

A Geoelectrical Forward Problem

Introduction

Electrical resistivity tomography (ERT) is a geophysical method used for the 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 archeology, 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 $100 \Omega\text{-m}$, 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

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

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, $R(C_1, C_2, P_1, P_2)_i$, $i = 1, \dots, N$. 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

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

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.

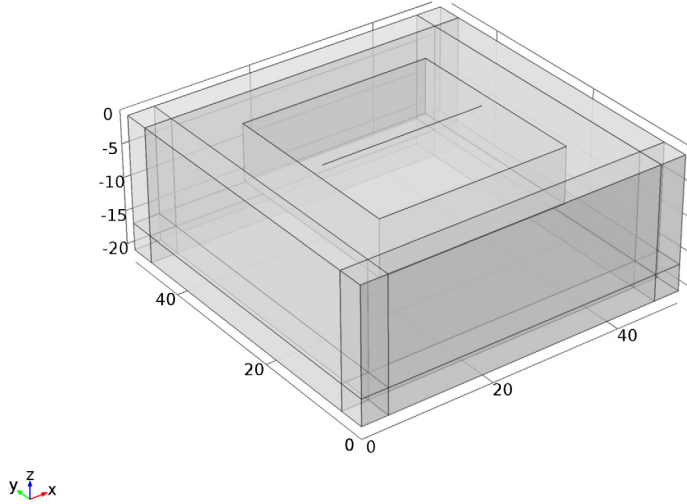


Figure 1: Computational domain. A line of 25 electrodes is placed at the top of a 50-by-50-by-20 meters box bounded by infinite element regions.

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 $100 \Omega\text{m}$.

The solution is compared to the analytical solution

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

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](#).

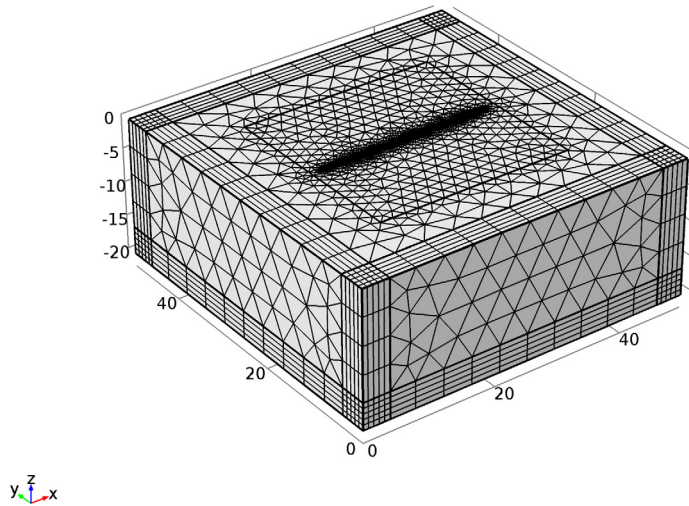


Figure 2: Refined mesh around the electrodes and swept mesh in infinite element regions.

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.

2: dom_C1=39, dom_C2=44 Surface: Electric potential (V) Contour: Electric potential (V)

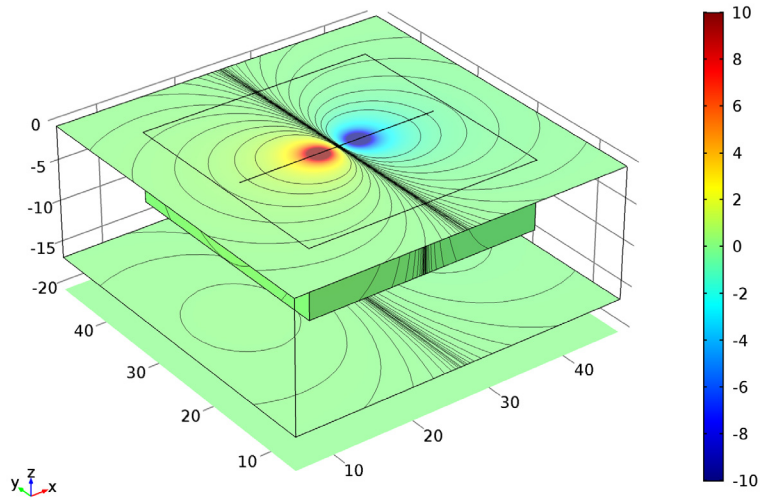


Figure 3: Electric potential around the active electrode pair.

