



RFE3
**Three-dimensional RF Electric Fields in
Conductive Media**

Field Precision
Copyright 2001
Internet: www.fieldp.com
E Mail: techninfo@fieldp.com
PO Box 13595, Albuquerque, New Mexico 87192 U.S.A.
Telephone: 505-220-3975, FAX: 505-294-0222

RFE3

Three-dimensional RF Electric Fields in Conductive Media

Table of contents

Chapter 1. Introduction to RFE3

- 1.1. Function of the program
- 1.2. Test run

Chapter 2. Theoretical background

- 2.1. Electric fields in conductive media
- 2.2. Interpretation of boundary conditions

Chapter 3. Organizing AMaze simulations

Chapter 4. Script commands for run control

- 3.1. Script conventions
- 3.2. Program control commands

Chapter 5. Commands to set material properties

Chapter 6. Modeling small details in a large-scale solution

Chapter 7. Running RFE3 interactively

Chapter 8. Automatic runs under batch file control

- 8.1. Command line operation
- 8.2. Batch files
- 8.3. Introduction to **GCon**
- 8.4. Running **GCon**

Chapter 9. RFE3View - file menu

Chapter 10. RFE3View - plane plots

Chapter 11. RFE3View - slice plots

- 10.1. Setting the slice view
- 10.2. Setting slice plot properties
- 10.3. Analyses in a slice

Chapter 12. RFE3View - surface plots

Chapter 13. RFE3View - analysis functions and script operation

- 12.1. Point calculation - interpolation method
- 12.2. Line scans and matrix files
- 12.3. Automatic analyses
- 12.4. Script operation

Chapter 14. Structure of the RFE3 output file

Chapter 1. Introduction to RFE3

1.1 Program function

RFE3 is a versatile numerical tool to find RF electric fields in conductive media for arbitrary three-dimensional geometries. The program applies finite-element methods on a conformal hexahedron mesh for high accuracy. You can define up to 127 regions to represent electrodes or different materials. The amplitude and phase of the harmonic potentials on electrodes can be set individually. Conductive materials are characterized by values of the volume resistivity and relative dielectric constant.

RFE3View, an interactive graphics analysis program, provides full information on the amplitude and phase of electric fields and current density throughout the solution region. The program generates a variety of two-dimensional slice plots as well as three-dimensional plots of surfaces of constant potential, field amplitude or resistive power density. Two programs are supplied with the package:

- **RFE3.EXE** – determines one or more solutions in a window or under batch file control

- **RFE3View.EXE** – generates plots and analyses of solutions from **RFE3**.

MetaMesh, the **AMaze** conformal mesh generator, is required to create solutions. **RFE3** features fast and accurate calculations in random-access memory. The programs use dynamic memory allocation; therefore, the size of the solution is limited only by the installed RAM. A computer with 1 GB of memory can handle over 10 million elements.

The remainder of this chapter describes a step-by-step run that will test your setup and familiarize you with the programs. Chapter 2 covers the theory of RF electric field solutions in the non-radiative limit. Chapter 3 reviews the programs used for a complete solution and the organization of input and output files. Chapter 4 covers the structure of the **RFE3** control script. The script sets parameters for program operation and defines the material properties of regions. Chapters 5 and 6 review commands that may appear in the script. Chapter 7 covers features of the program when run as an interactive Windows utility. Chapter 8 describes how to run complex simulations automatically under batch file control. The chapter reviews features of the Field Precision utility program **GCon**. Chapters 9 through 13 describe the analysis and plotting capabilities of the **RFE3View** postprocessor. Finally Chapter 14 summarizes the format of the **RFE3** output file. This information is useful if you want to write your own analysis programs.

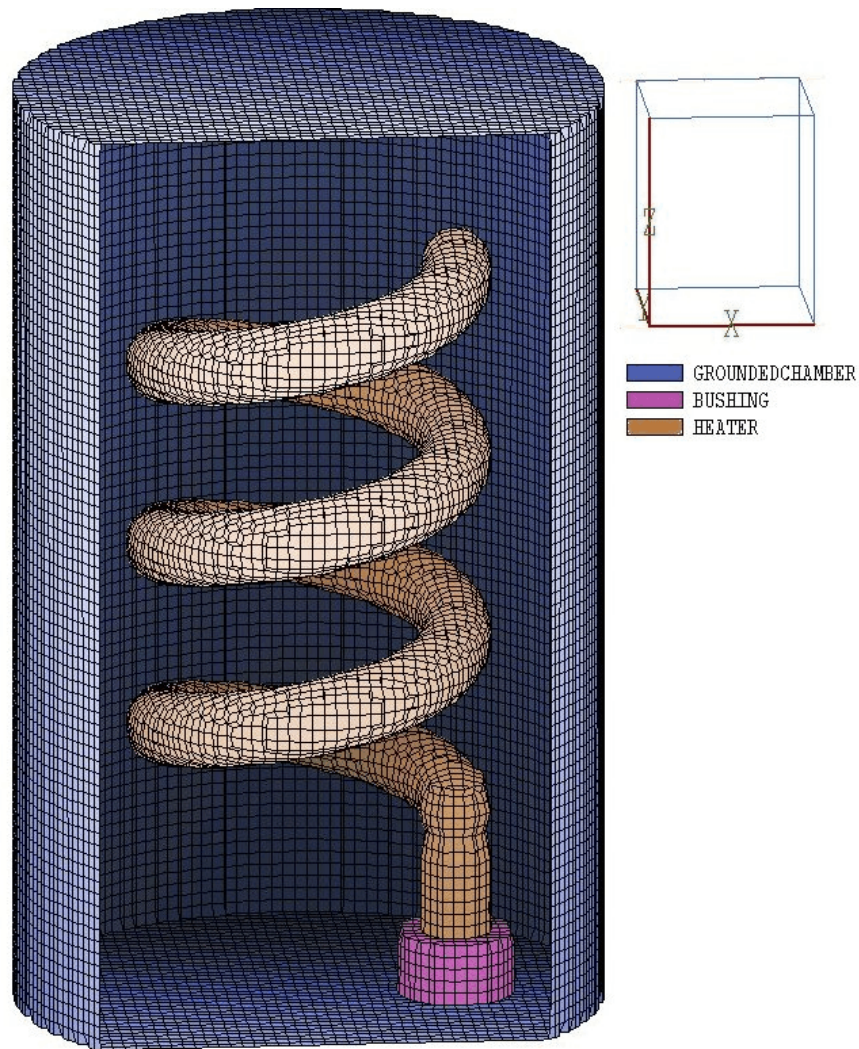


Figure 1.1. Geometry for the WALKTHROUGH example.

1.2. Test run

A quick way to understand the solution procedure is to step through an example. We want to find the capacitance and resistance between the helical electrode and the grounded chamber shown in Figure 1.1. The chamber is filled with purified water. This calculation would clearly be quite difficult with analytic methods. For reference, the conducting chamber has a radius of 5.0 cm and a height of 15.0 cm. To begin, transfer the examples files `WALKTHROUGH.MIN`, `WALKTHROUGH.RIN` and `WALKTHROUGH.SCR` to a working directory like `\AMAZE\BUFFER`. Run the program launcher `AMAZE.EXE` and make sure the program and data directories are set correctly (*i.e.*, to `\AMAZE` and `\AMAZE\BUFFER`).

Table 1.1. Contents of the file WALKTHROUGH.MIN

<pre>* File WALKTHROUGH.MIN * Units in cm GLOBAL XMesh -5.20 5.20 0.20 End YMesh -5.20 5.20 0.20 End ZMesh -0.20 15.20 0.20 End RegName(1) Chamber RegName(2) ConductiveMedium RegName(3) Bushing RegName(4) Heater END PART Region GroundedChamber Type Box Fab 10.40 10.40 15.40 Shift 0.00 0.00 7.50 END PART Region ConductiveMedium Type Cylinder Fab 5.00 15.00 Shift 0.00 0.00 7.50 Surface Region Chamber Coat Chamber Chamber END</pre>	<pre>PART Region Bushing Type Cylinder Fab 1.00 1.20 Shift 2.50 0.00 0.40 Surface Region Chamber Surface Region ConductiveMedium Coat Chamber Chamber END PART Region Heater Type Cylinder Fab 0.60 6.00 Shift 2.50 0.00 0.50 Surface Region Bushing Surface Region ConductiveMedium END PART Region Heater Type Helix Fab 2.50 0.60 9.00 3.00 Shift 0.00 0.00 3.00 Surface Region ConductiveMedium END ENDFILE</pre>
---	--

Run **MetaMesh** and click on the command *LOAD MIN FILE* in the *FILE* menu. Pick *WALKTHROUGH.MIN* in the dialog and click *OK*. You can view or edit the script with the *EDIT MIN FILE* command in the *FILE* menu. Table 1.1 shows the contents of the file. A relatively small number of commands is sufficient to define the complex geometry. Click on the *PROCESS MESH* command. **MetaMesh** sets up a foundation mesh, divides the volume into regions that represent physical entities, and then fits designated surfaces. The program requires about 5 seconds to create a mesh with 220,000 elements. Click on the *SAVE MESH* command in the *FILE* menu. The resulting file *WALKTHROUGH.MDF* will be used as input by **RFE3**. At this point you can experiment with some of the plotting capabilities of **MetaMesh** before proceeding to the next step. Figure 1.1 was created using commands in the *PLOT3D* menu.

Table 1.2. Contents of the file WALKTHROUGH.RIN

```
* File: WALKTHROUGH.RIN
Freq = 0.1
ResTarget = 1.5E-7
MaxCycle = 1500
DUnit = 100.0
NCheck = 25
Omega = 1.80
* Grounded chamber
Potential(1) = 0.0
* Purified water (0.1 MOhm-cm)
Epsi(2) = 81.0
Sigma(2) = 1.0E-3
* Busing
Epsi(3) = 2.71
Sigma(3) = 0.0
* Heater
Potential(4) = 1.0 (0.0)
ENDFILE
```

We shall perform two different calculations by making small changes to the **RFE3** control script `WALKTHROUGH.RIN`. Table 1.2 shows the file set up for a calculation of the resistance between the electrode and the chamber. We use low-frequency excitation so that the real current density is much higher than the displacement current density. The ratio of current densities is given by:

$$\frac{j_r}{j_d} = \frac{\sigma}{2\pi f \epsilon_r \epsilon_o} . \quad (1.1)$$

For $\sigma = 0.001$ S/m, $\epsilon_r = 81.0$ and $f = 0.1$ Hz, the ratio is $j_r/j_d = 2.220 \times 10^6$. From **AMaze** run **RFE3** and choose the command *START RUN* in the *RUN* menu. In the dialog pick the file `WALKTHROUGH.RIN` and click *OK*. The program reads the mesh file, analyzes parameters in the script, calculates coupling coefficients for the finite-element calculation, carries out a relaxation solution for the complex potential and then records values the data file `WALKTHROUGH.ROU`. The entire process takes less than 50 seconds.

Next run **RFE3View**. Before proceeding to plots, we shall check some numeric results. Click on *RUN SCRIPT* in the *FILE OPERATIONS* menu

Table 1.3. Contents of the file WALKTHROUGH.SCR

```
* File WALKTHROUGH.SCR
* NReg      RegName
* =====
*      1      GROUNDEDCHAMBER
*      2      CONDUCTIVEMEDIUM
*      3      BUSHING
*      4      HEATER
INPUT WALKTHROUGH.ROU
OUTPUT WALKTHROUGH
FULLANALYSIS
REGION 1
REGION 2
REGION 3
REGION 4
ENDFILE
```

and choose the file WALKTHROUGH . SCR. Table 1.3 shows the contents of the file. In response, the program automatically loads the data file WALKTHROUGH . ROU, opens a file WALKTHROUGH . DAT for output, performs calculations of volumetric quantities through the solution space and then analyzes volume and surface integrals for Regions 1 (surrounding chamber), 2 (conductive medium), 3 (bushing) and 4 (helical electrode). The program menu is inactive during the time (a few minutes) required for the calculations.

Choose *EDIT DATA FILE* in the *FILE OPERATIONS* menu and open WALKTHROUGH . DAT. The first section shows results from the global analysis. The time-average power dissipation is 6.26264×10^{-4} W for an applied voltage of amplitude $V_o = 1$ V. From the equation

$$\bar{P} = \frac{V_o^2}{2R}, \quad (1.2)$$

we infer a resistance of 798.4 Ω . The surface integrals in the region analysis of the conducting chamber (Region 1) give the following values for real and displacement current: $I_r = 1.273 \times 10^{-3}$ A and $I_d = 5.741 \times 10^{-10}$ A. The ratio of currents (2.222×10^6) agrees with the prediction of Eq. (1.1). The value of I_r implies a resistance of 785.5 Ω (a difference of 1.6% from the value determined from the volume integral). The integral over the surface of the helix gives a real current $I_r = 1.156 \times 10^{-3}$ A for an implied resistance of 865.1 Ω . The differences in the results illustrate

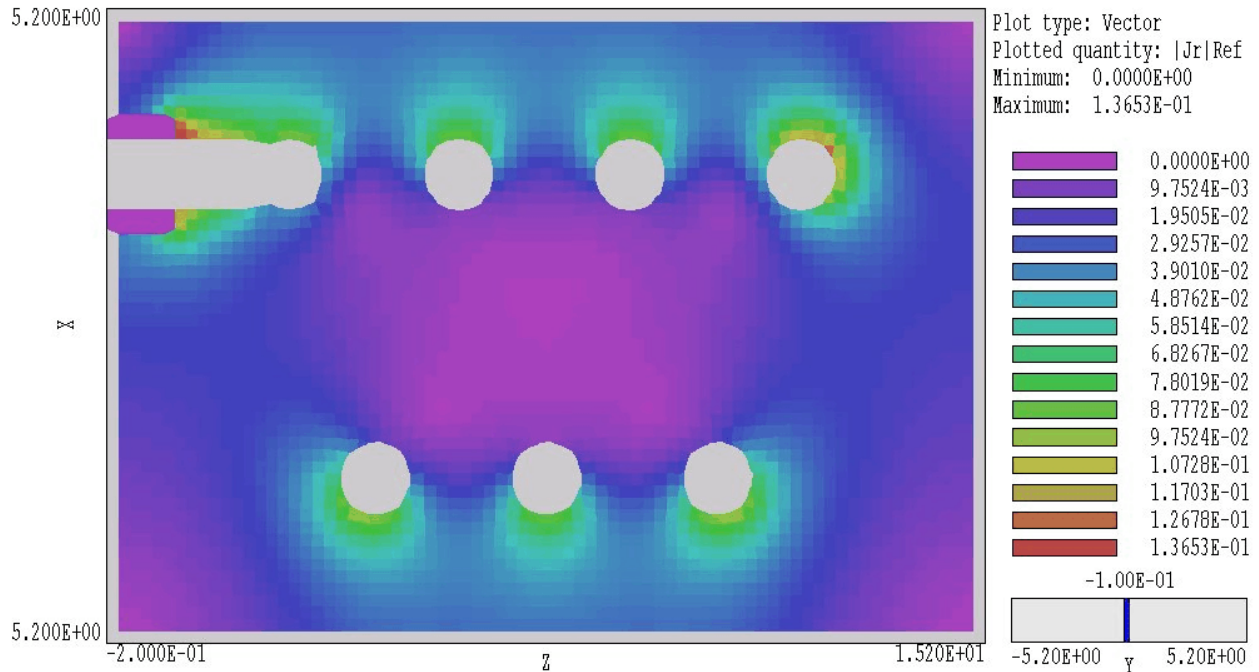


Figure 1.2. Element plot of the magnitude of real current at the reference phase of 0° in the plane $y = -0.1$)

accuracy limitations in numerical calculations. The volume integral involves an average over many elements and gives the highest accuracy. The surface integrals involve identification of and field interpolations over facets. The method achieves the best accuracy on flat or gently-curved surfaces (such as the chamber wall) and may encounter errors on complex sharply-curved surfaces (such as the helical electrode).

