infolytica
corporation

YOUR FASTEST SOLUTION TO A BETTER DESIGN

ElecNet

Version 7.8
Or later

Electrostatic / Current Flow / Time-harmonic / Transient

2D & 3D Tutorials

May 31, 2017

We welcome your comments regarding Infolytica Corporation documents. Please send comments or corrections to the following addresses:

email: docs@infolytica.com

fax: Documentation Department
(514) 849-4239

post: Documentation Department
Infolytica Corporation
300 Leo Pariseau, Suite 2222
Montreal, Quebec H2X 4B3
Canada

Part number EN7-T-010

Introduction to the tutorials

This document includes four tutorials to help you increase your skills with ElecNet. Each tutorial teaches you basic procedures that you can apply to your own models.

2D Tutorials

Translational geometry

- Electrostatic solution: Spheres
- Current Flow solution: Conductance of a trimmed resistor
- Time-harmonic and Transient solution: Microstrip

3D Tutorials

- Electrostatic and Transient solution: Microstrip

Additional information

The Getting Started Guide introduces you to the basic ElecNet concepts. More information on the procedures and concepts of model building with ElecNet is found in the Help, included with each package, and in the Live Docs section of our web site.

Features showcase

Tutorial #1
2D Electrostatic solution: Spheres

For a printed copy of this tutorial, print pages 6 through 15

Features you will learn

- Setting up the work environment by modifying initial settings, the construction grid, and the viewing area.
- Building the geometric model using the Keyboard Input Bar.
- Setting up the problem - this consists of making components, electrodes, and assigning boundary conditions.
- Generating the electrostatic field solution using ElecNet's 2D Electrostatic solver.
- Analyzing the results: this includes viewing the stored electric energy and charge of the model, the shaded plot of |E|, and the contour of V.
- Probing local fields at specific locations using the Field Line Graph feature.

Tutorial #2
2D Current Flow solution: Conductance of a trimmed resistor

For a printed copy of this tutorial, print pages 16 through 24

Features you will learn

- Setting up the work environment by modifying initial settings, the construction grid, and the viewing area.
- Building the geometric model using the Keyboard Input Bar.
- Setting up the problem - this consists of creating a new material, and of making components and electrodes.
- Generating the current flow field solution using ElecNet's 2D Current Flow solver.
- Analyzing the results: this includes viewing the power of the model, the current of the electrodes, the shaded plot of |E|, and the contours of V.
- Probing local fields at specific locations using the Field Line Graph feature.

Tutorial #3
2D Time-harmonic and Transient solution: Microstrip

For a printed copy of this tutorial, print pages 25 through 40

Features you will learn

- Setting up the work environment by modifying initial settings, the construction grid, and the viewing area.
- Building the geometric model using the Keyboard Input Bar.
- Setting up the problem - this consists of making components and electrodes, using parameterization, and assigning boundary conditions.
- Modifying the model so as to obtain the desired solution results.
- Generating the time-harmonic field solution using ElecNet's 2D Time-harmonic solver.
- Generating the transient field solution using ElecNet's 2D Transient solver.
- Analyzing the results: this includes viewing the contours of V at 0', animating the voltage contour field plot, and graphing the conduction and displacement current as a function of frequency and time.

Tutorial #4
3D Electrostatic and Transient solution: Microstrip

For a printed copy of this tutorial, print pages 41 through 58

Features you will learn

- Setting up the work environment by modifying initial settings, the construction grid, and the viewing area.
- Building the geometric model using the Keyboard Input Bar.
- Setting up the problem - this consists of making components and electrodes, and assigning boundary conditions.
- Modifying the model so as to obtain the desired solution results.
- Generating the electrostatic and transient field solution using ElecNet's 3D Electrostatic and Transient solvers.
- Generating streamlines using Infolytica's Streamline Generator extension.
- Analyzing the results: this includes viewing the contours of V and the shaded plot of |E| smoothed, from two viewing perspectives (Z=0 and X=500).

ElecNet

Tutorial #1

2D Electrostatic Tutorial

Spheres

Modeling plan

This problem is axisymmetric. Axisymmetric electric field problems usually have the axis itself unconstrained. Like axisymmetric magnetic field problems, only the model to the right of the axis is drawn. The boundaries can be nearer than in a Cartesian problem, simply because the field values fall as a $1/distance^3$, not $1/distance^2$. Commonly the potentials are specified on the device itself, with the boundaries set to Flux Tangential or Ground. The solver produces a set of scalar potentials as a solution.

In this example, as discussed above, the boundaries are Flux Tangential, except for the mid conducting plane set to Ground. It deals with the breakdown voltage between two spheres. The results match up neatly with experimental work in the laboratory. As a check of the computational method, it is instructive to calculate capacitance and attractive force between two spheres, and compare with published work. The figure shows a typical sphere-gap arrangement. The experiment establishes the relationship between breakdown voltage and sphere spacing and diameter. It is instructive for the student of high voltage to perform the real test in the laboratory, and then simulate it using a CAD package.

According to BBS, the voltage at breakdown is 131 kV rms. The peak value is thus 185 kV. We will apply this potential difference between the spheres and use ElecNet to calculate the maximum electric field strength (potential gradient) at breakdown.

Creating a new model

Opening a new model

- From your desktop, double-click the ElecNet icon.

  The Main window appears.
  or
- If ElecNet is already running, on the File menu, click *New* to open a new model.

Name the model

By default, ElecNet assigns a name to the model (e.g., ElecNet1) every time a new model is opened. As long as the application remains open, each new model number increments by one (e.g., if the new model you have opened is the fourth one in this session, ElecNet would assign the name ElecNet4). You can choose to retain this name, although it is recommended that you give the model a distinct name.

1. On the File menu, click *Save As*.
2. In the Save As dialog box, enter **Sphere** as the name of the model.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

Setting up the working environment

Initial Settings

Each new model reverts to the ElecNet default settings for the meshing method, preferred units for length, time and frequency, and for how curves will be displayed in the view. For our model, we are going to change only the preferred unit for length and accept all the other defaults.

1. From the Object page, select the model.
2. On the Edit menu, click *Properties*.
3. Click the Units tab and then from the Length drop down list, select *Millimeters*.
4. Click OK.

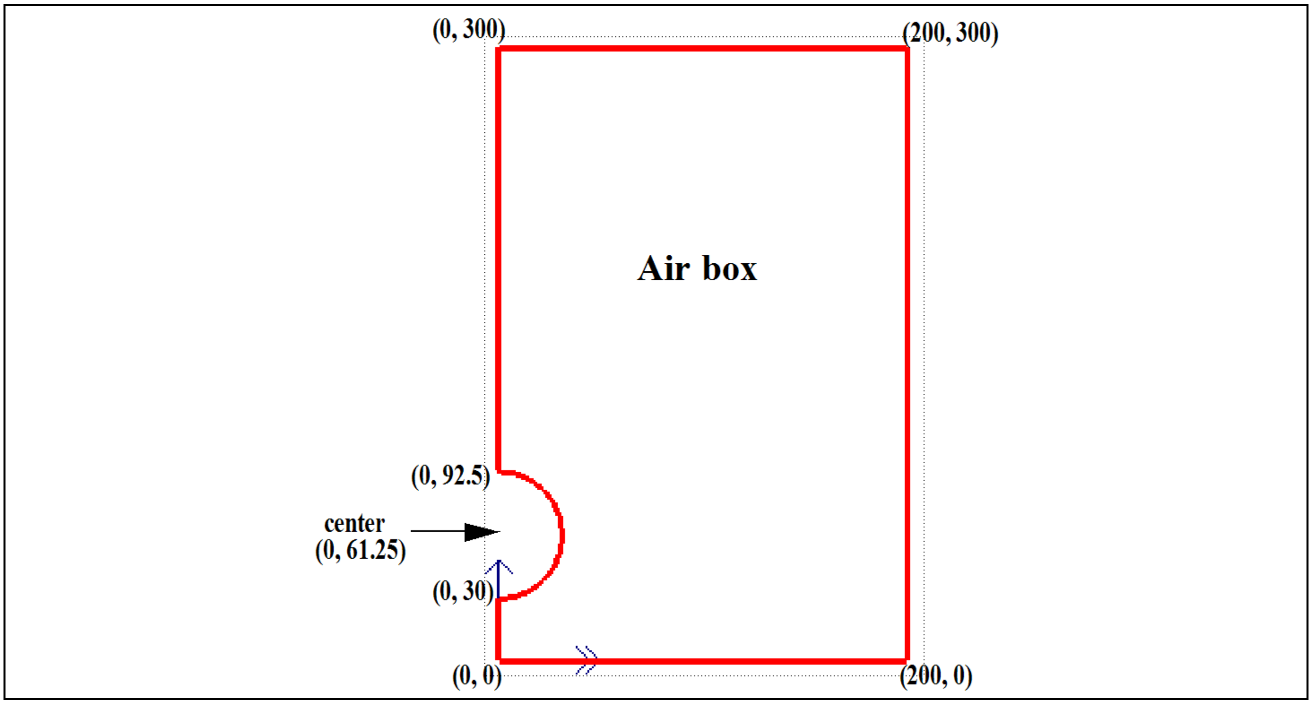
Set view to update automatically

Use of this feature resets the view automatically to include all of the model's geometry, as it is being drawn.

- On the View menu, click *Update Automatically*.

Build the geometric model

For the example chosen, the sphere is 62.5 mm in diameter. Since the problem has a plane of symmetry, it is possible to model only the upper sphere and half the air gap as shown, where the arc represents the sphere. Thus, the center is (31.25 +30) mm above the plane. The interior of the sphere need not be modeled.



Draw the geometric model of the components

1. On the Tools menu, click *Keyboard Input Bar*.
2. On the Draw menu, click *Arc (Center, Start, End)*.
3. In the Keyboard Input bar, enter the following coordinates to draw the sphere:

Center coordinates	**0, 61.25**	Press ENTER
Start coordinates	**0, 30**	Press ENTER
End coordinates	**0, 92.5**	Press ENTER

4. On the Draw menu, click *Line*.
5. In the Keyboard Input bar, enter the following coordinates to draw the air box:

Start coordinates	**0, 30**	Press ENTER
End coordinates	**0, 0**	Press ENTER
End coordinates	**200, 0**	Press ENTER
End coordinates	**200, 300**	Press ENTER
End coordinates	**0, 300**	Press ENTER
End coordinates	**0, 92.5**	Press ENTER

6. Press ESC.

Setting up the problem

Make the component

1. On the Edit menu, click *Select Construction Slice Surfaces*.
2. Select the region between the outer arc and the boundary.
3. On the Model menu, click *Make Component in an Arc*, and enter the following values:
   Name: **Air box**
   Material: **AIR**
   Angle: **-90** degrees
4. Click OK.

Make an electrode using the sphere

1. On the Edit menu, click *Select Component Surfaces*.
2. Select the half circle region, which represents the sphere surface.
3. On the Model menu, click *Make Electrode*.
4. In the Object page, select *Electrode #1*.
5. On the Edit menu, click *Properties* and modify Electrode#1 to read:
   Name: **Electrode (Sphere)**
6. Select the Waveform tab and enter the following values for ACDC:
   Magnitude (RMS): **92500**
   Phase (Deg): **0**
7. Click OK.

Assign boundary conditions

1. In the Object page, select *Face#3* of the *Air box* component.
   The surface of the bottom face is selected.
2. On the Boundary menu, click *Ground*.

Generating the electrostatic field solution

Set the solving options

1. On the Solve menu, click *Set Solver Options*, and verify that the following values are set:
   Material type: *Non-linear*
   Method: Newton-Raphson
   Max. Newton iterations: *20*
   Newton tolerance: *1 %*
   Polynomial order: Default (**Note** for axisymmetrical models the default is 2)
   CG tolerance: *0.01 %*
   Source frequency: *60 Hertz*

   Note CG Tolerance is the only value that has any significance for this electrostatic solution. All other values are listed for information purposes only.
2. Click OK.
3. On the Solve menu, click *Static 2D*.

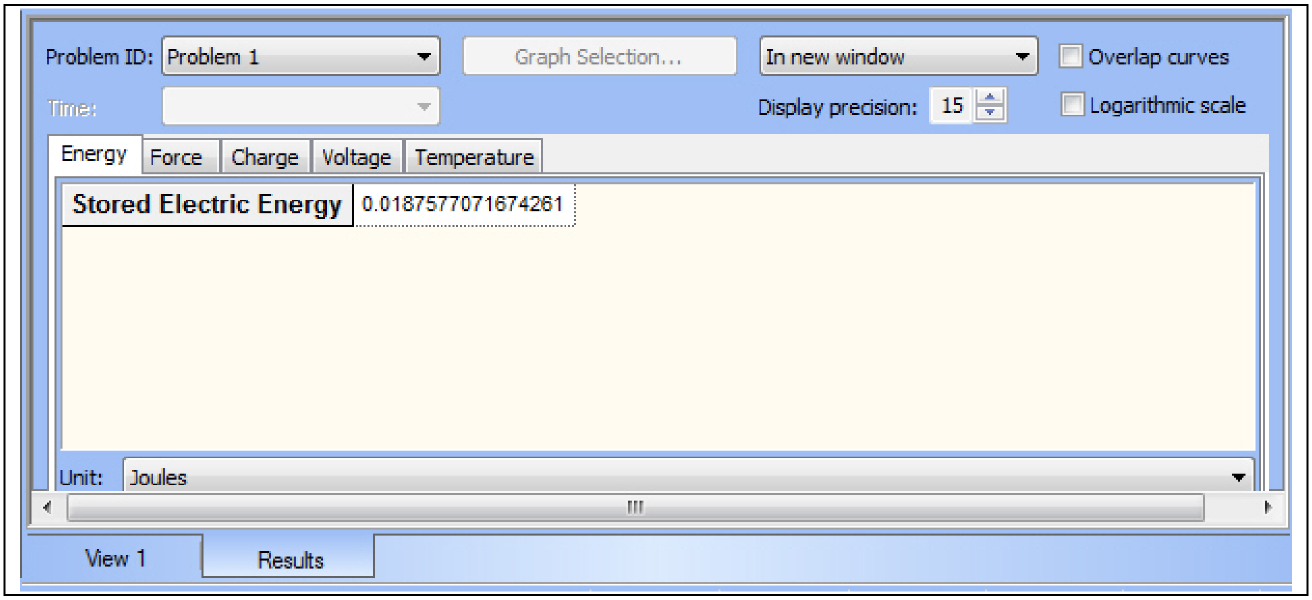
Analyzing the results

In this section, the following results are viewed:

- Stored electric energy of model
- Charge on electrode
- Shaded plot of |E| on the lowest part of the arc and the lower left hand corner surface of the model
- Contours of V

View the energy and charge

The Results window is automatically displayed when the Solver Progress dialog closes and the solution is complete.



1. Verify that the value for *Stored Electric Energy* is **0.0187577071674261** Joules.
2. Select the “Charge” tab.

   The Charge page is displayed.
3. Verify that the value for *Charge* is **4.0557204686326703e-007** Coulombs.
4. The capacitance between the spheres is simply the charge divided by the potential difference between them (i.e., 4.0557204686326703e-007/185x10e3 = 2.192pF). As a comparison, the analytical result is 2.3487 pF.

   Note To reduce the error in ElecNet’s value, use adaption or a higher polynomial order.

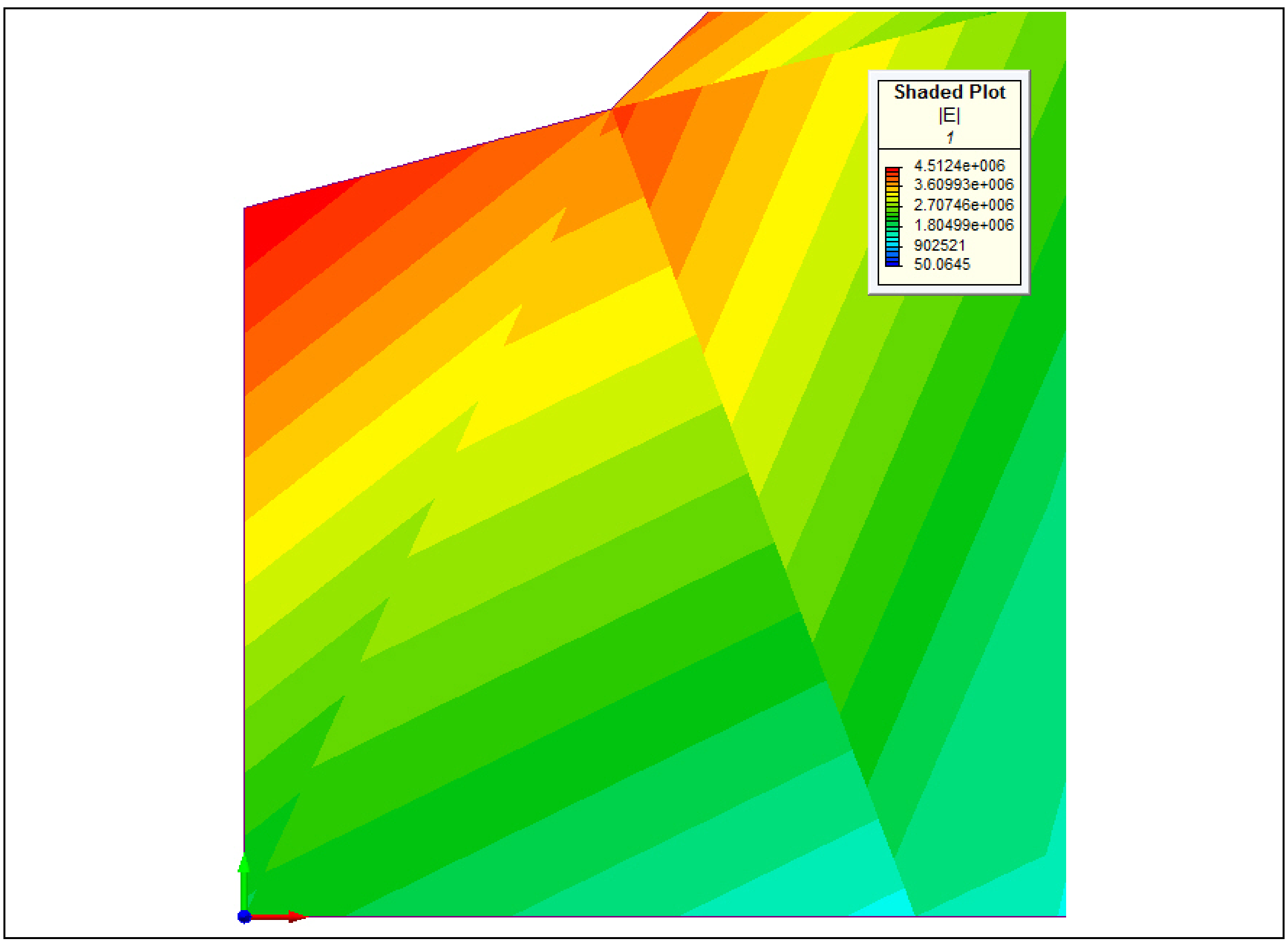
View the shaded plot of |E|

First, we will magnify the area of the model we want to place focus on, and then update the view to display the shaded plot of |E| for this area.

