Field Precision LLC

RFE3 4.0
Three-dimensional RF Electric Fields in Conductive Media

Contents

1 Introduction

1.1 Program function

RFE3 is a versatile numerical tool to find RF (radio-frequency) 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 250 regions to represent electrodes or different materials. The amplitude and phase of the harmonic potentials on electrodes may be set individually. Conductive materials are characterized by values of the volume resistivity and relative dielectric constant. Material properties may be uniform over a region of the solution volume or may vary following user-specified mathematical functions. The program can export a power density file that may be used in the **HeatWave** program for thermal analysis. **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 package includes two programs:

- `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 4 GB of memory can handle over 40 million elements.

RFE3 has the power to address the full range of your applications and keep pace with your growing expertise. Nonetheless, learning the program can be relatively painless if you proceed in manageable steps. We suggest the following sequence of activities:

- Follow the walkthrough example in the next section. This will give you quick introduction to the programs.

- Chapter 2 covers the theory of RF electric field solutions in the non-radiative limit.

- Read Chap. 3 to understand the procedure for **RFE3** calculations and the organization of data files.

- Try some of the ready-to-run solutions supplied in the example library. The examples illustrate many important techniques and may serve as templates for your applications.

- Chapters 8 through 11 give detailed descriptions of the capabilities of the **RFE3View** postprocessor.

- Tackle advanced topics when you have some experience with the programs. Chapter 4 describes how to modify **RFE3** scripts to invoke advanced functions. Other topics include creation of microscopic solutions to model small details (Sect. 4.3), definition of continuous
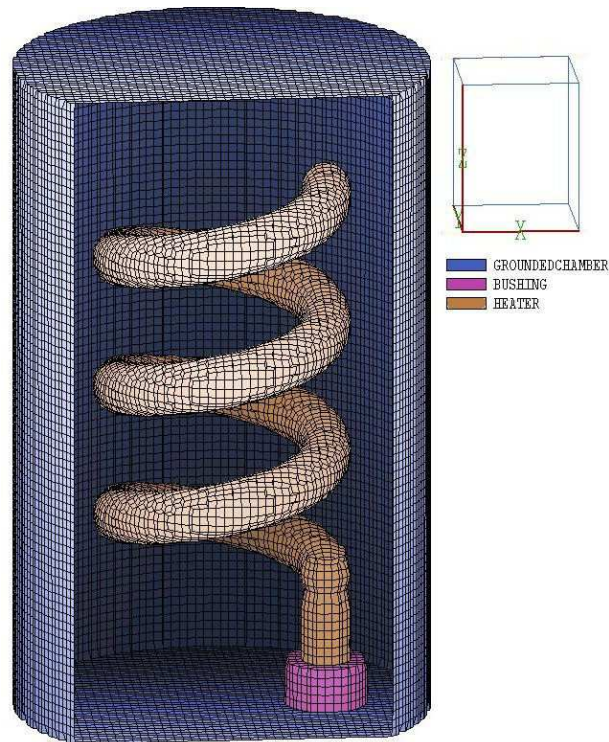
4

Figure 1: Geometry for the `WALKTHROUGH` example.

spatial variations of potential and material properties (Chap. 5) and automatic operation of programs in the background under batch file control (Sect. 6.2).

- Chapter 13 describes the **RFE3** output file format. The information is useful if you want to port data to own analysis programs.

1.2   Walkthrough example

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. The chamber is filled with purified water. This calculation would 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` and `WALKTHROUGH.SCR` to a working directory. Run the program launcher `AMAZE.EXE` and make sure the program and data directories are set correctly.

To begin, we shall use **Geometer** to confirm the geometry of the assembly. Run the program, choose the command *File/Load script* and pick `WALKTHRU.MIN`. The file content is listed in Table 1. Figure 2 shows a three-dimensional view of the assembly with the chamber show as a cutaway. Use the following operation to recreate the figure:

- Click on *View/Toggle parts/regions* to display objects by their region number.

- Click on *View/Orthogonal-perspective* to change to an interactive 3D view.

Table 1: File `WALKTHRU.MIN`, input to **MetaMesh**

```
GLOBAL                                  PART
  XMesh                                   Region Bushing
    -5.20   5.20   0.20                   Type Cylinder
  End                                     Fab 1.00 1.20
  YMesh                                   Shift  2.50  0.00  0.40
    -5.20   5.20   0.20                   Surface Region Chamber
  End                                     Surface Region ConductiveMedium
  ZMesh                                   Coat Chamber Chamber
    -0.20 15.20  0.20                   END
  End                                   PART
  RegName(1) Chamber                      Region Heater
  RegName(2) ConductiveMedium             Type Cylinder
  RegName(3) Bushing                      Fab  0.60  6.00
  RegName(4) Heater                       Shift 2.50 0.00 0.50
END                                       Surface Region Bushing
PART                                      Surface Region ConductiveMedium
  Region GroundedChamber                END
  Type Box                              PART
  Fab  10.40  10.40  15.40                Region Heater
  Shift  0.00  0.00  7.50                 Type Helix
END                                       Fab  2.50 0.60  9.00  3.00
PART                                      Shift 0.00 0.00 3.00
  Region ConductiveMedium                 Surface Region ConductiveMedium
  Type Cylinder                         END
  Fab  5.00  15.00                    ENDFILE
  Shift  0.00  0.00  7.50
  Surface Region Chamber
  Coat Chamber Chamber
END
```

- Use *View/Part clipping* to bring up the *Clip parts* dialog. In the row for Part 2, set *XLower* to zero.

- Move the mouse cursor into the display area and use the left and right buttons to rotate, shift or magnify the display.

Exit or minimize **Geometer** and run **MetaMesh**. Choose the command *File/Load MIN file* and pick `WALKTHRU.MIN`. Then choose *Process mesh* to generate the mesh of Fig. 1. Use the command *File/Save mesh* to create the file `WALKTHROUGH.MDF`.

We are now ready to use **RFE3**. Run the program from **AMaze**, choose the *Setup* command and pick the file `WALKTHROUGH.MDF`. The program displays the dialog of Fig. 3. Note that the grid contains a row for each mesh region. There are four columns in the grid where you can enter values: *Potential amplitude* and *phase* (fixed voltage), *EpsilonR* (relative dielectric constant) and *Sigma* (electrical conductivity in S/m).

The entries in the *Control parameter* section of the dialog have the following functions:

- FREQUENCY. The RF frequency in Hz. This entry is required. Enter a low, non-zero value for a DC solution.
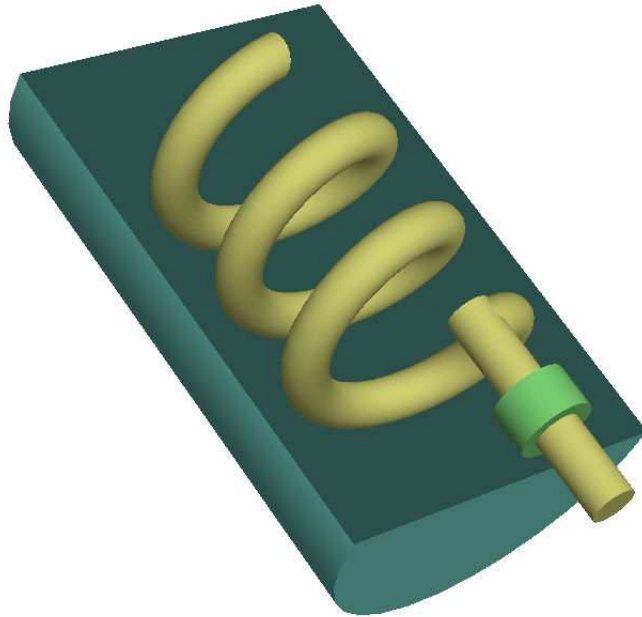
Figure 2: Three-dimensional view of the `WALKTHROUGH` assembly. Part clipping is applied so that the conductive medium is shown only in the region $x \leq 0.0$.

- DUNIT. The menu lists dimension choices. The dialog sets a factor to convert coordinate units used in **MetaMesh** to meters. The value is the number of mesh units per meter: 39.37 for inches, 100.0 for cm.

- RESTARGET. Accuracy tolerance for the iterative matrix solution of the finite-element equations.

- MAXCYCLE. Maximum number of cycles in the iterative solution.

- OMEGA. A parameter in the range 0.0 to 2.0 to control the iterative matrix solution.

- BOUNDARY and SUPERPOSITION. Advanced program capabilities described in Sect. 4.3.

The column options in the region grid box determine the material properties of the regions. An entry in the *Potential* column specifies that the region is a fixed-potential electrode in both solution types. The second column is the phase of the potential in degrees. If no value appears, the default is 0.0°. Entries in the third and/or fourth columns designate a material region. Enter the relative dielectric constant ϵ_r in the *EpsiR* column and the electrical conductivity σ in the *Sigma* column. The default values are 1.0 and 0.0 S/m. **RFE3** issues an error messages if material properties are applied to a fixed-potential region and vice versa. The values entered in Fig. 3 define a solution with the following region characteristics:
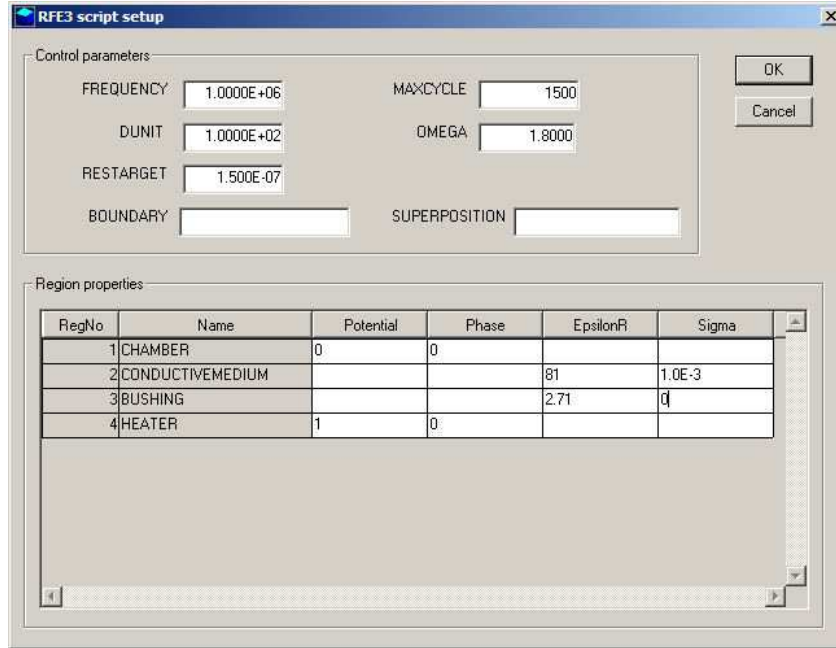
7

Figure 3: Dialog to set **RFE3** control parameters and region properties.

- Region 1. Fixed potential (boundary): $\phi = [0.0, 0.0]$ V.

- Region 2. Imperfect dielectric (fill medium): $\epsilon_r = 81.0$ and $\sigma = 1.0 \times 10^{-3}$ S/m.

- Region 3. Dielectric (bushing): $\epsilon_r = 2.71$ and $\sigma = 0.0$ S/m.

- Region 4. Fixed potential (coil): $\phi = [1.0, 0.0]$ V.

Set up the dialog with the values shown and click *OK*. **RFE3** uses the information in the dialog to create the script `WALKTHROUGH.RIN` shown in Table 2. Chapter 4 reviews the script format and advanced program capabilities.

We shall first perform a low-frequency calculation to compare results to theoretical predictions. The activity also shows how to make direct changes in the control script. Click on the File/Edit script file command and choose `WALKTHROUGH.RIN`. In the editor, make the following change to the script command:

```
* Freq =   1.0000E+06
Freq = 0.1
```

The asterisk at the beginning of the first line denotes a comment. Therefore, we shall perform the initial calculation at 0.1 Hz. At low frequency, the real current density is much higher than the displacement current density. The ratio of current densities is given by:

$$\frac{j_r}{j_d} = \frac{\sigma}{2\pi f \epsilon_r \epsilon_o}. \tag{1}$$

For $\sigma = 0.001$ S/m, $\epsilon_r = 81.0$ and $f = 0.1$ Hz, the predicted ratio is $j_r/j_d = 2.22 \times 10^6$.

Choose *Run* in the main **RFE3** menu and pick `WALKTHRU.RIN`. The program uses information in `WALKTHRU.MDF` and `WALKTHRU.RIN` to create finite-element equations appropriate for the

Table 2: **RFE3** script `WALKTHROUGH.RIN`

```
Mesh = walkthrough
Freq =   1.0000E+06
DUnit =   1.0000E+02
ResTarget =   1.5000E-07
Omega =   1.8000E+00
MaxCycle =    1500

* Region 1: CHAMBER
Potential(1) =   0.0000E+00  (     0.000)

* Region 2: CONDUCTIVEMEDIUM
Epsi(2) =   8.1000E+01
Sigma(2) =   1.0000E-03

* Region 3: BUSHING
Epsi(3) =   2.7100E+00

* Region 4: HEATER
Potential(4) =   1.0000E+00  (     0.000)

EndFile
```

conformal mesh and material properties and then solves the equations. The complete procedure takes less than a minute. **RFE3** creates the file `WALKTHRU.ROU` which contains information on the mesh geometry and the electrostatic potential at nodes.