Close WALKTHROUGH.DAT and return to the main menu. Click on *SLICE PLOTS* and then choose *SLICE NORMAL TO Y* in the *CHANGE VIEW* menu. In the *PLOT CONTROL* menu, click on *SLICE PLOT STYLE* and choose *ELEMENT*. Finally, under *PLOT QUANTITIES* in the *PLOT CONTROL* menu pick |Jr|REF. You should see a plot similar to that of Fig. 1.2. The figure shows variations of the amplitude of real current density in the plane $y = -0.10$. The plot is created at a reference phase of 0.0° for an applied voltage of the form $V(t) = V_o \cos(2\pi ft)$. As expected, the current density concentrates on the outside of the helical electrode and the equals zero inside the bushing.

To complete the analysis, we shall make small changes in the input script for a calculation of the assembly capacitance. Return to **RFE3** (or rerun it from **AMaze** if you closed the program). Choose *EDIT SCRIPT FILE* in the *FILE* menu and load `WALKTHROUGH.RIN`. Change the line `FREQ = 0.1` to `FREQ = 1.0E6` and change the line `SIGMA(2) = 1.0E-3` to `SIGMA(2) = 0.0`. Save the file, exit the editor and rerun the solution. Return to **RFE3View** and rerun the script `WALKTHROUGH.SCR` to analyze the revised field solution. The global analysis gives the following values for field energy U : 4.49165×10^{-10} J (water) and 1.23185×10^{-12} J (bushing). The field energy density is given by $\epsilon_0 \epsilon_r (\mathbf{E} \cdot \mathbf{E}^*)/2$, where \mathbf{E} is the complex electric field vector. Using the formula $U = CV^2/2$, the predicted total capacitance is 9.008×10^{-10} F. The water accounts for 99.7% of the capacitance. The predicted amplitude of the displacement current at 1 MHz is $I_{do} = 2\pi fCV_o = 5.660 \times 10^{-3}$ A. For comparison, the surface integral of displacement current density over the chamber wall gives 5.754×10^{-3} A.

In this section we have seen a few of the capabilities of **RFE3** and **RFE3View**. At this point, you can experiment with plotting capabilities of the post-processor. Figure 1.3 illustrates some 3D capabilities. The plot shows a cutaway view of the chamber and electrode with a surface that follows the contour $|\phi| = 0.6$ at a reference phase of 0.0° . It is important to note that the demonstration addressed two special cases where conduction was dominated by either real or displacement current. The true power of RFE3 lies in its ability to handle intermediate cases where both real and displacement currents play significant roles.

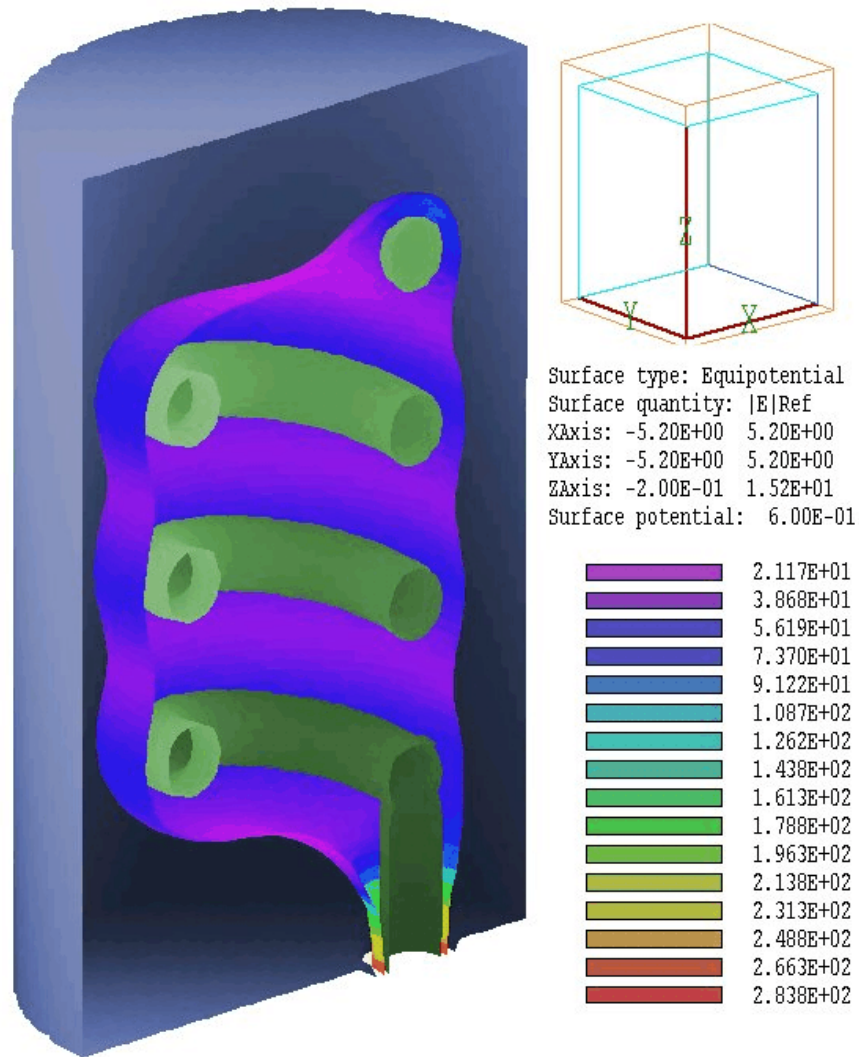


Figure 1.3. Three-dimensional cutaway view of the chamber and helical electrodes along with a calculated surface of $|\phi| = 0.6$ V. Color coding on the equipotential surface shows the magnitude of the electric field.

Chapter 2. Theoretical background

2.1. Electric fields in conductive media

Static electric fields can be written as the gradient of a scalar electrostatic potential:

$$\bar{E} = -\nabla\Phi. \quad (2.1)$$

Equation (2.1) also holds for AC electric fields at moderate frequency. In this case, time-varying magnetic fields make a small contribution to the electric field, so that

$$\nabla \times \bar{E} \approx 0. \quad (2.2)$$

Equation (2.2) is equivalent to Eq. (2.1). If the electric field varies as

$$\bar{E}(x,y,z,t) = \bar{E}(x,y,z)\cos\omega t = \bar{E}(x,y,z) \operatorname{Re}[\exp(i\omega t)], \quad (2.3)$$

then Eq. (2.1) is valid over scale lengths of order

$$L \ll \frac{1}{\omega\sqrt{\epsilon\mu_0}}. \quad (2.4)$$

In Eq. (2.4) ϵ is the dielectric constant of the material (in MKS units). As an example, taking $\epsilon = 10\epsilon_0$ and $L = 0.1$ m, the quasi-static approximation is valid for AC frequencies in the range $\omega \ll 9.5 \times 10^8$ Hz or $f \ll 151$ MHz.

For a system of ideal dielectrics (no conductivity), we can combine Eq. (2.1) with the divergence equation,

$$\nabla \cdot (\epsilon \bar{E}) = 0, \quad (2.5)$$

to derive the Laplace equation,

$$\nabla \cdot \epsilon \nabla \Phi = 0. \quad (2.6)$$

Equation (2.6) holds when there is no free space charge.

We can derive a similar equation for a conducting medium at zero frequency (no displacement currents). Because there is no time variation of the charge in the media, the divergence of current density equals zero,

$$\nabla \cdot \mathbf{j} = 0. \quad (2.7)$$

The current density is related to the electric field by

$$\mathbf{j} = \sigma \mathbf{E}, \quad (2.8)$$

where σ is the volume resistivity. Combining Eqs. (2.1), (2.7) and (2.8) gives an alternate form of the Laplace equation

$$\nabla \cdot [\sigma \nabla \Phi] = 0. \quad (2.9)$$

Note that Eq. (2.9) has the same form as Eq. (2.6) if we replace ϵ with σ .

We can consolidate the developments to describe low-frequency AC electric fields in imperfect dielectrics. Here the distribution of charge in a medium is determined by both real and displacement currents. We take a low angular frequency that satisfies Eq. (2.4) and assume that the time dependence of the electric field and all other time-varying quantities has the form

$$\mathbf{E}(x,y,z,t) = \mathbf{E}_0(x,y,z) \exp(i\omega t). \quad (2.10)$$

In Eq. (2.10) the quantity $\mathbf{E}_0(x,y,z)$ on the right hand side is a vector of three complex numbers that gives the local amplitude and phase of the field. Similarly $\Phi(x,y,z)$ is a complex number with information on the amplitude and phase of the potential. Take \mathbf{j}_r as the real current density in the conductive media and ρ_r as the space-charge density resulting from

this current. Conservation of charge implies that

$$\frac{\partial \rho_r}{\partial t} = -\nabla \cdot \mathbf{j}_r = i\omega \rho_r. \quad (2.11)$$

The divergence equation assumes the form

$$\nabla \cdot \epsilon \mathbf{E} = \rho_r = i \frac{\nabla \cdot \mathbf{j}_r}{\omega} = \nabla \cdot \left[\frac{i\sigma \mathbf{E}}{\omega} \right]. \quad (2.12)$$

The expression on the right-hand side proceeds from the relation $\mathbf{j}_r = \mathbf{E}/\rho$. Equation (2.12) can be written as

$$\nabla \cdot \left(\epsilon - \frac{i}{\rho\omega} \right) \mathbf{E} = 0, \quad (2.13)$$

or

$$\nabla \cdot \left[\left(\epsilon - \frac{i\sigma}{\omega} \right) \nabla \Phi \right] = 0. \quad (2.14)$$

Equation (2.14) is identical to Eq. (2.6) if we interpret the dielectric constant as a complex number,

$$\epsilon \rightarrow \left(\epsilon - \frac{i\sigma}{\omega} \right). \quad (2.15)$$

Equation (2.14) can be solved with standard solution methods by simply replacing the potential and dielectric constant with the appropriate complex numbers. This is the approach used in **RFE3**.

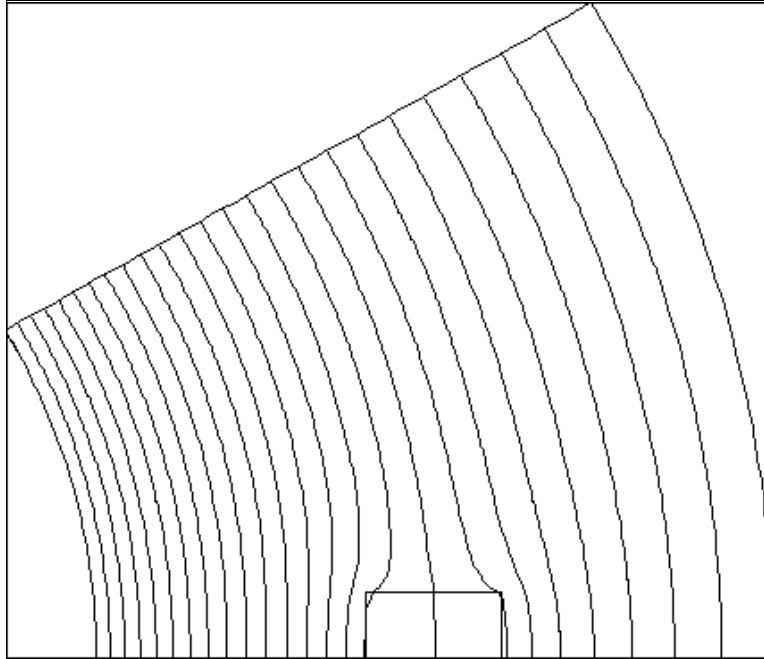


Figure 2.1. Special Neumann boundaries

5.2. Boundary conditions

The *boundary* is the outer edge of the solution volume. For RF electric field problems, node potentials on the boundary can be set to one of two conditions.

Dirichlet

Dirichlet boundary points have a fixed value of the complex potential that does not change as the **RFE3** relaxation proceeds. A region of uniform-potential vertices represents an electrode (equipotential volume). Electric field lines are normal to such a surface.

Neumann

A Neumann boundary is one where the normal derivative of the complex potential is specified. The boundaries in **RFE3** are limited to the special case where $\partial\phi/\partial n = 0$. The special Neumann condition implies that the electric field is parallel to the boundary. One of the advantages of the finite-element method is that all boundaries that are not fixed automatically satisfy the special Neumann condition, even if they are slanted or curved. Neumann boundaries are often used to reduce computation time for symmetric systems. For example, Fig. 2.1

shows an equipotential plot for a solution of the fields between two spherical electrodes with a dielectric inclusion. Accurate results can be obtained simulating only a portion of the inter-electrode volume. Symmetry applies on the bottom boundary and the top boundary is far enough from the object so that the field is almost purely radial. In conductive solutions, a special Neumann condition represents the surface of an ideal insulator (*i.e.*, current flow parallel to the surface).

Chapter 3. Organizing AMaze simulations

The **RFE3** package consists of the program **RFE3** that computes the physical solution and **RFE3View**, a dedicated post-processor for analyses of the results. The solution program can run in two modes: interactively in a window or autonomously in the background under batch file control. Similarly, **RFE3View** can run as an interactive application under user control or autonomously under script file control. The autonomous modes allow automatic processing of large or repetitive data sets.

Two input files are required for a **RFE3** solution:

- A **MetaMesh** output file that describes the conformal hexahedron mesh. The file contains node coordinates and the region numbers of elements and nodes.
- A script file that sets control parameters for the solution program and describes the physical properties associated with regions.

The **MetaMesh** output file always has a name of the form `FPREFIX.MDF`, where `FPREFIX` is a valid file prefix (1 to 20 characters). The script must have a name of the form `FPREFIX.RIN`, where `FPREFIX` is the same prefix as that of the mesh file and `RIN` stands for **RFE3 INput**. **RFE3** issues an error message if both input files are not available in the current working directory. To organize data, the resulting output file name begins with `FPREFIX`.

For review, a simulation generally consists of the following steps:

- Prepare a **MetaMesh** script (`FPREFIX.MIN`) that defines the solution space and describes the division into spatial regions.
- Run **MetaMesh** to create a file `FPREFIX.MDF` of standard mesh information. This file could be used as input for multiple **RFE3** simulations or for other **AMaze** programs.
- Prepare an **RFE3** script (`FPREFIX.RIN`) that sets control parameters and defines the material properties of regions.

