

# Geoelectrics

## *Introduction*

---

Electrical resistivity tomography (ERT) is a geophysical method aiming at the imaging of the resistivity of soil and rocks in the subsurface by electrical measurements taken at 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 for various 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 just a few. The method is closely related to electrical impedance tomography (EIT) as it is used in medical imaging or nondestructive material testing. One specialty 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 where the geoelectrical survey takes place is unbounded to the sides and the bottom. For this reason, one needs to care of modeling infinity at the computational domains.

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 model 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,  $\sigma$  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 method 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 an agreement with additional constraints (for example, smoothness or *a priori* information) are found.

Whereas 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

$$R_i = \int_{\Omega} S_i(\mathbf{r}, \rho) \rho(\mathbf{r}) dV \quad (2)$$

with the sensitivity function  $S_i$  defined as

$$S_i(\mathbf{r}, \rho) = \mathbf{J}(\mathbf{r}, \mathbf{r}_{C1, i}, \mathbf{r}_{C2, i}) \cdot \mathbf{J}(\mathbf{r}, \mathbf{r}_{P1, i}, \mathbf{r}_{P2, i}) \quad (3)$$

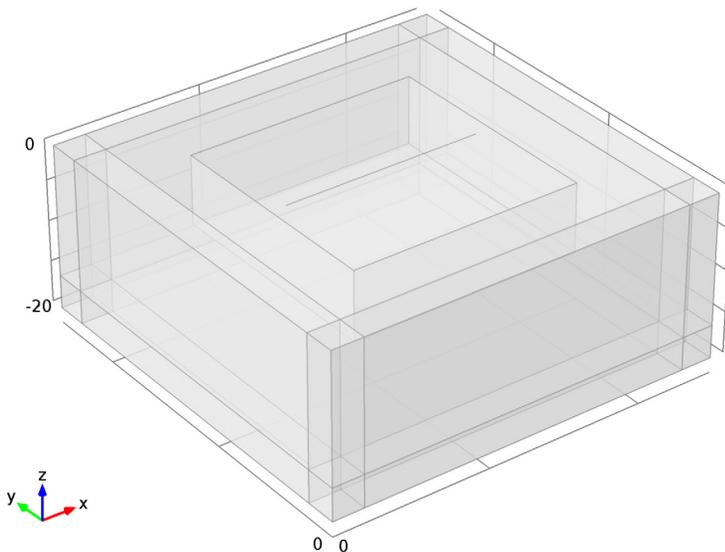
which is given by the reciprocity theorem (Ref. 4). Here,  $\mathbf{r}_{C1}$  and  $\mathbf{r}_{C2}$  are the positions for the electrodes injecting the electric current into the ground, and  $\mathbf{r}_{P1}$  and  $\mathbf{r}_{P2}$  are the position for the electrodes measuring the difference in electric 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 domains the infinite element domains account for the fact that the domain is not bounded.



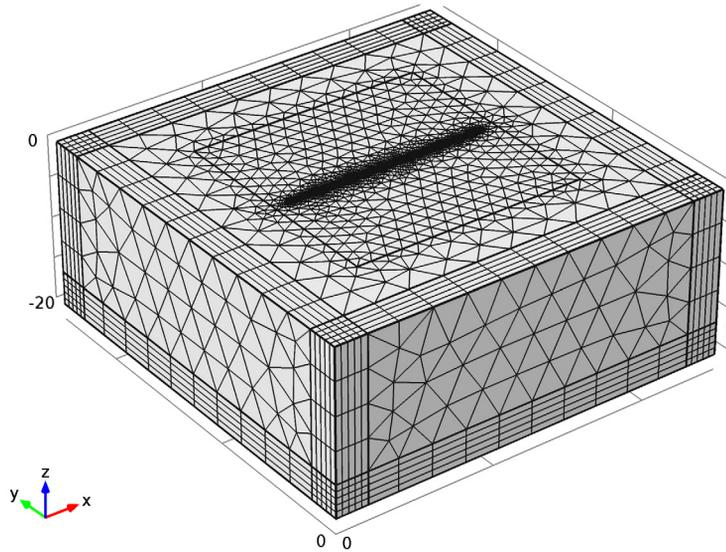
*Figure 1: Computational domain. A line of 25 electrodes is placed at the top of a 50-by-50-by-20 meters box bounded by infinite element regions.*

As a material, use a homogeneous half-space that often serves as a reference model for forward algorithms. 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

$$V = \frac{I}{2\pi\sigma} \left( \frac{1}{|\mathbf{r} - \mathbf{r}_{C1}|} - \frac{1}{|\mathbf{r} - \mathbf{r}_{C2}|} \right) \quad (4)$$

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 4](#).