1. Before viewing the shaded plot, switch back to the View window by clicking the View tab 

 located at the bottom of the window.
2. On the View toolbar, click  (Examine Model).
3. Hold down the CTRL key and left mouse button, and then drag the cursor down and across the lowest part of the arc and the lower left hand corner, releasing the button once the desired area is selected.
4. From the Project Bar, select the “Field” page, and then click the tab labelled “Contour”.
5. In the “Fields to Display” box, select *None*.
6. Click the tab labelled “Shaded”.
7. In the “Fields to Display” box, select |*E*|.
8. At the bottom of the “Field” page, click *Update View*.

   The shaded plot is displayed.

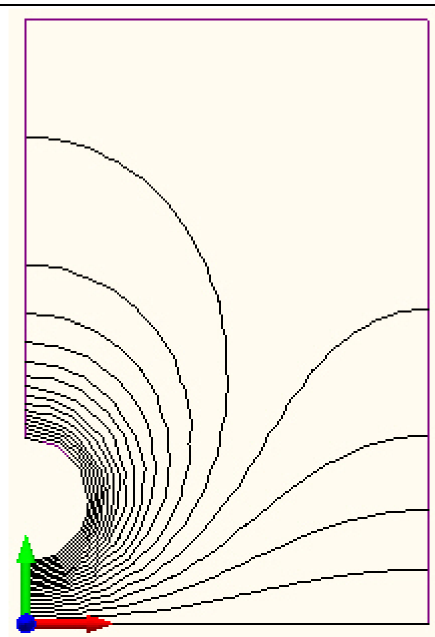


Note The maximum value of |E| (4.5124e+006 V/m) is just below the sphere.

View the contours of V

Here, we want to reset the viewing area so that the whole model is visible, and then update the view to display the contours of V.

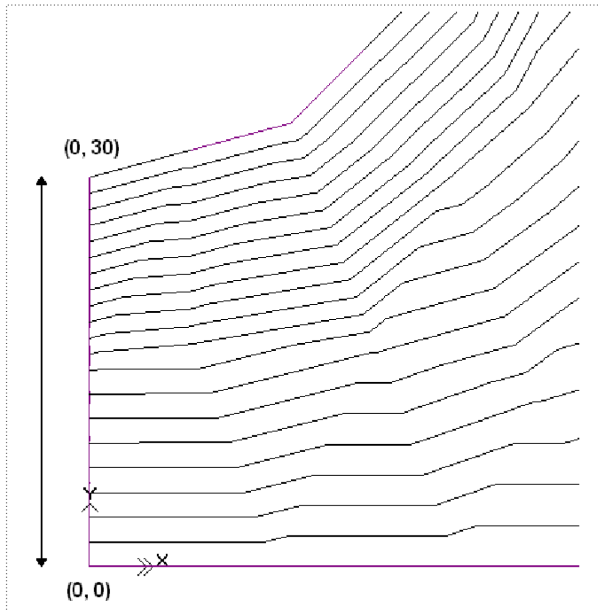
1. On the View menu, click *View All*.
2. In the “Fields to Display” box for the shaded plot, select *None*.
3. On the “Field” page, click the tab labelled “Contour”.
4. In the “Fields to Display” box, select *V*.
5. At the bottom of the Field page, click *Update View*.
   The contour is displayed.



Plotting V on a graph using the Field Contour Graphs feature

In this procedure we are going to use the ElecNet post-processing feature (Field Line Graph) that allows us to define a line segment, extract the 1000 field values along that segment, and then plot the field quantities on a graph.

Here we will use the *Field Line Graph* feature to extract **V** along a contour starting at (0, 0) and ending at (0,30)

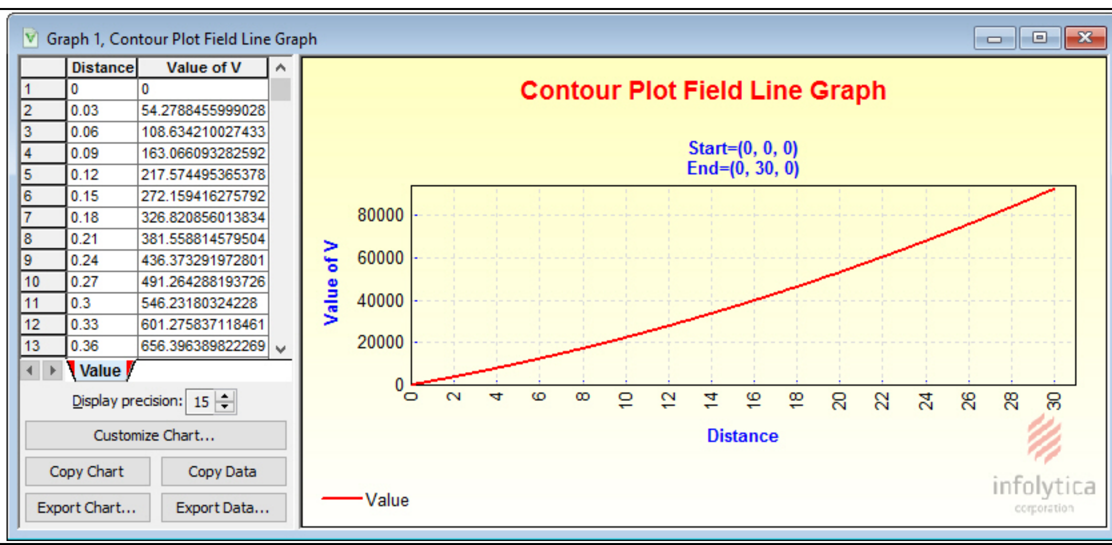


Note Although not shown in this procedure, *Field Arc Graph* and *Field Circle Graph* are also available for post-processing. Please refer to the Help for more information.

1. On the Tools menu, click *Field Line Graph*.
2. In the Keyboard Input bar, enter the coordinates (**0,0**) and (**0,30**), pressing *Enter* after each set of numbers.

   Note If you prefer, you can also use the mouse to draw the line graphically.

3. The graph is automatically displayed in a new Graph window.



Save the model

You have now completed the 2D Electrostatic tutorial.

1. On the File menu, click *Save*.
2. On the File menu, click *Close*.

Summary

In this tutorial, you completed the steps in creating a sphere model for an electrostatic solution. The skills you learned include:

- Setting up the work environment by modifying initial settings and the viewing area.
- Building the geometric model using the Keyboard Input Bar.
- Setting up the problem -- this consists of making components, electrodes, and assigning boundary conditions.
- Generating the electrostatic field solution using ElecNet's 2D Electrostatic solver.
- Analyzing the results, that includes viewing the stored electric energy and charge of the model, the shaded plot of |E|, and the contour of V.
- Probing local fields at specific locations using the Field Line Graph.

ElecNet
Tutorial #2

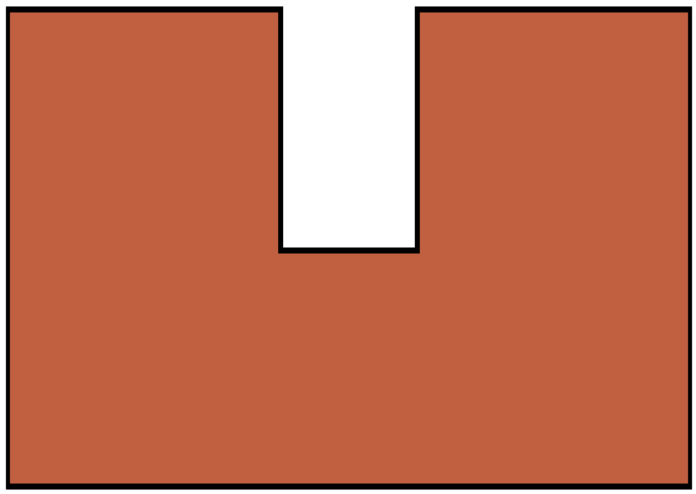
2D Current Flow Tutorial

Conductance of a trimmed resistor

Modeling plan

This problem consists of calculating the conductance of the shape shown below. It is a rectangle with a cut half way across it. The rectangle is 6 mm high, 10 mm long, with a slot 2 mm wide cutting half way across. Since the conductance per meter depth is totally independent of the units, it is only the ratio of the x and y dimensions that is important here.

The electric field E is the gradient of a scalar potential V.



Creating a new model

Opening a new model

- From your desktop, double-click the ElecNet icon.

  The Main window appears.
  or
- If ElecNet is already running, on the File menu, click *New* to open a new model.

Name the model

By default, ElecNet assigns a name to the model (e.g., ElecNet1) every time a new model is opened. As long as the application remains open, each new model number increments by one (e.g., if the new model you have opened is the fourth one in this session, ElecNet would assign the name ElecNet4). You can choose to retain this name, although it is recommended that you give the model a distinct name.

1. On the File menu, click *Save As*.
2. In the Save As dialog box, enter **Trim resistor** as the name of the model.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

Setting up the working environment

Initial Settings

Each new model reverts to the ElecNet default settings for the meshing method, preferred units for length, time and frequency, and for how curves will be displayed in the view. For our model, we are going to change only the preferred unit for length and accept all the other defaults.

1. From the Object page, select the model.
2. On the Edit menu, click *Properties*.
3. Click the Units tab and then from the Length drop down list, select *Millimeters*.
4. Click OK.

Setting the construction grid

1. On the View menu, click *Construction Grid*.
   A check mark next to the menu item indicates that it is enabled.
2. On the View menu, click *Set Construction Grid*.
3. In the Set Grid Extent and Spacing dialog, enter the following values:

Minimum X:	-1	Maximum X:	11
Minimum Y:	-1	Maximum Y:	9
X spacing:	1	Y spacing:	1

4. Click OK.

Set view to update automatically

Use of this feature resets the view automatically to include all of the model's geometry, as it is being drawn.

- On the View menu, click *Update Automatically*.

Build the geometric model

Draw the geometric model of the component

1. On the Tools menu, click *Keyboard Input Bar*.
2. On the Draw menu, click *Line*.
3. In the Keyboard Input bar, enter the following coordinates to draw the conductor:

Start coordinates	**0, 6**	Press ENTER
End coordinates	**0, 0**	Press ENTER
End coordinates	**10, 0**	Press ENTER
End coordinates	**10, 6**	Press ENTER
End coordinates	**6, 6**	Press ENTER
End coordinates	**6, 3**	Press ENTER
End coordinates	**4, 3**	Press ENTER
End coordinates	**4, 6**	Press ENTER
End coordinates	**0, 6**	Press ENTER

4. Press ESC.

Setting up the problem

Create a new material

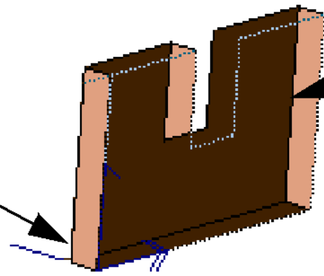
For the purposes of this tutorial, we need to assign a resistive material to the component we have just drawn. Since this type of material does not exist in the Material library, we will create a new material and save it to the database.

1. On the Tools menu, click *New User Material*.
2. On the General page, enter the following data:
    - Name: **Resistive Material**
    - Display color: Click *Display Color* and select an appropriate color
    - Transparency: *Optional*
    - Description: *Optional*
    - Categories: *Optional*
3. Click *Next*.
4. On the Options page, select the following:
    - Magnetic Permeability
    - Electric Conductivity
5. Using the Next button to advance to the appropriate pages, enter the following values:
    - Temperature Celsius = **20**
    - Relative Permeability = **1**
    - Coercivity Amps/m = **0**
    - Conductivity Siemens/m = **10**
6. Once you have entered all the values, click *Finish* in the Confirmation page to create the new material.
7. On the File menu, click *Save*.

Make the component

1. On the Edit menu, click *Select Construction Slice Surfaces*.
2. Select the interior region of the model you have drawn.
3. On the Model menu, click *Make Component in a Line*, and enter the following values:
    - Name: Conductor
    - Material: Resistive Material
    - Distance: 1 Millimeter
4. Click OK.

Making the electrodes



1. On the View menu, click  (Examine Model).
2. Holding down the left mouse button, place the cursor over the model and rotate it so that it appears as the illustration above.
3. In the Object page, select Face#4 of the Conductor component.
4. On the Model menu, click *Make Electrode*.
5. In the Object page, select *Face#10* of the *Conductor* component.
6. On the Model menu, click *Make Electrode*.
7. In the Object page, select *Electrode #2*.
8. On the Edit menu, click *Properties* and select the Waveform tab.
9. Enter the following values for ACDC:
   - Magnitude (RMS): **1**
   - Phase (Deg): **0**
10. Click OK.

Generating the current flow field solution

Set the solving options

1. On the Solve menu, click *Set Solver Options*, and enter (if necessary) the following values:
   - Material type: **Linear**
   - CG Tolerance: **0.01**
   - Source frequency: **60** Hertz
2. Click OK.

Solve the model

- On the Solve menu, click *Current Flow 2D*.

  The ElecNet Current Flow 2D Solver Progress dialog appears momentarily and then the Results page opens.

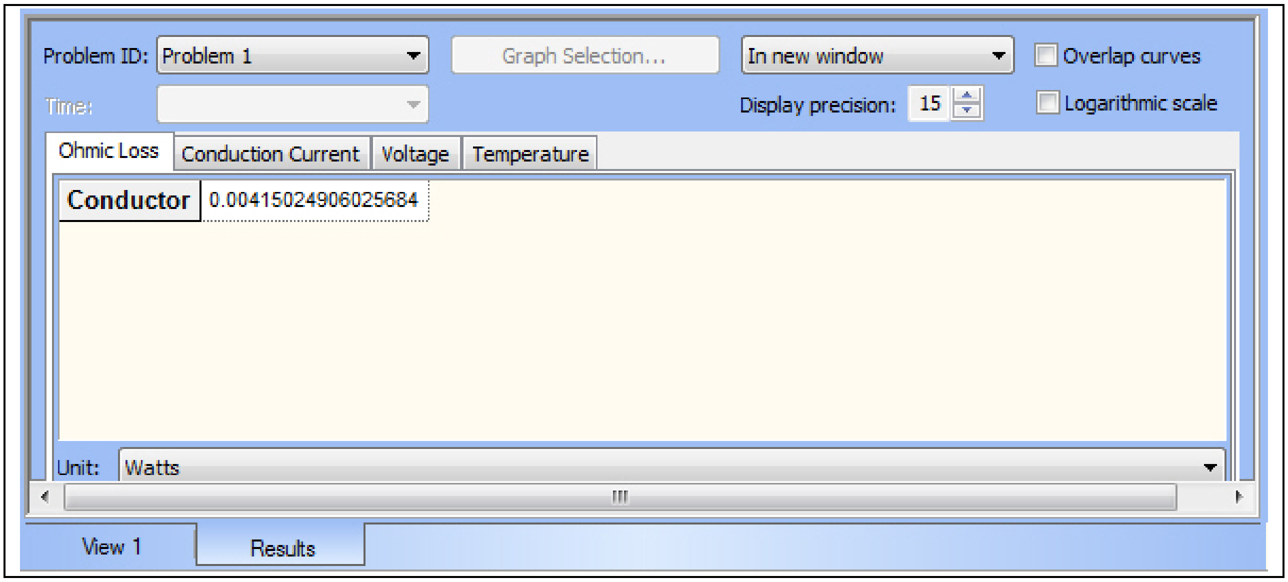
Analyzing the results

In this section, the following results are viewed:

- Ohmic loss
- Current on electrode
- Shaded plot of |E| of the model
- Contours of V

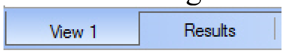
View the ohmic loss and current

The Results window is automatically displayed when the Solver Progress dialog closes and the solution is complete.