- Run **RFE3** to create a solution file `FPREFIX.ROU`. This file contains information on node spatial coordinates and the complex electrostatic potential ϕ .
- (*Optional*) Prepare a standard **AMaze** analysis script (`ANYNAME.SCR`) to control a **RFE3View** analysis session.
- Run **RFE3View** to create plots or to generate numerical data using the information in `FPREFIX.ROU`. The analysis can be performed interactively or automatically under the control of a script.

Table 3.1 summarizes the input and output file used in **RFE3**.

Table 3.1. RFE3 files		
Name form	Function	Status
<code>FPREFIX.MIN</code>	Description of simulation geometry, input to MetaMesh	<i>Required</i>
<code>FPREFIX.MDF</code>	Output from MetaMesh , input to RFE3	<i>Required</i>
<code>FPREFIX.RIN</code>	Description of material properties, input to RFE3	<i>Required</i>
<code>FPREFIX.ROU</code>	Output from RFE3 , input to RFE3View	<i>Required</i>
<code>ANYPREFIX.SCR</code>	Analysis control, input to RFE3View	<i>Optional</i>
<code>ANYPREFIX.DAT</code>	Analysis data output from RFE3View	<i>Optional</i>

Chapter 4. Script commands for run control

4.1. Script conventions

The input script for all **AMaze** solution programs is a text file (ASCII format) with data lines containing commands and parameters. Text editing is a critical activity working with Field Precision software, both for preparing input and for inspecting quantitative data output. The process can be frustrating with a rudimentary editor like Notepad. Here are some options:

- The solution programs and post-processors have a built-in editor with many useful features (*i.e.*, *Copy*, *Paste*, ...). The main limitation is that you can work on only one file at a time.
- We supply **SynEdit**, a good freeware program with multi-window capabilities.
- **TextPad** is an inexpensive and powerful shareware editor for programing and data analysis (<http://www.textpad.com>).

The script file must end with the *EndFile* command. The programs make no distinction between upper and lower case. Entries on a line can be separated by the following delimiters

```
Space, blank
Comma [,]
Tab
Colon [:]
Equal sign [=]
Left parenthesis [(]
Right parenthesis [)]
```

You can use any number of delimiters in a line. This means that you can add indentations and customize the appearance of the script. For example, the two lines

```
Epsi 2 5.56
Epsi(2) = 5.56
```

have the same effect.

RFE3 ignores blank lines and comment lines. Comment lines begin with the symbol [*] (asterisk). Most parameters are real numbers. The following formats are valid.

```
1.000
5.67E6
6.8845E+09
5
```

The final number is interpreted as 5.0.

RFE3 accepts commands in any order. The program reads and analyzes all commands before starting the solution. Generally, it is good practice to put control commands at the beginning and to group commands that set material properties by region. The following example illustrates a complete script for **RFE3**:

```
* File: CLAMP.MIN
DUnit = 1000.0
Freq = 5.0E6
ResTarget = 1.0E-7
MaxCycle = 200
Omega = 1.90
NCheck = 10

Epsi(1) = 1.0
Sigma(1) = 0.0
Epsi(2) = 500.0
Sigma(2) = 5.0
Epsi(3) = 800.0
Sigma(3) = 1.0
Potential(4) = 1.0 0.0
Potential(5) = 1.0 180.0
Potential(6) = 0.0 0.0

EndFile
```

You can place any amount of text in any format after the *EndFile* command. Therefore you can add documentation that can be helpful when you return to a simulation after a long period of time.

4.2. Program control commands

This section review commands control program operation. Each command is shown in a form that it might assume in a program.

FREQ 1.0E5

All electrode potentials in an **RFE3** solution have the same harmonic frequency f although they may have different phases. Enter the value of frequency in Hz. This is a required parameter – the program stops with an error message if a value is not supplied. For DC solutions, enter a small value for f rather than 0.0.

RESTARTGET 5.0E-6

The numerical calculation of RF electric fields requires the solution of a large set of coupled linear equations, one for each active node in the solution volume. **RFE3** uses an iterative technique based on corrections that reduce the error in ϕ at a point compared to predictions from values at neighboring nodes. The *residual* is an average of the relative errors over all nodes in the solution space during an iteration. This command sets a target value for the residual. The program stops if the error drops below the value. For good accuracy, the relative residual should be less than 10^{-6} . If the value of *ResTarget* is too low, the program may not converge because of roundoff errors. In this case, you can terminate a solution manually if you running **RFE3** in the interactive mode by using the *Stop and save run* or *Abandon run* menu options. Default value: *ResTarget* = 5.0×10^{-7} .

MAXCYCLE 2500

The integer parameter *MaxCycle* is the maximum number of iteration cycles. **RFE3** saves the solution and stops when it reaches *MaxCycle* even if *ResTarget* has not been attained. Default value: *MaxCycle* = 2500.

NCHECK = 20

The operations to calculate *ResTarget* take time; therefore, **RFE3** usually does not make a check on every iteration cycle. The integer parameter is the number of relaxation cycles between convergence checks. Default value: *NCheck* = 50.

OMEGA 1.92

The real-number parameter *Omega* is the over-relaxation factor used to correct potential errors during the solution. This quantity may have a significant effect on the solution time. If the *Set omega* command does not appear, **RFE3** automatically picks values that vary with the iteration cycle following the Chebyshev acceleration prescription. With this command, you can set a value manually. For numerical stability the value must be between 0.0 and 2.0. Higher values generally give faster convergence.

DUNIT 100.0

Internally **RFE3** uses SI units, with lengths in meters (m), potential in volts (V), electric field in volts/meter (V/m), and conductivity in siemens per meter (S/m). Use the *Set DUnit* command to work in alternate length units. The real-number quantity is the number of working units per meter. For example, if you used dimensions of centimeters in the **MetaMesh** script, set $DUnit = 100.0$. Alternatively, if the mesh dimensions are in inches, use $DUnit = 39.37$. The quantity *DUnit* is recorded in the output file and is used in **RFE3View** for the input and output of positions. Default: $DUnit = 1.0$.

FORMAT ASCII

By default, **RFE3** creates output files `FPREFIX.ROU` in binary format. This format occupies less disk space. Binary output files are required if you want to use **RFE3View** to analyze the solution. We have included an option to create output files in ASCII format to make it easier to port results to your own analysis programs. The string parameter can assume the values *BINARY* or *ASCII*. Chapter 14 describes the file structure for both formats.

NGAUSS 3

The parameter *NGauss* controls the number of intervals in Gaussian integrations to determine element matrices. Under normal circumstances, it should not be necessary to make any change from the default value of $NGauss = 4$. Lower values decrease the program setup time but may reduce accuracy.

Chapter 5. Commands to set material properties

Material properties in RF electric field solutions are relatively simple so only a few commands are required. Both symbolic and sample forms are given for the commands.

POTENTIAL RegNo Amp [Phase]

POTENTIAL(5) = 2500.0 (90.0)

This command sets all elements and nodes associated with a region to a fixed potential condition. The integer parameter is a region number defined in **MetaMesh**. The first real number is the value for the amplitude of the potential in volts (V). The optional second real number is the phase in degrees. RFE3 converts the number to radians for internal use. The resulting applied voltage has the form:

$$V(t) = Amp \times \cos(2\pi ft + Phase).$$

The fixed-potential condition can be applied to both filled and open regions.

EPSI RegNo EpsiRel

EPSI(4) = 12.6

This command sets a relative dielectric constant for all elements associated with a filled region. The integer parameter is the number of a region defined in **MetaMesh**. The second parameter is the value of relative dielectric constant ϵ/ϵ_0 (dimensionless). To model a floating electrode (conducting object whose potential is not previously known), assign a relatively large value of dielectric constant (*i.e.*, 10^4) to the region. Note that extremely high values of ϵ_r (*i.e.*, 10^8) may slow convergence. The default for all unspecified regions is $\epsilon/\epsilon_0 = 1.0$.

SIGMA RegNo ChgDens

SIGMA(5) = 0.682

This command assigns a conductivity to elements within the specified filled region. The real number parameter is the value of σ in siemens/m. To model a perfect insulator immersed in a conducting medium, set $\sigma = 0.0$. To model a floating electrode, assign a relatively high value of conductivity. The default for all unspecified regions is $\sigma = 0.0$ S/m. Both the *EPSI* and *SIGMA* commands may be used to define an imperfect dielectric.

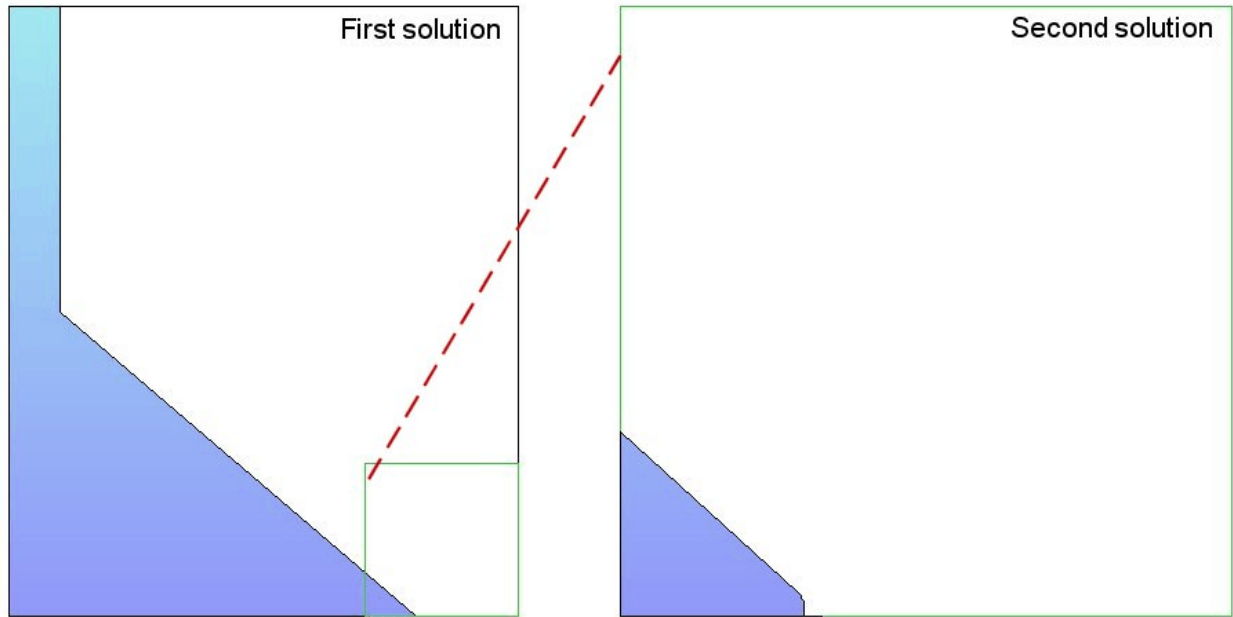


Figure 6.1. Creating a microscopic solution using the *BOUNDARY* command

Chapter 6. Modeling small details in a large-scale solution

RFE3 incorporates a powerful feature that enables accurate calculations of fields near small features in a large solution space. Suppose we wanted a precision calculation of the fields on a field emission tip. The radius of the tip is much smaller than the scale size of the electrodes that create the field and hold the tip. We need to do a large-scale calculation to find the macroscopic fields (Solution 1), but we would need very small elements near the tip to resolve the curvature. One approach is to use variable mesh resolution (**MetaMesh** manual, Section 4.2) to create small elements near the tip. A limitation to this technique arises from the structured mesh used in the **AMaze** programs – the region of small elements must extend the full length of the solution volume.

Figure 6.1 illustrates an alternate approach. We create a second solution that extends over a small subregion of the original solution (green outline). The microscopic solution contains any electrodes that are located within the subregion. The difference is that the electrode shapes may be resolved in much greater detail. The question is how to incorporate the macroscopic fields correctly into the microscopic solution. One resolution is to enclose the second solution in a variable-potential Dirichlet boundary. Values of ϕ

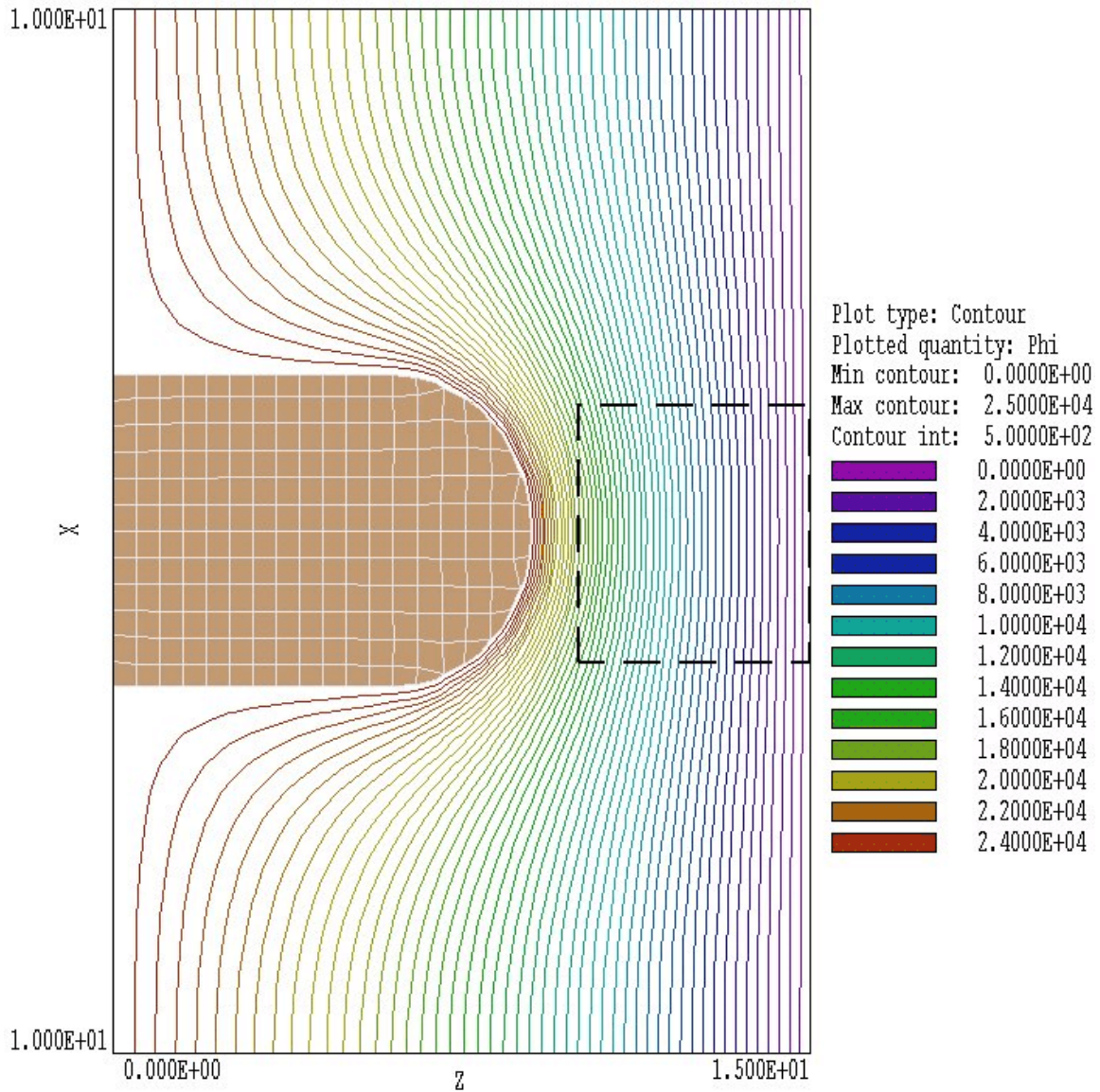


Figure 6.2. Equipotential lines and element resolution in the large-scale solution *BOUNDTEST01*. The dashed line shows the boundary of the small-scale solution.

on the boundary are calculated by interpolation at the corresponding point in the macroscopic solution (dashed red line). The total solution will be approximately correct as long as new features (such as the rounded tip) are well-removed from the variable-potential boundary.