Figure 5 shows the relative error of the computed solutions with respect to the analytical solutions given by Equation 4 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)

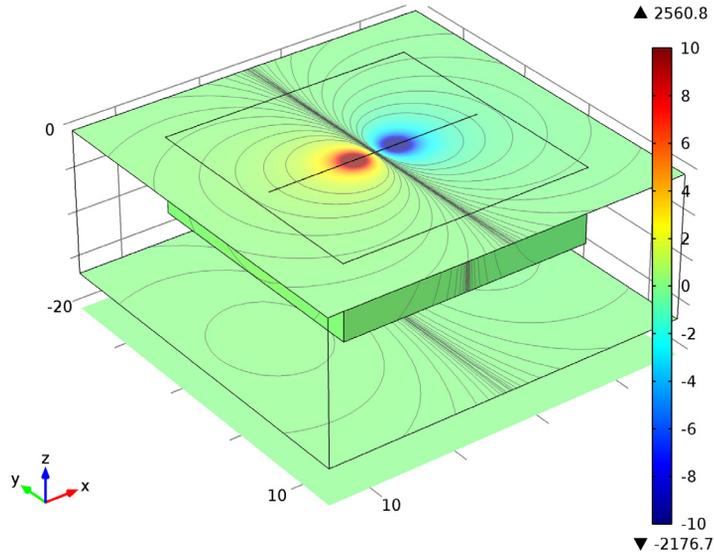


Figure 3: Electric potential around the active electrode pair.

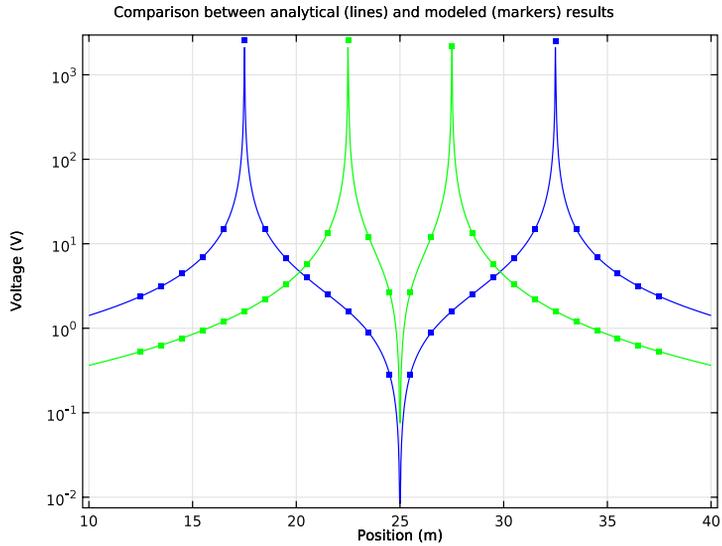


Figure 4: Comparison between analytical (lines) and modeled (markers) results.

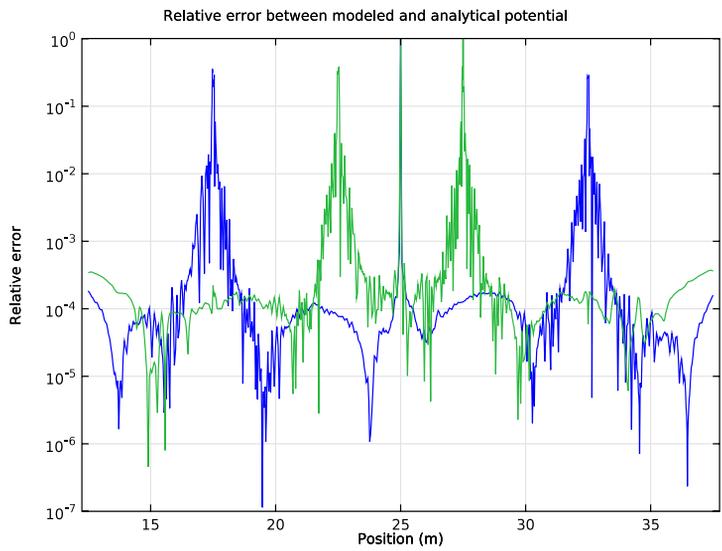


Figure 5: The relative error of the computed solutions with respect to the analytical solutions for the two electrode configurations. The plot is taken along the line joining the point sources.

