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** check box. In the associated text field, type `2`.

7 Click **Build All**.

The resulting mesh is shown in Figure 2.

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**, click to expand the **Study Extensions** section.

Add a parametric sweep on the charge source and sink. It can be seen that only one Jacobian is going to be computed, making the solution time for the parameter after the first one faster. A further time improvement can be achieved by forcing a direct solver, as the whole LU decomposition of first step would be reused in that case. This improved solution time would come at the cost of a larger memory usage.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dom_C1 (Positive electrode domain number)	34 39	

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dom_C2 (Negative electrode domain number)	49 44	

The continuation solver is not needed for this sweep.

8 From the **Run continuation for** list, choose **No parameter**.

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

RESULTS

Electric Potential (ec)

The default volume plot of the electric potential is useful when checking for general modeling errors. To adapt the plot for this application, modify it as follows:

Volume 1

1 In the **Model Builder** window, expand the **Electric Potential (ec)** node.

2 Right-click **Results>Electric Potential (ec)>Volume 1** and choose **Delete**.

Electric Potential (ec)

In the **Model Builder** window, under **Results** click **Electric Potential (ec)**.

Multislice 1

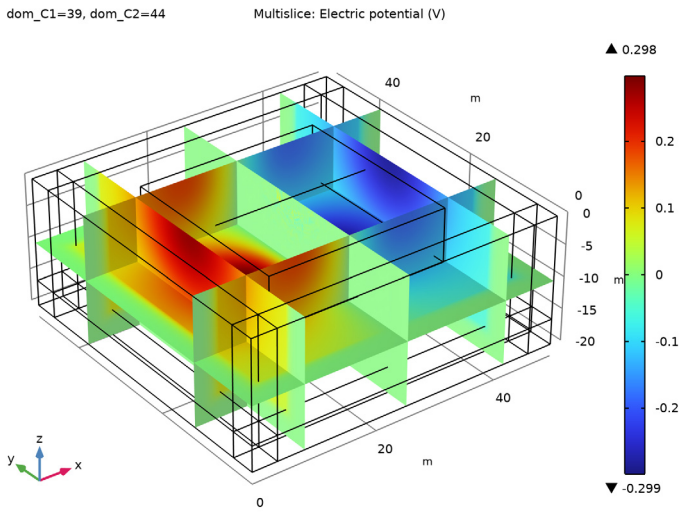
1 In the **Electric Potential (ec)** toolbar, click **More Plots** and choose **Multislice**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. In the **Planes** text field, type 3.

4 Find the **y-planes** subsection. In the **Planes** text field, type 2.

5 In the **Electric Potential (ec)** toolbar, click **Plot**.



Follow the instructions below to create tailored plots for this application.

DEFINITIONS

Hide for Physics 1

1 In the **Model Builder** window, right-click **View 1** and choose **Hide for Physics**.

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

3 From the **Selection** list, choose **Infinite Element Domains**.

Hide for Physics 2

1 Right-click **View 1** and choose **Hide for Physics**.

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

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

4 From the **Selection** list, choose **Infinite Elements Boundaries**.

Start by reproducing the sensitivity plot shown in Figure 3.

RESULTS

3D Plot Group 3

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

Slice 1

1 Right-click **3D Plot Group 3** and choose **Slice**.

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

3 In the **Expression** text field, type `with(1,ec.Jx)*with(2,ec.Jx)+with(1,ec.Jy)*with(2,ec.Jy)+with(1,ec.Jz)*with(2,ec.Jz)`.

4 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

5 From the **Entry method** list, choose **Coordinates**.

6 In the **y-coordinates** text field, type `25 27 29`.

7 Click to expand the **Range** section. Select the **Manual color range** check box.

8 In the **Minimum** text field, type `-1E-4`.

9 In the **Maximum** text field, type `1E-4`.

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

Sensitivity

1 In the **Model Builder** window, click **3D Plot Group 3**.

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

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Sensitivity plot`.

5 In the **Label** text field, type `Sensitivity`.

Next, plot the electric potential on the surface.

Study 1/Solution 1 (2) (sol1)

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

2 Right-click **Results>Datasets>Study 1/Solution 1 (sol1)** 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 Select Boundaries 37 and 50 only.

Electric Potential at the Surface

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

Surface 1

1 Right-click **Electric Potential at the Surface** and choose **Surface**.

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual color range** check box.

4 In the **Minimum** text field, type `-10`.

5 In the **Maximum** text field, type `10`.

Contour 1

1 In the **Model Builder** window, right-click **Electric Potential at the Surface** and choose **Contour**.

2 In the **Settings** window for **Contour**, locate the **Levels** section.

3 From the **Entry method** list, choose **Levels**.

4 In the **Levels** text field, type `-10^(range(0,-0.2,-3)) 10^(range(-3,0.2,0))`.

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

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

7 Clear the **Color legend** check box.

8 In the **Electric Potential at the Surface** toolbar, click **Plot**.

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

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.

3 From the **Plane** list, choose **zx-planes**.

4 In the **y-coordinate** text field, type `y0`.

Electric Potential, Slice

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Plane 1**.