Definition of the variable Dirichlet boundary is controlled by the following command that appears in the control script of Solution 2:

BOUNDARY FPrefix [BndScale]

BOUNDARY FEmitMacro

The quantity *FPrefix* (a string) is the prefix of the output file for Solution 1. The quantity *BndScale* (a real number) is an optional scaling factor applied to potential values transferred from Solution 1 to Solution 2. The default is *BndScale* = 1.0.

In response to the *BOUNDARY* command, **RFE3** loads Solution 1 into memory and sets up the apparatus for making second-order interpolations of complex potential in the solution. Then the program loops through all nodes of Solution 2. If the node is on the boundary of the solution box, **RFE3** marks it as a fixed-potential point and sets the potential equal to the interpolated value calculated at the position in Solution 1. **RFE3** issues an error message under the following conditions: 1) the output file for Solution 1 (*FPrefix.ROU*) is not available in the working directory, 2) Solution 2 does not fit completely inside Solution 1, or 3) an interpolation failed. The program makes no further validity checks. You must ensure that the geometry of Solution 2 represents a correct microscopic section of Solution 1.

Files included in the example library illustrate the process. The files *BOUNDTEST01.MIN* and *BOUNDTEST02.RIN* describes a large-scale solution, while *BOUNDTEST02.MIN* and *BOUNDTEST02.RIN* describe a microscopic solution, a subset of the first. Figure 6.2 shows the large-scale solution which covers a volume extending from -10.0" to 10.0" in *x* and *y* and from 0.0" to 15.0" in *z*. A shaped electrode creates a field on a flat grounded plate. We want to find precise fields near a small protrusion on the plate. The protrusion is smaller than the scale size of elements in the large-scale mesh (0.5"). The microscopic solution covers a portion of the original solution volume extending from -2.5" to 2.5 in *x* and *y* and from 10.0" to 15.0" in *z*. (dashed line in Fig. 5.2). The element scale size is 0.25" in the bulk of the solution and 0.05" in the region of the protrusion. The control script *BOUNDARYTEST02.RIN* has the following content:

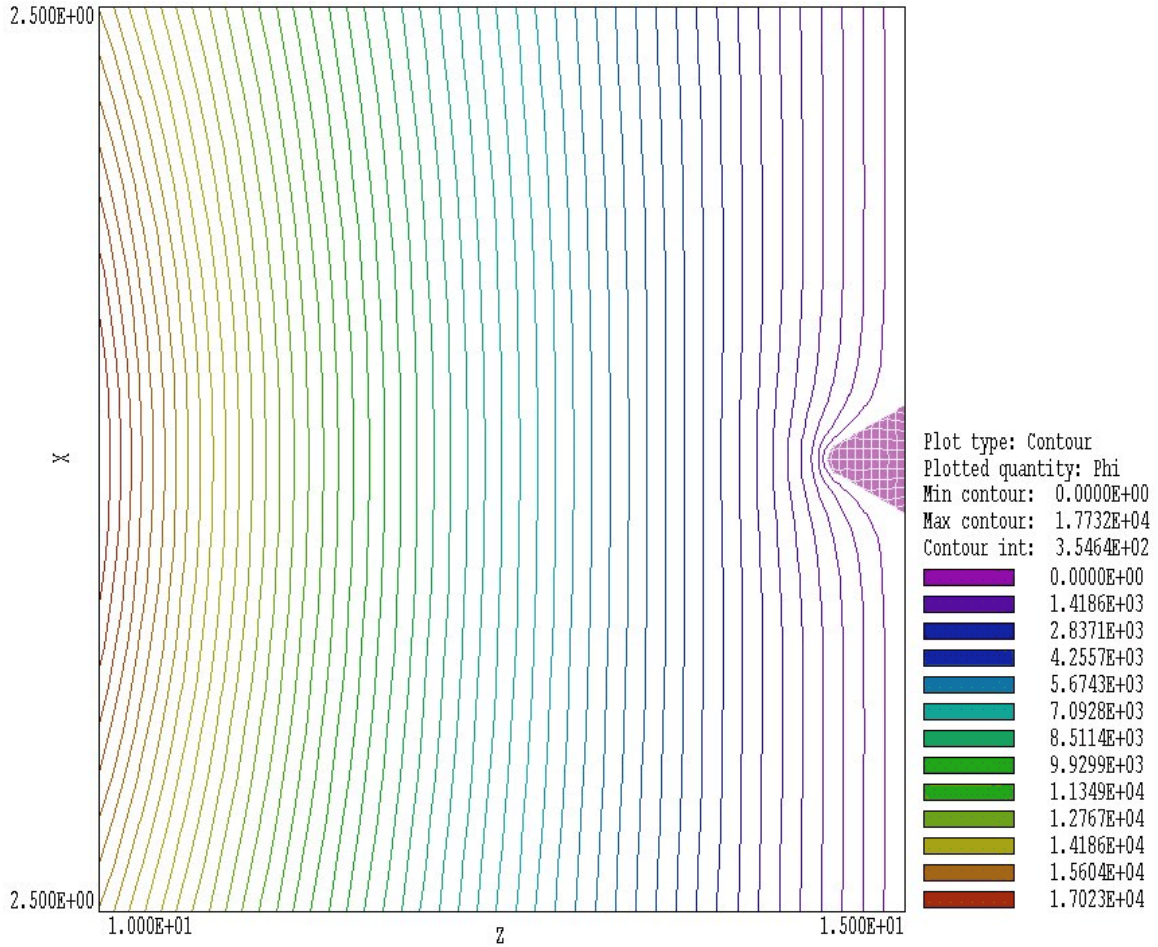


Figure 6.3. Equipotential lines and element resolution in the small-scale solution *BOUNDTEST02*.

```

* File BOUNDARYTEST02.RIN
*   NReg      RegName
*   =====
*     1       VACUUM
*     2       PROTRUSION
RESTARTET = 5.0E-8
MAXCYCLE = 2500
DUNIT = 39.37
NCHECK = 10
BOUNDARY BOUNDARYTEST01
EPSI(1) = 1.0
POTENTIAL(2) = 0.0
ENDFILE

```

Figure 6.3 shows equipotential lines and the element division of the protrusion in the microscopic electrostatic solution, a subset of the original. The surface of the protrusion is well-resolved and electric field interpolations in the vicinity will be accurate. The process is valid because the presence of the small protrusion does not significantly influence values of potential at the boundary of Solution 2.

In response to the *Superposition* command, **RFE3** superimposes values from a large-scale solution (Solution 1) on a small-scale solution (Solution 2). The following statement may appear in the command script for Solution 2.

**SUPERPOSITION FPrefix [SuperScale]
SUPERPOSITION UniField**

The quantity *FPrefix* (a string) is the prefix of the output file for Solution 1. The quantity *BndScale* (a real number) is a scaling factor applied to potential values transferred from Solution 1 to Solution 2. The default is *BndScale* = 1.0.

When this command is issued, **RFE3** opens the file *FPrefix.ROU* after Solution 2 has been completed. The program performs an interpolation in the space of Solution 1 to determine the potential ϕ_1 at each node location in Solution 2 and adjusts potential values according to

$$\phi_2' = \phi_2 + SuperScale \times \phi_1 , \quad (1)$$

before writing the output file.

You must ensure that the superposition is physically correct. In electrostatic solutions the presence of electrodes and dielectrics in Solution 2 can make significant local changes to the macroscopic field of Solution 1, so that a simple superposition would be invalid. Generally, the *Superposition* command is more useful for magnetostatic solutions.

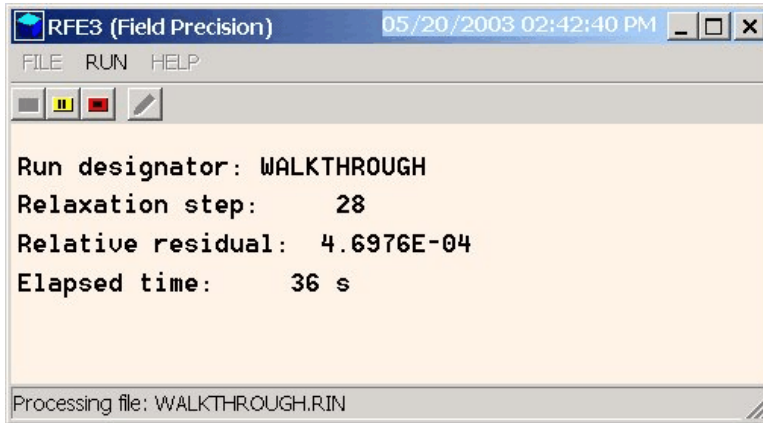


Figure 7.1. RFE3 display during a solution.

Chapter 7. Running RFE3 interactively

The program `RFE3.EXE` can be run interactively in a window. In this mode you can perform several solutions in a session and temporarily leave the program to work on other tasks. You can launch the program from **AMaze** or create your own shortcuts. Figure 7.1 shows the program window.

The program has three popup menus: *File*, *Run* and *Help*. The following commands appear in the *File* menu.

Edit script file

Edit listing file

Edit file

The commands call up the internal editor to inspect or to modify ASCII input and output files for the solution program. With the *Edit script file* command you can work on files with names of the form `FPREFIX.RIN`. With the *Edit listing file* command you can pick files with names of the form `FPREFIX.RLS`. The *Edit file* command shows all available files. Choosing a file from an alternate directory does not change the working directory of the program.

The *Run* menu has three commands.

Start run

Pick an input file with a name of the form `FPREFIX.RIN` to start a solution. The working directory changes if you pick a file from an alternate directory. The run begins if the file `FPREFIX.MDF` is available in the working directory. **RFE3** displays timer bars to show the status when the program is loading a mesh file, calculating element matrices and generating coupling coefficients of the linear equation set. When these tasks are completed, the program begins the relaxation solution and reports progress in the text window (Fig. 7.1).

Pause run

The intensive calculations of **RFE3** demand the full resources of your computer, causing other tasks to run slowly. If you need to perform critical work, you can pause the solution program during the relaxation process and restart it later without loss of data.

Stop run

This command terminates the program during the relaxation process and saves output data. For example, you may want to stop a run at a moderate value of convergence to check whether the problem has been correctly defined.

The *Help* menu has a single command.

Instructions

Displays the HTML help file using your default browser. In order for this command to operate correctly, the file `RFE3.HTML` must be in the same directory as `RFE3.EXE`.

Chapter 8. Automatic runs under batch file control

8.1 Command line operation

Batch file control is a useful option for running large technical programs like **RFE3**. You can prepare scripts to organize complex operations. The sequenced programs run automatically in the background or overnight.

To make a single **RFE3** simulation in the background, go to the Command Prompt from Windows and log to the data directory that contains the required MDF and RIN files.. For example, suppose the data files SWITCH.MDF and SWITCH.RIN are stored in \AMAZE\BUFFER and that the program RFE3.EXE is in the directory \AMAZE. From \AMAZE\BUFFER type

```
..\RFE3 SWITCH <Enter>
```

The program runs silently, writing detailed information in the listing file SWITCH.RLS. If the solution is successful, the program writes the output file SWITCH.ROU in the data directory. During lengthy runs you can perform other tasks in Windows. Note that the response of the computer will be considerably slower because **AMaze** programs seek to use the full power of the CPU.

8.2. Batch files

The main function of the command mode is autonomous operation under batch file control. As an example, assume you have prepared the input files SWITCH01.MIN, . . . , SWITCH08.MIN and SWITCH01.RIN, . . . , SWITCH08.RIN in the directory \AMAZE\BUFFER.. Next you create the following batch file SW_RUN.BAT in the data directory using an ASCII text editor.

```
@ECHO OFF
ECHO Main switch data run
START ..\METAMESH.EXE SWITCH01
START ..\RFE3.EXE SWITCH01
START ..\METAMESH.EXE SWITCH02
START ..\RFE3.EXE SWITCH02
. . .
START ..\METAMESH.EXE SWITCH08
START ..\RFE3.EXE SWITCH08
```

Type

```
SW_RUN <Enter>
```

to generate all solutions without the need for further keyboard input.

8.3. Introduction to GCon

Microsoft has released over thirty versions of its 32-bit operating system since Windows 95. There is considerable inconsistency in DOS emulation between versions. For example, Windows 2000 runs a set of programs in the proper sequential order for any valid batch format. Windows 95 starts all programs simultaneously unless you use the command form:

```
START /WAIT ProgName
```

Windows ME does not recognize the `START` command. And so on.... To ensure consistent batch file operation we supply the utility program **GCon** with all our software. To avoid problems we advise running batch scripts from **GCon** rather than from the Command Prompt.

GCon handles the following subset of DOS commands. Additional capabilities are described in the file `GCON.HTML`.

CD

Function: Displays the name of or changes the current directory.

Example: `CD C:\AMaze\Buffer`

COPY ... TO ...

Function: Copies a file to another location.

Example: `COPY Diode.min TO C:\AMaze\Diode.min`

ERASE

Function: Deletes a file.

Example: `ERASE diode.mls`

REM

Function: Displays the line, no action taken

MOVE ... TO ...

Function: Moves a file from one directory to another directory.

Example: MOVE Diode.mou TO C:\AMaze\Diode.mou

RENAME

Function: Renames a file or files.

Example: RENAME diode.mou diode_compare.mou

RUN

Function: Runs a specified program or command.

Example: RUN C:\tricom\mesh diode

As an example, the following script creates a mesh and an electrostatic solution using the input files DIODE.MIN and DIODE.RIN in the directory C:\EStudies\Inputs and stores the results in the directory C:\EStudies\Outputs

```
REM Sample GCon script
CD C:\EStudies\Inputs
RUN C:\AMAZE\METAMESH Diode
RUN C:\AMAZE\RFE3 Diode
MOVE Diode.MDF TO C:\EStudies\Outputs\Diode.MDF
```

8.4. Running GCon

GCon is simple to run. Start the program and click on the command *Run GCon script*. The dialog shows all files with the suffix GCN. The working directory is changed if you change directories in the dialog. Pick a batch file and click *OK*. The program executes the commands in the file in sequence and displays the current status.