The model solves for two current dipole situations with a common midpoint. Together they allow the calculation of the sensitivity of a Wenner- $\alpha$  configuration according to Equation 3.

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

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.

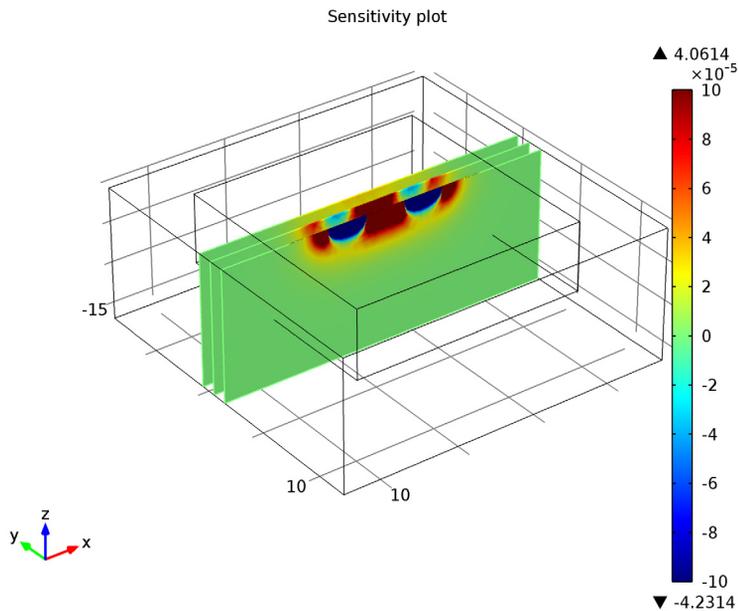


Figure 6: Sensitivity plot.

## References

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

---

**Model Library path:** ACDC\_Module/Other\_Industrial\_Applications/geoelectrics

---

### *Modeling Instructions*

---

#### **MODEL WIZARD**

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.
- 3 In the **Add physics** tree, select **AC/DC>Electric Currents (ec)**.
- 4 Click **Next**.
- 5 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.
- 6 Click **Finish**.

#### **GLOBAL DEFINITIONS**

##### *Parameters*

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

<b>Name</b>	<b>Expression</b>	<b>Description</b>
L	50[m]	Domain length
W	50[m]	Domain width
H	20[m]	Domain height
WI	4[m]	Infinite layer thickness
N	25	Number of electrodes
a	1[m]	Electrode separation
x0	$L/2 - N/2 * a$	First electrode x-position
y0	$W/2$	First electrode y-position
rho0	100[m/S]	Resistivity
conf	1	Active configuration

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

## GEOMETRY I

### Block 1

- 1 In the **Model Builder** window, under **Model I** right-click **Geometry I** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type L.
- 4 In the **Depth** edit field, type W.
- 5 In the **Height** edit field, type H.
- 6 Locate the **Position** section. In the **z** edit 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 the **Build Selected** button.

### Polygon 1

- 1 In the **Model Builder** window, right-click **Geometry I** and choose **More Primitives>Polygon**.
- 2 In the **Polygon** settings window, locate the **Coordinates** section.
- 3 In the **x** edit field, type  $\text{range}(x_0, a, x_0+N*a)$ .
- 4 In the **y** edit field, type  $y_0+0*\text{range}(0, 1, N)$ .
- 5 In the **z** edit field, type  $0*\text{range}(0, 1, N)$ .
- 6 Click the **Build Selected** button.

### Block 2

- 1 Right-click **Geometry I** and choose **Block**.

- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type  $(N+4) \cdot a$ .
- 4 In the **Depth** edit field, type  $(N+4) \cdot a$ .
- 5 In the **Height** edit field, type  $N \cdot a/3$ .
- 6 Locate the **Position** section. In the **x** edit field, type  $x0 - 2 \cdot a$ .
- 7 In the **y** edit field, type  $y0/2 - 2 \cdot a$ .
- 8 In the **z** edit field, type  $-N \cdot a/3$ .
- 9 Click the **Build Selected** button.