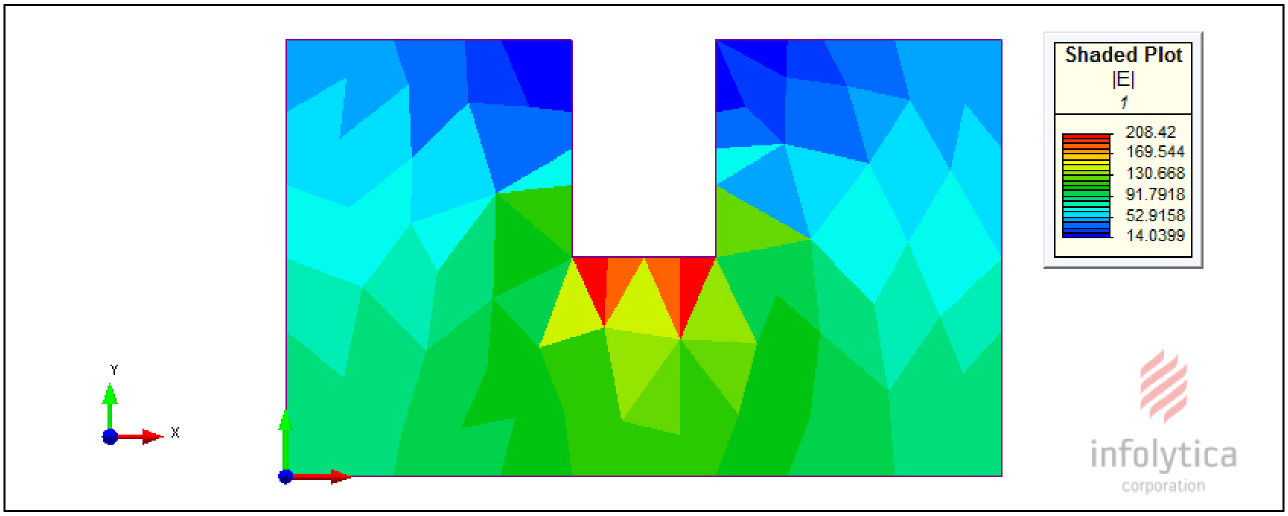


1. Verify that the value for Ohmic Loss is **0.00415024906025684** Watts.
2. Select the Conduction Current tab.
3. Verify that the values for each electrode is as follows:
    - Electrode#1 **-0.00415024906025684** Amperes
    - Electrode#2 **0.00415024906025684** Amperes

View the shaded plot of |E|

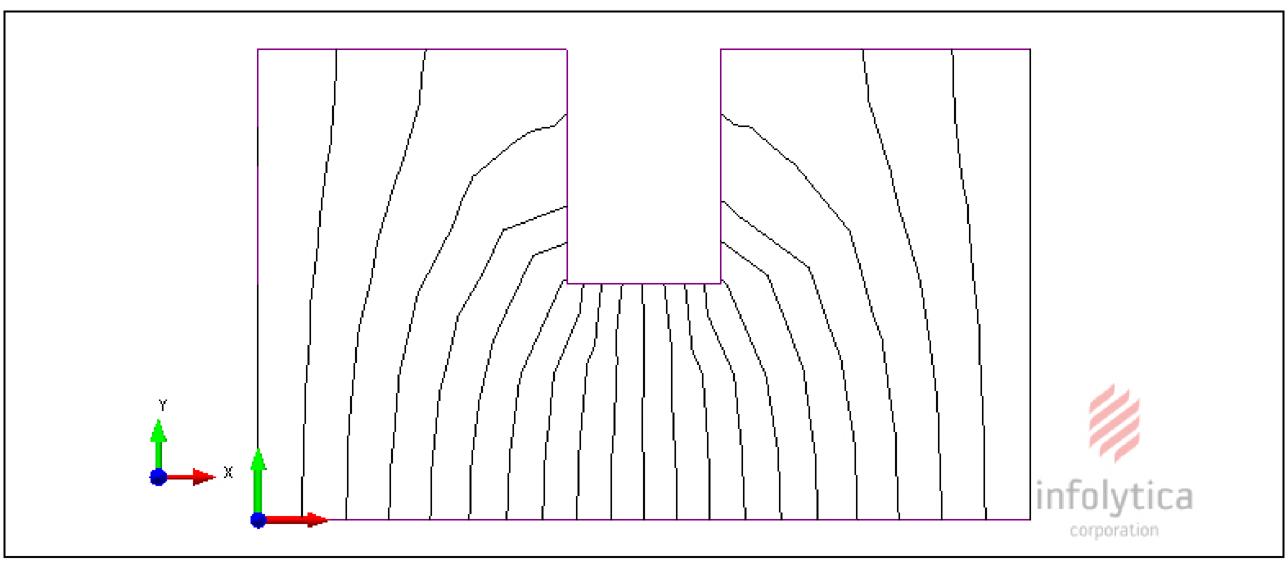
1. Before viewing the shaded plot, switch back to the View window by clicking the View tab 

located at the bottom of the window.
2. From the Project Bar, select the “Field” page, and then click the tab labelled “Contour”.
3. In the “Fields to Display” box, select *None*.
4. Click the tab labelled “Shaded”.
5. In the “Fields to Display” box, select |*E*|.
6. At the bottom of the Field page, click *Update View*.
   The shaded plot is displayed.



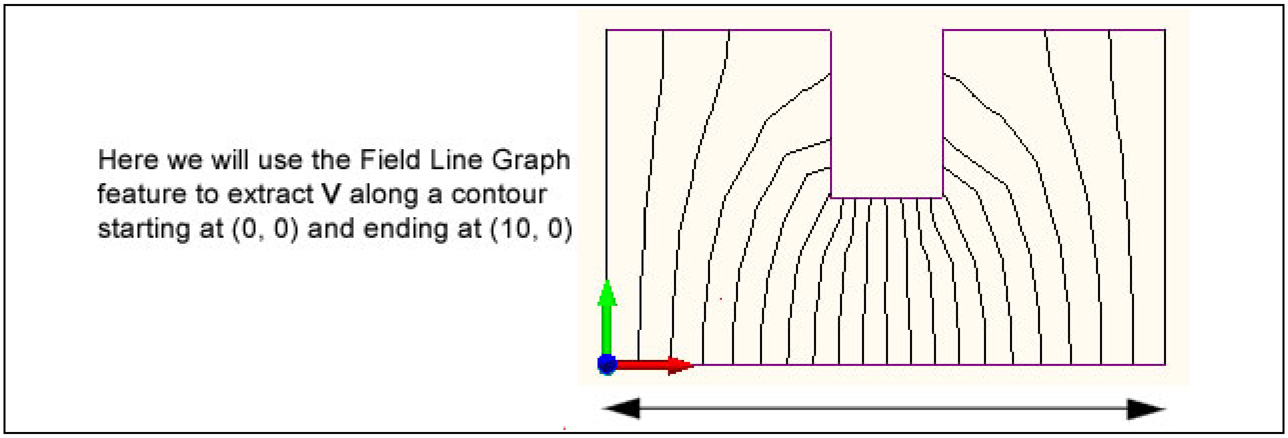
View the contour of V

1. In the “Fields to Display” box of the Shaded tab, select *None*.
2. Click the tab labelled “Contour”.
3. In the “Fields to Display” box, select *V*.
4. At the bottom of the Field page, Click *Update View*.
   The contour is displayed.



Plotting V on a graph using the Field Contour Graphs feature

In this procedure we are going to use the ElecNet post-processing feature (Field Line Graph) that allows us to define a line segment, extract the 1000 field values along that segment, and then plot the field quantities on a graph.

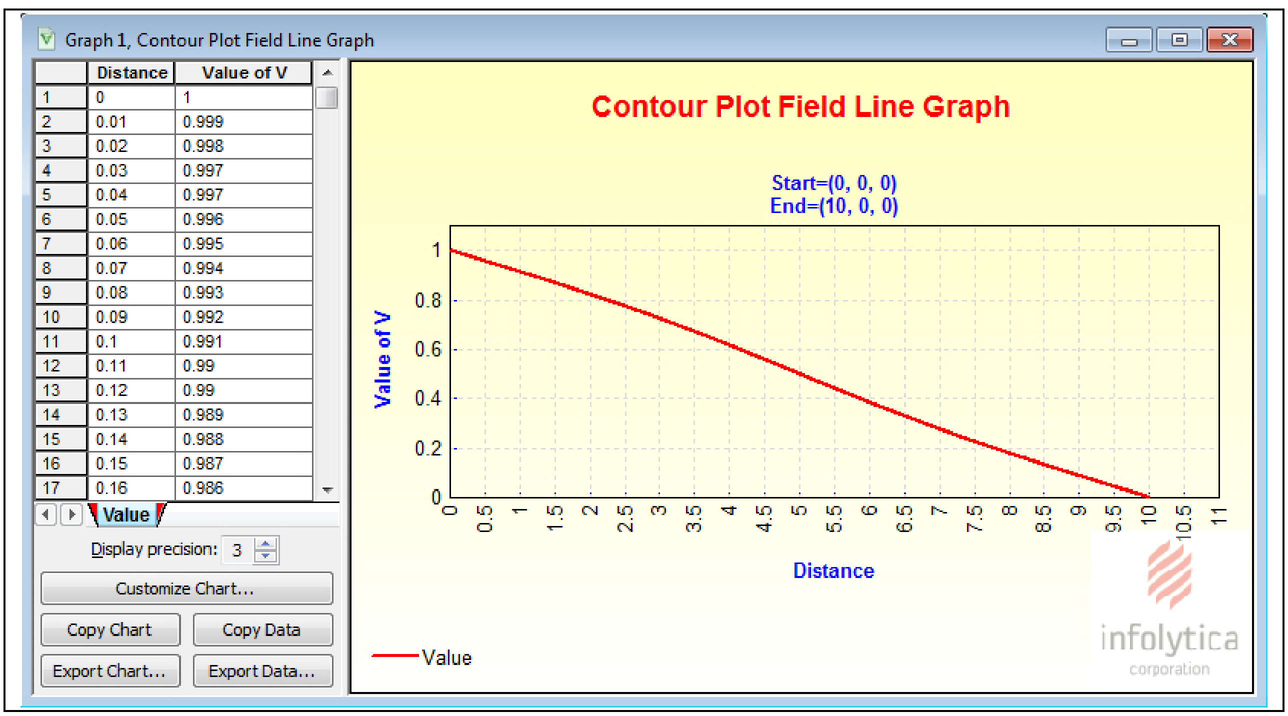


Note Although not shown in this procedure, *Field Arc Graph* and *Field Circle Graph* are also available for post-processing. Please refer to the Help for more information.

1. On the Tools menu, click *Field Line Graph*.
2. In the Keyboard Input bar, enter the coordinates (**0, 0**) and (**10, 0**), pressing *Enter* after each set of numbers.

   Note If you prefer, you can also use the mouse to draw the line graphically.

3. The graph is automatically displayed in a new window.



Save the model

You have now completed the 2D Current Flow tutorial.

1. On the File menu, click Save.
2. On the File menu, click Close.

Summary

In this tutorial, you completed the steps in creating a Trim Resistor model for a current flow solution. The skills you learned include:

- Setting up the work environment by modifying initial settings, the construction grid, and the viewing area.
- Building the geometric model using the Keyboard Input Bar.
- Setting up the problem -- this consists of creating a new material, and of making components and electrodes.
- Generating the current flow field solution using ElecNet's 2D Current Flow solver.
- Analyzing the results, which includes viewing the ohmic loss of the model, the current of the electrodes, the shaded plot of |E|, and the contours of V.
- Probing local fields at specific locations using the Field Line Graph feature.

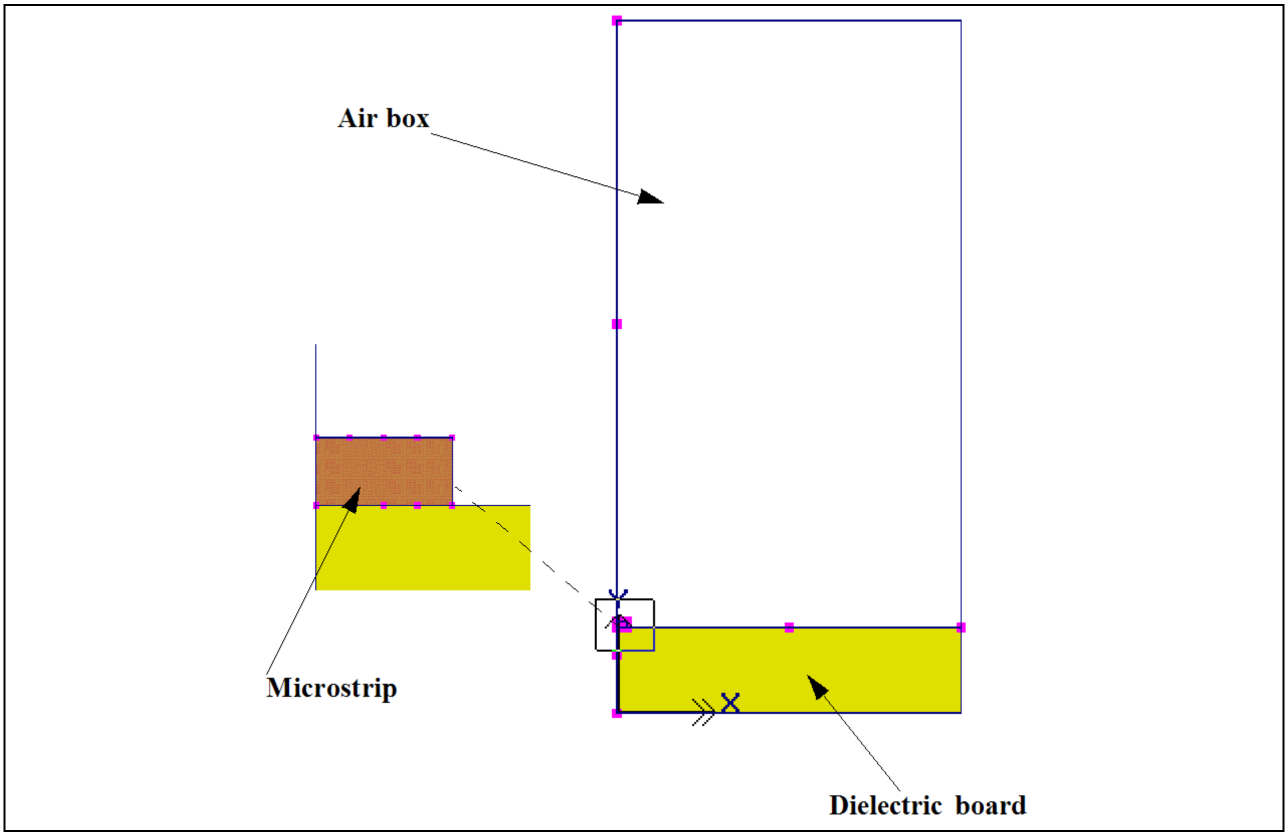
ElecNet

Tutorial #3

2D Time-harmonic and Transient Tutorial

Microstrip

Modeling plan



Creating a new model

Opening a new model

- From your desktop, double-click the ElecNet icon.
  The Main window appears.
  or
- If ElecNet is already running, on the File menu, click *New* to open a new model.

Name the model

By default, ElecNet assigns a name to the model (e.g., ElecNet1) every time a new model is opened. As long as the application remains open, each new model number increments by one (e.g., if the new model you have opened is the fourth one in this session, ElecNet would assign the name ElecNet4). You can choose to retain this name, although it is recommended that you give the model a distinct name.

1. On the File menu, click *Save As*.
2. In the Save As dialog box, enter **TH-Microstrip** as the name of the model.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

Setting up the working environment

Initial Settings

Each new model reverts to the ElecNet default settings for the meshing method, preferred units for length, time and frequency, and for how curves will be displayed in the view. For our model, we are going to change only the preferred unit for length and accept all the other defaults.

1. On the Tools menu, click *Set Units*.
2. From the Length drop down list, select *Millimeters*.
3. Click OK.

Set view to update automatically

Use of this feature resets the view automatically to include all of the model's geometry, as it is being drawn.

- On the View menu, click *Update Automatically*.

Build the geometric model and set up the problem for the dielectric board component

Create a new material

For the purposes of this tutorial, we need to assign a material that does not exist in the Material library. We will create a new material and save it to the database.

1. On the Tools menu, click *New User Material*.
2. On the General page, enter the following data:
    - Name: **Diel**
    - Display color: Click *Display Color* and select an appropriate color
    - Transparency: *Optional*
    - Description: *Optional*
    - Categories: *Optional*
3. Click Next.
4. On the Options page, select the following:
    - Electric ***Conductivity***
    - Electric ***Permittivity***
5. Using the Next button to advance to the appropriate pages, enter the following values:
    - Temperature Celsius = **20**
    - Conductivity Siemens/m = **1e-4**
    - Conductivity %IACS = **1.72e-010** (Automatic)

      Note Conductivity value entered in one unit will be automatically reflected in the other unit.
    - Relative Permittivity = **2**
6. Once you have entered all the values, click *Finish* in the Confirmation page to create the new material.
7. On the File menu, click *Save*.