The *Stop on error* command toggles a switch that determines whether the program will stop the batch process if there is an error in one of the batch commands. The command *Instructions* in the *Help* menu displays a help file in your default browser. The function requires that the file FPBATCH.HTML is in the same directory as FPBATCH.EXE.

Chapter 9. RFE3View - file operations menu

The **RFE3View** post-processor is designed for the analyses of the RF electric field simulations. The program has the following popup menus: *File operations*, *Analysis*, *Plane plots*, *Slice plots*, *Surface plots* and *Help*. When the program starts, only the *File* and *Help* menus are active. You must load a data file in order to create plots or to perform analyses. This section reviews options in the *File operations* menu.

Load solution file

The *Load solution file* command displays a dialog with a list of solution files with names of the form `FPREFIX.ROU`. Moving to a new directory in the dialog changes the program working directory. Pick an available file and click *OK*. The program loads the solution and updates the status bar. If data retrieval is successful, the analysis and plot menus become active.

Solution file information

The command displays information on the currently-loaded data file

Run script

Sometimes you may want to perform complex or repetitive analyses on a set of similar solutions. Script file operation is a powerful feature of **RFE3View**. This command displays a dialog with a list of analysis files that you have created with the suffix `SCR`. Pick a file and click *OK*. The script file can load data files, open and close history files, and perform any of the quantitative analysis functions described in this manual. Chapter 13 describes the standard script command language used in all **AMaze** postprocessors. Note that the script file must be in the same directory as the data files.

Create script

This command allows you to create script files using the internal editor. A dialog box requests a file prefix. The resulting script file will be saved as `FPREFIX.SCR`. Next, the program opens the file in the editor and writes a reference list of allowed commands. This list

follows the *EndFile* command and will be ignored. Enter analysis commands above the *EndFile* command. After saving the file, you can run it using the *Run script* command.

Edit script

Use this command to change an existing script file. The dialog lists files in the current directory with the subscript SCR. Changing directories does not change the working directory of the program.

Open data file

Several of the analysis commands like *Point calculation* and *Line scan* generate quantitative information. You can automatically record the data generated during an analysis session by opening a data file. Supply a file prefix in the dialog or accept the default. The data file has a name of the form FPREFIX.DAT and will be stored in the working directory. The file is in ASCII format. You can use a text editor to view the file or to extract information to send to mathematical analysis programs or spreadsheets. Table 1.3 shows examples of data file listings.

Close data listing file

Use this command if you want to start a new file to record data. The data file is automatically closed when you exit the postprocessor. Otherwise, you must close the file before using the *Edit data file* command or loading the file into another program. Failure to close the file may result in a Windows Resource Sharing Error.

Edit data file

Use this command to view or to modify files with names of the form FPREFIX.DAT.

Edit file

Use this command to view or to modify any ASCII file.

The *Help* menu shows program information and contains the following command:

Instructions

Displays the HTML help file using your default browser. In order for this command to operate correctly, the file RFE3.HTML must be in the same directory as RFE3View.EXE.

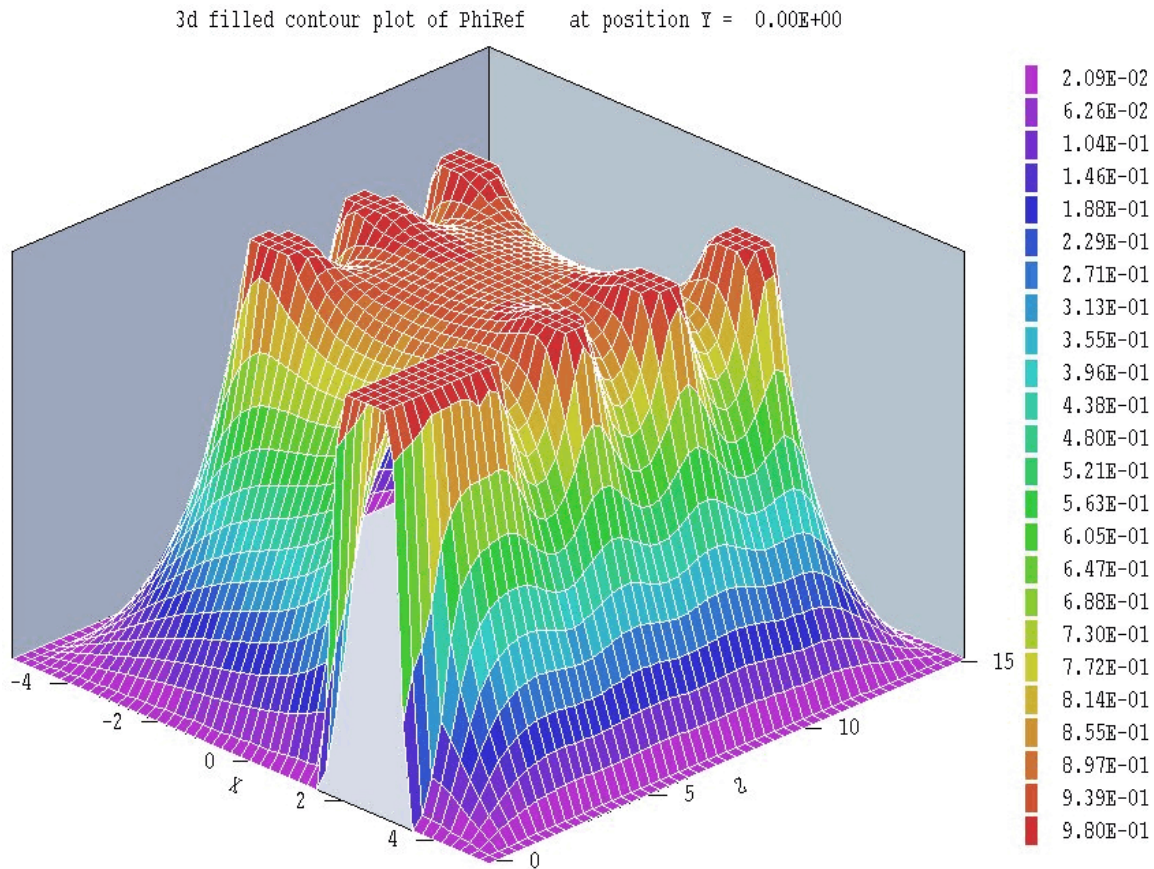


Figure 10.1. Illustration of a plane plot: potential at 0° phase for the WALKTHROUGH example.

Chapter 10. RFE3View - plane plots

Plane plots are two-dimensional plots that show the variation of quantities over a plane normal to one of the Cartesian axes. Plane plots provide a simple and quick views of the solution space. The technique is to create a rectangular mesh of values over a specified planar region and then to create plots in a variety of styles. No attempt is made to connect the plot mesh with the conformal mesh of the simulation. Slice plots, discussed in the next chapter, are also two-dimensional in a plane normal to a Cartesian axis. The difference is that slice plots are built using the computational mesh. They show precise region boundaries and equipotential lines in true scale.

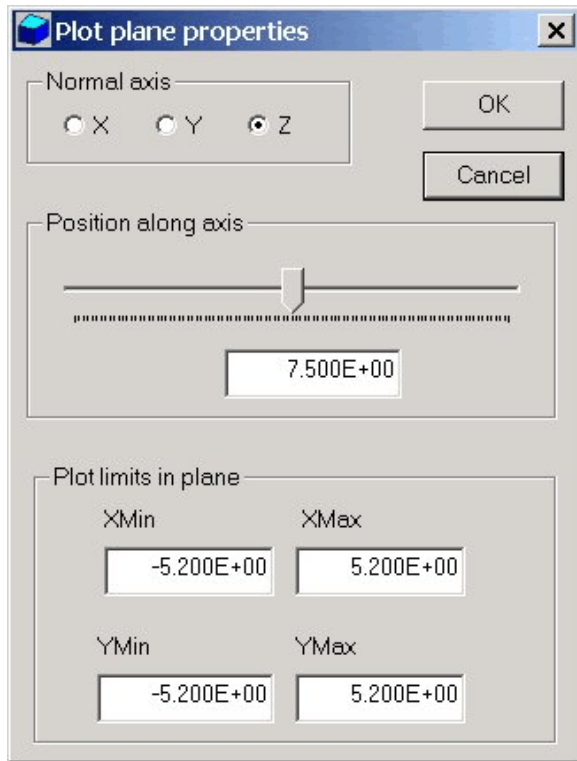


Figure 10.2. Set plane dialog

Set plane

This command brings up the dialog of Fig. 10.2 to set the plane for the plot. Set the normal axis with the radio buttons at the top. For example, for a choice of z the plot will be created in the x - y plane. You can use the slider bar to set the position along the normal axis or type a value in the box. The range of the slider bar is automatically set to the limits of the solution volume along the normal axis. The boxes at the bottom determine the plot range in the normal plane. The default settings are the limits of the solution volume. Note that plane plots are constructed to fill the maximum area. They do not preserve scaling in the normal plane.

The following commands are in the *Plot control* popup menu:

Plot style

This command brings up the dialog of Fig. 10.3 to set the plot style. As an example, Fig. 10.1 shows the *Filled contours 3D* plot style. The numbers at the bottom give the resolution of the mesh used to create

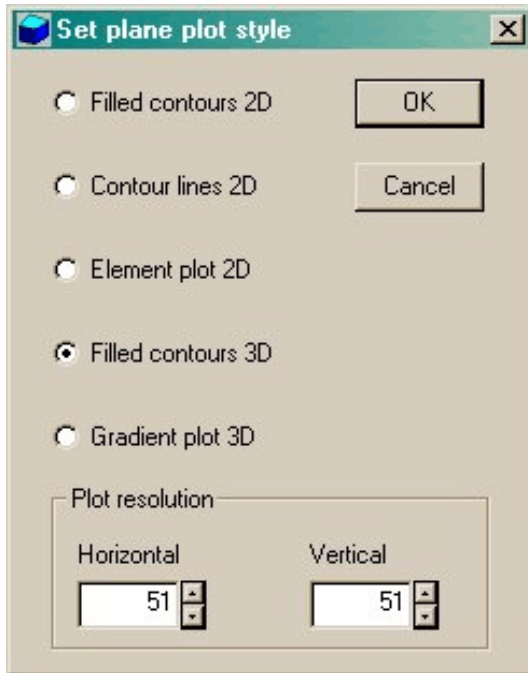


Figure 9.3. Plot style dialog

the plot. Higher values give more detail but require longer regenerate times. The default is a 51×51 mesh.

Plot quantity

Set the quantity to be plotted. The first choice, *PhiRef*, is the quasi-static potential ϕ at the reference phase. In this case the plot shows a snapshot of the spatial variation of ϕ at a particular phase. The default reference phase is 0.0° . You can change the value with the REFERENCE PHASE command. Take care in setting the value because the quantity of interest may have zero values everywhere at a particular phase. The next quantity is *|E|Ref*, the spatial magnitude of the electric field at the reference phase:

$$|E| = \sqrt{E_x^2 + E_y^2 + E_z^2} .$$

Other quantities include:

- *ExRef*, *EyRef*, *EzRef* (spatial components of the electric field at the reference phase),
- *|Jr|Ref*, *JxrRef*, *JyrRef*, *JzrRef* (spatial amplitude and components of the real part of the current density),

- $|Jd|Ref, JxdRef, JydRef, JzdRef$ (spatial amplitude and components of the displacement current density),
- $\langle E \rangle, \langle Jr \rangle, \langle Jd \rangle$ (time-averaged spatial amplitude of the electric field, real current density and displacement current density).
- *Power* (time averaged value of resistive power dissipation).

For example, suppose the electric field at a point varies as $E_o \cos 2\pi ft$, where

$$E_o = \sqrt{E_{xo}^2 + E_{yo}^2 + E_{zo}^2} .$$

Then the time-averaged resistive power density is given by

$$\langle P \rangle = \frac{\sigma E_o^2}{2} .$$

Note that electric field and current density plots may appear jagged at discontinuities because of the limited resolution of the mesh.

Plot limits

Set limits for the plotted quantity. When *Autoscale* is activated, **RFE3View** automatically sets limits based on the range of values in the current plot plane.

Rotate plot

This command is active only for the *Filled contours 3D* and *Gradient plot 3D* plot styles. You can rotate the plot in 90° increments for the best view.

The following command is used to changed the reference phase for displays:

Reference phase

Enter the reference phase in degrees

The commands of the *Export plot* menu are used to generate hardcopy or to create plot files.

Default printer

With this command, a **RFE3View** plot can be ported to any installed Windows printer (including network printers, postscript drivers, PDF drivers...). You can generate colored plots if you have a color printer. Note that the current screen plot is sent to the default Windows printer. If necessary, change the default using the *Settings* command of Windows *before* issuing the command.

Plot file (EPS)

Use this command to make a plot file of the current screen plot in Encapsulated PostScript format. Supply a file prefix in the dialog box. The plot file will be created in the current directory with the name `FPREFIX.EPS`.

Plot file (BMP)

Use this command to make a plot file of the current screen plot in Windows Bitmap format. Supply a file prefix in the dialog box. The plot file will be created in the current directory with the name `FPREFIX.BMP`.

Plot file (PNG)

Use this command to make a plot file of the current screen plot in Portable Network Graphics format (GIF equivalent). Supply a file prefix in the dialog box. The plot file will be created in the current directory with the name `FPREFIX.PNG`.

Copy to clipboard

The current plot is copied to the clipboard in Windows Metafile format. You can then paste the image into graphics software.

The screen capture capability of **MWSnap** (supplied with the program) is a useful alternative to transfer screen plots to documents. The method was used for the illustrations of this manual.