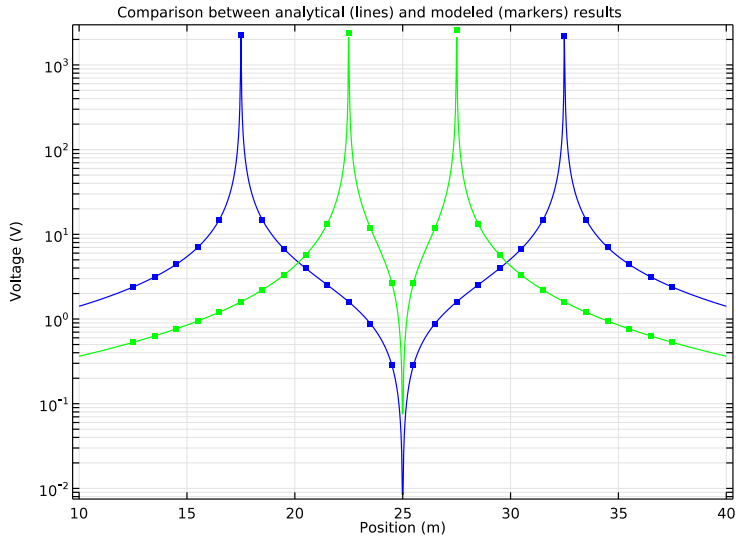


Figure 4: Comparison between analytical (lines) and modeled (markers) results.

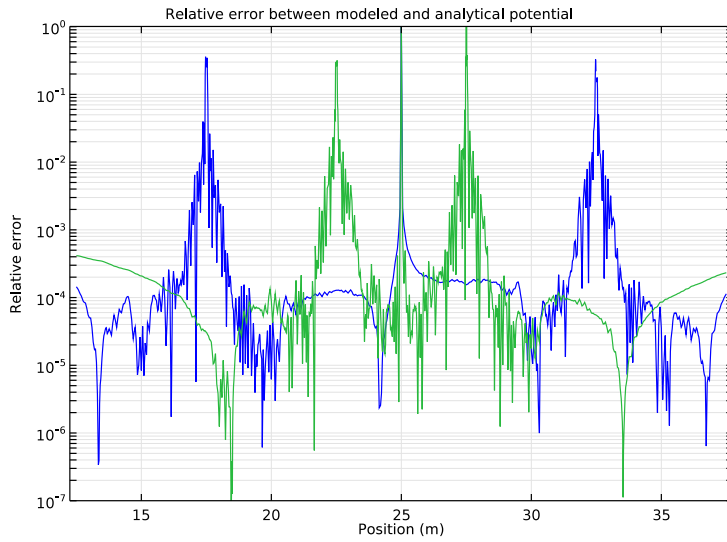


Figure 5: The relative error of the computed solutions with respect to the analytical solutions for the two electrode configurations. The plot is taken along the line joining the point sources.

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.

$$S = \text{with}(1, \text{ec.Jx}) * \text{with}(2, \text{ec.Jx}) + \text{with}(1, \text{ec.Jy}) * \text{with}(2, \text{ec.Jy}) + \text{with}(1, \text{ec.Jz}) * \text{with}(2, \text{ec.Jz})$$

The with operator makes reference to the modeled configuration (1 or 2 in the first argument of the operator). Figure 6 shows a plot of this expression.