Draw the dielectric board component

1. On the Tools menu, click *Keyboard Input Bar*.
2. On the Draw menu, click *Line*.
3. In the Keyboard Input bar, enter the following coordinates to draw the board:

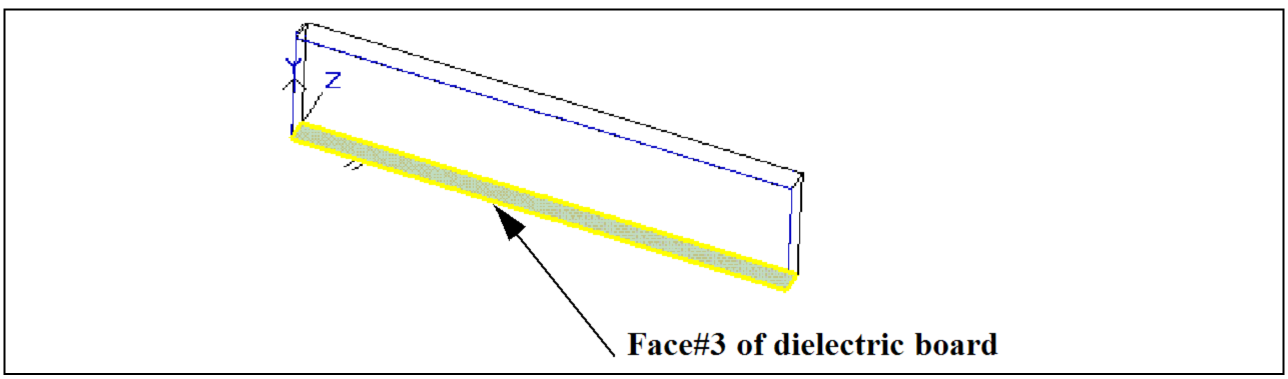
Start coordinates	**0, 0**	Press ENTER
End coordinates	**20, 0**	Press ENTER
End coordinates	**20, 5**	Press ENTER
End coordinates	**0, 5**	Press ENTER
End coordinates	**0, 0**	Press ENTER

4. Press ESC.

Make the dielectric board component

1. On the Edit menu, click *Select Construction Slice Surfaces*.
2. Select the interior region of the model you have drawn.
3. On the Model menu, click *Make Component in a Line*, and enter the following values:
   - Name: **Dielectric board**
   - Material: **Diel**
   - Distance: **1** Millimeters
4. Click OK.

Assign boundary conditions to the dielectric board



1. On the View menu, click *Wireframe Model*.
2. On the View menu, click  (Examine Model Dynamically).
3. Holding down the left mouse button, place the cursor over the model and rotate it so that it appears as the illustration above.
4. On the Edit menu, click *Select Component Surfaces*.
5. Expand the faces of the *Dielectric board* by clicking the + sign.
6. In the Object page, select *Face#3* of the *Dielectric Board* component.
   The bottom surface of the dielectric board is highlighted.
7. On the Boundary menu, click *Ground*.

Build the geometric model and set up the problem for the air box component

Draw the air box component

1. On the View menu, click *Preset Views*, and then *Positive Z Axis*.
2. On the Draw menu, click *Line*.
3. On the View menu, click *Update Automatically*.
4. In the Keyboard Input bar, enter the following coordinates to draw the air box:

Start coordinates	**0, 5**	Press ENTER
End coordinates	**20, 5**	Press ENTER
End coordinates	**20, 40**	Press ENTER
End coordinates	**0, 40**	Press ENTER
End coordinates	**0, 5**	Press ENTER

5. Press ESC.

Make the air box component

1. On the Edit menu, click *Select Construction Slice Surfaces*.
2. Select the interior region of the model you have drawn.
3. On the Model menu, click *Make Component in a Line*, and enter the following values:
   - Name: **Air box**
   - Material: **AIR**
   - Distance: **1** Millimeters
4. Click OK.

Build the geometric model and set up the problem for the microstrip component

Draw the microstrip component

1. On the View menu, click (Examine Model Dynamically).
2. Hold down the CTRL key and left mouse button, and then drag the cursor to form a rectangle (around the bottom left corner of the air box and the top left corner of the dielectric board), releasing the button once the desired area is selected.
3. On the Draw menu, click *Line*.
4. In the Keyboard Input bar, enter the following coordinates to draw the microstrip:

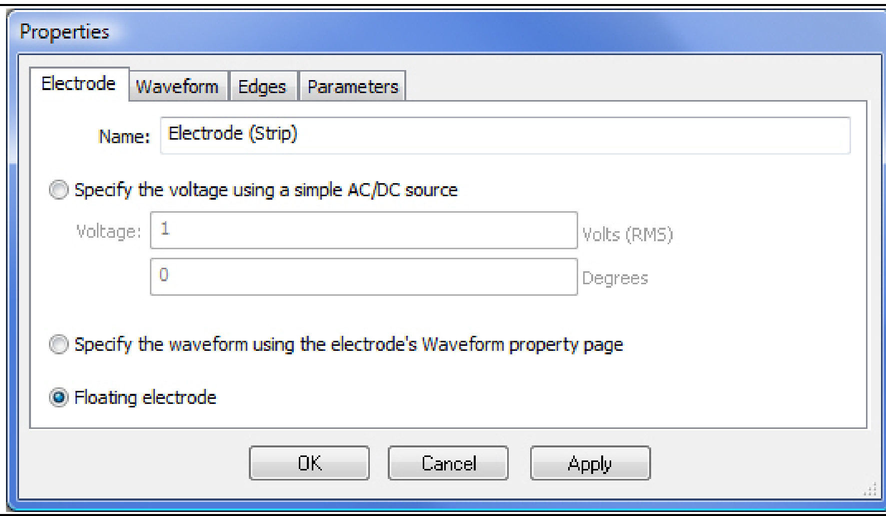
Start coordinates	**0, 5**	Press ENTER
End coordinates	**0.5, 5**	Press ENTER
End coordinates	**0.5, 5.25**	Press ENTER
End coordinates	**0, 5.25**	Press ENTER
End coordinates	**0, 5**	Press ENTER

5. Press ESC.

Make the microstrip component

1. On the Edit menu, click *Select Construction Slice Surfaces*.
2. Select the interior region of the model you have drawn.
3. On the Model menu, click *Make Component in a Line*, and enter the following values:
    - Name: **Microstrip**
    - Material: **Copper: 5.77e7 Siemens/meter**
    - Distance: **1** Millimeters
4. Click OK.

Make an electrode



1. In the Object page, select *Microstrip* component.
2. On the Model menu, click *Make Electrode*.
3. In the Object page, select *Electrode* #1.
4. On the Edit menu, click *Properties* and modify Electrode#1 to read:
    - Name: **Electrode (Strip)**
5. Select the Waveform tab and enter the following values for ACDC:
    - Magnitude (RMS): **1**
      Phase (Deg): **0**
6. Click OK.

Modifying the model prior to solving

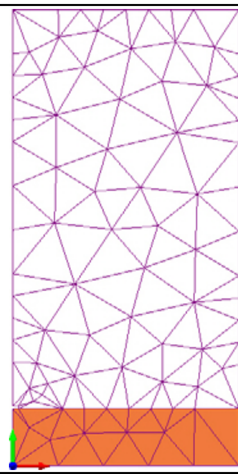
Modify the mesh

In the 2D finite element method of analysis, the model is divided into a mesh of triangular-shaped elements. The accuracy of the solution depends upon the nature of the field and the size of the mesh elements. In regions where the direction or magnitude of the field is changing rapidly, high accuracy requires small elements. One method of increasing mesh density is to set the *maximum element size* for a component volume or specific faces of a component. The following procedures will demonstrate this method.

View the initial mesh

Before changing the *maximum element size*, the default initial mesh can be viewed.

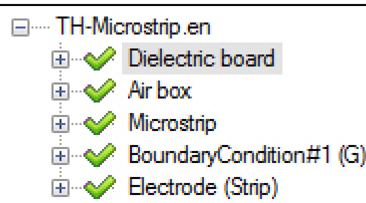
1. On the View menu, click *Update Automatically*.
2. On the View menu, click *Initial 2D Mesh*.
   The initial mesh appears in the View window.



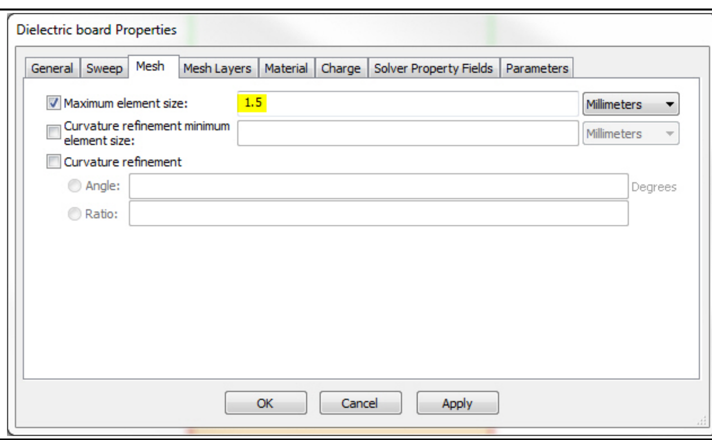
3. On the View menu, click *Solid Model*.

Set the maximum element size for each component

1. In the Object page of the Project bar, select the *Dielectric board* component.



2. On the Edit menu, click *Properties*.
   The Dielectric board Properties dialog appears.
3. Select the *Mesh* tab.
4. Click inside the *Maximum element size* checkbox, and then type **1.5** in the text box.



5. Click *Apply*.

   Tip Clicking Apply, instead of OK, keeps the dialog open and allows us to proceed to the next component without having to repeat steps 2 and 3.

6. In the Object page, select the *Air box* component.

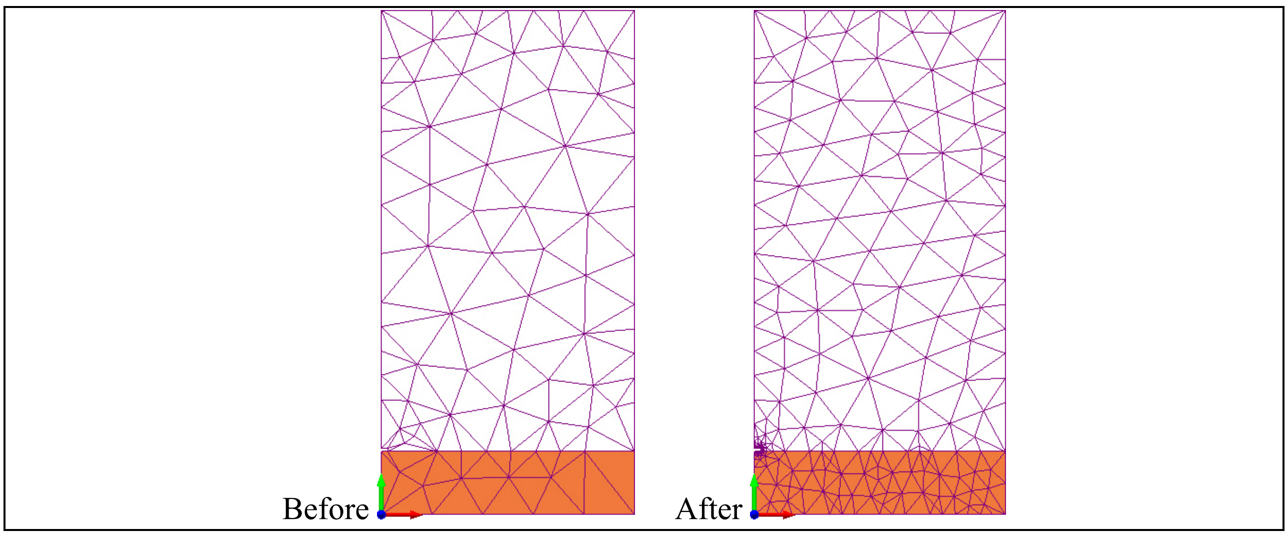
   Notice that the text in the Properties dialog Title Bar has now changed to read *Air box Properties*.

7. Click inside the *Maximum element size* checkbox, and then type **3** in the text box.
8. Click *Apply*.
9. In the Object page, select the *Microstrip* component.
10. Click inside the *Maximum element size* checkbox, and then type **0.1** in the text box.
11. Click OK.

View the changes to the mesh

1. On the View menu, click *Initial 2D Mesh*.

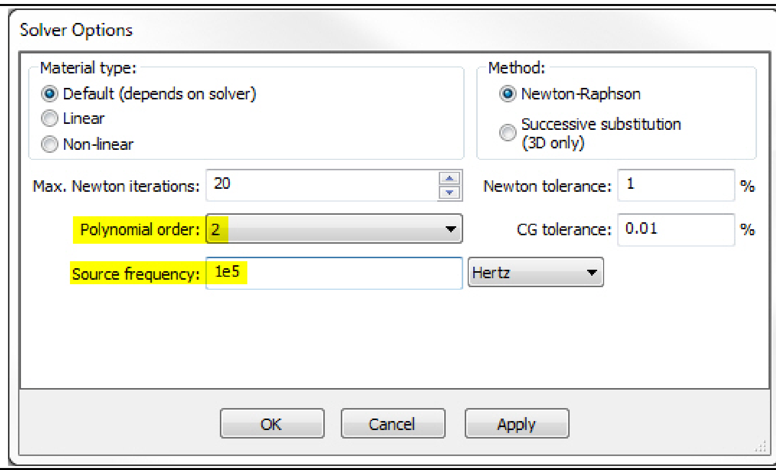
   The modified initial mesh appears in the View window and should look similar to the following illustration, on the right.



2. On the View menu, click *Solid Model*.

Generating the time-harmonic field solution

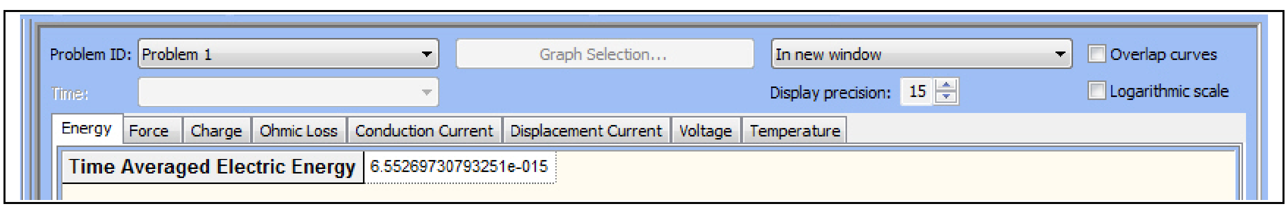
Set the solving options



1. On the Solve menu, click *Set Solver Options*, and verify that the following values are set:
    - Material type: *Default*
    - Method: *Newton-Raphson*
    - Maximum Newton iterations: *20*
    - Newton tolerance: *1%*
    - Polynomial order: ***2***
    - CG tolerance: *0.01%*
    - Source frequency: ***1e5*** *Hertz*
2. Click OK.

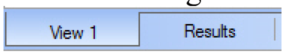
Solve the model

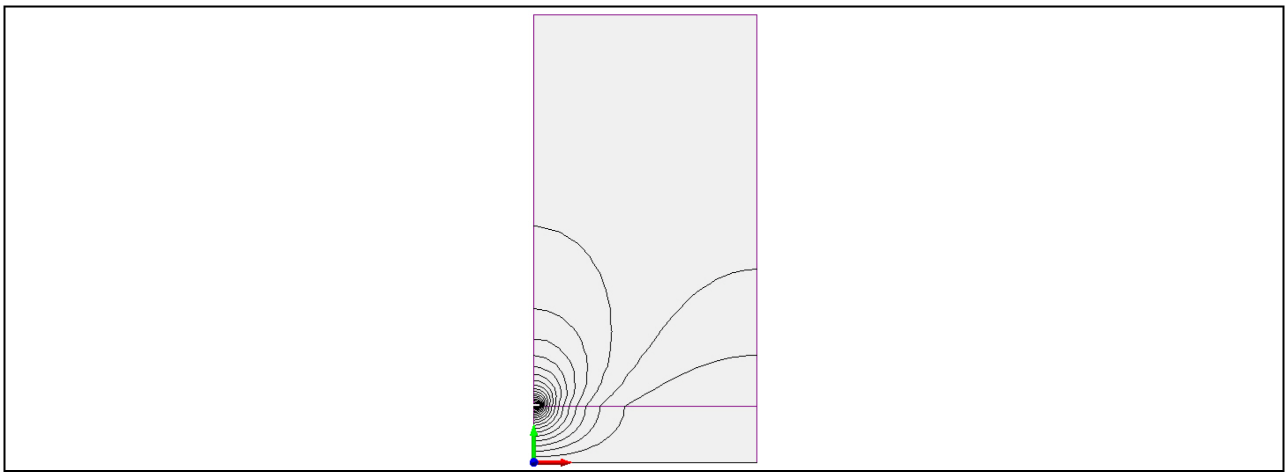
- On the Solve menu, click *Time Harmonic 2D*.
  The ElecNet Time-Harmonic 2D Solver Progress dialog appears briefly and then the Results window opens.



Analyzing the results

View the contours of V at 0'

1. Before viewing the contour plot, switch back to the View window by clicking the View tab  located at the bottom of the window.
2. From the Project Bar, select the "Field" page.
   The page appears with the Contour tab selected.
3. In the "Fields to Display" box, select *V at 0'*.
4. Select the "Shaded" tab.
5. In the "Fields to Display" box, select *None*.
6. At the bottom of the Field page, click *Update View*.
   The contour plot of *V at 0'* field is displayed.



Calculate the capacitance between the electrode and the ground

Calculating the capacitance requires taking the *imaginary part* of the total current (add the *conduction and displacement current* in the electrode) divided by the voltage, and then dividing it by the angular frequency w.