#### *Form Union*

- 1 In the **Model Builder** window, under **Model 1 > Geometry 1** right-click **Form Union** and choose **Build Selected**.
- 2 Click the **Go to Default 3D View** button on the Graphics toolbar.
- 3 Click the **Transparency** button on the Graphics toolbar to see the interior of the geometry.  
The completed geometry is shown in [Figure 1](#).
- 4 Click the **Transparency** button on the Graphics toolbar to return to the default state.

## DEFINITIONS

#### *Analytic 1*

- 1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Functions > Analytic**.
- 2 In the **Analytic** settings window, locate the **Function Name** section.
- 3 In the **Function name** edit field, type  $V_{ref}$ .
- 4 Locate the **Parameters** section. In the **Expression** edit field, type  $1[A] \cdot \rho_{h0} / (2 \cdot \pi) \cdot (1/abs(x-x1) - 1/abs(x-x2))$ .
- 5 In the **Arguments** edit field, type  $x$ ,  $x1$ ,  $x2$ .

Later, you will compare the simulation results with the analytical solution given by  $V_{ref}$ .

#### *Explicit 1*

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Selections > Explicit**.
- 2 In the **Explicit** settings window, 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 **Model 1>Definitions>Explicit 1** and choose **Rename**.
- 6 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.
- 7 Go to the **Rename Explicit** dialog box and type Ground boundaries in the **New name** edit field.
- 8 Click **OK**.

#### *Explicit 2*

- 1 Right-click **Definitions** and choose **Selections>Explicit**.
- 2 Select Domains 1–9, 11, 12, and 14–19 only.
- 3 Right-click **Model 1>Definitions>Explicit 2** and choose **Rename**.
- 4 Right-click **Model 1>Definitions>Explicit 2** and choose **Rename**.
- 5 Go to the **Rename Explicit** dialog box and type Infinite element domains in the **New name** edit field.
- 6 Click **OK**.

#### *Explicit 3*

- 1 Right-click **Definitions** and choose **Selections>Explicit**.
- 2 Select Domains 1–8, 11, 12, and 14–19 only.
- 3 In the **Explicit** settings window, locate the **Output Entities** section.
- 4 From the **Output entities** list, choose **Adjacent boundaries**.
- 5 Select the **Interior boundaries** check box.
- 6 Right-click **Model 1>Definitions>Explicit 3** and choose **Rename**.
- 7 Right-click **Model 1>Definitions>Explicit 3** and choose **Rename**.
- 8 Go to the **Rename Explicit** dialog box and type Infinite element boundaries in the **New name** edit field.
- 9 Click **OK**.

#### *Infinite Element Domain 1*

- 1 Right-click **Definitions** and choose **Infinite Element Domain**.
- 2 In the **Infinite Element Domain** settings window, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Infinite element domains**.

## MATERIALS

### *Material 1*

- 1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.
- 2 In the **Material** settings window, locate the **Material Contents** section.
- 3 In the table, enter the following settings:

Property	Name	Value
Electrical conductivity	sigma	$1/\rho_0$
Relative permittivity	epsilon <sub>r</sub>	1

- 4 Right-click **Model 1**>**Materials**>**Material 1** and choose **Rename**.
- 5 Go to the **Rename Material** dialog box and type 100 Ohmm homogeneous in the **New name** edit field.
- 6 Click **OK**.

## ELECTRIC CURRENTS

### *Point Current Source 1*

- 1 In the **Model Builder** window, under **Model 1** right-click **Electric Currents** and choose **Points**>**Point Current Source**.
- 2 Select Points 29–54 only.
- 3 In the **Point Current Source** settings window, locate the **Point Current Source** section.
- 4 In the  $Q_j$  edit field, type  $1*(\text{dom}==\text{dom}_{C1}) - 1*(\text{dom}==\text{dom}_{C2})$ .
- 5 Locate the **Point Selection** section. Click **Create Selection**.
- 6 Go to the **Create Selection** dialog box.
- 7 In the **Selection name** edit field, type Point sources.
- 8 Click the **OK** button.

### *Ground 1*

- 1 In the **Model Builder** window, right-click **Electric Currents** and choose **Ground**.
- 2 In the **Ground** settings window, 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 **Model 1** right-click **Mesh 1** and choose **Free Tetrahedral**.
- 2 In the **Free Tetrahedral** settings window, 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 **Model 1>Mesh 1>Free Tetrahedral 1** and choose **Size**.
- 2 In the **Size** settings window, 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 edit field, type 0.05.