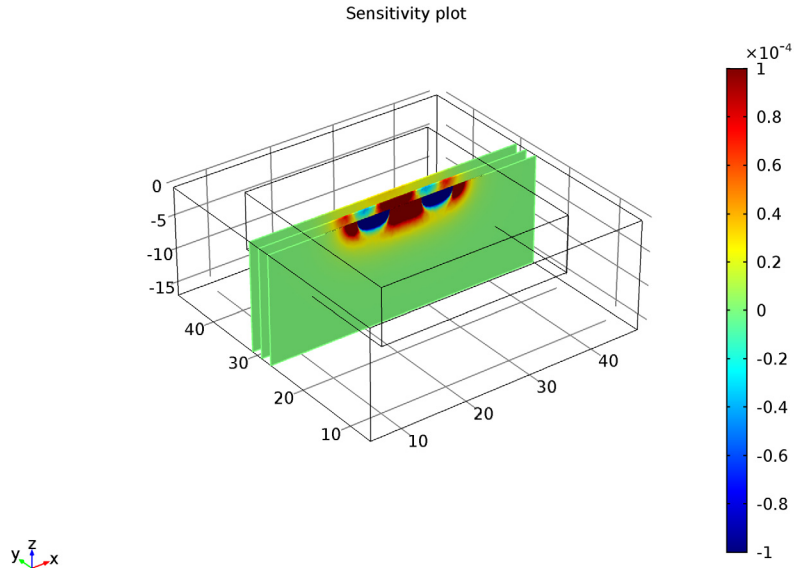


Figure 6: Sensitivity plot.

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/Other_Industrial_Applications/geoelectrics

Modeling Instructions

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

NEW

1 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 Currents (ec)**.
- 3 Click **Add**.
- 4 Click **Study**.
- 5 In the **Select study** tree, select **Preset Studies>Stationary**.
- 6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

- 1 On the **Home** toolbar, click **Parameters**.
- 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

Name	Expression	Value	Description
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 $\Omega\cdot\text{m}$	Resistivity
conf	1	1	Active configuration
dom_C1	39	39	Positive electrode domain number
dom_C2	44	44	Negative electrode domain number

GEOMETRY I

Block 1 (blk1)

- 1 On 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.

Polygon 1 (pol1)

- 1 Right-click **Block 1 (blk1)** and choose **Build Selected**.
- 2 On the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.
- 3 In the **Settings** window for Polygon, locate the **Coordinates** section.
- 4 In the **x** text field, type $\text{range}(x0, a, x0+N*a)$.
- 5 In the **y** text field, type $y0+0*\text{range}(0, 1, N)$.
- 6 In the **z** text field, type $0*\text{range}(0, 1, N)$.

Block 2 (blk2)

- 1 Right-click **Polygon 1 (pol1)** and choose **Build Selected**.
- 2 On the **Geometry** toolbar, click **Block**.
- 3 In the **Settings** window for Block, locate the **Size and Shape** section.
- 4 In the **Width** text field, type $(N+4) * a$.
- 5 In the **Depth** text field, type $(N+4) * a$.
- 6 In the **Height** text field, type $N * a / 3$.
- 7 Locate the **Position** section. In the **x** text field, type $x0 - 2 * a$.
- 8 In the **y** text field, type $y0 / 2 - 2 * a$.
- 9 In the **z** text field, type $-N * a / 3$.

Form Union (fin)

- 1 Right-click **Block 2 (blk2)** and choose **Build Selected**.
- 2 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.
- 3 Click the **Go to Default 3D View** button on the **Graphics** toolbar.
- 4 Click the **Transparency** button on the **Graphics** toolbar to see the interior of the geometry.

The completed geometry is shown in [Figure 1](#).

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

DEFINITIONS*Analytic 1 (an1)*

- 1 On the **Home** toolbar, click **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 , $x1$, $x2$.

Later, you will compare the simulation results with the analytical solution given by V_ref .

Explicit 1

- 1 On 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 Boundaries 1–5, 8, 10, 11, 15, 17, 18, 22, 23, 25, 26, 28, 33, 40, 45, 46, 54, 55, 57, 62, 69, and 74–81 only.
- 5 Right-click **Explicit 1** and choose **Rename**.
- 6 In the **Rename Explicit** dialog box, Right-click **Explicit 1** and choose **Rename**.
- 7 type Ground boundaries in the **New label** text field.
- 8 Click **OK**.