Note Adjust the display precision control on the Results page to 15 significant digits

voltage = 1
frequency = 1e5

$$\frac{-1.283726057337429e\text{-}009 + 9.5180883490292279e\text{-}009}{\text{voltage}} = \mathbf{8.2343622916917989e\text{-}009}$$

and

$$\frac{8.2343622916917989e\text{-}009}{2\pi f} = \mathbf{1.3105394619322571e\text{-}014}$$

Calculate the conductance through the lossy dielectric substrate

Calculating the conductance requires taking the *real part* of the total current (add the *conduction and displacement current* in the electrode) and then dividing it by the voltage.

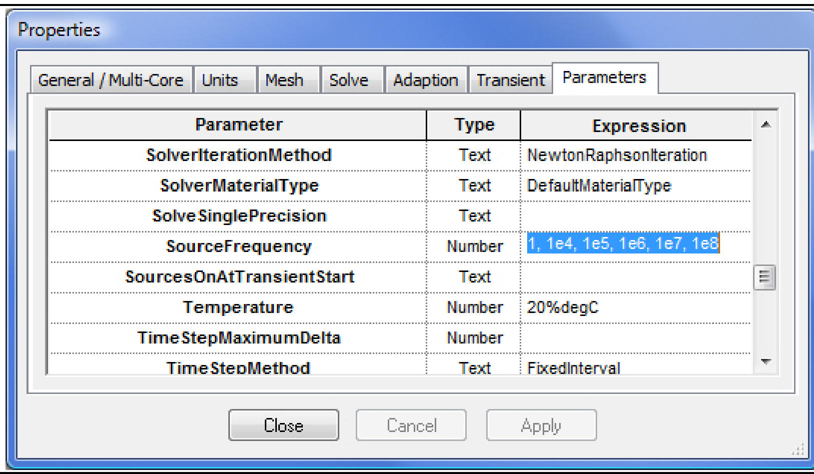
voltage = 1

$$\frac{4.867709461942146e\text{-}008 + 2.6640570893869313e\text{-}010}{\text{voltage}} = \mathbf{4.894350032836015e\text{-}008}$$

Parameterization

Parameterizing the solution frequency

Here we will use parameterization to study the behaviour as a function of frequency.

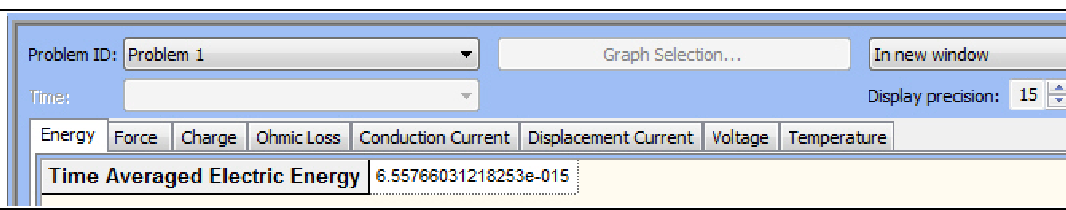


1. From the Object page, select the model (i.e. TH-Microstrip.en).
2. On the Edit menu, click *Properties*.
   The model properties dialog appears.
3. Click the “Parameters” tab.
4. Scroll down to the *SourceFrequency* parameter.
5. In the Expression column, highlight **100000%Hz** and then insert the following values: **1, 1e4, 1e5, 1e6, 1e7,** and **1e8**.
6. Press Tab.
7. Click OK.

Generating the time-harmonic field solution after parameterization

Solve the model

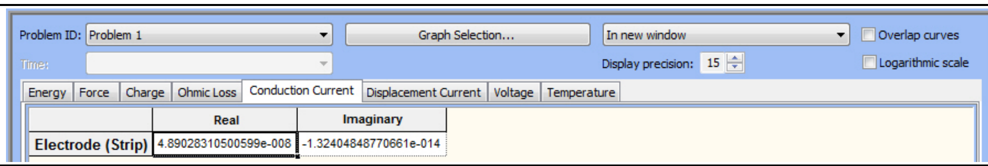
- On the Solve menu, click *Time Harmonic 2D*.
  The ElecNet Time-Harmonic 2D Solver Progress dialog appears briefly and then the Results window opens.



Analyzing the results after parameterization

Graph the conduction and displacement current as a function of frequency

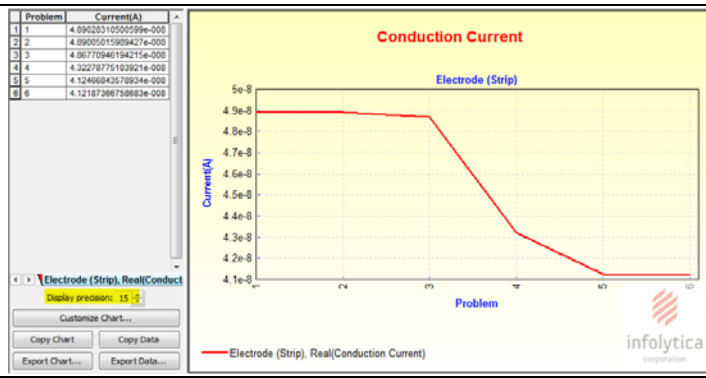
The Results window is automatically displayed when the Solver Progress dialog closes and the solution is complete. The effect of having parameterized *Source Frequency* with several values has created a multiple-problem analysis. The procedure that follows uses *Prob 1* as an example.



1. In the Results window, open the "Conduction Current" page for Problem 1.
2. Using the mouse pointer, click the *Real* value (i.e. 4.8902831050059879e-008) inside the Electrode (Strip) text box.
3. Click the *Graph Selection* button, located at the top of the Results window.

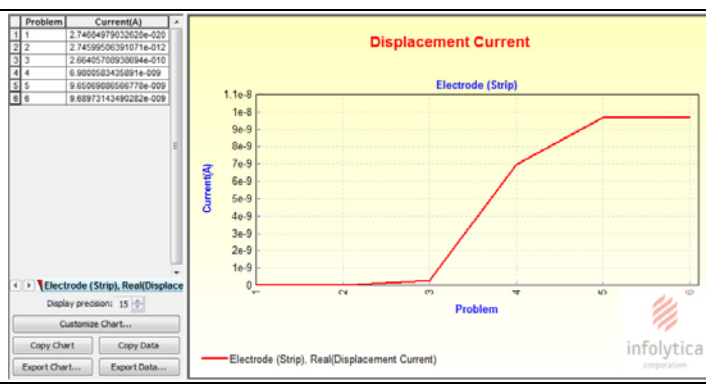
   The graph should look like the illustration below.

Note If required, adjust the display precision control on both Charts, illustrated below, to 15 significant digits to display the same values as shown in the image below.



4. In the Results window, open the "Displacement Current" page for Problem 1.
5. Using the mouse pointer, click the *Real* value (i.e. 2.7468497902597885e-020) inside the Electrode (Strip) text box.
6. Click the *Graph Selection* button, located at the top of the Results window.

   The graph should look like the illustration below.



7. On the File menu, click *Save*.

Reconfiguring the problem to perform a transient analysis

Removing the parameterization of the solution frequency

Before proceeding with the transient analysis, we will remove the values assigned to SourceFrequency. This will, in effect, return the tutorial to a single-problem analysis.

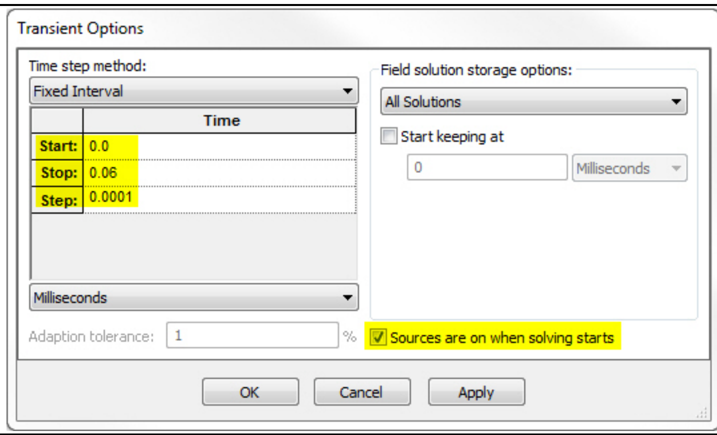
1. From the Object page, select the model (i.e. TH-Microstrip.en).
2. On the Edit menu, click *Properties*.
   The model properties dialog appears.
3. Click the "Parameters" tab.
4. Scroll down to the *SourceFrequency* parameter.
5. In the Expression column, highlight the following values:
   1, 1e4, 1e5, 1e6, 1e7, and **1e8**.
6. Press *Delete*.
7. Click Close.

Set the start, stop and step times

The start, stop, and step times are defined in the *Set Transient Options* dialog.

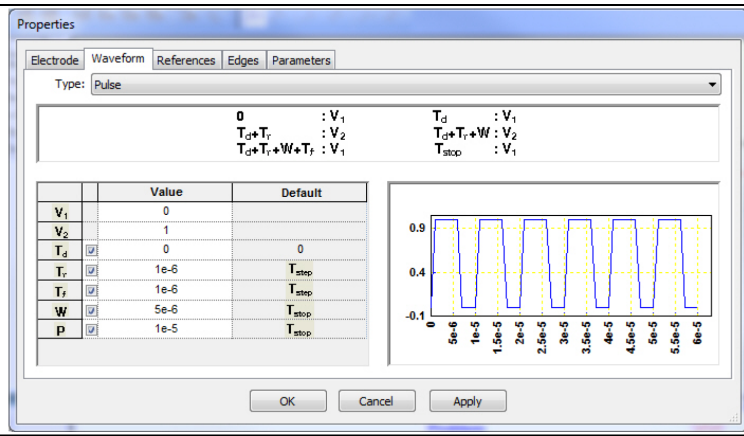
Note The transient solver assumes the source values before the start time are equal to the values at the start time only if the parameter SourcesOnAtTransientStart is set to Yes.

1. On the Solve menu, click *Set Transient Options*.
   The Transient Options dialog appears.



2. Make sure that Fixed Interval is selected as the Time Step method and Milliseconds as the unit for time, and then make the following modifications for Time:
   - Start = **0.0**
   - Stop = **0.06**
   - Step = **0.0001**
3. Click "Sources are on when solving starts" to set the SourcesOnAtTransientStart parameter to Yes.
4. Click OK.

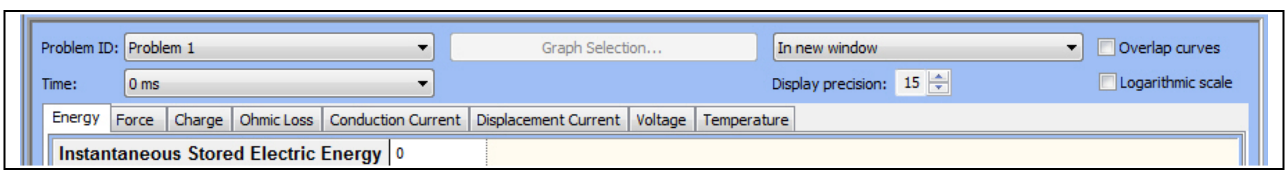
Defining a pulse waveform for the electrode



1. On the Project Bar, select the "Electrode" tab.
2. Place the cursor over *Electrode (Strip)* item and then right-click the mouse.
3. On the floating menu, click *Properties*.
   The *Properties* dialog appears.
4. Select the *Waveform* tab.
5. In the Type drop-down list, select *Pulse*.
6. Click the **P** checkbox to enable *Period in seconds* and all preceding optional values.
7. Insert the following values in the appropriate box:
   - V_1 --> **0**
   - V_2 --> **1**
   - T_d --> **0**
   - T_r --> **1e-6**
   - T_f --> **1e-6**
   - W --> **5e-6**
   - P --> **1e-5**
8. Click OK.

Solve

- On the Solve menu, click *Transient 2D*.
  The ElecNet Transient 2D Solver Progress dialog appears for a few seconds and then the Results window opens.



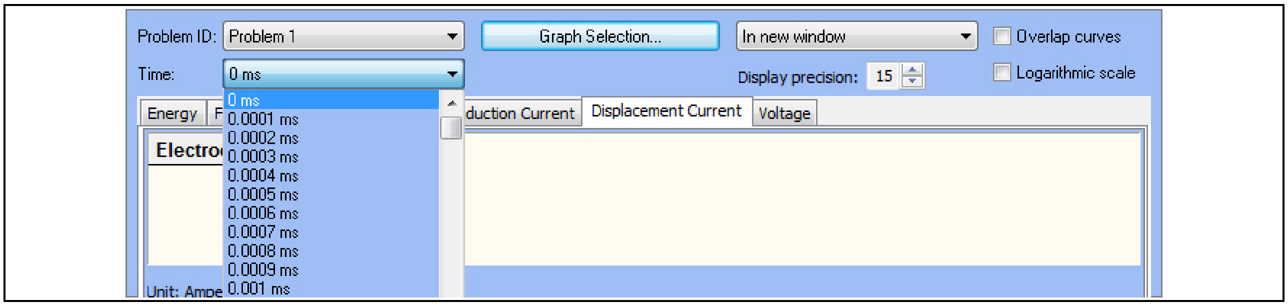
View the solution results

The following results will be reviewed in this section:

- A graph of the conduction current versus time
- A graph of the displacement current versus time

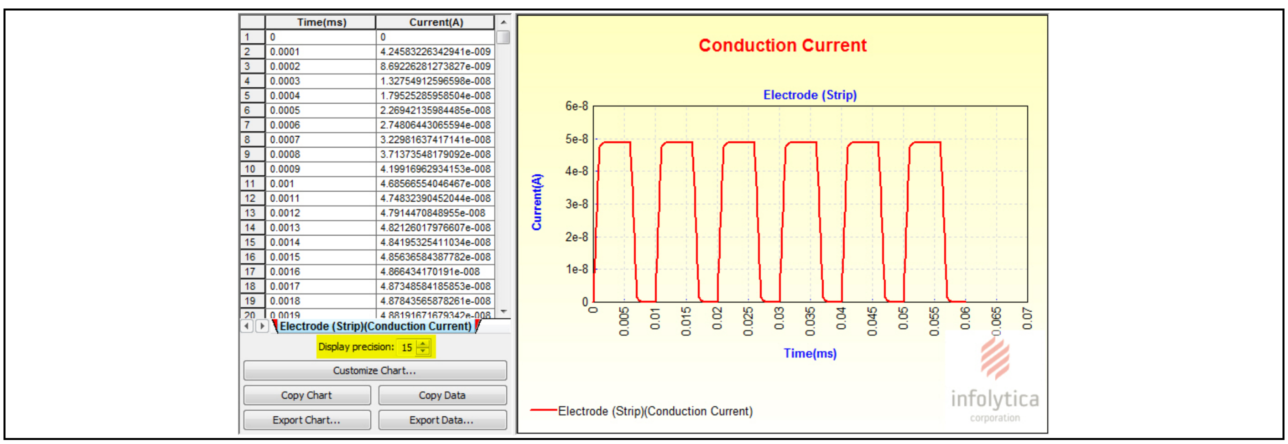
Graph the conduction and displacement current versus time

The Results window is automatically displayed when the Solver Progress dialog closes and the solution is complete. The Time Steps can be viewed by clicking the Time drop-down list box (as shown).



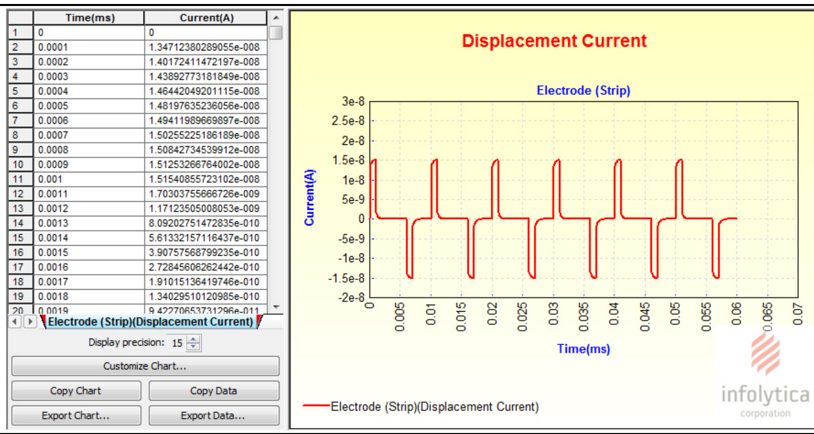
1. In the Results window, open the “Conduction Current” page.
2. Using the mouse pointer, click anywhere inside the *Electrode (Strip)* text box.
3. Click the *Graph Selection* button, located at the top of the Results window.

   The graphs that follow should look like the illustrations below once you have adjusted the display precision control on the Charts to 15 significant digits to display the same values.



4. In the Results window, open the “Displacement Current” page.
5. Using the mouse pointer, click anywhere inside the *Electrode (Strip)* text box.

6. Click the *Graph Selection* button, located at the top of the Results window. The graph should look like the illustration below.



Note The total current can be obtained by adding the displacement current with the conduction current.

Save the model

You have now completed the 2D Time Harmonic and 2D Transient tutorial.