### *Size 2*

- 1 Right-click **Free Tetrahedral 1** and choose **Size**.
- 2 Select Domain 13 only.
- 3 In the **Size** settings window, 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 edit field, type 2.

### *Swept 1*

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.
- 2 In the **Swept** settings window, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domains 4, 8, 9, 12, and 17 only.

### *Distribution 1*

- Right-click **Model 1>Mesh 1>Swept 1** and choose **Distribution**.

*Swept 2*

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.
- 2 In the **Swept** settings window, 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 **Settings** window, click **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 **Stationary** settings window, click to expand the **Study Extensions** section.
- 3 Select the **Continuation** check box.
- 4 Click **Add**.
- 5 In the table, enter the following settings:

Continuation parameter	Parameter value list
dom_C1	34 39

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

Continuation parameter	Parameter value list
dom_C2	49 44

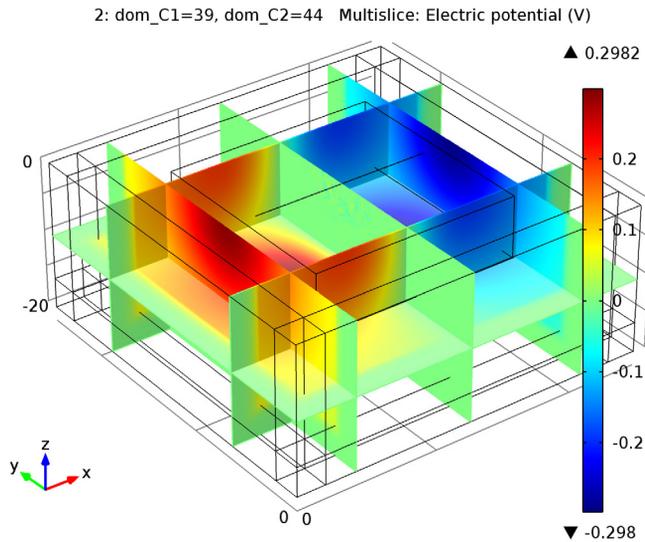
- 8 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

**RESULTS***Electric potential*

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

- 1 In the **Model Builder** window, expand the **Electric potential** node, then click **Multislice**.

- 2 In the **Multislice** settings window, locate the **Multipane Data** section.
- 3 Find the **x-planes** subsection. In the **Planes** edit field, type 3.
- 4 Find the **y-planes** subsection. In the **Planes** edit field, type 2.
- 5 Click the **Plot** button.



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

## DEFINITIONS

### View 1

- 1 In the **Model Builder** window, expand the **Model 1>Definitions** node.
- 2 Right-click **View 1** and choose **Hide Geometric Entities**.
- 3 In the **Hide Geometric Entities** settings window, locate the **Geometric Entity Selection** section.
- 4 From the **Selection** list, choose **Infinite element domains**.
- 5 In the **Model Builder** window, right-click **View 1** and choose **Hide Geometric Entities**.
- 6 In the **Hide Geometric Entities** settings window, locate the **Geometric Entity Selection** section.
- 7 From the **Geometric entity level** list, choose **Boundary**.
- 8 From the **Selection** list, choose **Infinite element boundaries**.

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

**RESULTS***3D Plot Group 2*

- 1** In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.
- 2** Right-click **3D Plot Group 2** and choose **Slice**.
- 3** In the **Slice** settings window, locate the **Expression** section.
- 4** In the **Expression** edit 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** edit field, type 25 27 29.
- 8** Locate the **Range** section. Select the **Manual color range** check box.
- 9** In the **Minimum** edit field, type  $-1.0\text{E}-4$ .
- 10** In the **Maximum** edit field, type  $1.0\text{E}-4$ .
- 11** Click the **Plot** button.
- 12** In the **Model Builder** window, click **3D Plot Group 2**.
- 13** In the **3D Plot Group** settings window, 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** Right-click **3D Plot Group 2** and choose **Rename**.
- 18** Go to the **Rename 3D Plot Group** dialog box and type Sensitivity in the **New name** edit field.
- 19** Click **OK**.