Next run **RFE3View**. Before proceeding to plots, we shall check some numeric results. Click the *Run script* command in the *File* menu and choose the file `WALKTHROUGH.SCR`. Table 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* menu and open `WALKTHROUGH.DAT`. The first section shows results from the global analysis. The time-averaged power dissipation is 6.249×10^{-4} W for an applied voltage of amplitude $V_0 = 1.0$ V. From the equation

$$\overline{P} = \frac{V_0^2}{2R}, \tag{2}$$

we infer a resistance of 769.3 Ω . The surface integrals of current over 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 (2.22×10^6) agrees with the prediction of Eq. 1. The value of I_r implies a resistance of 785.5 Ω (a difference of 2.0% 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

9

Table 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.DAT
VOLUMEINT
RFCURRENT 1
RFCURRENT 4

ENDFILE
```

results illustrate 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 *File operations/Load configuration file* and then choose `rfe3view_reference.cfg`. With this configuration, you can display a snapshot of RF quantities at a chosen reference phase. Click on *Slice plot*. In the slice plot menu, choose *Slice normal to Y* in the *Change view* menu. In the *Plot control* menu, click on *Plot style* and choose *Filled contour*. Under *Plot quantities* in the *Plot control* menu, pick *JcMagRef*. Finally, click on Vector tools/Vector arrow plot. You should see a plot similar to that of Fig. 4. The figure shows variations of the total conductive current at the default phase $\phi = 0.0^{\circ}$ in the plane $y = -0.10$. As expected, the current density concentrates on the outside of the helical electrode and equals zero inside the bushing.

To complete the analysis, edit `WALKTHROUGH.RIN` to change the frequency to $f = 1.0 MHz$ and change the conductivity for Region 2 from $\sigma = 0.001$ S/m to 0.0 S/m. Save the file, exit the editor and rerun the solution. Return to **RFE3View** and reload the configuration file `rfe3view_standard.cfg`. Click on the *Analysis/Volume integrals* command. The global peak field energy is $U = 4.509 \times 10^{-10}$ J. Using the formula $U = CV^2/2$ (where V is the amplitude of the applied voltage), the predicted total capacitance is 9.02×10^{-10} F. The predicted amplitude of the displacement current at $f = 1.0$ MHz is $I_{d0} = 2\pi f C V_0 = 5.660 \times 10^{-3}$ A. For comparison, the surface integral of displacement current density over the chamber wall gives 5.756×10^{-3} A.
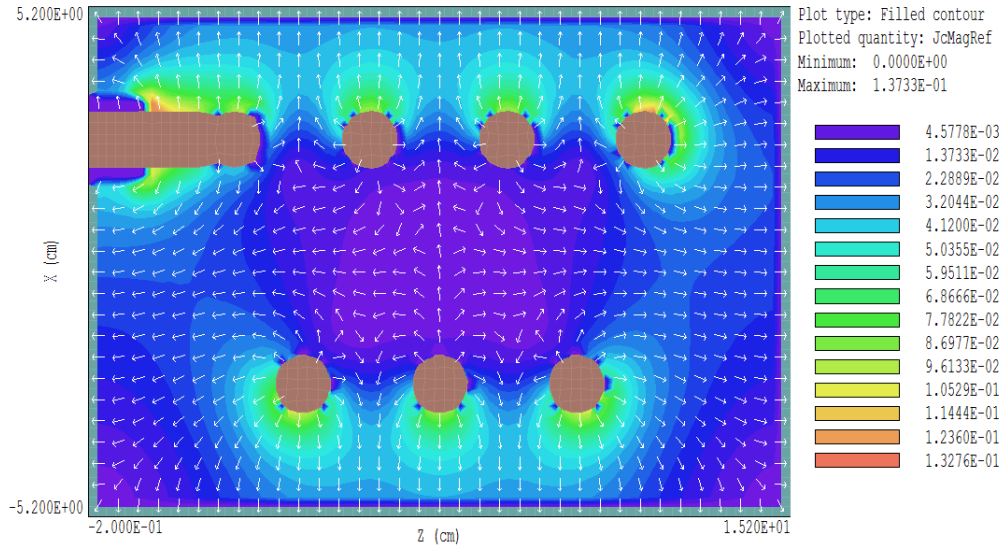
Figure 4: Filled-contour plot of the total conductive current density at the reference phase 0.0º in the plane $y = 0.1$.

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 5 illustrates some 3D capabilities. The plot shows a cutaway view of the chamber and electrode with values of $|\mathbf{E}|$. 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.
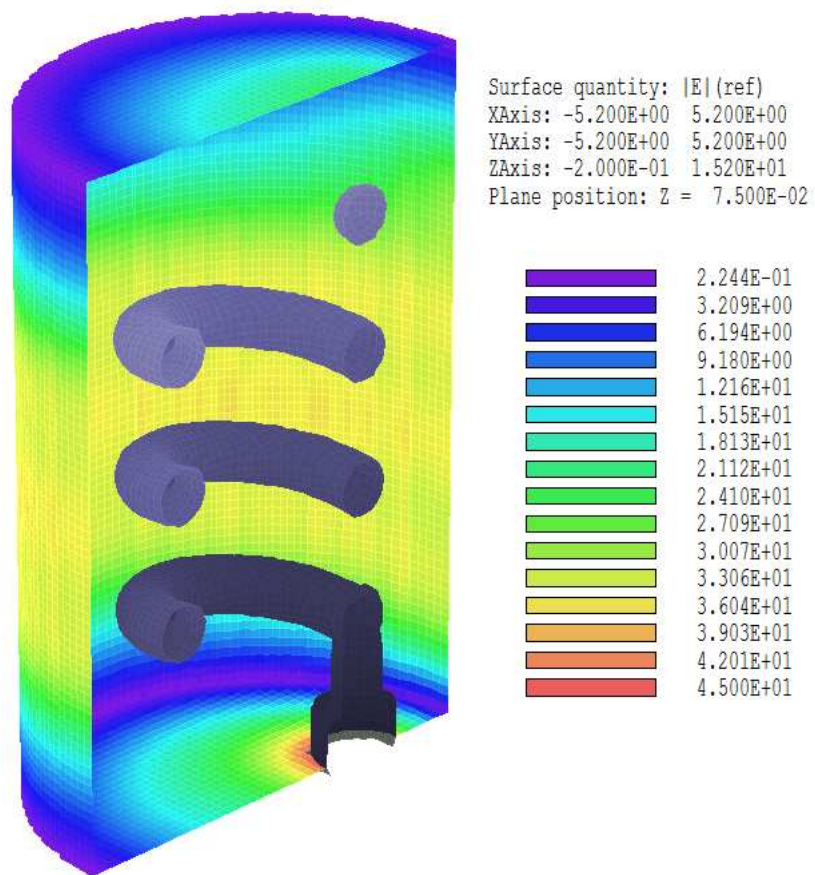
Figure 5: Three-dimensional cutaway view of the chamber and helical electrodes along with a calculated values of $|\mathbf{E}|$ on the chamber wall.

2 Theoretical background

2.1 Electric fields in conductive media

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

$$\mathbf{E} = \nabla\Phi. \tag{3}$$

Equation 3 also holds for RF electric fields at moderate frequency. In this case, time-varying magnetic fields make a small contribution to the electric field, so that

$$\nabla \times \mathbf{E} \cong 0. \tag{4}$$

Equation 4 is equivalent to Eq. 3. If the electric field varies as

$$\mathbf{E}(x, y, z, t) = \mathbf{E}_0(x, y, z)\cos(\omega t) = \mathbf{E}_0(x, y, z)\ Re[\exp(i\omega t)], \tag{5}$$

then Eq. 3 is valid over scale lengths of order

$$L \ll \frac{1}{\omega\sqrt{\epsilon\mu_0}}. \tag{6}$$

In Eq. 6, ϵ is the dielectric constant of the material. As an example, taking $\epsilon = 10\epsilon_0$ and $L = 0.1$ m, the quasi-static approximation is valid for RF frequencies in the range $f \ll 9.5 \times 10^8$ Hz or $f \ll 151$ MHZ.

For a system of ideal dielectrics (no conductivity), we can combine Eq. 3 with the divergence equation,

$$\nabla \cdot (\epsilon\mathbf{E}) = 0, \tag{7}$$

to derive the Laplace equation,

$$\nabla \cdot (\epsilon\nabla\Phi) = 0. \tag{8}$$

Equation 8 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,

$$\nabla \cdot \mathbf{j} = 0. \tag{9}$$

The current density is related to the electric field by

$$\mathbf{j} = \sigma\mathbf{E}, \tag{10}$$

where σ is the electrical conductivity. Combining Eqs. 3, 9 and 10 gives an alternate form of the Laplace equation

$$\nabla \cdot (\sigma \nabla \Phi) = 0. \tag{11}$$

Note that Eq. 11 has the same form as Eq. 8 if we substitute σ for ϵ. with σ.

We can consolidate the developments to describe low-frequency RF 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 in a region that satisfies Eq. 6 and assume that the time dependence of the electric field (and all other time-varying quantities) follows the form of Eq. 5. In the equation, the quantity $\mathbf{E}_0(x, y, z)$ 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. Taking $\mathbf{j}_r$ 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

$$\frac{\partial \rho_r}{\partial t} = i\omega \rho_r = -\nabla \cdot \mathbf{j}_r. \tag{12}$$

The divergence equation assumes the form

$$\nabla \cdot (\epsilon \mathbf{E}) = \rho_i = i \frac{\nabla \cdot \mathbf{j}_r}{\omega} = \nabla \cdot \left(\frac{i\sigma \mathbf{E}}{\omega} \right). \tag{13}$$

The expression on the right-hand side proceeds from the relation $\mathbf{j}_r = \sigma \mathbf{E}$. Equation 13 can be written as

$$\nabla \cdot \left(\epsilon - \frac{i\sigma}{\omega} \right) \mathbf{E} = 0, \tag{14}$$

or

$$\nabla \cdot \left[\left(\epsilon - \frac{i\sigma}{\omega} \right) \nabla \Phi \right] = 0. \tag{15}$$

Equation 15 is identical to Eq. 8 if we interpret the dielectric constant as a complex number,

$$\epsilon \Rightarrow \left(\epsilon - \frac{i\sigma}{\omega} \right). \tag{16}$$

Equation 15 can be solved with standard solution methods by simply replacing the potential and dielectric constant with the appropriate complex numbers.

2.2 Boundary conditions

The boundary is the outer edge of the solution volume. For RF electric field problems, node potentials on the boundary may 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 nodes represents an electrode (equipotential volume). Electric field lines are normal to such a surface.
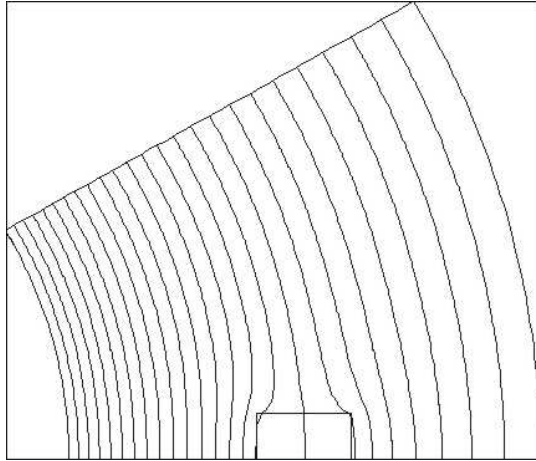
Figure 6: Special Neumann boundaries

- **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. 6 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, while 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).

3 Organizing RFE3 calculations

3.1 Procedures and files

The **RFE3** package consists of the program `rfe3.exe` that computes the physical solution and `rfe3view.exe`, a 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 (`MPrefix.MDF`) that describes the conformal hexahedron mesh. The file contains node coordinates and the region numbers of elements and nodes.

- A **RFE3** script (`FPrefix.RIN`) that sets control parameters for the solution program and describes the physical properties associated with regions.

The **RFE3** file is usually prepared with the dialog described in Sect.1.2. You can use an editor to compose a script or to add advanced features (Chap. 4).

A **RFE3** calculation generally consists of the following steps:

1. Use **Geometer** or an editor to prepare a **MetaMesh** script with a name of the form `MPREFIX.MIN` that defines the solution space and describes the division into spatial regions.

2. Run **MetaMesh** to create a file `MPREFIX.MDF` of standard mesh information. This file could be used as input for multiple **RFE3** simulations or for other **AMaze** programs.

3. Use the *Setup* command of **RFE3** to prepare a script (`FPREFIX.RIN`) that sets control parameters and defines the material properties of regions.

4. Use the **RFE3** *Process* command to create a solution file `FPREFIX.ROU`. This file contains information on node spatial coordinates and complex potential.

5. (Optional) Prepare a standard **AMaze** analysis script (`SPREFIX.SCR`) to control an **RFE3View** analysis session.

6. Use **RFE3View** to create plots or to generate numerical data using the information in `FPREFIX.ROU`. The analysis may be performed interactively or automatically under the control of a script.

Table 4 summarizes the input and output file used in **RFE3**.

Table 4: **RFE3** file types

Name form	Function	Status
MPREFIX.MIN	Simulation geometry, input to **MetaMesh**	Required
MPREFIX.MDF	Output from **MetaMesh**, input to **RFE3**	Required
FPREFIX.RIN	Material properties, input to **RFE3**	Required
FPREFIX.ROU	Output from **RFE3**, input to **RFE3View**	Required
CPREFIX.CFG	Configuration file for **RFE3View**	Required
SPREFIX.SCR	Analysis control, input to **RFE3View**	Optional
PREFIX.DAT	Analysis data output, from **RFE3View**	Optional

3.2   Script conventions

The **RFE3** input script is a text file with data lines containing commands and parameters. The script must end with the *EndFile* command. The programs make no distinction between upper and lower case. Entries on a line may be separated by the following delimiters:

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

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

4 Script commands

4.1 Program control

This chapter reviews the full set of commands that may be included in **RFE3** scripts. The first group, discussed in this section, controls program operation. Each command is shown symbolically and in a form that it might assume in a program.

MESH MPrefix
MESH = TriggerConfig
The prefix of the **MetaMesh** output file (`MPrefix.MDF`) to use for the definition of the solution geometry. If the command does not appear in a script `FPrefix.RIN`, **RFE3** searches for the default file `FPrefix.MDF`.

FREQ Freq
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.

DUNIT Unit
DUNIT = MM
DUNIT = 100.0
Use this command to set the units used in **MetaMesh** to create the input mesh. The *Unit* argument may be one of the following strings for common units: `angstrom`, `nanometer`, `micrometer`, `mil`, `mm`, `cm`, `inch`, `foot`, `yard`, `meter`, `kilometer` or `mile`. For custom units, enter a real number equal to the number of **MetaMesh** units per meter. For example, if you used dimensions of centimeters in the **MetaMesh** script, set $DUnit = 100.0$. The quantity $DUnit$ is recorded in the output file and is used in **E** for the input and output of positions. Default: $DUnit = 1.0$.

RESTARGET ResTarget
RESTARGET = 5.0E-6
The numerical calculation of RF 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 the complex-number potential 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* command. Default value: $ResTarget = 5.0 \times 10^{-7}$.

MAXCYCLE MaxCycle
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: $MaxCycle = 2500$.

OMEGA Omega
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 run time. If this command does not appear, **RFE3** picks values that vary with the iteration cycle following the Chebyshev prescription. With this command, you can set a value manually. The value must be between 0.0 and 2.0 for numerical stability. Higher values generally give faster convergence.

PARALLEL [NProc]
PARALLEL = 4

This command invokes multithread support for the 64-bit program running on a multi-core machine. When the command appears with no value of *NProc*, **RFE3** uses the full resources of the computer during the matrix inversion, giving a substantial reduction of the run time. Set a value of *NProc* to assign a specific number of threads. If the command does not appear, the operating system assigns the program to a single thread, leaving the others free for other tasks. Do not employ parallel processing if you are running multiple instances of **RFE3**. The command has no effect on the 32-bit program version.

The remaining control commands serve specialized functions and are used infrequently.

NCHECK NCheck
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: $NCheck = 50$.

FORMAT [Text, Binary]
Format = Text

By default, **RFE3** creates output files FPREFIX.ROU in binary format. This format preserves full numerical accuracy and minimizes the file size. Use text format if you want to port results to your own analysis programs. **PhiView** can read either format. The string parameter may be either *Binary* or *Text*.

INITVAL FPrefix
INITVAL = DistortionSwitch
In some circumstances, you can reduce the solution time for large meshes by using precomputed initial values of complex-number potential. In normal operation, **RFE3** initializes value of electrostatic potential to $\phi = [0.0, 0.0]$ at all nodes that do not have a fixed potential. When this command appears, the program sets initial values at variable nodes equal to those in the file `FPrefix.ROU`. One application for the command is to run additional relaxation cycles to improve the accuracy of an existing solution. You can also use the command if you make small changes to applied potentials or the geometry of electrodes or dielectrics. The solution file `FPrefix.ROU` should be available in the working directory and must have the same foundation mesh as the input file `MPrefix.MDF`. A necessary condition is that the values of I_{max}, J_{max}, K_{max}, x_{min}, x_{max}, y_{min}, y_{max}, z_{min} and z_{max} must match. Within this limitation, there may be small geometric differences between objects in the two files. If there are large differences, the procedure may not significantly reduce the solution time.

4.2 Basic material properties

Material properties in RF electric field solutions are relatively simple so only a few commands are required. The basic commands of this section set properties that are uniform throughout a region. Chapter 5 describes how to define material properties that vary according to a mathematical expression over the region volume.

POTENTIAL RegNo A [Phi]
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:

$$V(t) = A\cos(2\pi f t + \phi). \tag{17}$$

The fixed-potential condition may 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 $\epsilon_r = \epsilon/epsilon_0$ (dimensionless). To model a floating electrode (conducting object whose potential is not previously know), assign a relatively large value of dielectric constant (*i.e.*, 10^4) to the region. Note that extremely high values of ϵ_r may slow convergence. The default for all unspecified regions is $\epsilon_r = 1.0$.
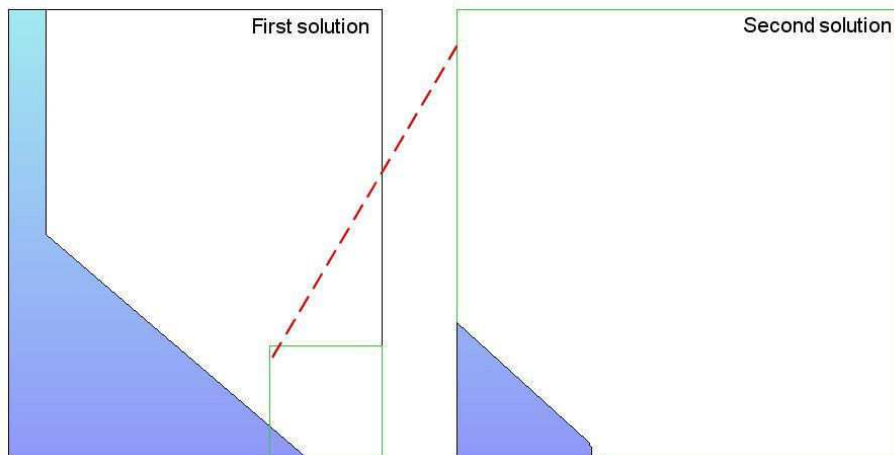
Figure 7: Creating a microscopic solution using the *Boundary* command.

SIGMA RegNo Cond
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 S/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 *Epsil* and *Sigma* commands may be used to define an imperfect dielectric.

4.3 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 precise 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 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 8 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 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 script of **Solution 2**:
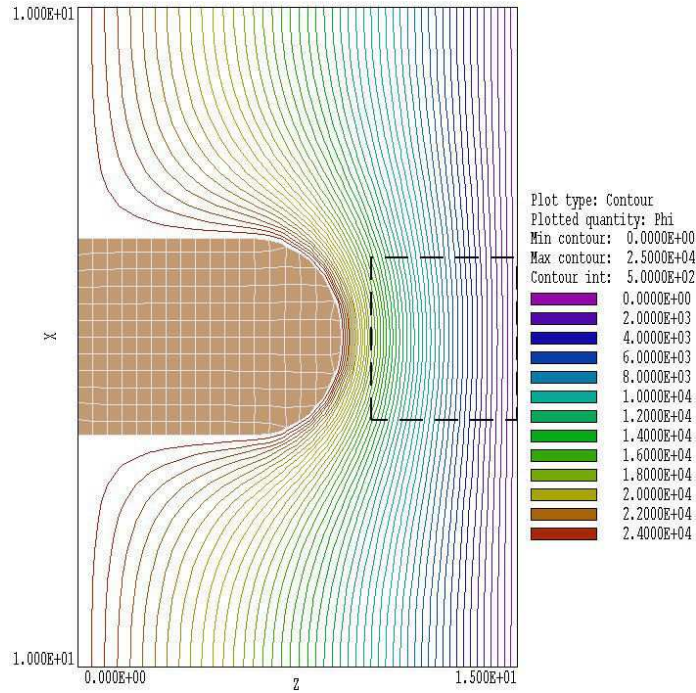
Figure 8: Equipotential lines and element resolution in the large-scale solution `BOUNDTEST01`. The dashed line shows the boundary of the small-scale solution.

BOUNDARY FPrefix [BScale] [BStatus]
BOUNDARY FEmitMacro 1.0 101111
The string quantity *FPrefix* is the prefix of the output file for **Solution 1**. The real-number quantity *BndScale* is an optional scaling factor applied to potential values transferred from **Solution 1** to **Solution 2**. The default is *BndScale* = 1.0. The optional parameter *BStatus* is a string with six characters, either '0' or '1'. It can be used to control whether a specific boundary is assigned the computed Dirichlet condition. The order of boundaries in the string is XDn, XUp, YDn, YUp, ZDn and ZUp. An entry of 1' indicates that the Dirichlet condition should be applied. The default is *BStatus* = '111111'.

In response to the *Boundary* command, **RFE3** loads **Solution 1** into memory and sets up the apparatus for making second-order interpolations of potential in the solution. Next, 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** 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**.

The example library contains files to illustrate the process. The file set `BOUNDTEST01.MIN` and `BOUNDTEST01.RIN` describes a large-scale solution, while `BOUNDTEST02.MIN` and `BOUNDTEST02.RIN` describe a microscopic solution, a subset of the first. Figure 9 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
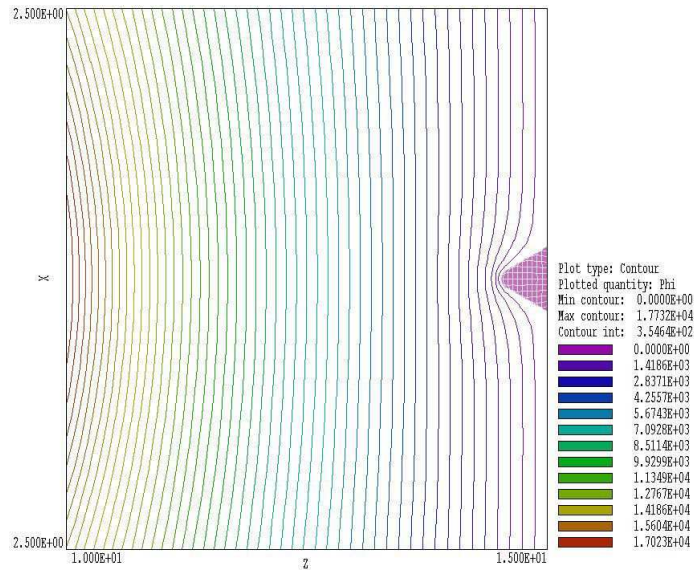
Figure 9: Equipotential lines and element resolution in the small-scale solution `BOUNDTEST02`.

a small protrusion on the plate. The protrusion is smaller than the scale size off 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. 9). The element scale size is 0.25" in the bulk of the solution and 0.05" in the region of the protrusion. The script `BOUNDTEST02.RIN` has the following content:

```
* File BOUNDARYTEST02.RIN
*   NReg      RegName
*   ==============================
*    1        VACUUM
*    2        PROTRUSION
FREQ = 0.10
RESTARGET = 5.0E-8
MAXCYCLE = 2500
DUNIT = 39.37
NCHECK = 10
BOUNDARY BOUNDARYTEST01
EPSI(1) = 1.0
POTENTIAL(2) = 0.0
ENDFILE
```

Figure 9 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** adds values from one solution (**Solution 1**) to another (**Solution 2**). The following statement may appear in the command script for **Solution 2**.

SUPERPOSITION FPrefix [SScale]
SUPERPOSITION UniField
The string *FPrefix* is the prefix of the output file for **Solution 1**. The restriction is that
Solution 1 must fit inside **Solution 2**. The real quantity *SScale* is a scaling factor applied to
potential values transferred from **Solution 1** to **Solution 2**. The default is $SuperScale = 1.0$.

If the *Superposition* commands appears in the script, **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
values according to

$$\phi_2' = \phi_2 + SScale \times \phi_1, \tag{18}$$

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.

5 Spatial variations of material quantities from mathematical functions

5.1 Program capabilities

Section 4.2 covered commands to define material properties (ϵ_r and/or σ) that are uniform throughout a region. This section describes how to represent quantities that vary continuously in space following a mathematical prescription. It is straightforward to assign variable region properties with following commands:

EPSI RegNo > Function
EPSI(7) > 1.0 + ($x - 0.25)/5.50
EPSI(4) > 1.0 - 0.625*$z∧2
Assign values of the relative dielectric constant ϵ_r to elements of the region according to a specified spatial variation. The spatial function is evaluated at the center of mass of each element. **RFE3** issues an error message if the value of the function in any element gives $\epsilon_r \leq 0.0$.

SIGMA RegNo > Function
SIGMA(7) > 0.5 + 2.0*(1.0 - cos(3.14156*$x/15.0))
SIGMA(4) > 100.0 - 50*exp(($z/10)∧2)
Assign values of electrical conductivity to elements of the region according to a specified spatial variation. The spatial function is evaluated at the center of mass of each element and should return values in units of S/m. **RFE3** issues an error message if the value of the function in any element is $\sigma < 0.0$.

5.2 Function syntax

RFE3 features a flexible and robust algebraic function interpreter. A function is a string (up to 230 characters) that may include the following entities:

- Spatial variables, `$x`, `$y` or `$z`.

- Real and/or integer numbers in any valid format (*e.g.*, `3.1415`, `476`, `1.367E23`, `6.25E-02`, `8.92E+04`,...). Integers are converted to real numbers for evaluation.

- Binary operations: `+` (addition), `-` (subtraction), `*` (multiplication), `/` (division) and `∧` (exponentiation).

- Functions: `abs` (absolute value), `sin` (sine), `cos` (cosine), `tan` (tangent), `ln` (normal logarithm), `log` (base 10 logorithm), `exp` (normal exponent) and `sqt` (square root).

- Up to 20 sets of parentheses to any depth.

- Any number of space delimiters.

The parser conforms to the standard algebraic rules and features comprehensive error checking. Errors may include unbalanced parentheses, unrecognized characters and sequential binary operations. To illustrate a valid example, the expression

```
1 - exp(-1.0*(($x^2 + $y^2 + $z^2)/24))
```

corresponds to

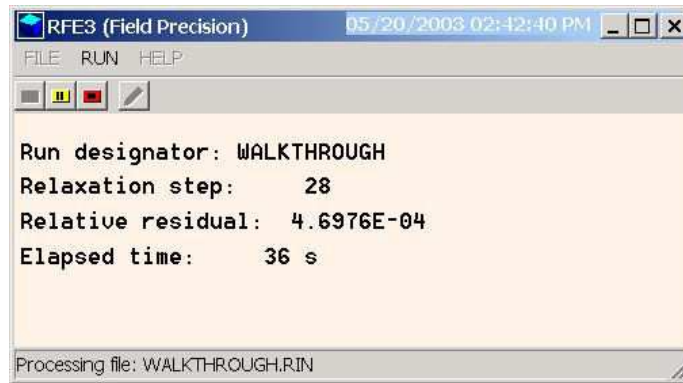$$1 - \exp\left[-\left(\frac{x^2 + y^2 + z^2}{24}\right)\right].$$ (19)

Figure 10: **RFE3** display during a solution.

6 Running the RFE3 program

6.1 Interactive operation

The program `rfe3.exe` can 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 you own shortcuts. Figure 10 shows the program window.

The program menu has four main commands: *File*, *Setup*, *Run* and *Help*. The following commands appear in the *File* popup menu.

EDIT SCRIPT
EDIT LISTING FILE
EDIT FILE
The commands call up the internal editor to inspect or to modify text input and output files for the solution program. With the *Edit script* 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. Note that the main program will be inactive until you exit the editor.

SETUP
In response to this command, **RFE3** prompts for the name of a **MetaMesh** file and brings up the dialog of Fig. 3. Fill in values for control parameters and material properties to create a basic script. You can use an editor to modify the script or to add the advanced functions described in Chap. 4.

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 requested **MetaMesh** file `MPREFIX.MDF` is available in the working directory. **RFE3** displays information in the window to show the progress of program operations (Fig. 10).

PAUSE RUN

The intensive calculations of **RFE3** make demands on the resources of your computer, possibly 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. Note that if you have a dual-processor machine, the Windows operating system will shunt tasks so that you have approximately the power of one processor when **RFE3** is running. To use the full power of the machine, you can launch two instances of **RFE3** to perform two independent calculations.

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 includes the following commands.

RFE3 MANUAL

The command displays this manual using your default PDF viewer. The file `rfe3.pdf` must be in the same directory as `rfe3.exe`.

PHYSICAL CONSTANTS

Display a table of useful physical constants. The file `physcons.pdf` must be in the same directory as `rfe3.exe`.

6.2   Automatic runs under batch file control

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. This feature is particularly attractive on dual-processor machines.

To run a single **RFE3** calculation 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 creates the output file `SWITCH.ROU` in the data directory.

The main function of the command mode is autonomous operation under batch file control. As an example, suppose 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 `SWRUN.BAT` in the data directory using a text editor:

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

Type