1. On the File menu, click *Save*.
2. On the File menu, click *Close*.

Summary

In this tutorial, you completed the steps in creating a microstrip model for time-harmonic and transient solutions. The skills you learned include:

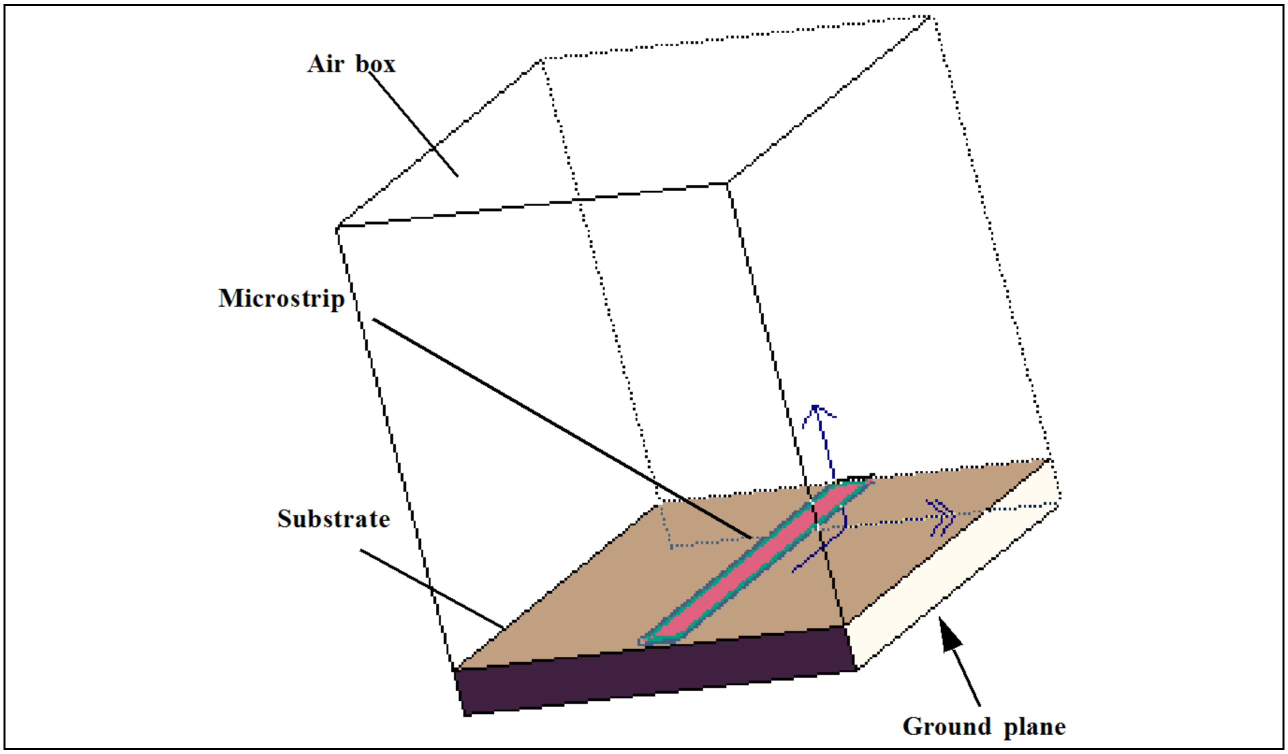
- Setting up the work environment by modifying initial settings, the construction grid, and the viewing area.
- Building the geometric model using the Keyboard Input Bar.
- Setting up the problem -- this consists of making components and electrodes, using parameterization, and assigning boundary conditions.
- Modifying the model to obtain the desired solution results.
- Generating the time-harmonic field solution using ElecNet's 2D Time-harmonic solver.
- Generating the transient field solution using ElecNet's 2D Transient solver.
- Analyzing the results, which includes:
    - viewing the contours of V at 0'
    - calculating the conductance and capacitance in the electrode
    - graphing the conduction and displacement current as a function of frequency and time

ElecNet
Tutorial #4

3D Electrostatic and Transient Tutorial
Microstrip

Modeling plan

This problem consists of a substrate, a microstrip, a ground plane, and an air box.



Creating a new model

Opening a new model

- From your desktop, double-click the ElecNet icon.
  The Main window appears.
  or
- If ElecNet is already running, on the File menu, click *New* to open a new model.

Name the model

By default, ElecNet assigns a name to the model (e.g., ElecNet1) every time a new model is opened. As long as the application remains open, each new model number increments by one (e.g., if the new model you have opened is the fourth one in this session, ElecNet would assign the name ElecNet4). You can choose to retain this name, although it is recommended that you give the model a distinct name.

1. On the File menu, click *Save As*.
2. In the Save As dialog box, enter **Microstrip** as the name of the model.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

Setting up the working environment

Initial Settings

Each new model reverts to the ElecNet default settings for the meshing method, preferred units for length, time and frequency, and for how curves will be displayed in the view. For our model, we are going to change only the preferred unit for length and accept all the other defaults.

1. From the Object page, select the model.
2. On the Edit menu, click *Properties*.
3. Select the *Units* tab.
4. From the Length drop down list, select *Microns*.
5. Click OK.

Set view to update automatically

Use of this feature resets the view automatically to include all of the model's geometry, as it is being drawn.

- On the View menu, click *Update Automatically*.

Build the geometric model and set up the problem for the substrate component

Draw the substrate component

1. On the Tools menu, click *Keyboard Input Bar*.
2. On the Draw menu, click *Line*.
3. In the Keyboard Input bar, enter the following coordinates to draw the substrate:

Start coordinates	**-4500, 0**	Press ENTER
End coordinates	**5500, 0**	Press ENTER
End coordinates	**5500, 1000**	Press ENTER
End coordinates	**-4500, 1000**	Press ENTER
End coordinates	**-4500, 0**	Press ENTER

4. Press ESC.

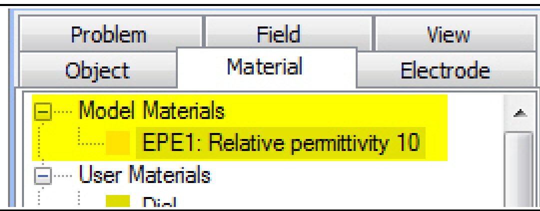
Make the substrate component

1. On the Edit menu, click *Select Construction Slice Surfaces*.
2. Select the interior region of the model you have drawn.
3. On the Model menu, click *Make Component in a Line*, and enter the following values:
    - Name: **Substrate**
    - Material: **EPE1: Relative permittivity 10**
    - Distance: **10000** Microns
4. Click OK.

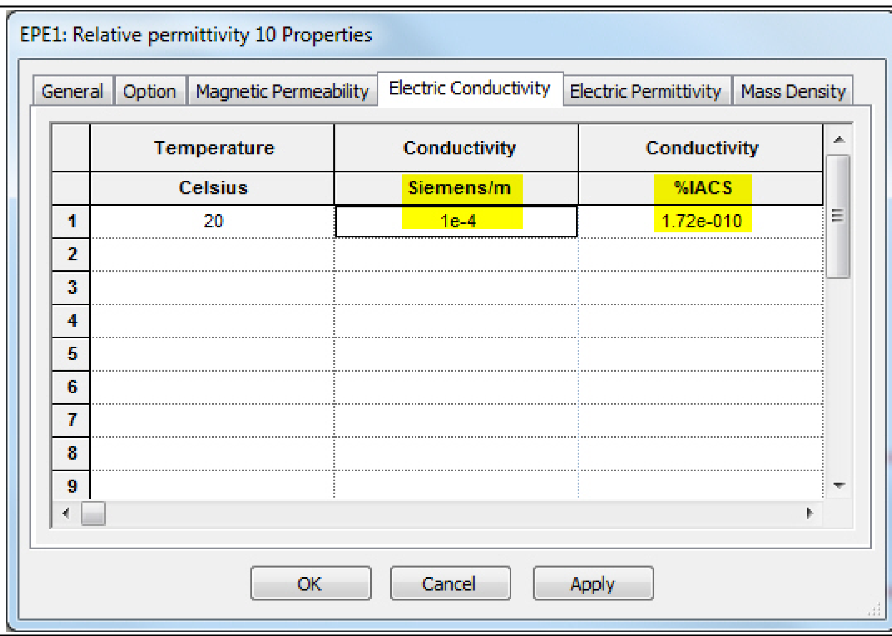
5. From the Object page, select the *Substrate* component.
6. On the Edit menu, click *Properties*.
7. Select the *Mesh* tab.
8. Click the *Maximum Element Size* checkbox and then type **500** in the text box.
   Verify that units are in Microns.
9. Select the *Parameters* tab.
10. Scroll down to *PolynomialOrder* and type **2** in the Expression text box.
11. Click OK.

Set the electric conductivity of the material "EPE1"

1. In the Project Bar, select the *Material* tab.
2. Select "EPE1: Relative Permittivity 10" in the Model Materials database.

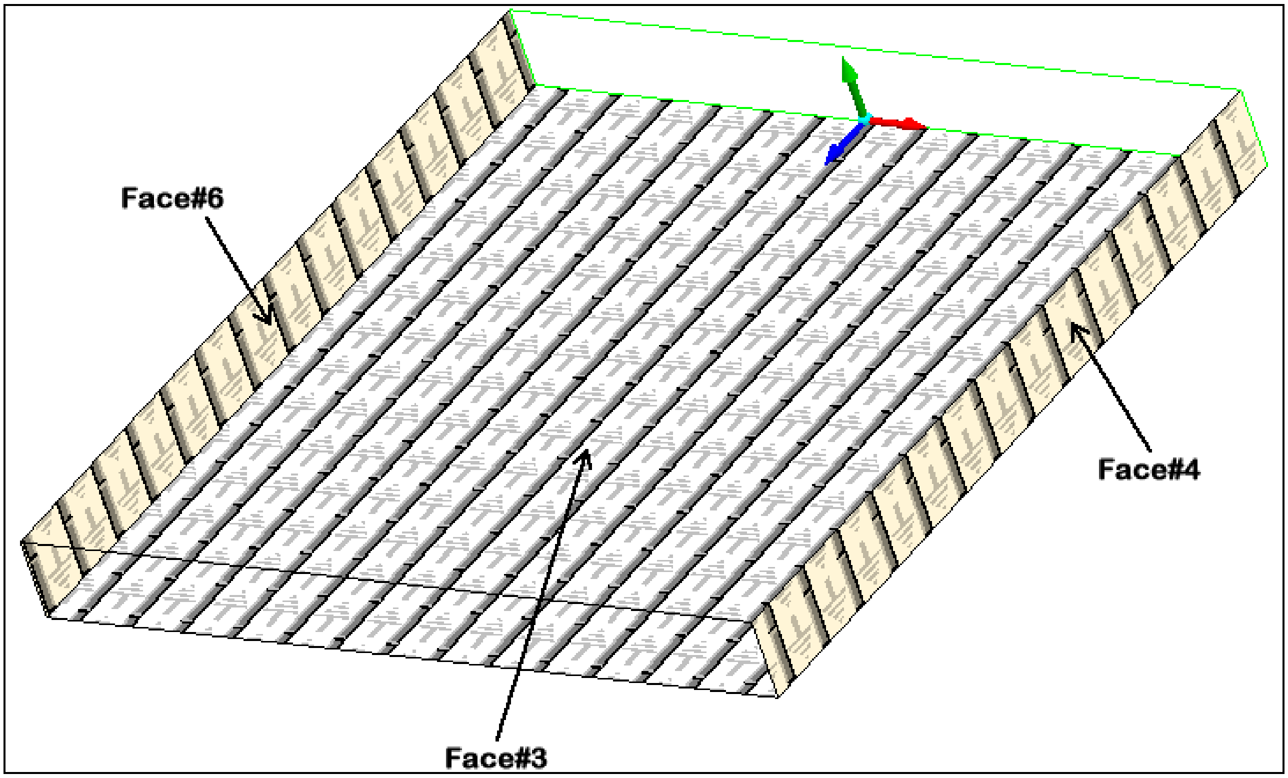


3. Right-click and then select *Properties*.
   The "EPE1: Relative Permittivity 10" Properties dialog appears.
4. Select the "Electric Conductivity" tab.
5. In the *Conductivity* Siemens/m column, type **1e-4** and then press Tab.
   The value for %IACS is automatically generated.



6. Click OK.

Assign boundary conditions to the substrate



1. On the View menu, click *Examine Model Dynamically*.
2. Place and hold the cursor over the model and rotate upward until it appears similar to the sample above.
3. In the Object page, select *Face#3* of the *Substrate* component.
   The bottom surface of the substrate is selected.
4. On the Boundary menu, click *Ground*.
5. Repeat procedures 3 (substituting Face#3) and 4 for *Face#4* and then *Face#6*.

Move the construction slice to draw the microstrip

Move the construction slice

1. In the Object page, select Face#5 of the *Substrate* component.
2. On the Draw menu, click *Move Construction Slice*.
3. Ensure that the "To the currently selected surface" option is selected, and click OK.
4. On the Draw menu, click *Move Construction Slice*.
5. Select the "Along an arc" option and enter the following values, if required:
   - Angle: **180** degrees
   - Center: **(0, 0)** Microns
   - Axis vector: **(0, -1)**
6. Click OK.

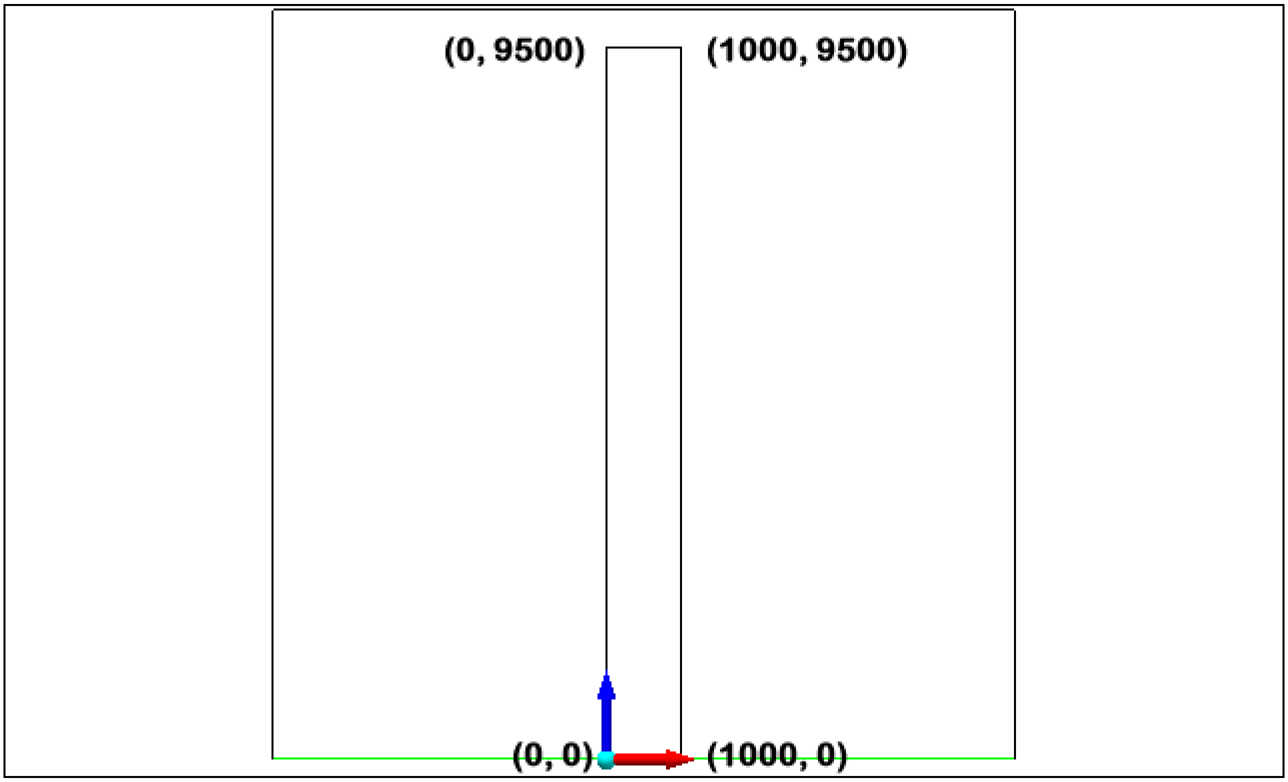
Delete construction slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

1. On the Edit menu, click *Select Construction Slice Edges*.
2. On the Edit menu, click *Select All*.
3. Press the *Delete* key.

Build the geometric model and set up the problem for the microstrip component

Draw the microstrip component



1. On the View menu, click *Wireframe Model*.
2. On the View menu, click *Preset Views*, and then click *Negative Y Axis*.
3. On the Draw menu, click *Line*.
4. In the Keyboard Input bar, enter the following coordinates to draw the microstrip:

Start coordinates	**0, 0**	Press ENTER
End coordinates	**0, 9500**	Press ENTER
End coordinates	**1000, 9500**	Press ENTER
End coordinates	**1000, 0**	Press ENTER
End coordinates	**0, 0**	Press ENTER

5. Press ESC.

Make the microstrip component

1. On the Edit menu, click *Select Construction Slice Surfaces*.
2. Select the interior region of the model you have drawn.
3. On the Model menu, click *Make Component in a Line*, and enter the following values:
   - Name: **Strip**
   - Material: **AIR**
   - Distance: **0** Microns
4. Click OK.
5. From the Object page, select the *Strip Face#1 (Start face)* surface.
6. On the Edit menu, click *Properties*.
7. Select the *Mesh* tab.
8. Click the *Maximum Element Size* checkbox and then type **100** in the text box.
   Verify that units are in Microns.
9. Click OK.

Make an electrode

1. In the Object page, select *Strip* component.
2. On the Model menu, click *Make Electrode*.
3. In the Object page, select *Electrode #1*.
4. On the Edit menu, click *Properties* and modify Electrode#1 to read:
   - Name: **Electrode (Strip)**
5. Select the Waveform tab and enter the following values for ACDC:
   - Magnitude (RMS): **0.000707107**
     Phase (Deg): **0**
6. Click OK.