*3D Plot Group 3*

- 1** Right-click **Results** and choose **3D Plot Group**.
- 2** In the **Model Builder** window, under **Results** right-click **3D Plot Group 3** and choose **Surface**.
- 3** In the **Surface** settings window, click to expand the **Range** section.
- 4** Select the **Manual color range** check box.
- 5** In the **Minimum** edit field, type  $-10$ .
- 6** In the **Maximum** edit field, type  $10$ .

- 7 In the **Model Builder** window, right-click **3D Plot Group 3** and choose **Contour**.
- 8 In the **Contour** settings window, locate the **Levels** section.
- 9 From the **Entry method** list, choose **Levels**.
- 10 In the **Levels** edit 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 Click the **Plot** button.
- 15 Click the **Go to Default 3D View** button on the Graphics toolbar.

#### *Data Sets*

- 1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Cut Plane**.
- 2 In the **Cut Plane** settings window, locate the **Plane Data** section.
- 3 From the **Plane** list, choose **zx-planes**.
- 4 In the **y-coordinates** edit field, type  $y_0$ .

#### *3D Plot Group 4*

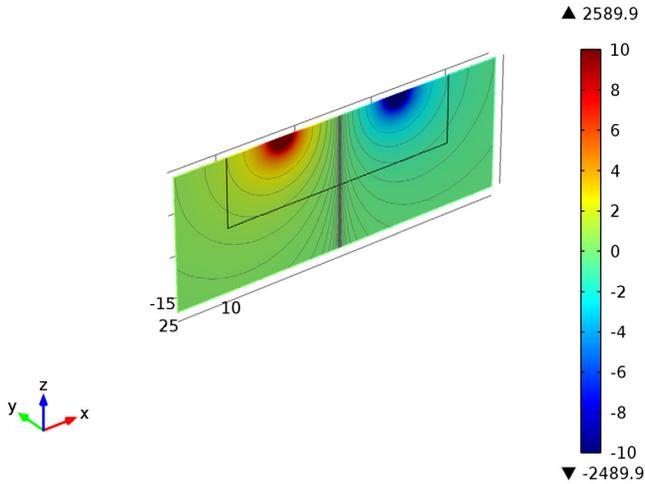
- 1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.
- 2 In the **3D Plot Group** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Cut Plane I**.
- 4 From the **Parameter value (dom\_C1,dom\_C2)** list, choose **I: dom\_C1=34, dom\_C2=49**.
- 5 Right-click **Results>3D Plot Group 4** and choose **Surface**.
- 6 In the **Surface** settings window, locate the **Range** section.
- 7 Select the **Manual color range** check box.
- 8 In the **Minimum** edit field, type  $-10$ .
- 9 In the **Maximum** edit field, type  $10$ .
- 10 In the **Model Builder** window, right-click **3D Plot Group 4** and choose **Contour**.
- 11 In the **Contour** settings window, locate the **Levels** section.
- 12 From the **Entry method** list, choose **Levels**.
- 13 In the **Levels** edit 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** Click the **Plot** button.

1: dom\_C1=34, dom\_C2=49 Surface: Electric potential (V)  
Contour: Electric potential (V)



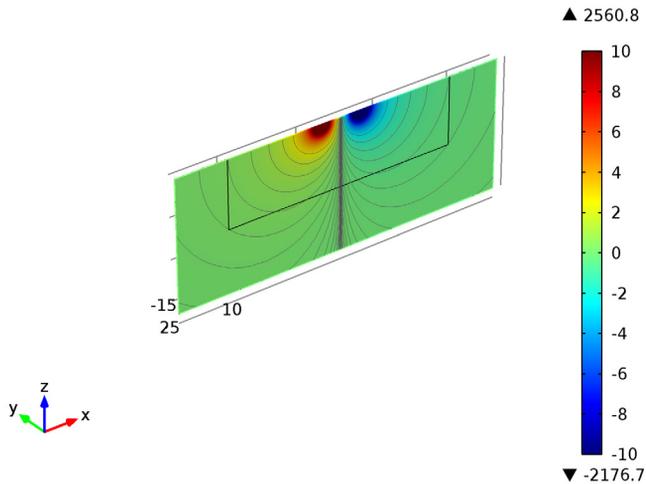
**18** In the **Model Builder** window, click **3D Plot Group 4**.