4 From the **Parameter value (dom_C1,dom_C2)** list, choose **1: dom_C1=34, dom_C2=49**.

Surface 1

1 Right-click **Electric Potential, Slice** and choose **Surface**.

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

3 Select the **Manual color range** check box.

4 In the **Minimum** text field, type `-10`.

5 In the **Maximum** text field, type `10`.

Contour 1

1 In the **Model Builder** window, right-click **Electric Potential, Slice** and choose **Contour**.

2 In the **Settings** window for **Contour**, locate the **Levels** section.

3 From the **Entry method** list, choose **Levels**.

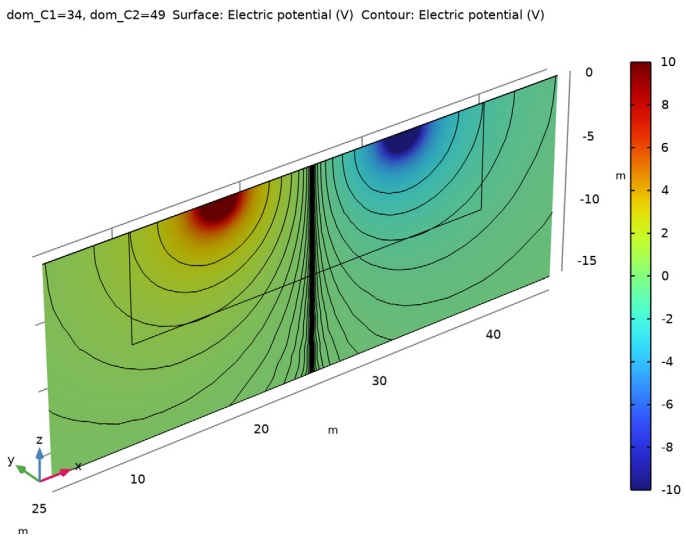
4 In the **Levels** text field, type `-10^(range(0.4,-0.2,-3)) 10^(range(-3,0.2,0.4))`.

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

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

7 Clear the **Color legend** check box.

8 In the **Electric Potential, Slice** toolbar, click **Plot**.



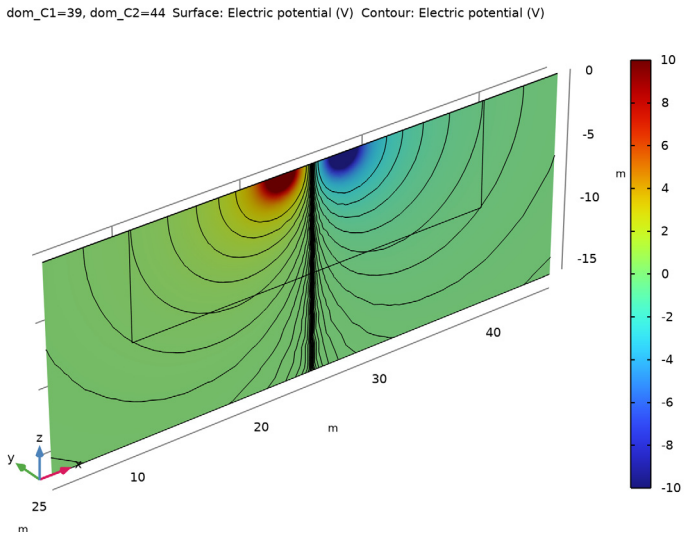
Electric Potential, Slice

1 In the **Model Builder** window, click **Electric Potential, Slice**.

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

3 From the **Parameter value (dom_C1,dom_C2)** list, choose **2: dom_C1=39, dom_C2=44**.

4 In the **Electric Potential, Slice** toolbar, click **Plot**.



Proceed to reproduce the comparison plot in Figure 4. Begin by adding a Function dataset for the analytic solution.

Grid 1D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Grid>Grid 1D**.

2 In the **Settings** window for **Grid 1D**, locate the **Data** section.

3 From the **Source** list, choose **Function**.

4 From the **Function** list, choose **Analytic 1 (V_ref)**.

5 Locate the **Parameter Bounds** section. In the **Minimum** text field, type `10`.

6 In the **Maximum** text field, type `40`.

Result Comparison

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

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

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Comparison between analytical (lines) and modeled (markers) results`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `Position (m)`.

7 Select the **y-axis label** check box. In the associated text field, type `Voltage (V)`.

Point Graph 1

1 Right-click **Result Comparison** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (1) (sol1)**.

4 From the **Parameter selection (dom_C1, dom_C2)** list, choose **First**.

5 Locate the **Selection** section. From the **Selection** list, choose **Point sources**.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(V)`.

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

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

9 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Blue**.

10 Click to expand the **Legends** section.

Point Graph 2

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

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

3 From the **Parameter selection (dom_C1, dom_C2)** list, choose **Last**.

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

Line Graph 1

1 In the **Model Builder** window, right-click **Result Comparison** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Grid 1D 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(V_ref(x,17.5, 32.5))`.

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

6 In the **Expression** text field, type `x`.

7 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Blue**.

Line Graph 2

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

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

3 In the **Expression** text field, type `abs(V_ref(x,22.5,27.5))`.

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

5 In the **Result Comparison** toolbar, click **Plot**.

Change to log scale.

6 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

7 In the **Result Comparison** toolbar, click **Plot**.

Finally, reproduce the relative error plots in Figure 5.

Relative Error

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

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

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Relative error between modeled and analytical potential`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `Position (m)`.

7 Select the **y-axis label** check box. In the associated text field, type `Relative error`.

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

9 Select the **y-axis log scale** check box.

10 In the **x minimum** text field, type `12.25`.

11 In the **x maximum** text field, type `37.75`.

12 In the **y minimum** text field, type `1e-7`.

Line Graph 1

1 Right-click **Relative Error** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (1) (sol1)**.

4 From the **Parameter selection (dom_C1, dom_C2)** list, choose **First**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(V-V_ref(x,17.5,32.5))/abs(V)`.

6 Select Edges 67–91 only.

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

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

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 **Parameter selection (dom_C1, dom_C2)** list, choose **Last**.
4. Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(V-V_ref(x,22.5, 27.5))/abs(V)`.
5. In the **Relative Error** toolbar, click **Plot**.