Explicit 2

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 Select Domains 1–9, 11, 12, and 14–19 only.
- 3 Right-click **Explicit 2** and choose **Rename**.
- 4 In the **Rename Explicit** dialog box, Right-click **Explicit 2** and choose **Rename**.
- 5 type Infinite element domains in the **New label** text field.
- 6 Click **OK**.

Explicit 3

- 1 On 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 Right-click **Explicit 3** and choose **Rename**.
- 7 In the **Rename Explicit** dialog box, Right-click **Explicit 3** and choose **Rename**.
- 8 type Infinite element boundaries in the **New label** text field.
- 9 Click **OK**.

Infinite Element Domain 1 (iel)

- 1 On 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**.

MATERIALS*Material 1 (mat1)*

- 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	Name	Value	Unit	Property group
Electrical conductivity	sigma	1/rho0	S/m	Basic
Relative permittivity	epsilon	1	1	Basic

- 4 Right-click **Component 1 (comp1)>Materials>Material 1 (mat1)** and choose **Rename**.
- 5 In the **Rename Material** dialog box, Right-click **Component 1 (comp1)>Materials>Material 1 (mat1)** and choose **Rename**.
- 6 type 100 Ohmm homogeneous in the **New label** text field.
- 7 Click **OK**.

ELECTRIC CURRENTS (EC)*Point Current Source 1*

- 1 On the **Physics** toolbar, click **Points** and choose **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 text field, type $1*(\text{dom}==\text{dom_C1}) - 1*(\text{dom}==\text{dom_C2})$.
- 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 On 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

Free Tetrahedral 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.
- 2 In the **Settings** window for Free Tetrahedral, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domains 10 and 13 only.

Size 1

- 1 Right-click **Component 1 (comp1)>Mesh 1>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 **Point**.
- 4 From the **Selection** list, choose **Point sources**.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 7 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. Select the **Maximum element size** check box.
- 6 In the associated text field, type 2.

Swept 1

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.
- 2 In the **Settings** window for Swept, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domains 4, 8, 9, 12, and 17 only.
- 5 Right-click **Component 1 (comp1)>Mesh 1>Swept 1** and choose **Distribution**.

Swept 2

- 1 Right-click **Mesh 1** and choose **Swept**.
- 2 In the **Settings** window for Swept, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domains 2, 3, 6, 7, 11, 15, 16, and 19 only.

Swept 3

- 1 Right-click **Mesh 1** and choose **Swept**.
- 2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Swept 3** and choose **Build All**.
- 3 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

The resulting mesh is shown in [Figure 2](#).

STUDY 1*Step 1: Stationary*

- 1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.
- 2 In the **Settings** window for Stationary, click to expand the **Study extensions** section.
- 3 Locate the **Study Extensions** section. 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	34 39	

- 6 Click **Add**.
- 7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dom_C2	49 44	

The continuation solver is not needed for this sweep.

- 8 From the **Run continuation for** list, choose **No parameter**.
- 9 On the **Home** toolbar, click **Compute**.

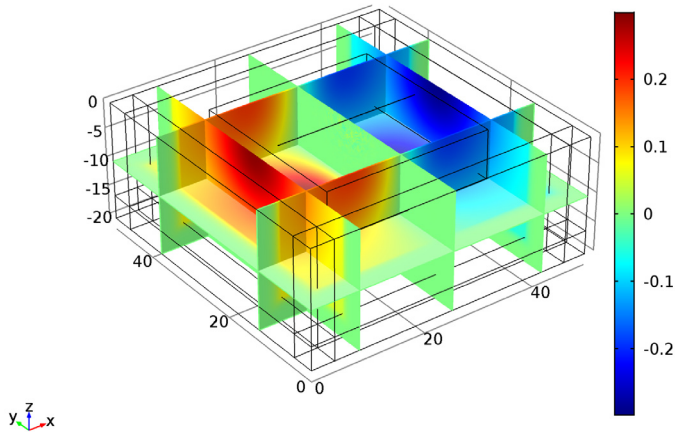
RESULTS

Electric Potential (ec)

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

- 1 In the **Model Builder** window, expand the **Electric Potential (ec)** node, then click **Multislice 1**.
- 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 On the **Electric Potential (ec)** toolbar, click **Plot**.