**19** In the **3D Plot Group** settings window, locate the **Data** section.

**20** From the **Parameter value (dom\_C1,dom\_C2)** list, choose **2: dom\_C1=39, dom\_C2=44**.

21 Click the **Plot** button.

2: dom\_C1=39, dom\_C2=44 Surface: Electric potential (V)  
Contour: Electric potential (V)



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

#### Data Sets

- 1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **More Data Sets>Function ID**.
- 2 In the **Function ID** settings window, locate the **Function** section.
- 3 From the **Function** list, choose **Analytic I**.
- 4 Locate the **Parameter Bounds** section. In the **Minimum** edit field, type 10.
- 5 In the **Maximum** edit field, type 40.

#### ID Plot Group 5

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 In the **ID Plot Group** settings window, 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 edit field, type Position (m).

- 7 Select the **y-axis label** check box.
- 8 In the associated edit field, type `Voltage (V)`.
- 9 Right-click **Results>ID Plot Group 5** and choose **Point Graph**.
- 10 In the **Point Graph** settings window, locate the **Data** section.
- 11 From the **Data set** list, choose **Solution 1**.
- 12 From the **Parameter selection (dom\_C1, dom\_C2)** list, choose **First**.
- 13 Locate the **Selection** section. From the **Selection** list, choose **Point sources**.
- 14 Locate the **y-Axis Data** section. In the **Expression** edit field, type `abs(V)`.
- 15 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 16 In the **Expression** edit field, type `x`.
- 17 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Color** list, choose **Blue**.
- 18 Click to expand the **Legends** section. In the **Model Builder** window, under **Results>ID Plot Group 5** right-click **Point Graph 1** and choose **Duplicate**.
- 19 In the **Point Graph** settings window, locate the **Data** section.
- 20 From the **Parameter selection (dom\_C1, dom\_C2)** list, choose **Last**.
- 21 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Color** list, choose **Green**.
- 22 In the **Model Builder** window, right-click **ID Plot Group 5** and choose **Line Graph**.
- 23 In the **Line Graph** settings window, locate the **Data** section.
- 24 From the **Data set** list, choose **Function ID 1**.
- 25 Locate the **y-Axis Data** section. In the **Expression** edit field, type `abs(mod1.V_ref(s,17.5,32.5))`.  
The prefix `mod1` instructs COMSOL Multiphysics that this function is defined in Model 1.
- 26 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 27 In the **Expression** edit field, type `s`.
- 28 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Color** list, choose **Blue**.
- 29 Right-click **Results>ID Plot Group 5>Line Graph 1** and choose **Duplicate**.
- 30 In the **Line Graph** settings window, locate the **y-Axis Data** section.
- 31 In the **Expression** edit field, type `abs(mod1.V_ref(s,22.5,27.5))`.

- 32** Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Color** list, choose **Green**.
- 33** Click the **Plot** button.  
Change to log scale.
- 34** Click the **y-Axis Log Scale** button on the Graphics toolbar.
- 35** Click the **Plot** button.

#### *ID Plot Group 6*

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

- 1** In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2** In the **ID Plot Group** settings window, 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 edit field, type Position (m).
- 7** Select the **y-axis label** check box.
- 8** In the associated edit 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** edit field, type 12.25.
- 12** In the **x maximum** edit field, type 37.75.
- 13** In the **y minimum** edit field, type 1.0e-7.
- 14** Right-click **Results>ID Plot Group 6** and choose **Line Graph**.
- 15** In the **Line Graph** settings window, locate the **Data** section.
- 16** From the **Data set** list, choose **Solution 1**.
- 17** From the **Parameter selection (dom\_C1, dom\_C2)** list, choose **First**.
- 18** Locate the **y-Axis Data** section. In the **Expression** edit field, type  $\text{abs}(V - \text{mod}1.V_{\text{ref}}(x, 17.5, 32.5)) / \text{abs}(V)$ .
- 19** Select Edges 67–91 only.
- 20** Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 21** In the **Expression** edit field, type  $x$ .

- 22** In the **Model Builder** window, under **Results>ID Plot Group 6** right-click **Line Graph 1** and choose **Duplicate**.
- 23** In the **Line Graph** settings window, locate the **Data** section.
- 24** From the **Parameter selection (dom\_C1, dom\_C2)** list, choose **Last**.
- 25** Locate the **y-Axis Data** section. In the **Expression** edit field, type  $\text{abs}(V - \text{mod}1.V\_ref(x, 22.5, 27.5)) / \text{abs}(V)$ .
- 26** Click the **Plot** button.