```
SWRUN <Enter>
```

to generate all solutions without the need for further keyboard input. The option `/WAIT` ensures that one process finishes before the next begins.

7 RFE3View: calculated quantities

RFE3 uses complex number procedures to find steady-state solutions for RF electric fields. Here, all quantities vary harmonically at frequency f over the solution volume. A general expression for the variations of potential is

$$V(x, y, z, t) = V_0(x, y, z) \, \cos\left[\omega t + \phi(x, y, z)\right], \tag{20}$$

where V_0 is the amplitude, ϕ is the phase and the angular frequency is $\omega = 2\pi f$. Equation 20 can be represented in the complex form

$$V(x, y, z, t) = \mathrm{Re}\left\{V_0(x, y, z) \, \exp\left[j\phi(x, y, z)\right] \, \exp\left[j\omega t\right]\right\}. \tag{21}$$

We can identify the spatially varying part of Eq. 21 as the *phasor*

$$\overline{V} = V_0 \, \exp\left(j\phi\right). \tag{22}$$

The variation of potential in space and time can be written as

$$V(x, y, z, t) = \mathrm{Re}\left\{\overline{V}(x, y, z) \, \exp(j\omega t)\right\}. \tag{23}$$

The complex-value phasor carries complete information on the spatial variation of the potential amplitude and phase. RFE3 records the real and imaginary parts of the phasor in the solution output file `FPREFIX.ROU`:

$$\begin{aligned}
\overline{V}_r(x, y, z) &= V_0(x, y, z) \, \cos[\phi(z, y, z)], \\
\overline{V}_i(x, y, z) &= V_0(x, y, z) \, \sin[\phi(x, y, z)].
\end{aligned} \tag{24}$$

This chapter discusses how **RFE3View** uses phasor to determine physical quantities.

The main quantities of interest are the potential, electric field, real current density and displacement current density. We can write the time variation of potential in terms of the real and imaginary parts of its phasor:

$$V(x, y, z, t) = Re\left\{(\overline{V}_r(x, y, z) + j\overline{V}_i(x, y, z)) \exp(j\omega t)\right\}. \tag{25}$$

The x component of the harmonically-varying electric field follows from the gradient of Eq. 24:

$$E_x = -\frac{\partial V}{\partial x} = Re\left\{\left(\frac{\partial \overline{V}_r}{\partial x} + j\frac{\partial \overline{V}_i}{\partial x}\right)) \exp(j\omega t)\right\}. \tag{26}$$

Following Eq. 25, the electric-field components in phasor form are

$$\begin{aligned}
\overline{E_{xr}} &= \frac{\partial \overline{V}_r}{\partial x}, \quad \overline{E_{xi}} = \frac{\partial \overline{V}_i}{\partial x}, \\
\overline{E_{yr}} &= \frac{\partial \overline{V}_r}{\partial y}, \quad \overline{E_{yi}} = \frac{\partial \overline{V}_i}{\partial y}, \\
\overline{E_{zr}} &= \frac{\partial \overline{V}_r}{\partial z}, \quad \overline{E_{zi}} = \frac{\partial \overline{V}_i}{\partial z}.
\end{aligned} \tag{27}$$

The conductive and displacement current densities are related to the electric field by

$$\mathbf{j_c} = \sigma \mathbf{E},$$
$$\mathbf{j_d} = \epsilon_r \epsilon_0 \, \frac{\partial \mathbf{E}}{\partial t}. \tag{28}$$

The real and imaginary parts of the phasor for the x component of the conductive current is

$$\overline{j}_{cxr} = \sigma \, \overline{E_{xr}},$$
$$\overline{j}_{cxi} = \sigma \, \overline{E_{xi}}. \tag{29}$$

The product of the electric field phasor times $j\epsilon_r\epsilon_0\omega$ gives the time derivative of electric field. Therefore, the real and imaginary parts of the phasor for the x component of the conductive current is

$$\overline{j}_{dxr} = -\epsilon_r\epsilon_0\omega\overline{E_{xi}},$$
$$\overline{j}_{dxi} = \epsilon_r\epsilon_0\omega\overline{E_{xr}}. \tag{30}$$

The phasor for the x component of total current is

$$\overline{j}_{txr} = \sigma \, \overline{E_{xr}} - \epsilon_r\epsilon_0\omega\overline{E_{xi}},$$
$$\overline{j}_{txi} = \sigma \, \overline{E_{xi}} + \epsilon_r\epsilon_0\omega\overline{E_{xr}}. \tag{31}$$

We shall denote the amplitude (maximum value in time) of a harmonic quantity Q as $Qamp$. It is given in terms of the phasor components by

$$Qamp = \sqrt{\overline{Q_r}^2 + \overline{Q_i}^2}. \tag{32}$$

The phase of Q is

$$Qphase = \tan^{-1}\left(\frac{\overline{Q_r}}{\overline{Q_i}}\right). \tag{33}$$

The magnitude of a vector quantity (in space) is the total vector length derived from the components. As an example, the amplitude of the electric-field magnitude is given in terms of phasor components by

$$|Eamp| = \sqrt{\mathbf{E} \cdot \mathbf{E}^*} = \sqrt{\overline{E_{xr}}^2 + \overline{E_{xr}}^2 + \overline{E_{xr}}^2 + \overline{E_{xr}}^2 + \overline{E_{xr}}^2 + \overline{E_{xr}}^2}. \tag{34}$$

Sometimes we may want to display a snapshot of the spatial variation of Q at a particular reference phase $\omega t_0 = \phi_0$. Applying Eq. 20, the calculated quantity is

$$Q(x, y, z, t_0) = Q_0(x, y, z) \, \cos(\phi_0 + \phi). \tag{35}$$

We can rewrite Eq. 35 as

$$Q = Q_0 \ (\cos \phi_0 \cos \phi - \sin \phi_0 \sin \phi). \tag{36}$$

Comparison to Eqs. 24 leads to the relation:

$$Q(x, y, z, t_0) = \overline{Q_r} \ \cos \phi_0 - \overline{Q_i} \ \sin \phi_0. \tag{37}$$

8   RFE3View – file operations and plane plots

8.1   File operations

The function of the **RFE3View** post-processor is create plots and to calculate numerical quantities from **RFE3** binary solution files. The program has the following popup menus: *File operations*, *Analysis*, *Plane plots*, *Slice plots*, *Surface plots* and *Help*. Initially, only the *File operations* 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
RFE3View displays a dialog with a list of solution files with names of the form `FPREFIX.ROU`. Changing the 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 shows a message box with information on the currently-loaded data file.

LOAD CONFIGURATION FILE
Configuration files determine the plot and analysis quantities of **RFE3View**. The files have names of the form `FPREFIX.CFG`. Four files are supplied with the program:

- `rfe3view_standard.cfg` – standard analysis, amplitudes of RF quantities.

- `rfe3view_reference.cfg` – snapshot of RF quantities at a user-defined reference phase.

- `rfe3view_phase.cfg` – plots and calculations of the amplitude and phase of the RF potential and electric-field components.

- `rfe3view_complex.cfg` – direct display of the real and imaginary parts of the phasors of RF quantities.

Chapter 12 describes how to create your own custom configurations.

RUN SCRIPT
Sometimes you may want to perform complex or repetitive analyses on a set of similar solutions. Analysis script operation is a powerful feature of **RFE3View**. This command displays a dialog with a list of analysis scripts (suffix `SCR`) that you have created. Pick a file and click *OK*. Changing directories in the dialog changes the working directory of the program. The analysis script can load data files, open and close history files, and perform any of the numerical functions described in this manual. Section 9.2 reviews the analysis script language.

CREATE SCRIPT

Use this command to create scripts using the internal editor. A 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 the reference list of allowed commands shown in Table 5. Enter commands in the space above *EndFile*. 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 text data file has a name of the form FPREFIX.DAT and will be stored in the working directory. You can use an editor to view the file or to extract information for mathematical analysis programs or spreadsheets.

CLOSE DATA 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

View or modify files with names of the form FPREFIX.DAT.

EDIT FILE

Use the program editor to view or to modify any text file.

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

INSTRUCTION MANUAL

Displays this document in your default PDF viewer. The file rfe3.pdf must be in the same directory as rfe3view.exe.

Table 5: Create script - default file content

```
* Analysis script

* Insert commands here...

ENDFILE

      --- Command summary ----

INPUT InFileName
 [Close current solution file and load FileName]
OUTPUT OutFileName
 [Close current data file and open OutFileName]
CONFIGURATION ConfigFileName
 [Load a new configuration, ConfigFileName]
POINT xp yp zp
 [Point field calculation at the given coordinates]
SCAN xp1 yp1 zp1 xp2 yp2 zp2
 [Scan along a line between the given coordinates]
GENSCAN
   xp1 yp1 zp1
   ...
   xpn ypn zpn
END
 [Write field values at listed coordinates]
VOLUMEINT [RegNo]
 [Volume integrals, optionally for region RegNo]
SURFACEINT Reg1 Reg2 ... RegN
 [Surface integrals around the region set]
MATRIX FileName XMin XMax NX YMin YMax NY ZMin ZMax NZ
 [Write a matrix of field values to the file FileName]
NSCAN  100
 [Set the number of points in a line scan]
INTERPOLATION [LSQ,Linear]
 [Set the interpolation method]
ENDFILE
 [Terminate the analysis]
```
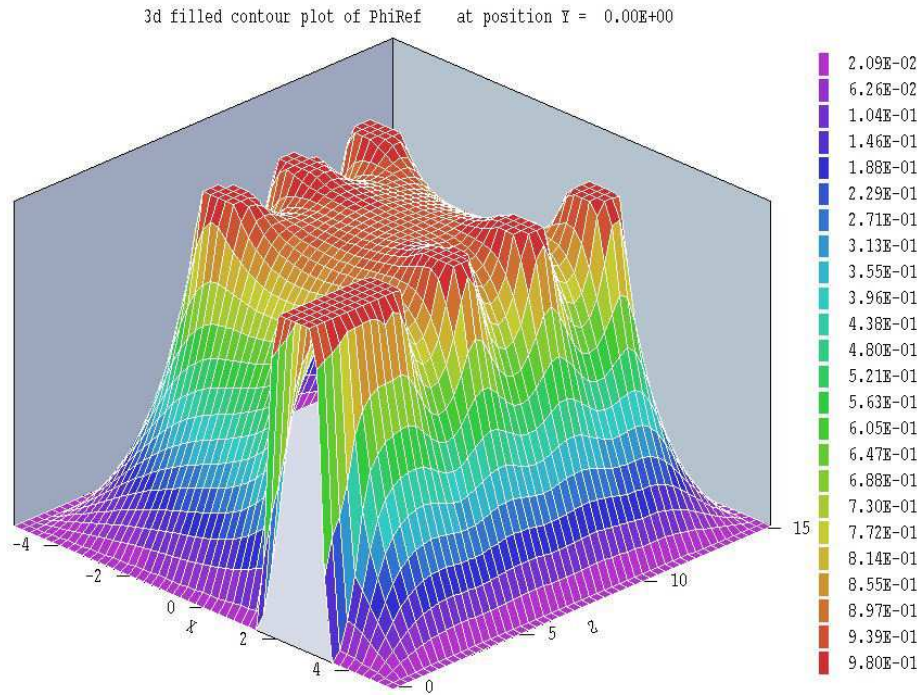
Figure 11: Illustration of a plane plot: potential at 0.0° phase for the `WALKTHROUGH` example.

8.2 Plane plots

Plane plots (Fig. 11) are two-dimensional plots that show the variation of quantities over planes that cut the solution volume. There are three differences from the slice plots discussed in the next section:

- Plane plots are created from a rectangular mesh of values while slice plots built using the computational mesh. Slice plots show precise region boundaries and equipotential lines in true scale.

- Because they are simpler to construct, plane plots support a wider variety of plot styles.

- Plane plots are not limited to slice normal to a Cartesian axis.

SET PLANE

This command brings up the dialog of Fig. 12 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 field group at the bottom-left determines 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 final command group at the bottom-right control an optional rotation of the plot plane. The radio buttons set the pivot axis, the quantity *Rotate* is the rotation angle in degrees and *Position* is the pivot position along the direction normal to the pivot axis.

37
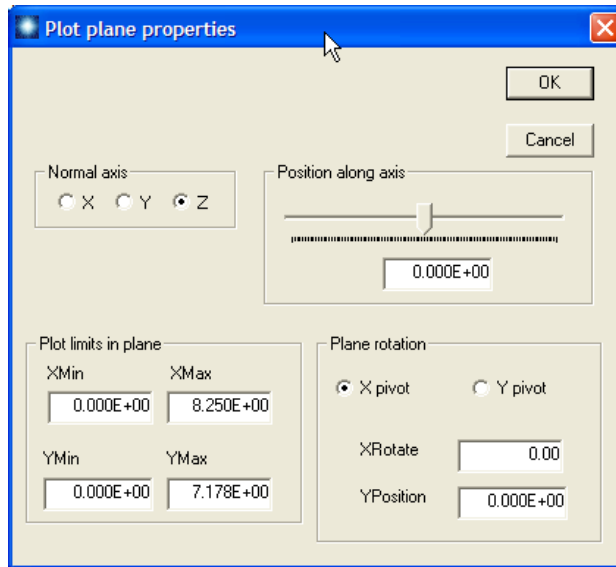
Figure 12: Set plane dialog.

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

PLOT STYLE
This command brings up the dialog of Fig. 13 to set the plot style. As an example, Fig. 11 shows the *Filled contours 3D* plot style. The numbers at the bottom give the resolution of the mesh used to create the plot. Higher values give more detail but require longer regenerate times. The default is a 100×100 mesh.

PLOT QUANTITY
Set the quantity to be plotted. the choice depends on the current configuration file. Chapter 7 reviews options for calculated quantities in a solution for RF electric fields. The file rfe3view_standard.cfg defines the amplitude (peak value in time) of the quasi-static potential ϕ_{amp}, the amplitudes of the total electric field ($|\mathbf{E}|_{amp}$), the conductive current ($|\mathbf{J}_c|_{amp}$) and displacement current ($|\mathbf{J}_d|_{amp}$), the time-averaged power density (p), the relative dielectric constant (ϵ_r) and the electrical conductivity (σ)

PLOT LIMITS
Set limits for the plotted quantity. When *Autoscale* is active, **RFE3View** automatically sets limits based on the range of values in the solution file.

ROTATE PLOT
This command is active only for the *Filled contours 3D* and *Gradient plot 3D* styles. You can rotate the plot in 90º increments for the best view.
    The following command is used to changed the reference phase for displays:
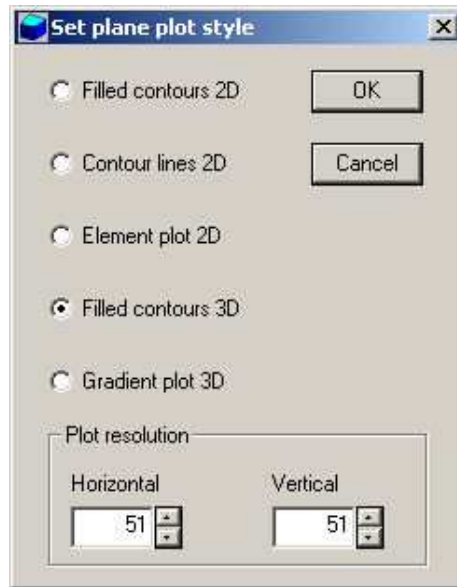
Figure 13: Plot style dialog.

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)
PLOT FILE (BMP)
PLOT FILE (PNG)
Use this command to make a plot file of the current screen plot in either encapsulated PostScript, Windows bitmap or portable network graphics formats. Supply a file prefix in the dialog box. The plot file will be created in the current directory with a name FPREFIX.EPS, FPREFIX.BMP or 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.

9   RFE3View – numerical 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.

9.1   Automatic analyses operations

POINT CALCULATION

The command brings up a dialog where you enter the x, y and z coordinates of any 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*, **REF3View** displays an information box and also writes the results to the data file in the following format:

```
   --- Point Field Analysis ---
Position: [  0.0000E+00,   0.0000E+00,   0.0000E+00]
Region number:    2
PhiAmp:    1.028953E+02
MagEAmp:   4.181282E+06
MagJrAmp:   0.000000E+00
MagJdAmp:   2.773703E-01
PDensAvg:   0.000000E+00
EpsiR:   2.710000E+00
Sigma:   0.000000E+00
```

The set of values depends on the quantities defined in the `INTERPOLATION` section of the currently-loaded configuration file.

LINE SCAN

In the *Analysis* menu, scans may be performed along arbitrary straight lines between any two points in the solution volume. When you click the *Line scan* command a dialog appears in which 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, **REF3View** creates a screen plot of a selected quantity determined by the *Set scan quantity* command. Features of the scan plot environment are discussed in Sect. 10.3. The following is an example of a listing.

```
 ------- Field scan between points -------
   XStart:   0.0000E+00   YStart:   0.0000E+00   ZStart:   0.0000E+00
    XEnd:   5.0000E+00    YEnd:   0.0000E+00    ZEnd:   0.0000E+00

 X              Y                Z                NReg   PhiAmp
  0.000000E+00  0.000000E+00   0.000000E+00      2   1.028953E+02
  3.333333E-02  0.000000E+00   0.000000E+00      2   2.454171E+03
  6.666667E-02  0.000000E+00   0.000000E+00      2   4.973895E+03
  1.000000E-01  0.000000E+00   0.000000E+00      2   7.509723E+03
     ...

        MagEAmp       MagJrAmp       MagJdAmp       PDensAvg
        4.181282E+06  0.000000E+00   2.773703E-01   0.000000E+00
        4.183350E+06  0.000000E+00   2.775075E-01   0.000000E+00
        4.190152E+06  0.000000E+00   2.779587E-01   0.000000E+00
        4.201749E+06  0.000000E+00   2.787280E-01   0.000000E+00
             ...
```

VOLUME INTEGRALS

In response to the command, **RFE3View** performs volume integrals of quantities defined in the VOLUME section of the configuration file over all elements of the mesh. The calculations are intensive, so there may be a delay for large meshes. **RFE3View** records results to the data file in the following form:

```
---------- Volume Integrals ----------
   Global volume:   6.299225E-04
   RegNo    Region volume
   ======================
       1  1.670099E-04
       2  3.646254E-04
       3  6.002472E-05
       4  1.279744E-05
       5  1.273257E-05
       6  1.273257E-05

Quantity: FEnergy
   Global value:   5.047845E-02
   RegNo    Region volume
   ======================
       2  3.990090E-02
       3  1.057755E-02

Quantity: Power
   Global value:   3.503341E+01
   RegNo    Region volume
   ======================
       3  3.503341E+01
```
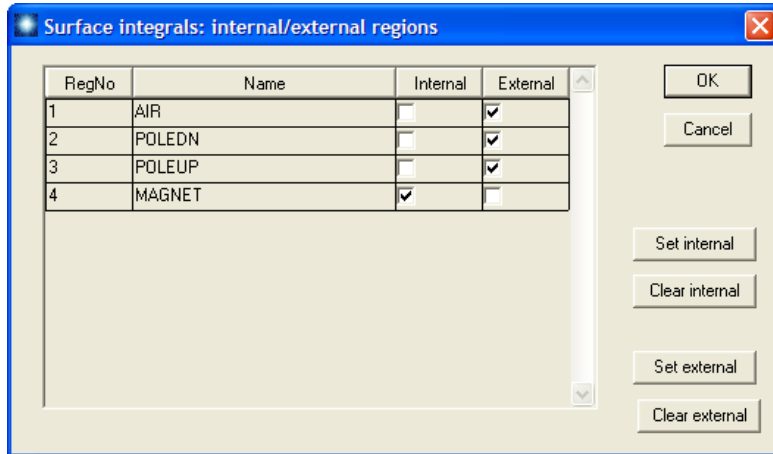
Figure 14: Dialog to pick the region set for surface integrals.

SURFACE INTEGRALS

In response to this command, **REF3View** takes integrals of defined quantities over the selected region surfaces. The program displays the dialog of Fig. 14 which includes a row for each region along with the name assigned in **MetaMesh** and forwarded by **RFE3**. Activate the check boxes in the *Internal* column to specify the regions to include in the set. The default is to include the full exterior surface of the region set (*i.e.*, all remaining regions are *External*). Use the check boxes to specify *External* regions. For example, to take a surface integral over the boundary between Regions 2 and 5, set Region 2 as *Internal* and Region 5 as *External*. The calculated quantities are defined in the SURFACE section of the configuration file. Note that there are no surface quantities defined in rfe3view_standard.cfg. Use the *RF current command* to find the total conductive and displacement current from a region set.

MATRIX FILE

This command controls a 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 much easier to use results in this form than to deal with the conformal mesh directly. The command calls up the dialog of Fig. 15. 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

$$x = x_{min} + \frac{n \left(x_{max} - x_{min} \right)}{n_x}, \tag{38}$$

where $n = 0, 1, 2, ..., n_x$. For example, if you set $x_{min} = 0.5$, $x_{max} = 1.5$ and $n_x = 10$, the calculations are performed at points with $x = 0.5, 0.6, ..., 1.4, 1.5$. Quantities are listed in the order that they appear in the INTERPOLATION section of the configuration file, up to a maximum of *NRecord* values. Change the order of quantities or *NRecord* to limit the entries. The format of the matrix file is described in Chap. 13.
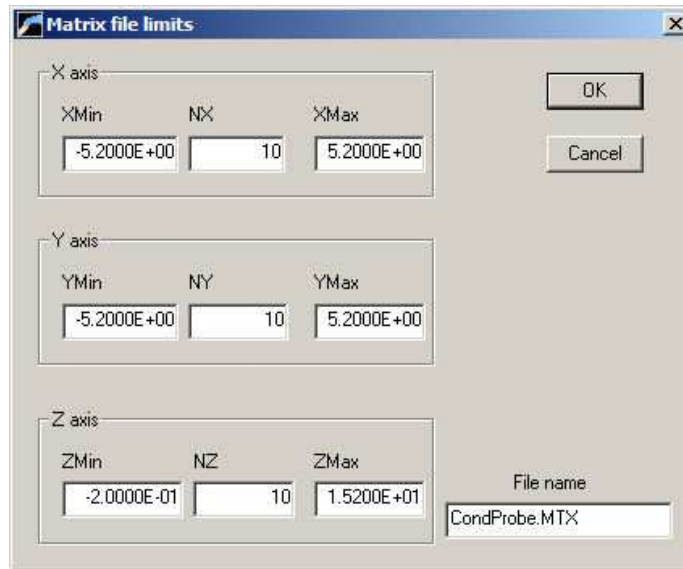
Figure 15: Create matrix file dialog.

RF POINT

This is the first of a set of three commands included for analysis of complex-number solutions. As in the simple point command, define a position in the dialog. The main difference is that **RFE3View** writes values for both the amplitude and phase of the potential and components of the electric field, conductive current density and displacement current density. The format is `Amplitude [Phase]`, where the phase is in degrees. The output is displayed in a message box and written to a data file (if open) in the following format:

```
   --- RF Point Analysis ---
Position: [  1.8626E-07,  -2.0000E-01,   2.0000E+00]
Region number:   2
EpsilonR:   0.0000E+00
Sigma:   4.4250E-05 (S/m)
Phi:   5.9634E-01 [  26.55] (V)
Ex:   3.9308E-06 [ 122.01] (V/m)
Ey:   4.7883E-06 [ -44.12] (V/m)
Ez:   5.9634E+01 [  26.55] (V/m)
Jrx:   1.7394E-10 [ 122.01] (A/m2)
Jry:   2.1188E-10 [ -44.12] (A/m2)
Jrz:   2.6388E-03 [  26.55] (A/m2)
Jdx:   0.0000E+00 [-147.99] (A/m2)
Jdy:   0.0000E+00 [  45.88] (A/m2)
Jdz:   0.0000E+00 [ 116.55] (A/m2)
```

RF SCAN

The command operate in the same way as the *Line scan* command except that field quantities are written in `Amplitude [Phase]` format. In the interactive mode, the output is written to a data file but not displayed on the screen. The data listing includes the listed quantities are

43

```
------- Field scan between points -------
   XStart:   1.8626E-07   YStart:  -2.0000E-01   ZStart:   6.0000E-01
    XEnd:    1.8626E-07    YEnd:   -2.0000E-01    ZEnd:    2.2000E+00

      X               Y               Z           NReg         Phi
=====================================================================
 1.862645E-07 -2.000000E-01  6.000000E-01      1   9.338572E-01 [   9.863]
 1.862645E-07 -2.000000E-01  6.106667E-01      1   9.329490E-01 [  10.050]
 1.862645E-07 -2.000000E-01  6.213333E-01      1   9.320508E-01 [  10.237]
 1.862645E-07 -2.000000E-01  6.320000E-01      1   9.311626E-01 [  10.425]
 1.862645E-07 -2.000000E-01  6.426667E-01      1   9.302844E-01 [  10.613]
 1.862645E-07 -2.000000E-01  6.533334E-01      1   9.294162E-01 [  10.801]
  ...

           Ex                    Ey                    Ez
    ===============================================================
    1.583684E-06 [  -1.206]  1.189428E-05 [-170.442]  2.980369E+01 [ -63.445]
    1.009640E-06 [  -9.195]  1.236187E-05 [-168.242]  2.980368E+01 [ -63.445]
    5.018101E-07 [ -35.210]  1.284634E-05 [-166.206]  2.980368E+01 [ -63.445]
    5.400537E-06 [ 160.849]  1.651028E-05 [-147.713]  2.980368E+01 [ -63.445]
    5.009105E-06 [ 157.317]  1.634658E-05 [-147.571]  2.980368E+01 [ -63.445]
    4.639878E-06 [ 153.205]  1.618298E-05 [-147.427]  2.980368E+01 [ -63.445]
      ...
```

RFCURRENT

This command performs a sum over facets between a selected set of internal and external regions to determine the current. **RFE3View** takes a vector sum of differential currents to find the amplitude and phase of the conductive, displacement and total currents. As with the *Surface integral command*, specify the internal and external regions in the dialog of Fig. 14. In the interactive mode, results are reported in a message box. Information is also written to a data file if open.