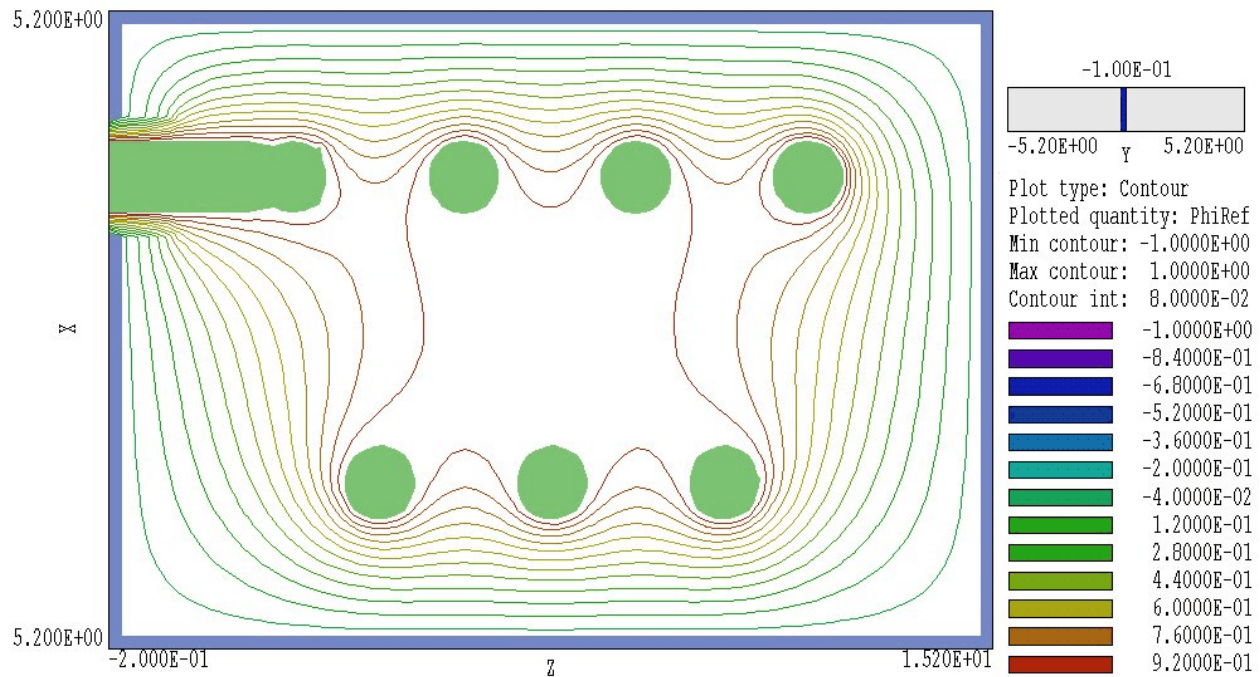


Figure 11.1. Slice plot – contours of the quasi-static potential at the reference phase.

Chapter 11. RFE3View - slice plots

11. 1. Setting the slice view

Slice plots are two-dimensional plots that show the variation of quantities over a plane normal to one of the Cartesian axes. In contrast to plane plots, slice plots are based on the structure of the mesh projected to a slice plane. This structure may be quite complex for a conformal mesh; therefore, slice plots require more computational effort. To facilitate the process, slices are constructed at discrete locations along the normal axis corresponding roughly to element planes in the foundation mesh. The precise rendering of spatial information makes it possible to apply mouse point-and-click analysis operations (point calculation, line scan,...) in the slice.

The *Change view* popup menu contains commands to set the slice plane and to adjust the dimensions of the plot.

Set slice plane properties

This command calls up the same dialog as the *Set plane* command in the plane plot menu (Fig. 10.2). You can change the normal axis, change the position along the normal axis, and set plot limits in the normal plane.

Slice normal to X**Slice normal to Y****Slice normal to Z**

Quick commands to change the normal axis.

Jump forward**Step forward****Step backward****Jump backward**

Move along the slice axis by small or large steps. The small step is approximately one layer of the foundation mesh and the large step is 5 layers. The term *forward* implies motion toward higher indices of the normal axis. The slider bar in the orientation area to the right of the plot (Fig. 11.1) shows the current location.

Zoom window

As an alternative to the entries in the *Set slice plane* dialog, you can interactively change plot limits in the normal plane using the mouse. Choose the command and move the mouse pointer into the plot area. The status bar enters coordinate mode. It shows the current mouse position in the plot and the snap mode. Use the left button to pick one corner and then move the mouse to create a view box. Click the left button again, and the plot regenerates. On any coordinate operation, press the *F1* key if want to enter values from the keyboard. The normal plane box in the orientation area to the right of the plot shows the dimensions of the slice plane and the outline of the current zoomed view.

Zoom in

Enlarge the plot about the current view center.

Expand view

Expand the plot about the current view center.

Global view

Enlarge the plot boundaries to show the entire normal plane.

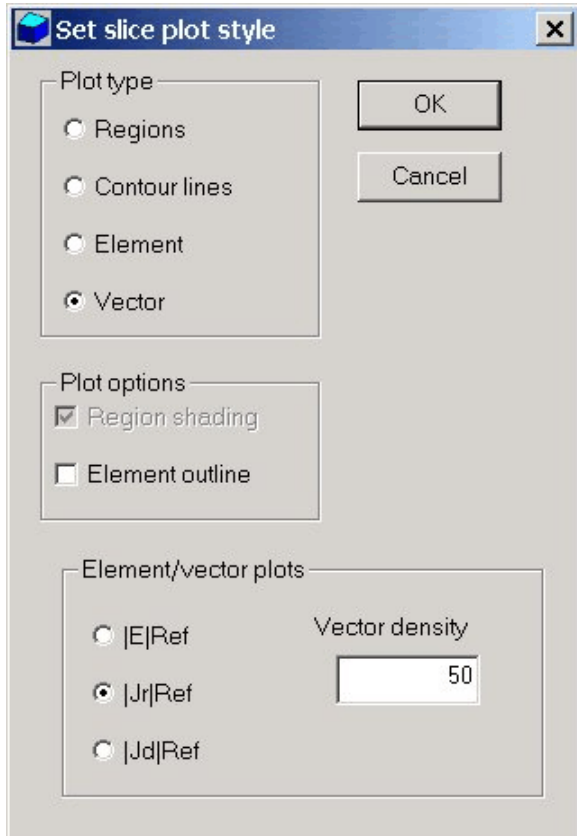


Figure 11.2. Slice plot style dialog.

Pan

When the plot is zoomed, you can use this operation to shift the current view center. Use the mouse to define a relative starting and ending point for the shift.

11.2. Setting slice plot properties

The commands in the *Plot control* popup menu are used to set the plot style and mouse options.

Set slice plot style

This command brings up the dialog of Fig. 11.2. The *Region plot* style is a cross-section view of the mesh element divisions color-coded by region. In contrast to the logical plane plot of **MetaMesh**, **RFE3View** attempts to resolve the exact mesh structure in the plane. The *Contour plot* style shows lines of constant potential at the reference phase. The vector component of electric field in the normal plane is normal to the equipotential lines and $|\mathbf{E}|$ is inversely proportional to the distance between lines. The two options at the bottom of the dialog are active

for contour plots. *Region shading* adds light background colors to show the boundaries between different dielectric or conductive regions. Only electrodes are colored when region shading is inactive. When *Element outline* is active the boundaries of elements are added in fixed-potential regions. The *Element plot* style shows elements in the slice color-coded according to a variety of computed quantities. A *Vector plot* superimposes vector lines on an element plot. The radio buttons at the bottom of the dialog set the plotted quantity (electric field, real current density or displacement current density at the reference phase). For electric field, color-coding follows the $|\mathbf{E}|$ at the reference phase and the vector lines lie along the direction of \mathbf{E} projected in the slice plane.

Plot limits

In the Auto-scale mode, **RFE3View** chooses defaults for the minimum and maximum potential values of contours. Deactivate Auto-scale to set the values manually.

Number of contour lines

Change the number of lines for contour plots.

Toggle snap mode

Mouse coordinates for commands such as *Zoom window*, *Pan*, and *Scan in slice* can be entered in two modes. In the normal mode, the returned position corresponds to the mouse position on the screen. In the snap mode, the program picks a point at an even interval close to the mouse position. The returned point depends on the value of the parameter *DSnap*. For example if $DSnap = 0.1$ and the mouse is at position (6.2345,-5.6113), the returned position is (6.2000,-5.6000). The status bar displays the actual or snapped position of the mouse.

Set snap distance

Change the value of *DSnap* from the default value determined by the program.

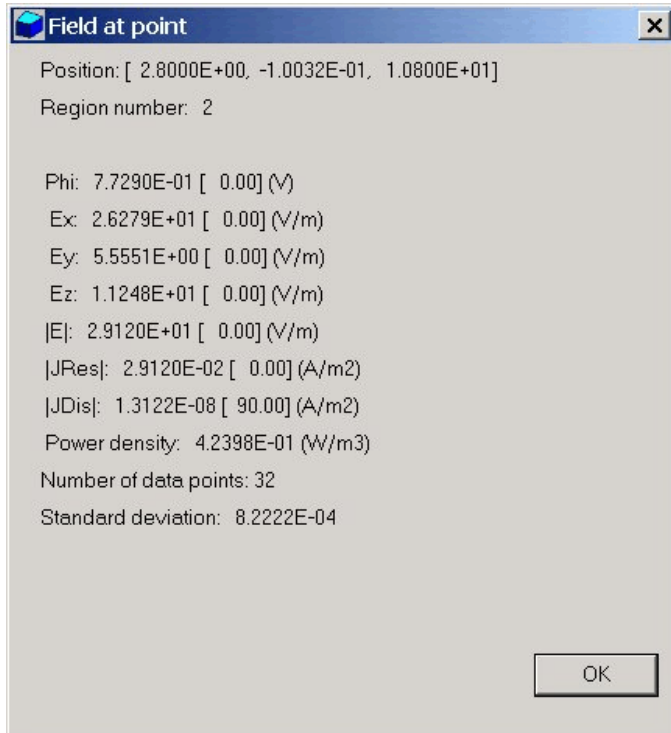


Figure 11.3. Information display — point calculation

Toggle grid

A set of dashed grid lines can be superimposed on slice plots. **RFE3View** automatically chooses intervals and positions so that the lines occur at convenient values along the horizontal and vertical directions (for example, 0.01 rather than 0.01153). Listings of the grid intervals are included in the axis labels. Grids corresponding to the normal plane axes are plotted as solid lines.

11.3. Analyses in a slice

You can determine field values at points and along scan lines with the commands of the *Analysis* popup menu.

Point calculation

This command is useful to make quick checks of fields in the solution volume. After you click the *Point calculation* command, move the mouse into the plot area. The mouse pointer changes to a cross-hair pattern and the status bar enters coordinate mode. Click the left button to specify a point or press the *F1* key to enter the coordinates from the keyboard. Note that mouse coordinates will shift between discrete values if snap mode is active. **RFE3View** calculates potential and field

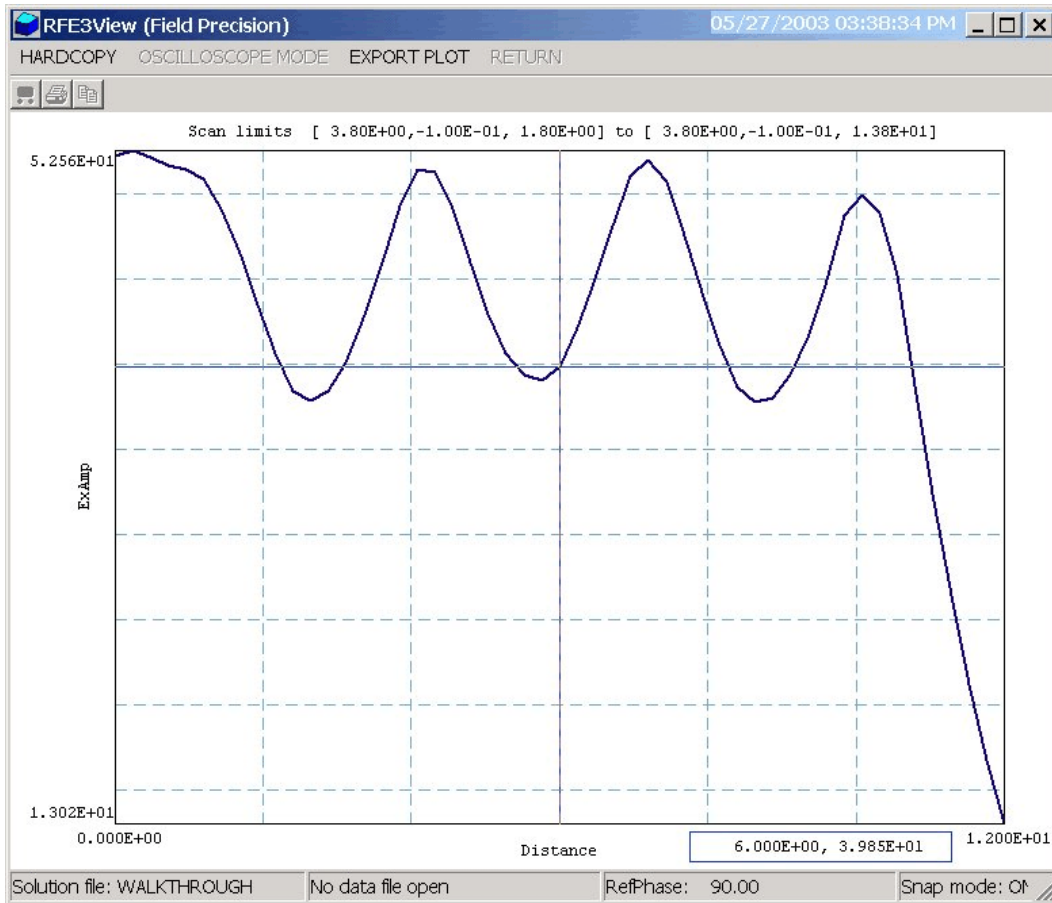


Figure 11.4. Scan plot screen in the oscilloscope mode.

values at the point in the normal plane given by the coordinates. The following chapter describes the interpolation method. Figure 11.3 shows the information display. RFE3View calculates complex values of the potential, electric field spatial components, electric field magnitude, resistive current magnitude and displacement current magnitude. A complex number is represented by an amplitude (in SI units) and a bracketed phase in degrees. The results are recorded if a data file is open.

Line scan

Line scans are one of the most useful **RFE3View** capabilities. After clicking on the command, supply two points with the mouse to define a scan line (or press the F1 key to enter coordinates manually). The snap mode is useful in this application (for example, you may want the scan to extend from 0.000 to 5.000 rather than 0.067 to 4.985.) The program computes a series of values of field quantities in the normal

plane at equal intervals along the line. Complete information is recorded if a data file is open. The program also makes a screen plot of the currently-selected quantity versus distance along the scan and activates the *Scan plot* menu (Fig. 11.4). **RFE3View** adds fiducial lines to the plot using intelligent grid selection. This means that the plot is adjusted to fill the screen and grids are drawn at useful intervals (*i.e.*, 0.05 or 2.00).

Set scan quantity

With this command you can pick the quantity that will be displayed in screen and exported plots of line scans. Pick the quantity from the list box and click *OK*. A wide variety of quantities are available in RFE3View:

- ϕ , E_x , E_y , E_z , $|\mathbf{E}|$, $|\mathbf{J}_{\text{real}}|$ and $|\mathbf{J}_{\text{disp}}|$ at the reference phase,
- the amplitude of ϕ , E_x , E_y , E_z , $|\mathbf{E}|$, $|\mathbf{J}_{\text{real}}|$ and $|\mathbf{J}_{\text{disp}}|$,
- the time-averaged power density.

Set number of scan points

This command sets the number of line scan points in the screen plot and data file listing. The default value is 50 and the maximum number is 250.

In addition to the standard *Export plot* options, the *Scan plot* menu contains the following command:

Oscilloscope mode

In oscilloscope mode, a scan plot assumes characteristics of a digital oscilloscope. **RFE3View** superimposes a cross-hair pattern on the graph. Plot values at the intersection are displayed in the information window. Move the marker along the plot by moving the mouse. If you click the left mouse button at a point, the program records information when a data file is open. Press the right mouse button to exit the oscilloscope mode.