Move the construction slice to draw the boundaries

Move the construction slice

1. In the Object page, select *Face#1 (Start Face)* of the *Substrate* component.
2. On the Draw menu, click *Move Construction Slice*.
3. Ensure that the "To the currently selected surface" option is selected, and click OK.

Delete construction slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

1. On the Edit menu, click *Select Construction Slice Edges*.
2. On the Edit menu, click *Select All*.
   Make sure that the View window is the active window.
3. Press the *Delete* key.

Build the geometric model and set up the problem for the air box component

Draw the air box component

1. On the View menu, click *Preset Views*, and then click *Positive Z Axis*.
2. On the Draw menu, click *Line*.
3. In the Keyboard Input bar, enter the following coordinates to draw the air box:

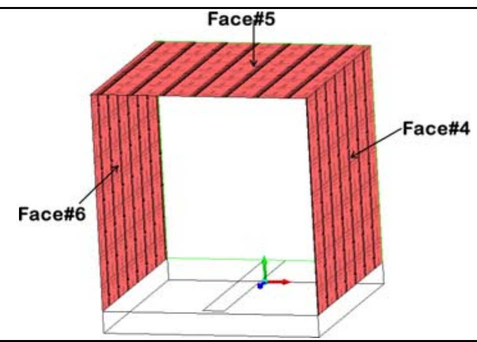
Start coordinates	**-4500, 1000**	Press ENTER
End coordinates	**5500, 1000**	Press ENTER
End coordinates	**5500, 11000**	Press ENTER
End coordinates	**-4500, 11000**	Press ENTER
End coordinates	**-4500, 1000**	Press ENTER

4. Press ESC.

Make the air box component

1. On the Edit menu, click *Select Construction Slice Surfaces*.
2. Select the interior region of the model you have drawn.
3. On the Model menu, click *Make Component in a Line*, and enter the following values:
   - Name: **Air box**
   - Material: **AIR**
   - Distance: **10000** Microns
4. Click OK.
5. From the Object page, select the *Air box* component.
6. On the Edit menu, click *Properties*.
7. Select the *Parameters* tab.
8. Scroll down to *PolynomialOrder* and type **2** in the Expression text box.
9. Click OK.

Assign boundary conditions to the air box



1. On the View menu, click *Solid Model*.
2. On the View menu, click *Examine Model Dynamically*.
3. Place and hold the cursor over the model and rotate upward until it appears similar to the sample above.
4. In the Object page, select *Face#4* of the *Air box* component.
5. On the Boundary menu, click *Ground*.
6. Repeat procedures 3 and 4 for *Face#5* and *Face#6*.

Modifying the model prior to solving

Create new slices for X=500 and Z=0

In this procedure, we are going to create two new slices, with one running through the microstrip component (X=500), and the other perpendicular to it (Z=0). This will allow us to view the field plot for each slice.

Note Slices are only visible in a solution mesh.

1. On the Tools menu, click *New Slice*.
2. Select the "Based on the point-normal definition of a plane" option, and enter the following data:
   - Point in the slice: (**500, 0, 0**) Microns
   - Normal on the slice: (**1, 0, 0**)
3. Click OK.
   The slice (i.e. Slice#1) for X=500 is created.
4. On the Tools menu, click *New Slice*.
5. Select the "Based on the point-normal definition of a plane" option, and enter the following data:
   - Point in the slice: (**0, 0, 0**) Microns
   - Normal on the slice: (**0, 0, 1**)
6. Click OK.
   The slice (i.e. Slice#2) for Z=0 is created.

Generating the electrostatic field solution

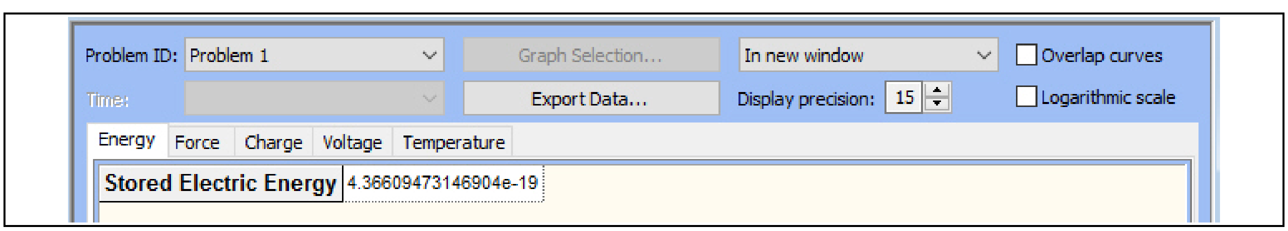
Set the solving options

1. On the Solve menu, click Set Solver Options, and verify that the following values are set:
   - Material type: *Non-linear*
   - Method: *Newton-Raphson*
   - Maximum Newton iterations: *20*
   - Newton tolerance: *1 %*
   - CG tolerance: *0.01 %*
   - Source frequency: *1e5 Hertz*

   Note CG Tolerance is the only value that has any significance for electrostatic solutions. All other values are listed for information purposes only.
2. Click OK.

Solve the model

- On the Solve menu, click *Static 3D*.
  The Static 3D Solver Progress dialog appears briefly and then the Results window opens.



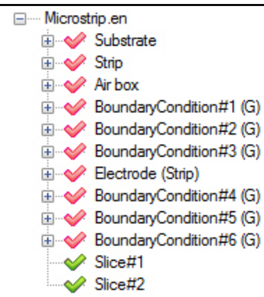
Analyzing the results

In this section, the following results are viewed:

- Contours of V and shaded plot of |E| smoothed at Z=0
- Contours of V and shaded plot of |E| smoothed at X=500

Hide all the components

Hiding all of the components and the boundary conditions will allow us to view the fields on the two slices.



1. On the Object page, keeping the Shift key down, first select *Substrate* and then select *BoundaryCondition#6 (G)*.

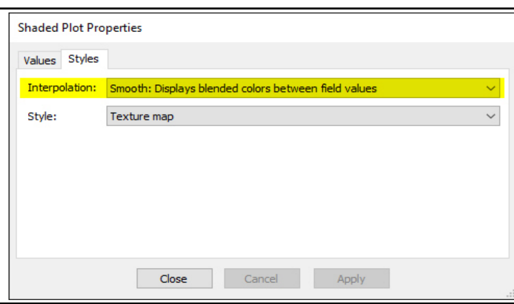
   All the required components and boundary conditions are selected.
2. On the Edit menu, click *Toggle Visibility of Selected Components*.
3. The symbol appearing next to each component object indicates that they are hidden.

Set the color interpolation and style of the shaded plot

This procedure will set the default for shaded plots to smooth instead of discrete, which is the default.

1. On the View menu, click *Default Fields*.
2. On the Project Bar, select the *View* tab.
3. From the View tree, click *Shaded Plot*.
4. On the Edit menu, click *Properties*.

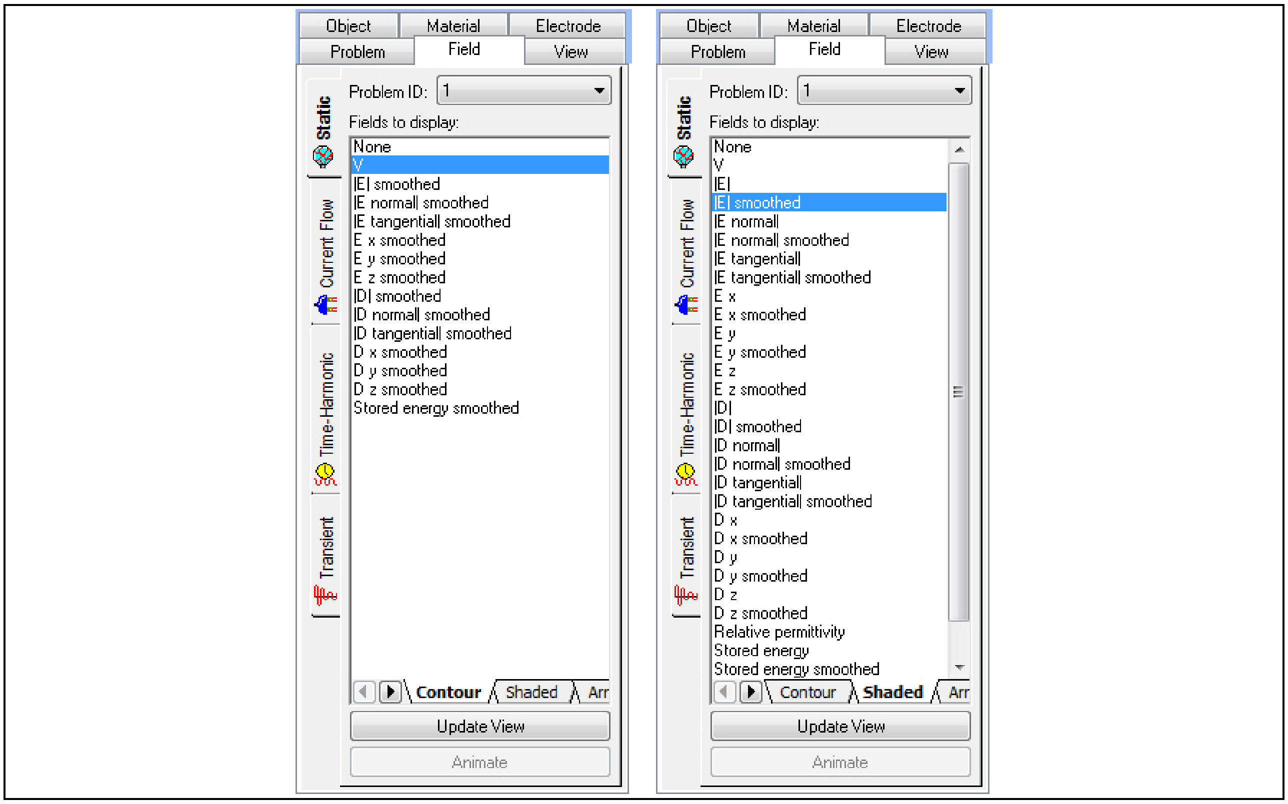
   The Shaded Plot Properties page appears.



5. Click the Styles tab and then from the Interpolation drop-down menu, select "Smooth: Displays blended colors between field values".
6. Click OK.

View the contours of V and shaded plot of |E| smoothed

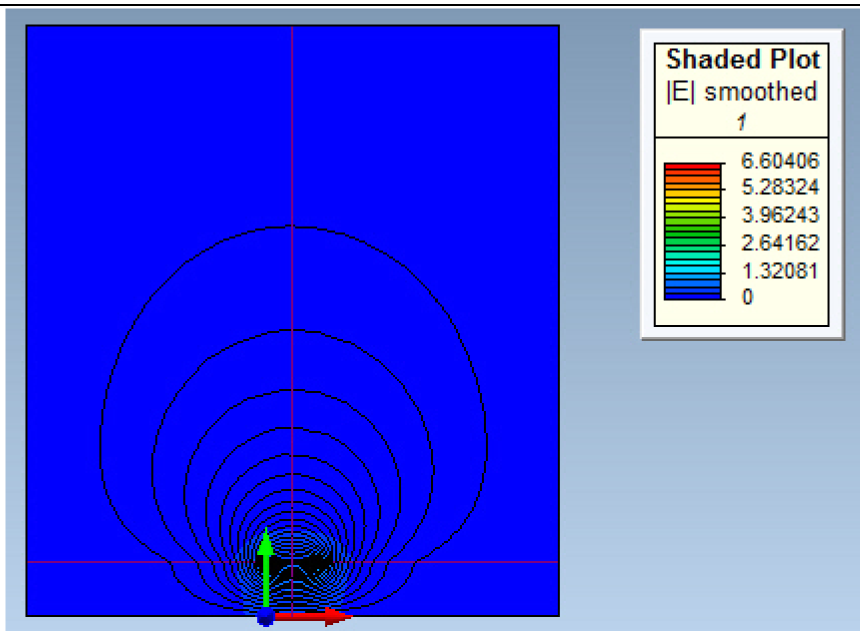
Since V and |E| smoothed are the default fields for ElecNet, this procedure is pretty straightforward. To plot these fields on the slice we created will require only one additional step, that is, changing the view for X=500.



1. Before viewing the contour and shaded plots, switch back to the View window by clicking the View tab 

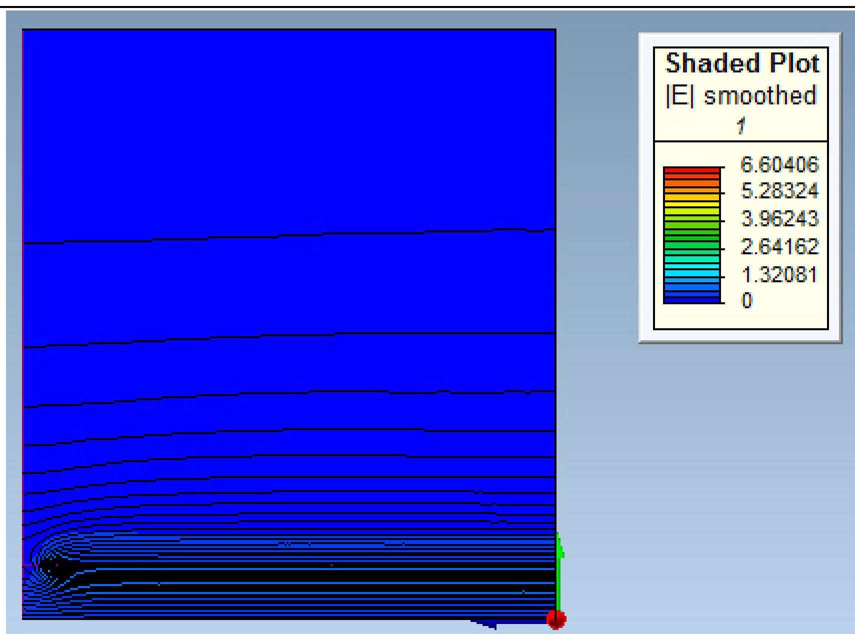
located at the bottom of the window.
2. On the View menu, click *Preset Views*, and then click *Positive Z Axis*.
3. From the Project Bar, select the “Field” page.
   The page appears with the “Contour” tab selected.
4. In the “Fields to Display” box, verify that *V* is selected.
   Optionally, you can select the Shaded tab and verify that |E| smoothed is selected. As mentioned above, since both fields are the defaults, this step is for verification purposes, only.

5. At the bottom of the Field page, click *Update View*.
   The view of the combined fields (contours of V and the shaded plot of |E| smoothed) at Z=0 is displayed.



Note The boundary of the electrode represents a singularity – the more the boundary is refined, the higher |E| will be.

6. On the View menu, click *Preset Views*, and then click *Positive X Axis*.
   The view of the combined fields (contours of V and the shaded plot of |E| smoothed) at X=500 is displayed.

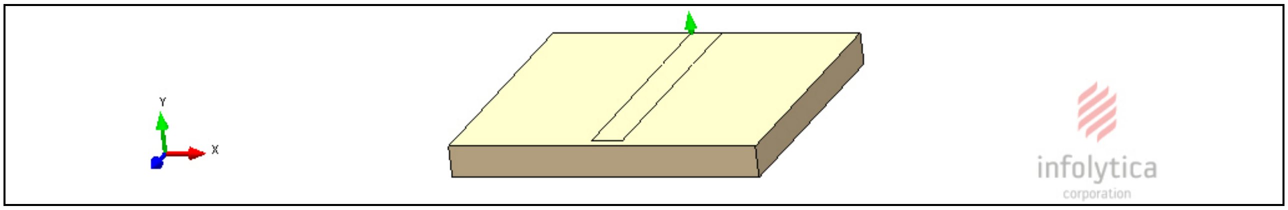


Note The boundary of the electrode represents a singularity – the more the boundary is refined, the higher |E| will be.

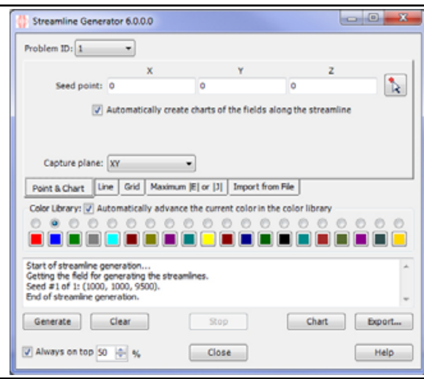
Generating streamlines

The following procedure demonstrates how the *Streamline Generator* extension can be used to create "Flux function" contour plots (i.e. streamlines) in a 3D model.

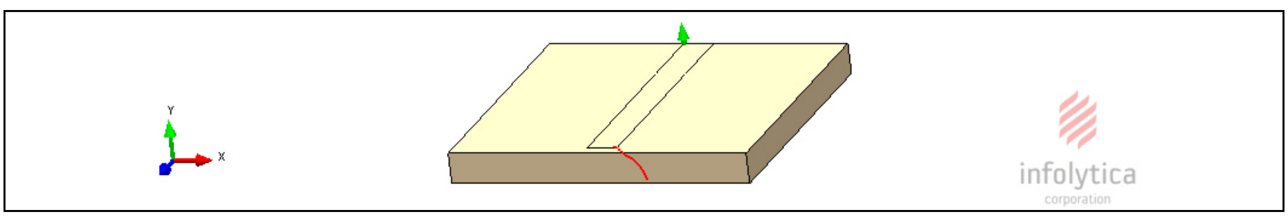
1. On the Object page, keeping the Ctrl key down, first select *Substrate* and then select *Strip*.
2. On the Edit menu, click *Toggle Visibility of Selected Components*.
   A ✓ symbol appears next to the selected components to signify that they are now visible.
3. On the View menu, click *Solid Model*.
4. On the View menu, click *Examine Model Dynamically*.
5. Place and hold the cursor over the model and rotate it until it appears similar to the sample below.