2: dom_C1=39, dom_C2=44 Multislice: Electric potential (V)



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

DEFINITIONS

View 1

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

Hide Geometric Entities 1

On the **View 1** toolbar, click **Hide Geometric Entities**.

View 1

- 1 In the **Settings** window for Hide Geometric Entities, locate the **Geometric Entity Selection** section.
- 2 From the **Selection** list, choose **Infinite element domains**.
- 3 In the **Model Builder** window, click **View 1**.

Hide Geometric Entities 2

On the **View 1** toolbar, click **Hide Geometric Entities**.

View 1

- 1 In the **Settings** window for Hide Geometric Entities, locate the **Geometric Entity Selection** section.
- 2 From the **Geometric entity level** list, choose **Boundary**.
- 3 From the **Selection** list, choose **Infinite element boundaries**.

Start by reproducing the sensitivity plot shown in [Figure 6](#).

RESULTS*3D Plot Group 2*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
- 2 In the **Model Builder** window, right-click **3D Plot Group 2** and choose **Slice**.
- 3 In the **Settings** window for Slice, locate the **Expression** section.
- 4 In the **Expression** text field, type

$$\text{with}(1, \text{ec.Jx}) * \text{with}(2, \text{ec.Jx}) + \text{with}(1, \text{ec.Jy}) * \text{with}(2, \text{ec.Jy}) + \text{with}(1, \text{ec.Jz}) * \text{with}(2, \text{ec.Jz}).$$
- 5 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.
- 6 From the **Entry method** list, choose **Coordinates**.
- 7 In the **y-coordinates** text field, type 25 27 29.
- 8 Click to expand the **Range** section. Select the **Manual color range** check box.
- 9 In the **Minimum** text field, type -1E-4.
- 10 In the **Maximum** text field, type 1E-4.
- 11 On the **3D Plot Group 2** toolbar, click **Plot**.
- 12 In the **Model Builder** window, click **3D Plot Group 2**.
- 13 In the **Settings** window for 3D Plot Group, click to expand the **Title** section.
- 14 From the **Title type** list, choose **Manual**.

- 15 In the **Title** text area, type **Sensitivity plot**.
- 16 Right-click **3D Plot Group 2** and choose **Rename**.
- 17 In the **Rename 3D Plot Group** dialog box, Right-click **3D Plot Group 2** and choose **Rename**.
- 18 type **Sensitivity** in the **New label** text field.
- 19 Click **OK**.

3D Plot Group 3

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
- 2 In the **Model Builder** window, right-click **3D Plot Group 3** and choose **Surface**.
- 3 In the **Settings** window for **Surface**, click to expand the **Range** section.
- 4 Select the **Manual color range** check box.
- 5 In the **Minimum** text field, type **-10**.
- 6 In the **Maximum** text field, type **10**.
- 7 Right-click **3D Plot Group 3** and choose **Contour**.
- 8 In the **Settings** window for **Contour**, locate the **Levels** section.
- 9 From the **Entry method** list, choose **Levels**.
- 10 In the **Levels** text field, type $-10^{(\text{range}(0, -0.2, -3))} \ 10^{(\text{range}(-3, 0.2, 0))}$.
- 11 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 12 From the **Color** list, choose **Black**.
- 13 Clear the **Color legend** check box.
- 14 On the **3D Plot Group 3** toolbar, click **Plot**.
- 15 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Cut Plane 1

On the **Results** toolbar, click **Cut Plane**.