Click *Return* to exit the scan plot and return to the slice plot.

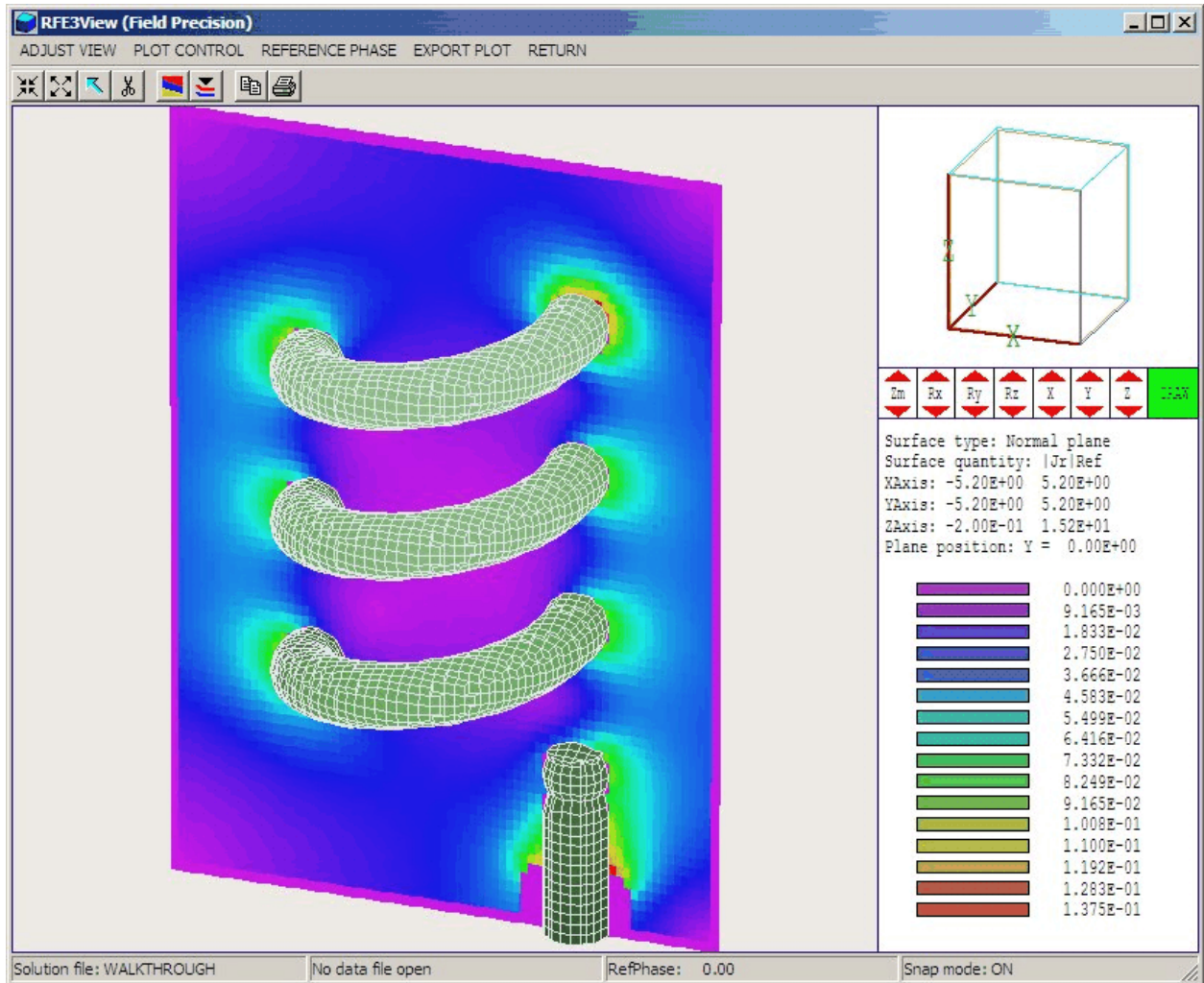


Figure 12.1. Three-dimensional plot with onscreen view controls.

Chapter 12. RFE3View - surface plots

Surface plots are three-dimensional views of the solution space. Three types of information can be displayed: 1) boundaries of regions, 2) computed quantities in a slice plane normal to one of the Cartesian axes (Fig. 12.1) and 3) equipotential surfaces (at the reference phase) with color-coding by $|E|$. Boundary plots can be superimposed on either of the data plot types. Surface plots are created from the conformal mesh and preserve true spatial scaling.

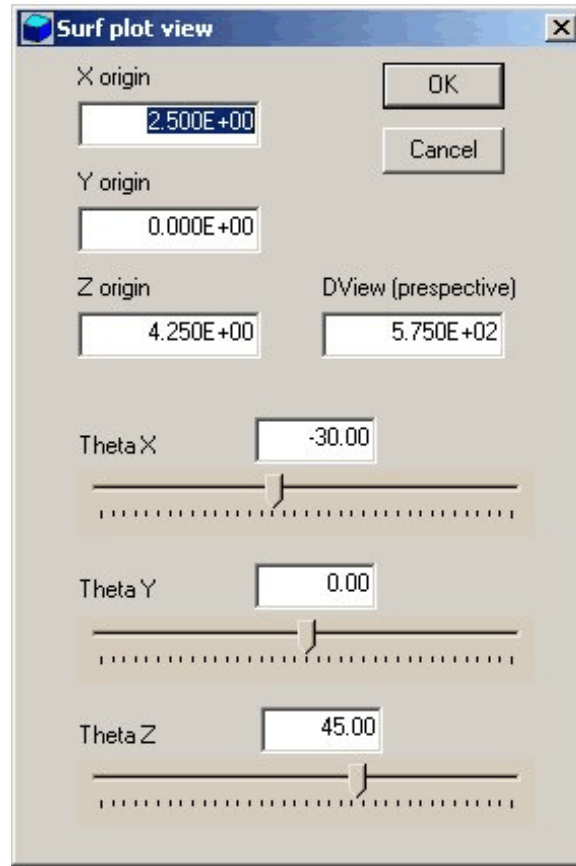


Figure 12.2. Set surface view dialog

The three-dimensional display of the current plot style is controlled either with the onscreen arrows or with the commands of the *Adjust view* popup menu. The commands are identical to those used in **MetaMesh** for projection and surface plots. You can change view angles, displacements and the zoom factor from the *Set surface view* dialog (Fig. 12.2). Alternatively, you can use the following quick menu commands or tools for individual functions.

ThetaX(Up)

Rotate about the *x* axis in the direction of positive angle. The default angular step is 22.5°. You can change the angular step in the *Set surface view* dialog. Related commands: **ThetaY(Up)**, **ThetaZ(Up)**.

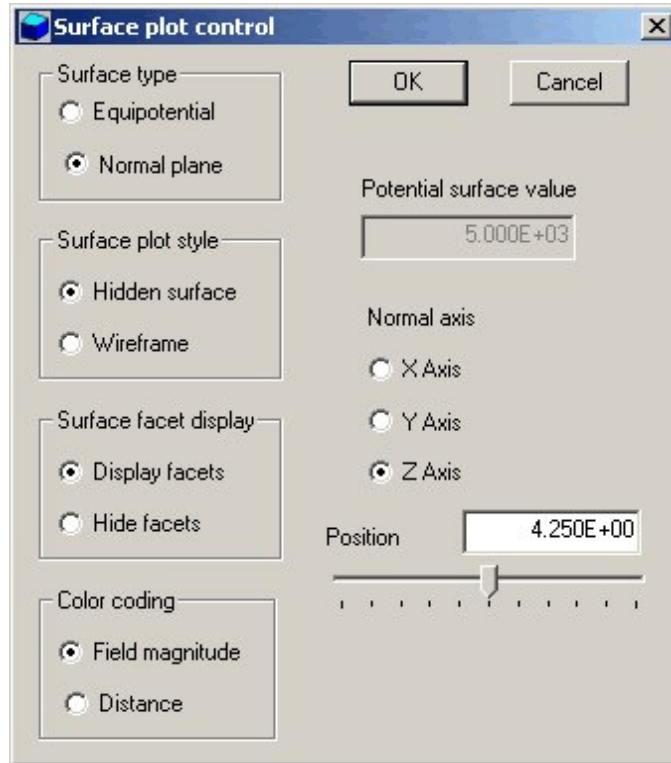


Figure 12.3. Surface plot style dialog

ThetaX(Dn)

Rotate about the x axis in direction of negative angles. Related commands: **ThetaY(Dn)**, **ThetaZ(Dn)**.

XDisp(Up)

Move the view point in the direction of positive x. You can change the displacement step in the *Set surface view* dialog. Related commands: **YDisp(Up)**, **ZDisp(Up)**.

XDisp(Dn)

Move the view in the direction of negative x. Related commands, **YDisp(Dn)**, **ZDisp(Dn)**.

Zoom in

Magnify the view about the current view center.

Expand view

Expand the view about the current view center.

The commands of the *Plot control* popup menu control the appearance of the plot.

Surface plot style

This command brings up the dialog of Fig. 12.3 that controls the data plot. The first group sets the data plot type. The *Normal plane* plot shows color-coded information on a variety of quantities in a plane normal to one of the Cartesian axes (Fig. 12.1). It is the three-dimensional equivalent of the plane plot discussed in Chap. 9 except that it preserves true spatial scaling. For *Normal plane* plots the commands to choose the normal plane on the right-hand side of the dialog become active. You can set the normal axis and pick the plane position along the axis. The *Equipotential* surface is a much more complex plot. **RFE3View** analyzes all element facets in the mesh to create a precise view. The surface is defined by the condition that the reference value of potential ϕ equals a constant value ϕ_0 . By default, **RFE3View** picks ϕ_0 as the average of the minimum and maximum values of ϕ in the solution space. You can set ϕ_0 in the box marked *Potential surface value*. The radio buttons *Hidden surface* and *Wireframe* control whether the facets are filled. The wireframe view regenerates faster but may be difficult to understand. You can use the wireframe mode to set up a three-dimensional view and then switch to the hidden-surface mode to create the final plot. The radio buttons *Display facets* or *Hide facets* controls whether the edges of facets are shown for region boundaries or equipotential surfaces. The final set of radio buttons the method for color-coding equipotential surfaces, either by $|\mathbf{E}|$ or distance from the viewpoint.

Displayed regions

The command brings up a dialog where you can pick region boundaries to superimpose on a plot. The default is that no regions are displayed.

Set cut planes

In a hidden surface plot, internal details may be obscured by surrounding parts or regions. This command brings up a dialog that

allows you to adjust the display areas along the x , y and z axes.

RFE3View does not display facets that lie outside the limits. With this feature you can create cutaway views. The default is that cut limits are set equal to the dimensions of the solution volume so that all facets are included.

Set surface plot quantity

Use this command to set the quantity for color-coding in normal plane plots. The choices are ϕ , $|\mathbf{E}|$, $|\mathbf{J}_r|$ and $|\mathbf{J}_d|$ at the reference phase, the amplitude of ϕ , $|\mathbf{E}|$, $|\mathbf{J}_r|$ and $|\mathbf{J}_d|$, and the time-averaged power dissipation.

Set surface plots limits

Manually set limits on color-coded quantities for normal plane plots.

Chapter 13. Analysis

The commands of the *Analysis* popup menu are used to determine numerical values for the solution. In the interactive mode, the menu is active only when a data file is open.

13.1. Point calculation - interpolation method

The *Point calculation* command brings up a dialog where you enter the x , y and z coordinates of a general point in the solution space. In contrast to the *Point calculation* command of the *Slice plots* menu, the point is not limited to a slice plane. When you click *OK*, **RFE3View** displays the information box of Fig. 11.3 and also writes the results to the data file in the following format:

```
--- Point Field Analysis ---
Position: [ 4.5000E+00, 0.0000E+00, 7.5000E+00]
Phi: 1.7482E-01 [ 0.00] (V)
Ex: 3.6473E+01 [ 0.00] (V/m)
Ey: 1.7692E-03 [ 0.00] (V/m)
Ez: 1.1051E-02 [ 0.00] (V/m)
|E|: 3.6473E+01 [ 0.00] (V/m)
|JRes|: 3.6473E-02 [ 0.00] (A/m2)
|JDis|: 1.6435E-08 [ 90.00] (A/m2)
Power density: 6.6513E-01 (W/m3)
Number of data points: 32
Standard deviation: 1.0010E-04
```

Note that the results are displayed in complex-number format (amplitude with the phase in degrees shown in brackets). The components E_x , E_y and E_z give the spatial direction of the electric field, real current density and displacement current density.

RFE3View must perform a complex sequence of operations to make the calculation

- Estimate the indices of the element that contains the target point.
- Check this element and its neighbors using a normal coordinate inversion to determine the precise element that contains the target point.
- Collect the nodes of the target element as well as neighboring nodes

that border on at least one element with the same region number as the target element

- Fit a three-dimensional, second-order function to the potential values of the data set using a least-squares method.
- Find the potential at the target point using the interpolation function.
- Find the electric field values by taking derivatives of the interpolation function.

Although the procedure sounds complicated, **RFE3View** performs it with high reliability. The effort pays dividends in the accuracy of the interpolated values. The second-order least-squares method gives electric field estimates that are substantially better than those derived from the direct use of the quasi-linear finite-element form functions. Furthermore, the requirement that the nodes of the data set border on elements of the target region ensures that data points will be collected from one side of a dielectric boundary. Therefore, **RFE3View** correctly handles the discontinuity in electric field at such a boundary.