```
---------- RF Surface Integrals ----------

Region status
RegNo    Status     Name
==================================
    1   External   GROUNDSHIELD
    2   External   DIELECTRIC
    3   External   COATINGS
    4   Internal   WIRE1
    5   External   WIRE2
    6   External   WIRE3
Surface area of region set (m2):   2.053691E-03
Conductive current
  ICond(Real):   8.768053E-05 (A)   ICond(Imag):  -7.409918E-04 (A)
   ICond(Mag):   7.461613E-04 (A)   ICond(Phase):  -83.252 (deg)
Displacement current
  IDisp(Real):   6.144322E-04 (A)   IDisp(Imag):   7.270490E-05 (A)
   IDisp(Mag):   6.187188E-04 (A)   IDisp(Phase):    6.748 (deg)
Total current
   ITot(Real):   7.021127E-04 (A)    ITot(Imag):  -6.682869E-04 (A)
    ITot(Mag):   9.693140E-04 (A)    ITot(Phase):  -43.586 (deg)
```

RFMATRIX FILE

This command is similar to the *Matrix file* command. The difference is that program records native quantities of the RF solution that could be useful for your own analysis programs. Each data line includes the real and imaginary parts of the potential ϕ and field components E_x, E_y and E_z.

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*. The listed quantities depend on the entries in the `INTERPOLATION` section of the configuration file. This setting has no effect on the data file listing.

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 150 and the maximum number is 500.

LSQ/LINEAR INTERPOLATION

Set the interpolation method for analysis and plots. The default is a second-order least-squares-fit procedure that uses data collected from nearby elements. Under the *Linear* option, only node values from the target element are included. This option is useful in tight spaces near material boundaries.

REFERENCE PHASE

Use this command to set the reference phase for field calculations and plots. Enter the value in degrees. The command has an effect only when the configuration file `RFE3VIEW_REFERENCE.CFG` has been loaded.

9.2 RFE3View analysis scripts

RFE3View scripts are useful for automating complex or repetitive analysis procedures. For example, you may want to compare a line scan of values between two points in a set of similar solutions. In the interactive mode, it would be necessary to type in the same coordinates for each scan. An analysis script can load each solution file, perform the scan and record all information in a single data file. In the **RFE3View** main menu, the following commands in the *File* popup menu (Sect. 8.1) apply to analysis scripts.

RUN SCRIPT

Pick and run a script file with a name of the form `FPREFIX.SCR`. The script can operate on the currently-loaded solution file or load a different file. The last solution file loaded by the script becomes the current file for plots and interactive analysis.

CREATE SCRIPT

Use this command to create an analysis script using the internal program editor.

EDIT SCRIPT
Use this command to change an existing script with the internal program editor.

RFE3View may be called as a command line task. The calling syntax is

```
[progpath\]RFE3VIEW [scriptpath\]SPREFIX
```

where the script has the file name `SPREFIX.SCR`. You should supply the full path `[scriptpath]` if the script is not in the current directory.

Command line operation opens several possibilities.

- **RFE3View** runs quickly and silently in the background when launched from the *Command Prompt* in a terminal window. To minimize typing, open the terminal in the working directory. Use the DOS `CHDIR` command to change directories.

- The program may be called by DOS batch files with commands of the form

```
START /w [progpath]\RFE3VIEW [scriptpath\]SPREFIX
START [progpath]\RFE3VIEW [scriptpath\]SPREFIX
```

Use batch files to organize calculations that run autonomously. The option `/w` in the first example signals that the batch file waits for completion of the program to continue. This form is useful if data from **RFE3View** is required for the next operation. Omit the option if you want to launch several instances of **RFE3View** to take advantage of a multiple-core machine. Use the `CHDIR` command to set the working directory.

- Run **RFE3View** as a subtask from your own compiled or interpreted programs. One application is to integrate finite element solutions and analysis into an optimization loop.

9.3   Script commands

Entries on a command line may be separated by the following set of delimiters: space, tab, equal sign [=], colon [:], left parenthesis [(] or right parenthesis [)]. Each command listed below is shown in symbolic form along with an example of how they might appear in a script.

INPUT [datapath\] FileName
INPUT = E:\FEData\AprilRuns\HVCross.ROU
Close the currently-opened solution file and load a solution file from the current directory. If `[datapath]` is not specified, the solution file must be in the working directory.

OUTPUT [datapath\] FileName
OUTPUT: WGUIDE1.DAT
Close the current data record file and open a new one. If a `[datapath]` is not specified, the file will be written in the working directory. A data file must be opened to use the following analysis commands.

CONFIGURATION [datapath\] FileName
CONFIGURATION = C:\PROGDIR\RFE3VIEW_REFERENCE.CFG
Load a new **RFE3View** configuration file to change the analysis characteristics. If a [datapath]
is not specified, the configuration file must be available in the working directory. If this command does not appear, the program uses the current configuration or the one in force in the
last run.

POINT xp yp zp
POINT = (0.00, 0.05, 4.67)
Perform a point calculation and write values of the quantities defined in the INTERPOLATION
section of the configuration file to the data record file. Enter coordinates in the units set by
$DUnit$.

SCAN xp1 yp1 zp1 xp2 yp2 zp2
LINE = (0.00, 0.00, 15.00) (12.00, 0.00, 15.00) Perform $(NScan + 1)$ calculations along a
line in space. At each point, write the first $NRecord$ quantities defined in the INTERPOLATION
section of the configuration file to the data record file. Enter coordinates in units set by $DUnit$.

GENSCAN
Perform any number of calculations along an arbitrary path in space. At each point, write the
first $NRecord$ quantities defined in the INTERPOLATION section of the configuration file to the
data record file. The command must be part of a structure with the following form:

```
GENSCAN
  xp1 yp1 zp1
  xp2 yp2 zp2
  ...
  xpn ypn zpn
END
```

Each data line contains three real numbers separated by spaces to define a point in the solution
volume. Enter coordinates in units set by $DUnit$.

VOLUMEINT [NReg]
VOLUMEINT
VOLUMEINT 8
Perform a volume integral of quantities defined in the VOLUME section of the configuration file
and write the results to the data record file. If a region number does not appear, the integral is
taken over the full solution volume. Otherwise, the integral extends over elements with region
number $NReg$. Volume integrals may take a long time, so use the region-number option if there
are several regions that are not of interest.

SURFACEINT Reg1 Reg2 ... RegN
SURFACEINT 5 7 9 -12

Perform a surface integral of quantities defined in the `SURFACE` section of the configuration file and write the results to the data record file. Positive integer values correspond to *Internal* regions and negative to *External* regions. With no specification, all regions that are not *Internal* are taken as *External*. The integral is taken over the surface facets between any *Internal* and *External* elements. The command may include any number of regions. The regions in the *Internal* set may or may not be contiguous. Note that **RFE3View** does not include facets on the boundary of the solution volume in surface integrals.

MATRIX FName XMin XMax Nx YMin YMax Ny ZMin ZMax Nz
MATRIX WGUIDE.MAT 1.00 1.00 10.00 2.00 2.00 12.00

Write values calculated at an array of location to a data file in text format. The file is named $FileName$ and is created in the current directory. The real-number values x_{min} and x_{max} are the limits of the listing volume along the x direction. The parameter N_x is an integer. Listings are made at $(N_x + 1)$ evenly spaced positions along x. The program lists the first $NRecord$ quantities defined in the `INTERPOLATION` section of the configuration file.

RFPOINT xp yp zp
RFPOINT = (0.00, 0.05, 4.67)

The command functions like *Point* command. The program records values of ϕ, E_x, E_y and E_z at the point in amplitude/phase format. The phase is given in degrees.

RFSCAN xp1 yp1 zp1 xp2 yp2 zp2
RFSCAN = (0.00, 0.00, 15.00) (12.00, 0.00, 15.00)

The function is similar to the *Scan* command. The program records values of ϕ, E_x, E_y and E_z at the point in amplitude/phase format. The phase is given in degrees.

RFCURRENT Reg1 Reg2 ... RegN
RFCURRENT 2 3 -1 -4

This command is similar to that of the *SurfaceInt* command. **RFE3View** sums complex quantities to find the total conductive and displacement currents through surfaces between the internal and external region set and records the result in amplitude/phase format.

RFMATRIX FName XMin XMax Nx YMin YMax Ny ZMin ZMax Nz
RFMATRIX Therm.MTX 1.0 1.0 10 10.0 2.0 10 2.0 12.0 20

The function is similar to the *Matrix* command. The difference is that **RFE3View** records values of ϕ, E_x, E_y and E_z in real/complex format.

NSCAN NScan
NSCAN = 100

Set the number of intervals for line scans. The default is $NScan = 150$ and the maximum value is $NScan = 500$.

INTERPOLATION [LSQ,Linear]
INTERPOLATION = Linear
Set the interpolation type to a multi-element least-squares fit or a single-element linear fit.

ENDFILE
Terminate the analysis
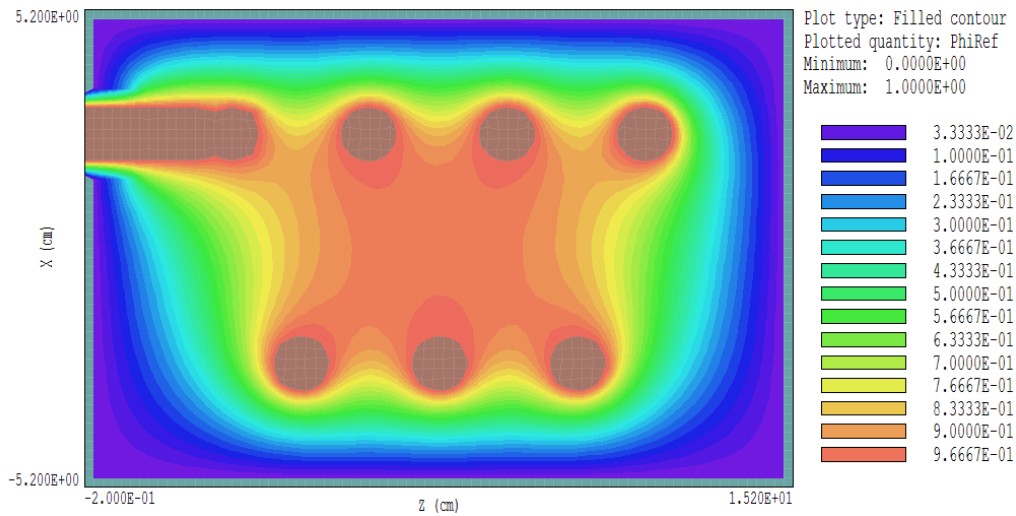
Figure 16: Slice plot – filled contour style.

10 RFE3View – slice plots

10.1 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 the planes of the foundation mesh. The precise rendering of mesh information enables 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. 12). 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. 16) 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. 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. Note that the normal plane box in the orientation area to the right of the plot (Fig. 16) 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.

PAN
When the plot is zoomed, you can use this operation to shift the current view center. Use the mouse to define relative start and end points for the shift.

10.2   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. 17. 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 Filled contour and *Contour* plot styles shows lines of constant value of the current plot quantity. The *Element* plot style shows elements in the slice color-coded according to a variety of computed

Figure 17: Slice plot style dialog.

quantities. The *Region shading* option at the bottom adds light background colors to show the boundaries between different dielectric or conductive regions in contour line plots. . 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 *Region shading* options at the bottom 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.

PLOT QUANTITY
Use this command to set the quantity for color-coding in *Filled contour* and *Element* plots and line values in *Contour* plots. The choices depend on entries in the currently-loaded configuration file.

PLOT LIMITS
In the *Autoscale* mode, **RFE3View** chooses defaults for the minimum and maximum values for colors and contours based on the range of values of the current plot quantity in the current slice. Deactivate *Autoscale* to set the values manually. The manual mode is useful for comparing field distributions between different slices.

RESET PLOTS TO DEFAULT
 Return to the default view that applied when the solution was loaded

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.
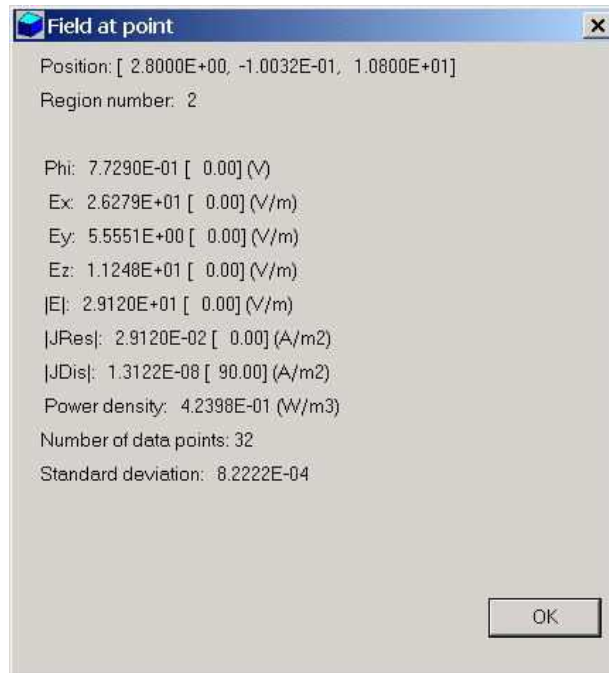
Figure 18: Information display point calculation

GRID INTERVALS

This command brings up a dialog to set intervals for grid lines in the horizontal and vertical directions. Deactivate the *AutoGrid* box and enter real numbers in the boxes. The values should be specified in the length units set by the *DUnit* parameter.

NUMBER OF CONTOUR LINES

Change the number of lines for *Filled contour* and *Contour* plots.

TOGGLE SNAP MODE

Mouse coordinates for commands such as *Zoom window*, *Pan*, and *Scan in slice* may 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.

SNAP DISTANCE

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

10.3 Analyses in a slice

You can determine field values at points and along scan lines with the commands of the *Analysis* popup menu.
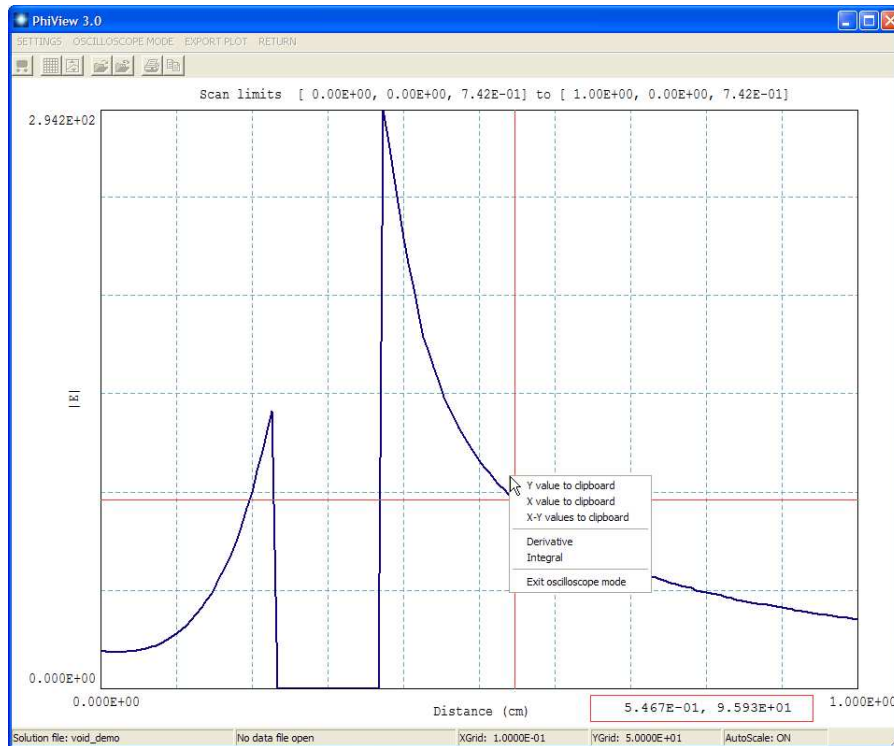
Figure 19: Scan plot display.

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 values at the point in the normal plane given by the coordinates. Figure 18 shows the information display. **RFE3View** calculates the quantities defined in the `INTERPOLATION` section of the current configuration file.

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* display (Fig. 19). **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*. The listed quantities depend on the entries in the INTERPOLATION section of the configuration file. This setting has no effect on the data file listing which includes up to *NRecord* field quantities.

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 150 and the maximum number is 500.

RF POINT

The command functions in the same way as the *Point calculation* command. The quantities listed in the message box and data file are ϕ, E_x, E_y and E_z in amplitude/phase format.

RF SCAN The command operation is the same as the *Line scan* command. **RFE3View** lists the quantities ϕ, E_x, E_y and E_z in amplitude/phase format in the data file. The program does not open the scan menu.

10.4   Scan menu

As shown in Fig. 19, the current scan quantity is plotted as a function of distance from the start point. The coordinates of the start and end points are listed above the graph. Coordinates are plotted in units set by the *DUnit* parameter. In the default mode, **RFE3View** sets limits so that the plot fills the screen. The program includes grid lines at even intervals. The status bar at the bottom lists the solution file, the data file (if open) and the grid intervals. The *Export plot* menu includes that standard set of commands described in the previous chapter. The *Settings* menu contains the following commands.

SYMBOL DISPLAY

Toggle the plot display to either a thick line or a set of symbols at calculated points connected by a thin line.

VERTICAL LIMITS

Override automatic selection and set vertical limits for the current quantity. To set manual limits, uncheck the *AutoScale* box and enter real-numbers values for the lower and upper limits of the plotted quantity. Check the *Autoscale* box to return to the default mode.

TOGGLE GRID

Suppress or activate the display of grid lines.

GRID INTERVALS

Override automatic selection of grid intervals. To set values for the horizontal and vertical

grid intervals, uncheck the *AutoGrid* box and enter real-number values. For the horizontal direction, enter the interval in length units set by the *DUnit* parameter. Check the *Autogrid* box to return to the default mode.

OPEN DATA FILE

In the oscilloscope mode, you can record a variety of numerical values for the scan. The information will be written to a data file if already opened. Use this command to open a new data file.

CLOSE DATA FILE

Close the current data file. This command should be invoked before attempting to use the file in another program.

OSCILLOSCOPE MODE

In response to this command, the scan plot becomes interactive with many of the features of a digital oscilloscope. The commands and tools of the *Scan plot* menu are deactivated and **RFE3View** displays the red cross-hair lines shown in Fig. 19. Values of the distance and plot quantity at the cross-hair intersection are displayed in the red box below the plot. The intersection point follows the mouse when the cursor is inside the plot area. Note that the intersection moves discontinuously between the data points. To check points at the end of the interval, move the mouse slowly toward the left or right-hand boundaries.

If a data file is open, **RFE3View** records values at the current intersection point when you click the left mouse button:

```
 Scan point calculation
    Distance:  9.506666E-01 (cm)
    Phi:  0.384693E+04
```

Clicking the right button activates the pop-up menu shown in Fig. 19. The menu contains the following commands:

Y VALUE TO CLIPBOARD

Copy the value of the plotted quantity at the cross-hair intersection to the Windows clipboard. You can then past the number into a text document, spreadsheet or calculator program.

X VALUE TO CLIPBOARD

Copy the value of the distance (in units set by *DUnit*) at the cross-hair intersection to Windows clipboard.

XY VALUES TO CLIPBOARD

Copy the values of the distance and plotted quantity to the clipboard. The x value is on the first line and the y value on the second.

DERIVATIVE

Calculate and record the numerical derivative at the intersection point. The dimensions are determined by the *DUnit* parameter. For example, if the scan quantity is potential in V and the distance is shown in centimeters, the derivative has units V/cm. **RFE3View** writes the value to the clipboard and also displays the results in a dialog. If a data file is open, the program makes a record like the following one:

```
Derivative
    Location:  1.694667E+00 (cm)
    Phi/(cm):  3.654382E+05
```

INTEGRAL

Calculate and record the definite integral of the plotted quantity from the start point of the scan to the cross-hair intersection point. The dimensions are determined by the *DUnit* parameter. For example, if the scan quantity is the electric field in V/cm and the distance is shown in centimeters, the integral has units (V/cm)-cm = V. **RFE3View** writes the value to the clipboard and also displays the results in a dialog. If a data file is open, the program makes a record like the following one:

```
Definite integral
    Location:  1.908000E+00 (cm)
    Phi-(cm):  9.679801E+03
```

EXIT OSCILLOSCOPE MODE

Activate commands and tools and return to the standard *Scan plot* mode. You can also press the *Esc* key to exit the oscilloscope mode.

10.5   Slice vector tools

Vector electric-field lines are defined only for reference mode displays (i.e, a snapshot of the field distribution at a specific). Therefore, the following commands function only under the configuration file `rfe3view_reference.cfg`. In this case, **RFE3View** can display the direction of the vector quantities $\mathbf{E}$, $\mathbf{J}_c$ and $\mathbf{J}_d$ in slice plots. Figure 20 shows the corresponding entries on the toolbar.

E PROBE

This feature was inspired by the familiar Magnaprobe illustrated in Fig. 20. When you click on the tool and move the cursor into the slice-plot area, it changes to a semi-transparent probe that rotates about a pivot point to show the direction of the electric field or current density. The probe functions in all plot styles and may be combined with field-line traces and scatter plots. The status bar shows the coordinates and $|\mathbf{E}|$, $|\mathbf{J}_c|$ or $|\mathbf{J}_d|$ at the pivot point.
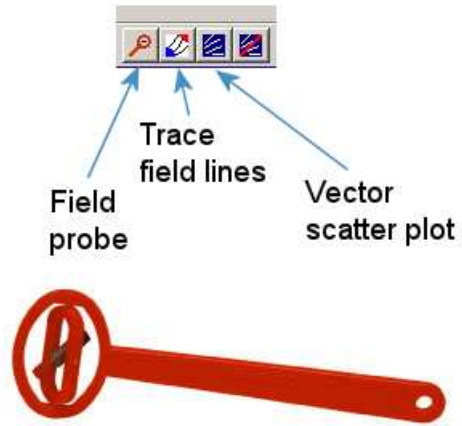
Figure 20: Tools to display vector information in the slice plot menu.
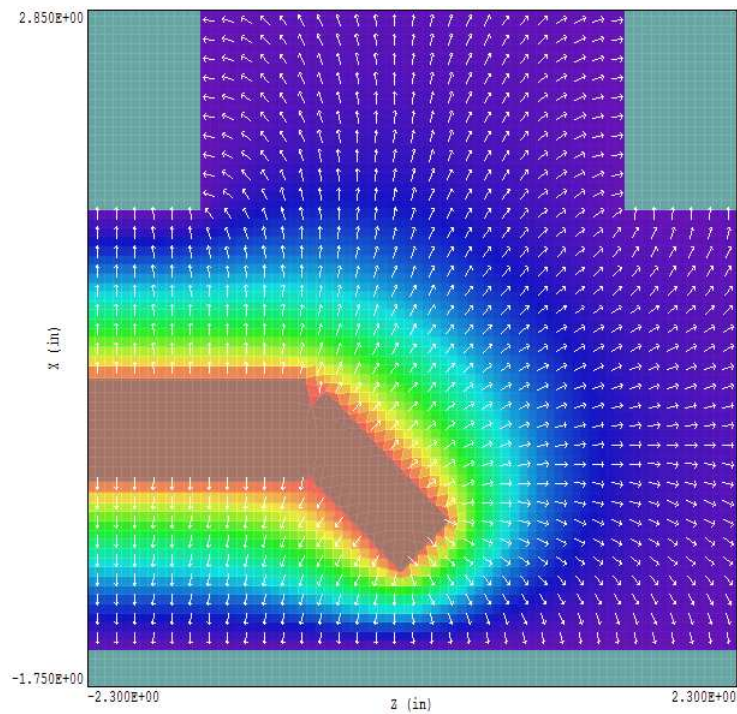
Figure 21: Element plot with electric field vectors.

FIELD LINES AT POINTS

Use this command to add lines of electric field or current density projected to the slice plane. The program enters coordinate entry mode when you click the command. Move the mouse to a point in the solution volume and click the left button. **RFE3View** calculates the three-dimensional path of an electric field line that passes through the point and plots the projection in the slice plane. You can continue to add any number of lines. Click the right mouse button or press *ECS* to exit coordinate mode. The lines are not included in hardcopy or plot file exports and disappear if you change the slice view. Use a screen capture utility to record them. It is important to recognize the nature of the plots. The lines are three-dimensional curves projected to the slice plane. They may be difficult to interpret if the line does not lie close to the slice plane. For full three-dimensional field line plots, see the *Field line plot file* command in Chap. 11.

VECTOR SCATTER PLOT

Superimpose a uniform distribution of vector arrows pointing in the direction of **B** and **H** in the slice plane (Fig. 21). Arrows may be added to any of the plot types, including *Region.* They are included in hardcopy and plot file export. The arrows are preserved and adjusted if you change the slice view.

REMOVE VECTORS

Use this command to turn off the vector scatter plot mode and to remove vector tool displays from the current plot.
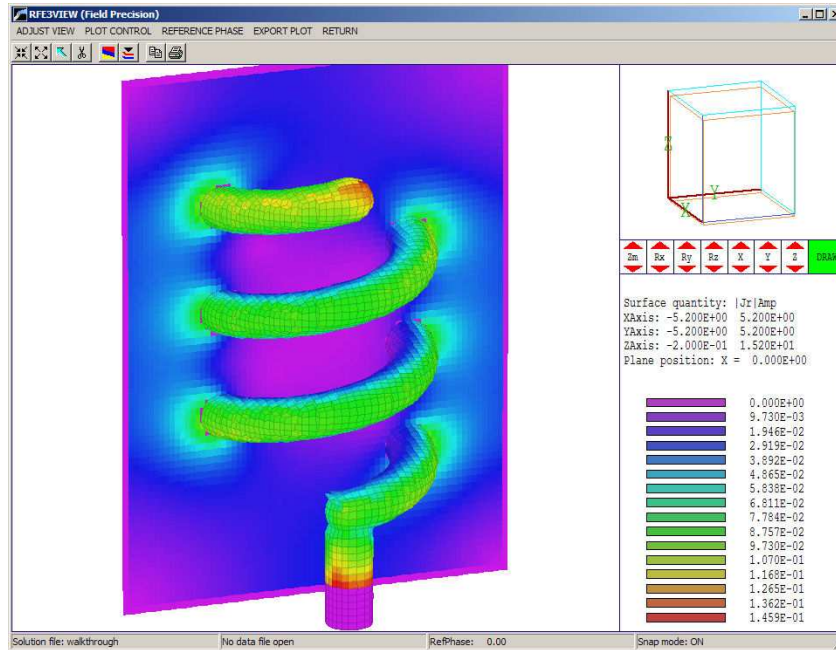
Figure 22: Three-dimensional surface view of region boundaries with color coding by the magnitude of the real current density.

11   RFE3View – surface plots

Surface plots are three-dimensional views of the solution space (Fig. 22). Three types of information can be displayed: 1) boundaries of regions color coded by region number and shaded with distance from the view, 2) region boundaries color coded by the values of computed quantities on the surface and 3) computed quantities in a slice plane normal to one of the Cartesian axes. Slice plane plots may be superimposed on either of the region boundary plot types. Surface plots are created from the conformal mesh and preserve true spatial scaling.

The method to control the three-dimensional display with the mouse is identical to that used in **MetaMesh**. Figure 23 shows the active areas of the screen. The central zone (A) is used for zooming in (left button) and out (right button). Hold down the left mouse button in zones B, C, D and E to walk around the object. Hold down the right mouse button in zones B, C, D and E to move the viewpoint to the right, upward, to the left and downward. Note that changes are reflected in the orientation box in the upper-right portion of the screen. The plot is updated when you release the mouse button. You can control additional aspects of the three-dimensional view with the commands of the *Adjust view* popup menu.

You can control additional aspects of the three-dimensional view with the commands of the *Adjust view* popup menu.

SET VIEW DIALOG

This command brings up a dialog (Fig. 24) where you can set specific view angles, displacements and the distance to the viewpoint, *DView*. The parameter *DView* controls perspective. Set it
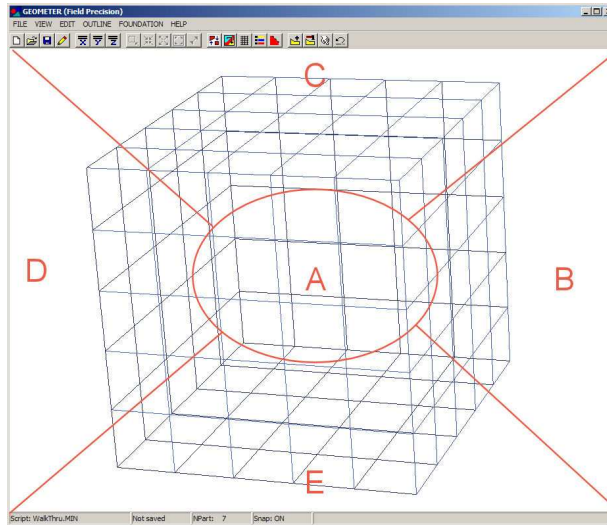
Figure 23: Active areas for mouse control of the 3D view.

to a large value for an orthographic view. A value smaller than the dimensions of the system may result in an incoherent display.

RESTORE DEFAULT VIEW
This command is useful if you loose your orientation after several rotations and translations. The view is returned to the default: $\theta_x = -30°$, $\theta_y = 0°$ and $\theta_z = 45°$ with the origin at the center.

+X VIEW
+Y VIEW
+Z VIEW
Rotate to views from the $+x$, $+y$ or $+z$ directions. Origin shifts are not affected

CENTER VIEW
Remove shifts by setting the origin to the center of the solution volume.

ORTHOGONAL/PERSPECTIVE VIEW
By default, **RFE3View** includes perspective correction in surface plots. Click on this command to move the view point to infinity. Click the command a second time to restore the view point.

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

SURFACE PLOT CONTROL
This command brings up the dialog of Fig. 25. The group of commands on the left-hand side controls the plot style. A three-dimensional plot is constructed from color-coded facets. There are two types of facets: 1) element facets on the boundary of a region with color coding by region

61
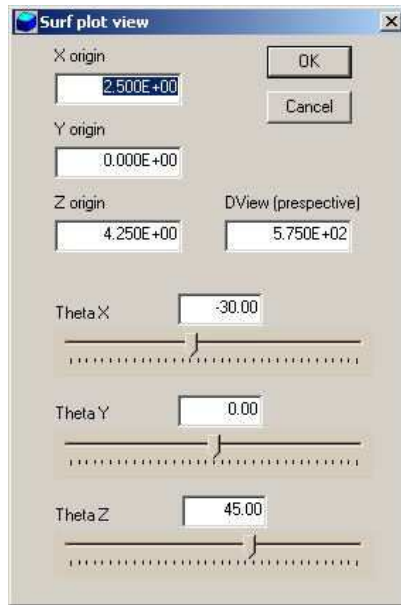
Figure 24: Tool area to adjust the 3D view.

number or the selected computed quantity and 2) rectangular facets comprising a slice plane with color-coding by the values of the computed field quantity. Slice plane and region boundary information may be superimposed. The *Surface plot style* radio buttons control whether the facets of region boundaries are plotted as solid plates (hidden surface) or wireframe outlines. The *Facet display* check box determines whether the boundaries of region facets are plotted in the hidden-surface mode. (Note that the facets of the slice plane are always plotted as solids with no outline). **RFE3View** plots a reference grid along the boundaries of the solution volume when the *Reference grid* box is checked. The group of commands on the right-hand side controls the slice plane. The *Include slice plot* box determines whether a slice plot is included. The radio buttons determine the axis normal to the slice plane. You may move the plane along the chosen axis by entering values in the box or moving the slider.
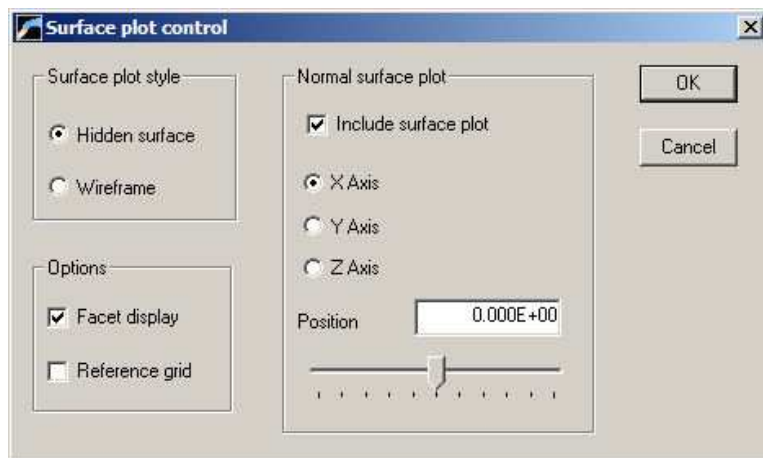
Figure 25: Surface plot control dialog.

62
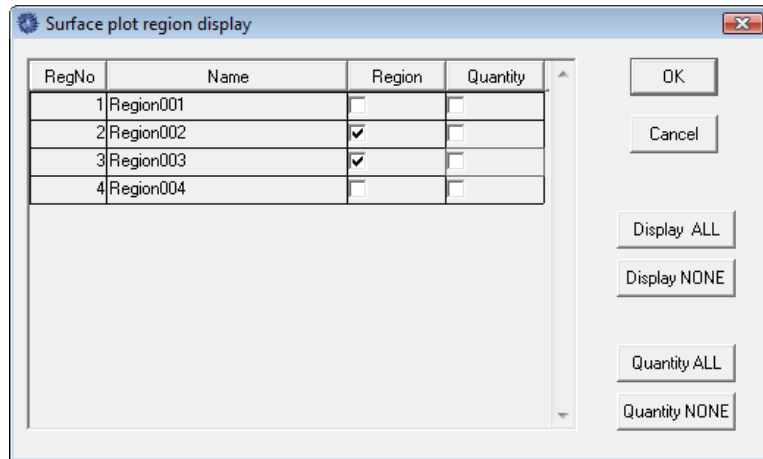
Figure 26: Region display dialog.

DISPLAYED REGIONS

The command brings up the dialog of Fig. 26 where you can pick region boundaries to include in the plot. Depress a button in the *Display* column to activate a region. The buttons in the column marked $|E|$ determine the presentation style for boundary facets. If the box is uncheck, RFE3View colors facets by region number. If one or more buttons are depressed in the $|E|$ column, color code is by the computed field quantity. The information window shows the correspondence between color and the quantity. Note that the field quantity is always calculated at a point near the facet **outside** the chosen region. If the region is a fixed-potential electrode, the color coding gives the computed quantity at the surface in the adjacent vacuum/dielectric region. If a boundary separates two dielectric regions, the computed quantity may depend on which region is chosen for display.

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 displayed area 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 of facets on region boundaries or in the normal plane. Quantities defined in the `INTERPOLATION` section of the configuration file are available.

SET PLOT LIMITS

By default the limits of the plotted quantity are automatically computed by **RFE3View**. With this command you can set limits manually. Facets where the computed quantity is outside the set range are plotted in gray. The feature is useful for emphasizing a certain range of values (*i.e.*, portions of an electrode surface where $|\mathbf{E}|$ exceeds a value). The command is essential on

occasions when a distorted element on a region surface leads to a inaccurate field interpolation, skewing the plot range.

The following three commands function only when the current configuration is `rfe3view_reference.c`

FIELD LINE PLOT FILE

You can add electric field lines (at the reference phase) to three-dimensional plots with this command. To use it, you must prepare a text file that contains any number of point coordinates. **RFE3View** reads the file an adds field lines that pass through each point. The file consists of data lines, where each line contains three real numbers (x, y, z) separated by any of the standard delimiters. Enter the coordinates in units set by *DUnit*. The file may contain comment lines that start with an asterisk and should end with the *ENDFILE* command. Lines are not plotted for target points within fixed-potential regions or outside the solution volume.

RECORD FIELD LINE PLOT

Use this command to record calculated field lines. **RFE3View** prompts for the name of an output text file with a name of the form `FNAME.FLP`. A file extract is shown below. The command functions only if a file of starting points has been opened with the *Field line plot file* command.