Data Sets

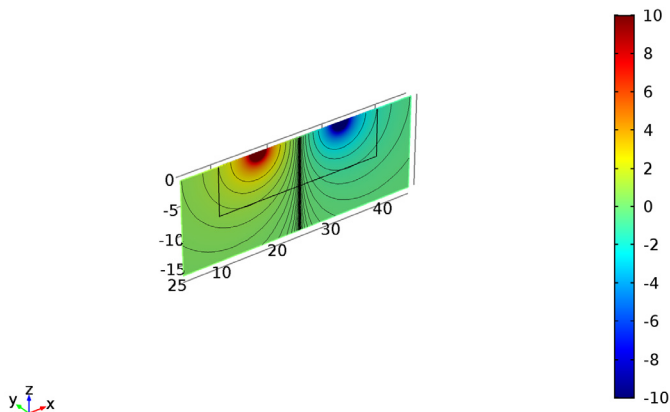
- 1 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.
- 2 From the **Plane** list, choose **zx-planes**.
- 3 In the **y-coordinate** text field, type **y0**.

3D Plot Group 4

- 1 On the **Results** toolbar, click **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

- 3 From the **Data set** list, choose **Cut Plane I**.
- 4 From the **Parameter value (dom_C1,dom_C2)** list, choose **1: dom_C1=34, dom_C2=49**.
- 5 Right-click **3D Plot Group 4** and choose **Surface**.
- 6 In the **Settings** window for Surface, locate the **Range** section.
- 7 Select the **Manual color range** check box.
- 8 In the **Minimum** text field, type -10.
- 9 In the **Maximum** text field, type 10.
- 10 Right-click **3D Plot Group 4** and choose **Contour**.
- 11 In the **Settings** window for Contour, locate the **Levels** section.
- 12 From the **Entry method** list, choose **Levels**.
- 13 In the **Levels** text field, type $-10^{(\text{range}(0.4, -0.2, -3))}$
 $10^{(\text{range}(-3, 0.2, 0.4))}$.
- 14 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 15 From the **Color** list, choose **Black**.
- 16 Clear the **Color legend** check box.
- 17 On the **3D Plot Group 4** toolbar, click **Plot**.

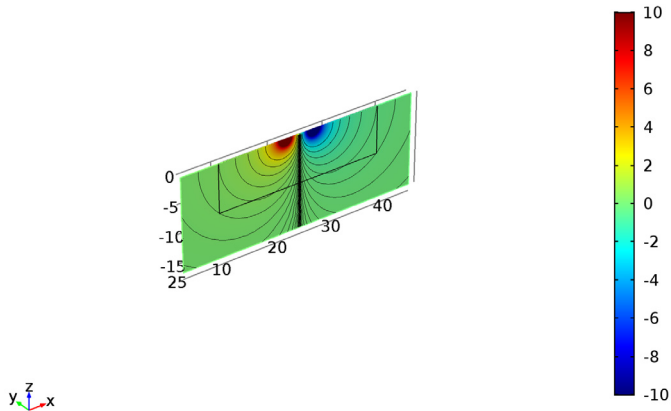
1: dom_C1=34, dom_C2=49 Surface: Electric potential (V)
 Contour: Electric potential (V)



- 18 In the **Model Builder** window, click **3D Plot Group 4**.
- 19 In the **Settings** window for 3D Plot Group, locate the **Data** section.
- 20 From the **Parameter value (dom_C1,dom_C2)** list, choose **2: dom_C1=39, dom_C2=44**.

2 On the **3D Plot Group 4** toolbar, click **Plot**.

2: dom_C1=39, dom_C2=44 Surface: Electric potential (V)
Contour: Electric potential (V)



Proceed to reproduce the comparison plot in [Figure 4](#). Begin by adding a Function data set for the analytic solution.

Grid ID 1

On the **Results** toolbar, click **More Data Sets** and choose **Grid>Grid ID**.

Data Sets

- 1** In the **Settings** window for Grid ID, locate the **Data** section.
- 2** From the **Function** list, choose **Analytic 1 (V_ref)**.
- 3** Locate the **Parameter Bounds** section. In the **Minimum** text field, type 10.
- 4** In the **Maximum** text field, type 40.