13.2. Line scans and matrix files

In the *Analysis* menu scans can be performed along arbitrary straight lines between any two points in the solution volume. When you click the *Line scan* command a dialog appears where you can specify the start and end points. Enter values in units set by *DUnit*. In other words, if you used dimensions of cm in **MetaMesh** and the **RFE3** script contains the command $DUnit = 100.0$, then enter the dimensions in cm. In the interactive mode, **RFE3View** creates a screen plot of a selected quantity determined by the *Set scan quantity* command. The command has no effect on the data file listing which contains all calculated quantities. The following listing example has been divided into multiple lines for clarity. The tabulated quantities are complex values of ϕ , E_x , E_y , E_z , $|\mathbf{E}|$, $|\mathbf{J}_r|$, $|\mathbf{J}_d|$ and the time-averaged power

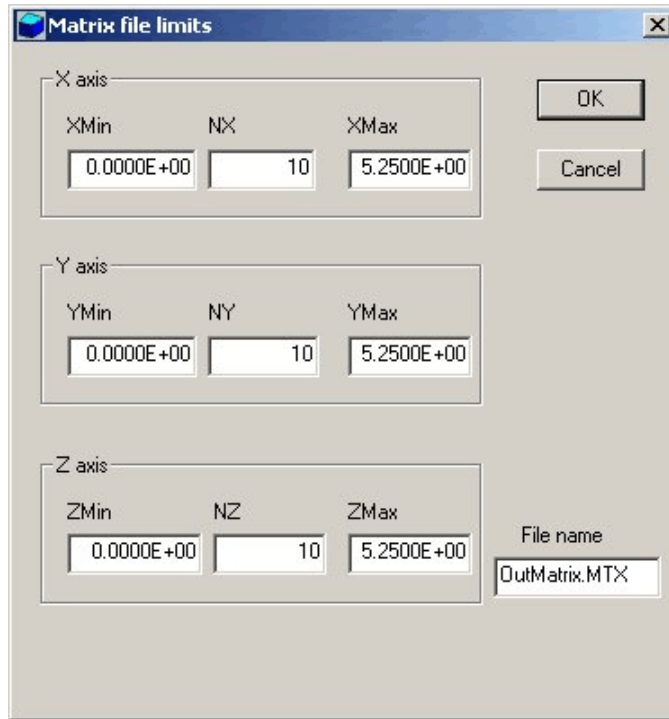


Figure 13.2. Create matrix file dialog

```

----- Field scan between points -----
XStart:  4.5000E+00  YStart:  0.0000E+00  ZStart:  2.0000E+00
XEnd:    4.5000E+00  YEnd:    0.0000E+00  ZEnd:    8.0000E+00

      X           Y           Z           Phi
=====
4.500E+00  0.000E+00  2.000E+00  1.574E-01 [ 0.000]
4.500E+00  0.000E+00  2.120E+00  1.614E-01 [ 0.000]
4.500E+00  0.000E+00  2.240E+00  1.647E-01 [ 0.000]
4.500E+00  0.000E+00  2.360E+00  1.675E-01 [ 0.000]
...

      |E|           Ex           Ey
=====
3.477E+01 [ 0.000]  3.458E+01 [ 0.000]  7.978E-01 [ 180.000]
3.543E+01 [ 0.000]  3.529E+01 [ 0.000]  8.565E-01 [ 180.000]
3.603E+01 [ 0.000]  3.593E+01 [ 0.000]  9.268E-01 [ 180.000]
3.649E+01 [ 0.000]  3.642E+01 [ 0.000]  9.951E-01 [ 180.000]
...

      Ez           |Jr|           |Jd|
=====
3.573E+00 [ 180.000]  3.477E-02 [ 0.000]  1.567E-08 [ 90.000]
3.042E+00 [ 180.000]  3.543E-02 [ 0.000]  1.596E-08 [ 90.000]
2.536E+00 [ 180.000]  3.603E-02 [ 0.000]  1.624E-08 [ 90.000]
2.082E+00 [ 180.000]  3.649E-02 [ 0.000]  1.645E-08 [ 90.000]
...

```


Power	NReg
6.045E-01	2
6.275E-01	2
6.490E-01	2
6.659E-01	2
...	

The *Create matrix file* command controls a **RFE3View** feature that is useful if you want to write your own analysis routines or port results to mathematical software. In response to the command, **RFE3View** performs interpolations over a specified box region on a regular grid of values. It is easier to use results in this form than to deal with the conformal mesh directly. Clicking on *Create matrix file* calls up the dialog of Fig. 13.2. Specify the dimensions of the box along each axis (in units set by *DUnit*) and the number of calculation intervals. To illustrate, calculations are performed at positions with x coordinates given by

$$x = x_{\min} + \frac{n (x_{\max} - x_{\min})}{n_x}, \quad (13.4)$$

where $n = 0, 1, 2, \dots, n_x$. For example, if you set $x_{\min} = 0.5$, $x_{\max} = 1.5$ and $n_x = 10$, the calculations are performed at points with $x = 0.5, 0.6, \dots, 1.4, 1.5$. You can also specify an output file name. The ASCII file is created in an ordered format with labels. The stored quantities depend on the solution type. You can inspect the file with an editor.

13.3. Automatic analyses

In response to the command *Full analysis*, **RFE3View** performs volume integrals over all elements of the mesh. For each element the program determines the electrostatic field energy and the resistive power. The calculations are intensive, so there may be a delay for large meshes. **RFE3View** organizes energy by region and write the results to the data file in the format shown below. Fixed-potential regions are listed separately because they have zero field energy.

----- Global analysis of the solution volume -----

Fixed potential regions

NReg	Potential Amp (V)	Phase (deg)	Volume (m3)
1	0.00000E+00	0.000	4.86546E-04
4	1.00000E+00	0.000	5.73522E-05

Dielectric regions

NReg	Epsilon (relative)	Sigma (S/m)	E (min) (V/m)	E (max) (V/m)
2	8.10000E+01	1.00000E-03	3.81470E-06	1.95896E+02
3	2.71000E+00	0.00000E+00	6.46640E+01	3.12082E+02

Energy (J)	Power (W)	Volume (m3)
4.49148E-10	6.26264E-04	1.11972E-03
1.26676E-12	0.00000E+00	2.38688E-06

Total field energy: 4.50415E-10 (J)
 Total power dissipation: 6.26264E-04 (W)
 Total volume: 1.66600E-03 (m3)

The walkthrough example of Chap. 1 showed how to use values of electrostatic energy and power dissipation to determine capacitance and resistance.

The volume integral method does not give information on mutual capacitance and resistances when there are several electrodes. In this case, you can use the *Region* command to perform surface integrals over selected regions. In response to the *Region* command with a specified region number, **RFE3View** performs the following operations:

- Carries out volume integrals over the region using the same routines as the *Full analysis* command.
- Determines all element facets on the region surface, excluding those on the outer surface of the solution volume.
- Performs a surface integral over each facet. At an integration point representing a differential surface area dA , the program determines the unit vector \mathbf{n} normal to the surface. **RFE3View** also calculates the electric field \mathbf{E} at a nearby position in the adjacent element with relative dielectric constant ϵ_r and σ .

- Forms the sumz of $\epsilon_r \mathbf{E} \cdot \mathbf{n} dA$ and $\sigma \mathbf{E} \cdot \mathbf{n} dA$.

The end result is a calculation of the free surface charge on the region and total current flow. Information in the data file has the following format:

```
----- Analysis of Region Number 4 -----
Fixed potential: 1.000E+00[ 0.000]
Volume integral quantities
Volume: 5.73522E-05 (m3)
Surface integral quantities
Surface area: 1.96319E-02 (m2)
Induced charge: 8.317E-10[ 0.000] (coulombs)
Real current: 1.156E-03[ 0.000] (A)
Displacement current: 5.226E-10[ 90.000] (A)
Total current: 1.156E-03[ 0.000] (A)
```

With information on surface charge (or current), we can calculate the matrix of mutual capacitances and resistances. To illustrate, suppose we have two electrodes and a ground plane and we want to find the capacitance between Electrode 1 and Electrode 2 as well as the capacitance between Electrode 1 and ground. We apply a voltage $V_o = 1V$ to Electrode 1 and determine the surface charge on Electrode 2 (Q_2) and the ground electrodes (Q_g). The mutual capacitance between Electrode 1 and 2 is $C_{12} = Q_2$ and the capacitance to ground is $C_{1g} = Q_g$. We could then do a second solution with a voltage on Electrode 2 to find C_{2g} .

13.4. Script operation

You can create a script that will automatically call the commands of the *Analysis* menu. **RFE3View** scripts have names of the form `SCRIPTNAME.SCR`. You can initiate an automatic analysis using the *Run script* command in the interactive mode or starting **RFE3View** from a batch file with a command of the form:

```
START \AMAZE\RFE3View SCRIPTNAME
```

We discussed the following interactive script commands in Chapter 8: *Run script*, *Create script* and *Edit script*. The *Create script* command generates a template file with the content shown in Table 13.1. In this section, we shall discussed the allowed commands in detail.

Table 13.1. Create script - default content

```
* AMaze script file
* Insert commands here...

ENDFILE

    --- Script command summary ----

INPUT FileName
  [Close current solution file and load FileName]
OUTPUT FPrefix
  [Close current data file and open FPrefix.DAT]
NSCAN 100
  [Set the number of points in a line scan]
POINT xp yp zp
  [Point field calculation at the given coordinates]
LINE xp1 yp1 zp1 xp2 yp2 zp2
  [Scan along a line between the given coordinates]
FULLANALYSIS
  [Write analyses for all region to the data file]
REGION RegNo
  [Write an analysis for region RegNo to the data file]
MATRIX FileName XMin XMax NX YMin YMax NY ZMin ZMax NZ
  [Write a matrix of field values to the file FileName]
ENDFILE
  [Terminate the analysis]
```

Symbolic command: INPUT FPrefix.ROU

Example: INPUT WGUIDE1.ROU

Function: Close the current solution file and load a binary solution file from the current directory

Comments: The solution file must be in the same directory as the script

Symbolic command: OUTPUT FPrefix

Example: OUTPUT WGUIDE1

Function: Close the current data file and open a new data file.

Comments: The data file has a name of the form FPrefix.DAT.

Symbolic command: NSCAN NScan

Example: NSCAN 100

Function: Set the number of intervals for line scans

Comments: The default value is $NScan = 50$, the maximum value is $NScan = 250$.

Symbolic command: POINT xp yp zp

Example: POINT 0.00 0.05 4.67

Function: Perform a point field calculation and write the result to the data file. Enter coordinates in the units set by *DUnit*.

Symbolic command: LINE xp1 yp1 zp1 xp2 yp2 zp2

Example: LINE 0.00 0.00 15.00 12.00 0.00 15.00

Function: Perform ($NScan+1$) calculations along a line in space and write the results to the data file. Enter coordinates in units set by *DUnit*.

Command: FULLANALYSIS

Function: Write volume integrals for the full solution space to the data file.

Symbolic command: REGION RegNo

Example: REGION 7

Function: Write volume and surface integrals for region *RegNo* to the data file.

Symbolic command: MATRIX FileName XMin XMax NX YMin YMax NY ZMin ZMax NZ

Example: MATRIX WGUIDE.MAT 1.00 1.00 10.00 2.00 2.00 12.00

Function: Write solution values computed field values at an array of location to a data file in ASCII format. The file is named *FileName* and is created in the current directory.

Command: ENDFILE

Function: Terminate the analysis

Chapter 14. Structure of the RFE3 output file

RFE3 creates output files in binary or ASCII format. The mode is controlled by the *FORMAT* command (Sect. 4.2). The *BINARY* mode must be used to communicate results to **RFE3View**. The ASCII mode is useful to transfer results to your own analysis programs.

As in **MetaMesh**, nodes are referenced with the indices $[I, J, K]$ where I (the index along the x axis) extends from 0 to I_{\max} , J (y axis) from 0 to J_{\max} , and K (z axis) from 0 to K_{\max} . The number of elements is approximately equal to the number of nodes. A single element (in the direction of positive x , y and z) is associated with each node for storage. Nodes and elements have integer *region numbers* to associate them with structures in the solution space.

The format of the **RFE3** binary output file is simple and compact, making it easy to transfer information to other programs. The code extract shown in Table 14.1. comprises the entire output algorithm. In the header, the quantities I_{\max} , J_{\max} and K_{\max} are 4-byte integers. *DUnit* is a 4-byte real, *Frequency* is a 4-byte real and *NCQuant* is a 1-byte integer. Here *NCQuant* is the number of stored quantities for the solution program. In **RFE3** $NCQuant = 2$ – the recorded node quantities are the real and imaginary parts of the complex electrostatic potential (ϕ) at the node. **RFE3** then records the name of the quantity as a string of length 12. This information is used to create labels in **RFE3View**.

The next step is to record information for each node. The quantities *RegNo* (the region number of the node) and *RegUp* (the region number of the adjacent element in the direction of increasing I , J and K) are 1-byte integers. The coordinates (X, Y and Z) and the stored quantities (ϕ_r and ϕ_i) are 4-byte real numbers. The final step is to record region information. The quantity *NRQuant*, the number of physical quantities stored per region, is equal to 4 in **RFE3**. The quantities are the relative dielectric constant ϵ_r , the conductivity σ in S/m, the amplitude of ϕ for fixed-potential regions in volts, and the phase of ϕ in degrees. The region quantity names (*RQuantName*) are strings of length 12. The variable *NFix* is a 1-byte integer and the region quantities are 4-byte real numbers.

Table 14.1. Code to create the RFE3 binary output file

```
! Mesh size parameters
WRITE (OutField) IMax,JMax,KMax,DUnit,Frequency,NCQuant
! Names of calculated quantities
DO N=1,NCQuant
  WRITE (OutField) CQuantName(N)
END DO
! Node and element information, potential values
DO K=0,KMax
  DO J=0,JMax
    DO I=0,IMax
      MC = M(I,J,K)
      WRITE (OutField) &
        C(MC).RegNo,C(MC).RegUp,C(MC).x,C(MC).y,C(MC).z, &
        REAL(C(MC).Phi),IMAG(C(MC).Phi)
    END DO
  END DO
END DO
! Number of region quantities and names of quantities
WRITE (OutField) NRQuant
DO N=1,NRQuant
  WRITE (OutField) RQuantName(N)
END DO
! Number of regions
WRITE (OutField) NRegMax
! Region quantities
DO N=1,NRegMax
  IF (Reg(N).Fixed) THEN
    NFix = 1
  ELSE
    NFix = 0
  ENDIF
  WRITE (OutField) NFix,Reg(N).EpsiR,Reg(N).Sigma, &
    Reg(N).PotAmp,Reg(N).PotPhase
END DO
```

Table 14.2. Code to create the RFE3 ASCII file

```
WRITE (OutField,5000)
WRITE (OutField,5100) OutFieldName,IMax,JMax,KMax, &
  DUnit,Frequency,NCQuant
WRITE (OutField,5200) (CQuantName(N),N=1,NCQuant)
DO K=0,KMax
  DO J=0,JMax
    DO I=0,IMax
      MC = M(I,J,K)
      WRITE (OutField,5300) &
        C(MC).RegNo,C(MC).RegUp,C(MC).x,C(MC).y,C(MC).z, &
        REAL(C(MC).Phi),IMAG(C(MC).Phi)
    END DO
  END DO
END DO
WRITE (OutField,5400) (RQuantName(N),N=1,NRQuant)
DO N=1,NRegMax
  IF (Reg(N).Fixed) THEN
    NFix = 1
  ELSE
    NFix = 0
  ENDIF
  WRITE (OutField,5500) N,NFix,Reg(N).EpsiR,Reg(N).Sigma, &
    Reg(N).PotAmp,Reg(N).PotPhase
END DO
5100 FORMAT (/ &
  ' File: ',A/ &
  ' IMax: ',I6/ &
  ' JMax: ',I6/ &
  ' KMax: ',I6/ &
  ' DUnit: ',1P,E12.4/ &
  ' Frequency: ',E12.4/ &
  ' NCQuant: ',I3)
5200 FORMAT (/ &
  ' RegNo ', &
  ' RegUp ', &
  ' X ', &
  ' Y ', &
  ' Z ',10A/)
5300 FORMAT (I5,I7,' ',1P,5E12.4)
5400 FORMAT (/ &
  ' RegNo ', &
  ' Fixed ',10A/)
5500 FORMAT (I5,I7,' ',1P,4E12.4)
O
```