6. On the Extensions menu, click *Streamline generator*.
   The Streamline Generator extension appears.



7. Select the *Maximum* |*E*| *or* |*J*| tab.
8. Click Generate.
   Streamlines are generated using seed points that are the coordinates of the points in the solution where maximum |E| occurs, as shown (red line) in the illustration below.

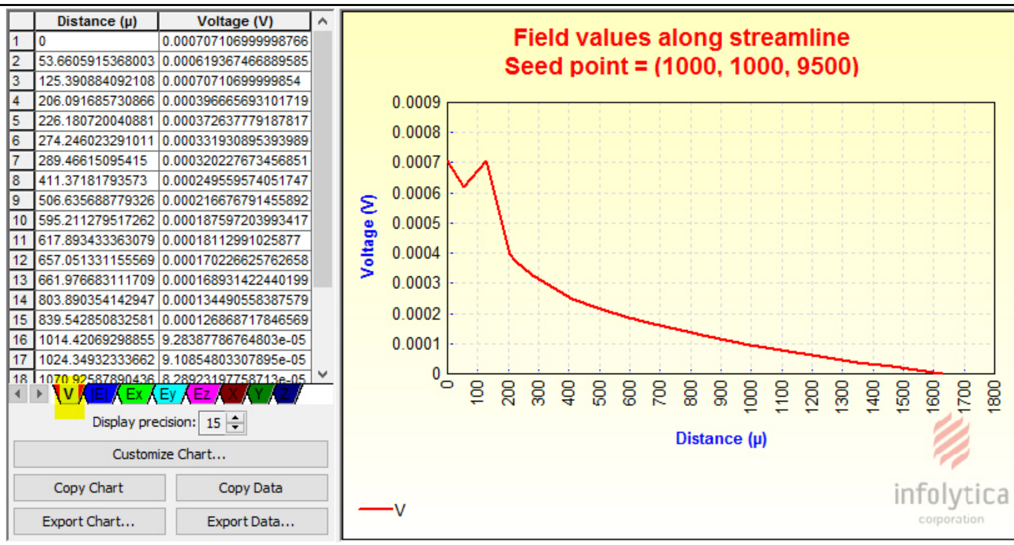


The report displays the following information:

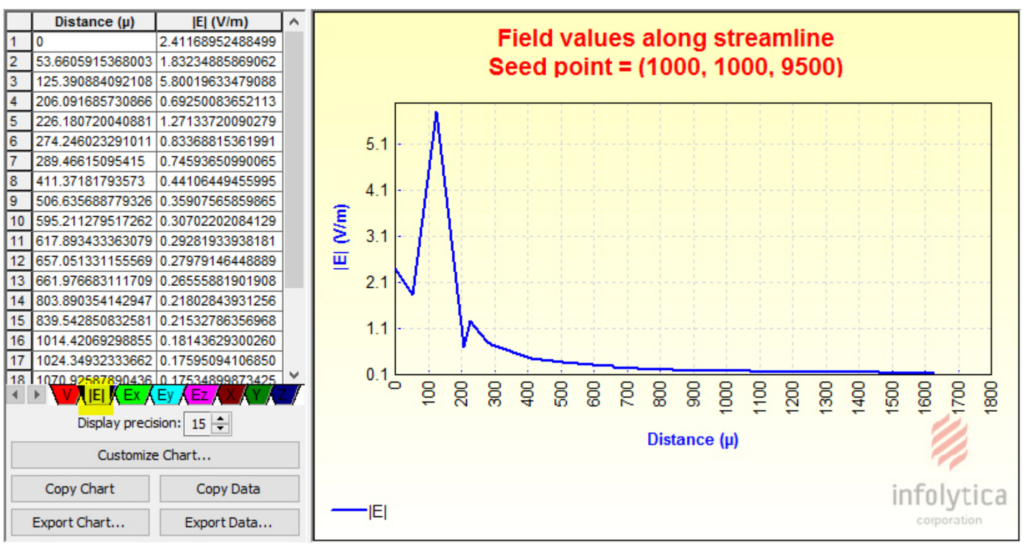
Start of streamline generation...
Getting the field for generating the streamlines.
Seed #1 of 1: (1000, 1000, 9500).
End of streamline generation.

9. Select the *Point & Chart* tab.
10. Using the coordinates [i.e. Seed #1 of 1: (1000, 1000, 9500)] listed in the report, enter these values in the X, Y, and Z text boxes, and make sure that "Automatically create charts of the fields along the streamline" is checked ☑.
11. Click Generate.

    Since we are using the same seed points, an identical streamline path to the one generated for Maximum |E| is produced. In addition to the streamlines, charts showing the field values of "Voltage (V) vs Distance (μ)" and "|E| (V/m) vs Distance (μ)" along streamlines (1000, 1000, 9500) are displayed.



Voltage (V) vs Distance (μ) chart



|E| (V/m) vs Distance (μ) chart

12. Click *Clear* to remove the streamlines from the model and then click *Close* to exit the Streamline Generator extension.
13. Close the Chart window and then maximize the model view.

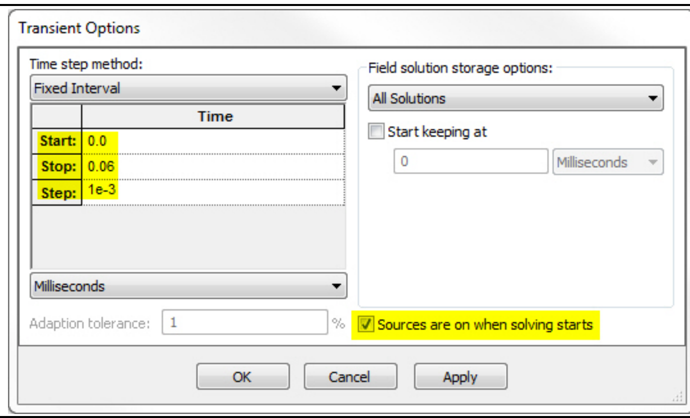
Reconfiguring the problem to perform a transient analysis

Set the start, stop and step times

The start, stop, and step times are defined in the Set Transient Options dialog.

Note The transient solver assumes the source values before the start time are equal to the values at the start time only if the parameter SourcesOnAtTransientStart is set to Yes.

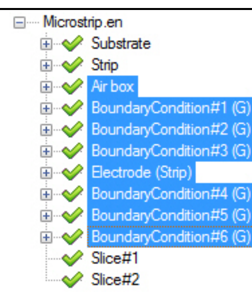
1. On the Solve menu, click *Set Transient Options*.
   The Set Transient Options dialog appears.



2. Make sure that *Fixed Interval* is selected as the Time Step method and *Milliseconds* as the unit for time, and then make the following modifications for Time:
   - Start = **0.0**
   - Stop = **0.06**
   - Step = **1e-3**
3. Click “Sources are on when solving starts” to set the SourcesOnAtTransientStart parameter to Yes, if required.
4. Click OK.

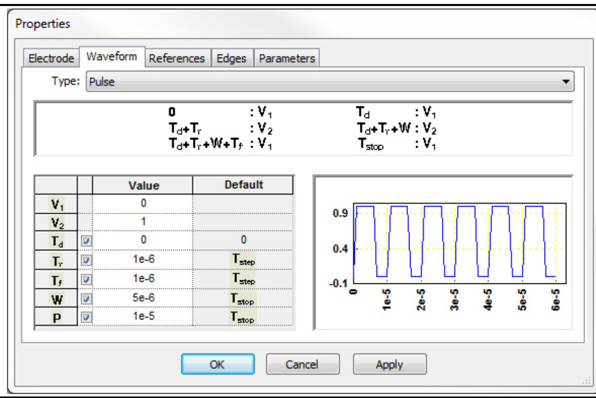
Defining a pulse waveform for the electrode

1. On the Object page, keeping the Shift key down, first select *Air box* and then select *BoundaryCondition#6 (G)*.



2. On the Edit menu, click *Toggle Visibility of Selected Components*.
   A ✓ symbol appears next to the selected items to signify that they are now visible.

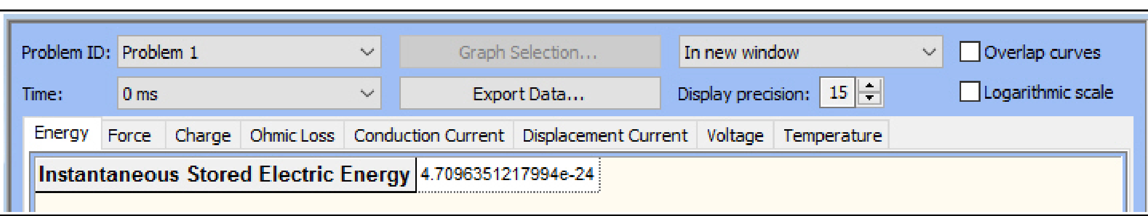
3. On the Project Bar, select the "Electrode" tab.
4. Place the cursor over *Electrode (Strip)* item and then right-click the mouse.
5. On the floating menu, click *Properties*.
   The Electrode properties dialog appears.
6. Select the "Waveform" tab.



7. In the Type drop-down list, select *Pulse*.
8. Click the **P** checkbox to enable *Period in seconds* and all preceding optional values.
9. Insert the following values in the appropriate box:
   - V_1 --> **0**
   - V_2 --> **1**
   - T_d --> **0**
   - T_r --> **1e-6**
   - T_f --> **1e-6**
   - W --> **5e-6**
   - P --> **1e-5**
10. Click OK.

Solve

- On the Solve menu, click *Transient 3D*.
  The ElecNet Transient 3D Solver Progress dialog appears for a minute or two and then the Results window opens. The Solver Progress dialog automatically exits when the solution is complete.



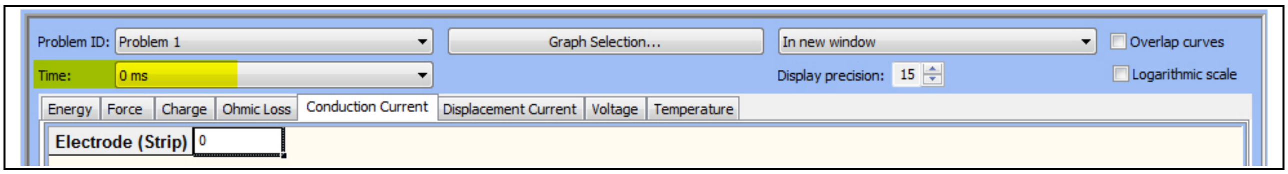
View the solution results

The following results will be reviewed in this section:

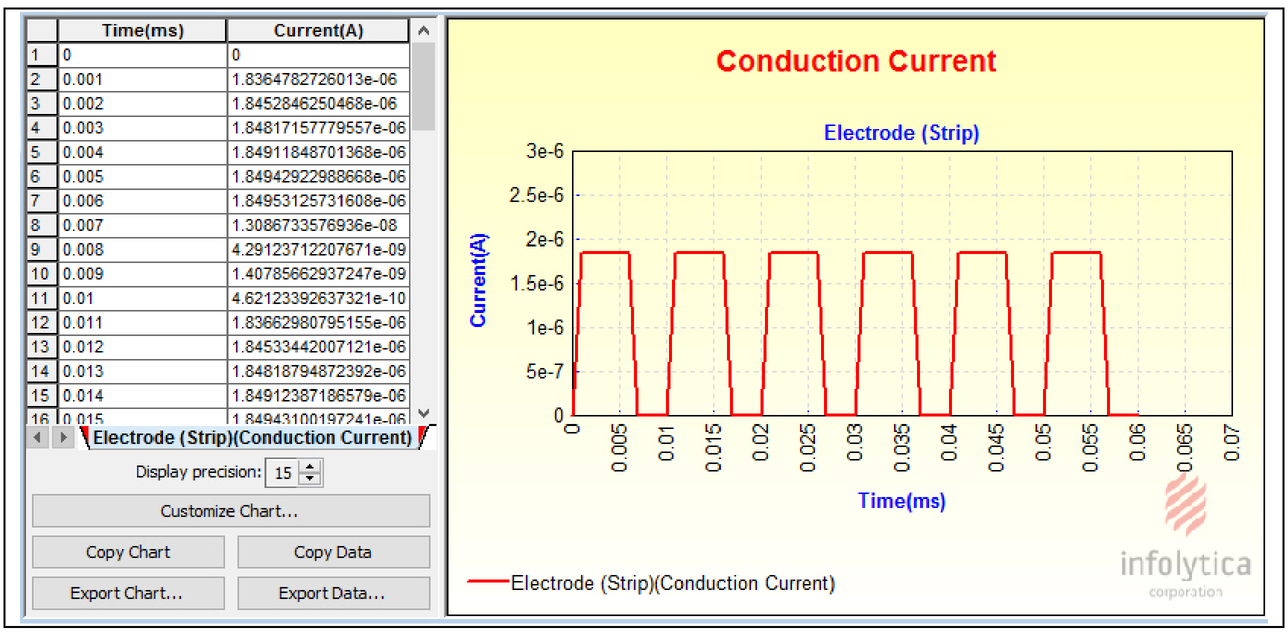
- A graph of the conduction current versus time
- A graph of the displacement current versus time

Graph the conduction and displacement current versus time

The Results window is automatically displayed when the Solver Progress dialog closes and the solution is complete. The Time Steps can be viewed by clicking the Time drop-down list box (highlighted in the image below).

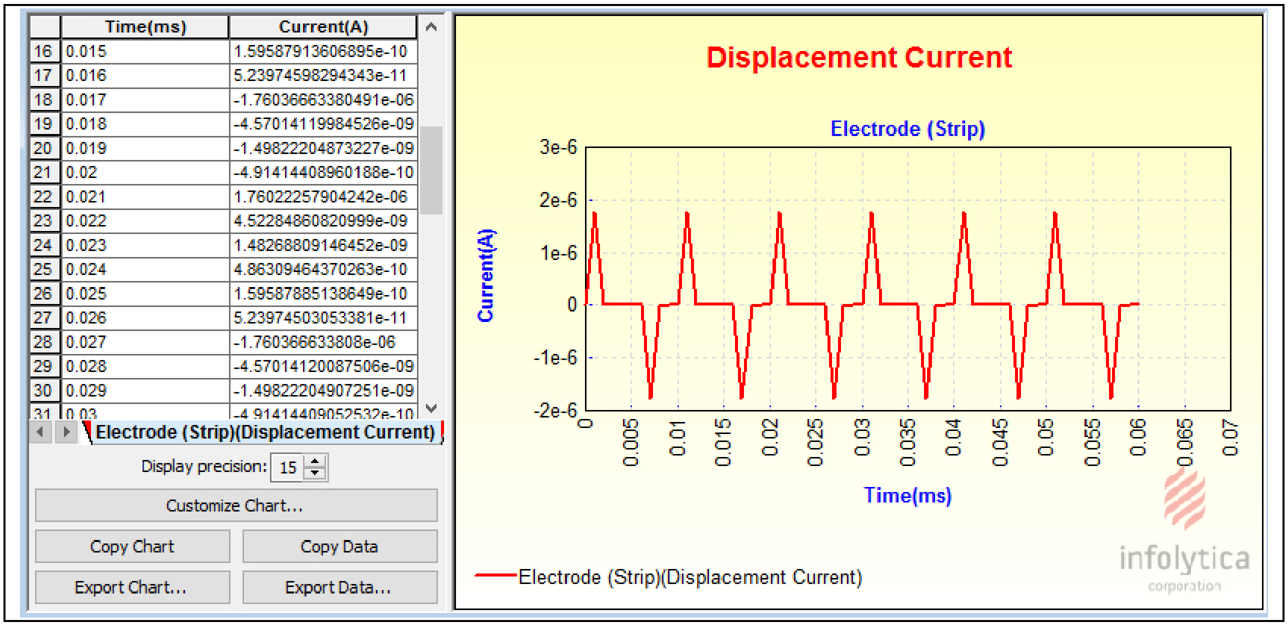


1. In the Results window, open the "Conduction Current" page.
2. Using the mouse pointer, click anywhere inside the *Electrode (Strip)* text box.
3. Click the *Graph Selection* button, located at the top of the Results window.
   The graph should look like the illustration below.



4. In the Results window, open the "Displacement Current" page.
5. Using the mouse pointer, click anywhere inside the *Electrode (Strip)* text box.

6. Click the *Graph Selection* button, located at the top of the Results window.
   The graph should look like the illustration below.



Note The total current can be obtained by adding the displacement current with the conduction current.

Save the model

You have now completed the 3D Transient portion of the tutorial.

1. On the File menu, click *Save*.
2. On the File menu, click *Close*.

Summary

In this tutorial, you completed the steps in creating a microstrip model for electrostatic and transient solutions. The skills you learned include:

- Setting up the work environment by modifying initial settings and the viewing area.
- Building the geometric model using the Keyboard Input Bar.
- Setting up the problem -- this consists of making components and electrodes, and assigning boundary conditions.
- Modifying the model so as to obtain the desired solution results.
- Generating the electrostatic and transient field solution using ElecNet's 3D Electrostatic and Transient solvers.
- Generating streamlines using Infolytica's Streamline Generator extension.
- Analyzing the results, which includes viewing the contours of V and the shaded plot of |E| smoothed, from two viewing perspectives (Z=0 and X=500).