1D Plot Group 5

- 1** On the **Results** toolbar, click **1D Plot Group**.
- 2** In the **Settings** window for 1D Plot Group, click to expand the **Title** section.
- 3** From the **Title type** list, choose **Manual**.
- 4** In the **Title** text area, type Comparison between analytical (lines) and modeled (markers) results.
- 5** Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 6** In the associated text field, type Position (m).
- 7** Select the **y-axis label** check box.

8 In the associated text field, type `Vo1tage (V)`.

Point Graph 1

On the **ID Plot Group 5** toolbar, click **Point Graph**.

ID Plot Group 5

- 1 In the **Settings** window for Point Graph, locate the **Data** section.
- 2 From the **Data set** list, choose **Study 1/Solution 1 (sol1)**.
- 3 From the **Parameter selection (dom_C1, dom_C2)** list, choose **First**.
- 4 Locate the **Selection** section. From the **Selection** list, choose **Point sources**.
- 5 Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(V)`.
- 6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 7 In the **Expression** text field, type `x`.
- 8 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
- 9 Click to expand the **Legends** section. Right-click **Point Graph 1** and choose **Duplicate**.
- 10 In the **Settings** window for Point Graph, locate the **Data** section.
- 11 From the **Parameter selection (dom_C1, dom_C2)** list, choose **Last**.
- 12 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.
- 13 In the **Model Builder** window, click **ID Plot Group 5**.

Line Graph 1

On the **ID Plot Group 5** toolbar, click **Line Graph**.

ID Plot Group 5

- 1 In the **Settings** window for Line Graph, locate the **Data** section.
- 2 From the **Data set** list, choose **Grid ID 1**.
- 3 Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(V_ref(x, 17.5, 32.5))`.
- 4 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Geometry>Coordinate>x - x-coordinate**.
- 5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
- 6 Right-click **Line Graph 1** and choose **Duplicate**.
- 7 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.
- 8 In the **Expression** text field, type `abs(V_ref(x, 22.5, 27.5))`.

- 9 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.
- 10 On the **ID Plot Group 5** toolbar, click **Plot**.
Change to log scale.
- 11 Click the **y-Axis Log Scale** button on the **Graphics** toolbar.
- 12 Click **Plot**.

Finally reproduce the relative error plots in [Figure 5](#).

ID Plot Group 6

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for ID Plot Group, locate the **Title** section.
- 3 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. Select the **x-axis label** check box.
- 6 In the associated text field, type Position (m).
- 7 Select the **y-axis label** check box.
- 8 In the associated text field, type Relative error.
- 9 Click to expand the **Axis** section. Select the **Manual axis limits** check box.
- 10 Select the **y-axis log scale** check box.
- 11 In the **x minimum** text field, type 12.25.
- 12 In the **x maximum** text field, type 37.75.
- 13 In the **y minimum** text field, type $1e-7$.

Line Graph 1

On the **ID Plot Group 6** toolbar, click **Line Graph**.

ID Plot Group 6

- 1 In the **Settings** window for Line Graph, locate the **Data** section.
- 2 From the **Data set** list, choose **Study 1/Solution 1 (sol1)**.
- 3 From the **Parameter selection (dom_C1, dom_C2)** list, choose **First**.
- 4 Locate the **y-Axis Data** section. In the **Expression** text field, type $\text{abs}(V - V_{\text{ref}}(x, 17.5, 32.5)) / \text{abs}(V)$.
- 5 Select Edges 67–91 only.
- 6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

- 7** In the **Expression** text field, type x .
- 8** In the **Model Builder** window, under **Results> ID Plot Group 6** right-click **Line Graph 1** and choose **Duplicate**.
- 9** In the **Settings** window for Line Graph, locate the **Data** section.
- 10** From the **Parameter selection (dom_C1, dom_C2)** list, choose **Last**.
- 11** Locate the **y-Axis Data** section. In the **Expression** text field, type $\text{abs}(V - V_{\text{ref}}(x, 22.5, 27.5)) / \text{abs}(V)$.
- 12** On the **ID Plot Group 6** toolbar, click **Plot**.