```
Electric field line plots for file: walkthru.HOU
   DUnit(unit conversion factor):   1.00000E+02


Start point
    X:    8.00000E+00
    Y:    2.50000E-01
    Z:    3.00000E+00
Forward
  8.00000E+00   2.50000E-01   3.00000E+00
  8.00599E+00   2.50385E-01   3.00003E+00
  8.01197E+00   2.50768E-01   3.00006E+00
  8.01796E+00   2.51151E-01   3.00009E+00
  8.02395E+00   2.51532E-01   3.00011E+00
  8.02994E+00   2.51912E-01   3.00014E+00
```

CLOSE FIELD LINE PLOT

Close the field line plot file and remove field lines from the current plot.

12   Customizing RFE3View

You can create configuration files to customize the operation of **RFE3View** for your application. You can then switch between configurations for different types of analyses. **RFE3View** is supplied with four default configuration files in the same directory as the executable program:

- `rfe3view_standard.cfg` – standard analysis, amplitudes of RF quantities.

- `rfe3view_reference.cfg` – snapshot of RF quantities at a user-defined reference phase.

- `rfe3view_phase.cfg` – plots and calculations of the amplitude and phase of the RF potential and electric-field components.

- `rfe3view_complex.cfg` – direct display of the real and imaginary parts of the phasors of RF quantities.

We recommend that you do not change these files. Instead, make a copy to use as a template and rename it. You can put your new configuration file anywhere, but we suggest you store all configurations in the same directory as `rfe3view.exe`. Note that **RFE3View** remembers the last configuration used and attempts to reload it at the next run.

When you have created a template file, open it in an editor. Note that it is divided into the ten sections shown in Table 12. With few exceptions, do not make changes to the sections `HEADER`, `STOREDQUANT REGIONQUANT`, `PROGPARAM`, `RUNPARAM` and `REALTIMEPARAM`. Entries in these sections coordinate the operation of **RFE3** and **RFE3View**. Modifications may cause file load errors.

12.1   Defining calculated quantities

You can define custom quantities for display and analysis in the last four sections of the configuration file:

- **Interpolation**. Scalar quantities for multiple uses: 1) displays in slice, plane and surface plots, 2) results of point and scan calculations and 3) values written to matrix output files.

- **Vector**. Vector quantities for the display of field or flux lines in slice and surface plots.

- **Volume**. Scalar quantities for volume integrals over regions.

- **Surface**. Vector quantities for surface integrals over the boundaries of region sets.

Run speed is a critical issue in **RFE3View**. The generation of a single slice plot may require over 50,000 interpolations. To ensure fast operation, expressions for calculated quantities in the configuration file are written in reverse Polish notation (RPN). **RFE3View** parses the function strings once while loading the configuration file and saves them in a coded form. Thereafter, operations are performed at the speed of compiled code. The following section reviews the basics of RPN notation. Section 12.3 covers parameters and variables that may appear in expressions. Section 12.4 reviews the form of the data sections.

Table 6: General organization of the **RFE3View** configuration file

```
HEADER
  Control parameters (Required)
END
STOREDQUANT
  Stored quantities in the solution file (Required)
END
REGIONQUANT
  Physical properties of regions (Optional)
END
PROGPARAM
  Program parameters (Optional)
END
RUNPARAM
  Parameters set by the solution program (Optional)
END
REALTIMEPARAM
  Parameters set by the user (Optional)
END
INTERPOLATION
  Calculated quantities for plots and data output  (Required)
END
VECTOR
  Calculated quantities for vector display (Optional)
END
VOLUME
  Calculated quantities for volume integrals (Optional)
END
SURFACE
  Calculated quantities for surface integrals (Optional)
END
ENDFILE
```
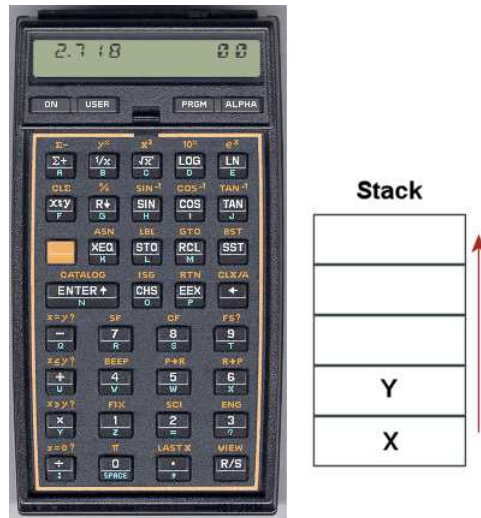
Figure 27:

12.2 Introduction to RPN

Consider the expression

$$5.0 \times 4.5^3 + \left[9.2 + 0.6 \times 0.9^{(2+0.67)}\right]. \qquad (39)$$

Equation 39 could be represented in algebraic notation by the string:

```
5.0 * 4.5^3 + (9.2 + 0.6 * 0.9 ^ (2 + 0.67))
```

The rules for parsing such a string are involved, requiring an interpreter with recursive logic. The equation has the following form in RPN:

```
0.9 0.67 2 + ^ 0.6 * 9.2 + 4.5 3 ^ 5.0 * +
```

The string is parsed in strict order from left to right for expressions of any degree of complexity. A calculator with RPN logic uses the stack shown in Fig. 27. Numbers are pushed and popped at the bottom. The first two stack registers have the special names X and Y. Three simple rules govern the evaluation of RPN expressions:

- If the entry is a number, push it on the stack.

- If the entry is a unary operator (exp, ln, sin,...), apply it to the number in the X register.

- If the entry is a binary operator $(+,*,\hat{}\,,...)$ combine the numbers in the X and Y registers and move the stack down.

The final result is given by the number remaining in the X register. By convention, the binary operators act in the following way:

- Addition(+): $Y + X$

- Subtraction (-): $Y - X$

- Multiplication (*): $Y \times X$

- Division (/): Y/X

- Exponentiation (^): Y^X

Reverse Polish notation eliminates all issues with regard to parsing order. Furthermore, it is simple to compile expressions. Entries are represented either by variables or operator codes.

12.3 Variables, operations and parameters

Expressions to define quantities in **RFE3View** may contain numbers, unary and binary operations, standard variables, standard calculated quantities and parameters. The standard variables are

```
$X, $Y, $Z, $NREG, $DUNIT
```

Note that the name of variables and parameters must start with a dollar sign. Calls to defined functions for plots, line scans and other operations are always made at a specific location in the solution space. **RFE3View** sets the current position [$X,$Y,$Z] before any calls are made. To illustrate, the following expression gives the distance from the origin in meters:

```
RDIST = $X 2 ^ $Y 2 ^ $Z ^ 2 + + @SQRT
```

The standard variable `$DUNIT` is a dimension conversion factor passed from the solution program. For example, suppose dimensions in **MetaMesh** were defined in centimeters and converted to meters for use in the solution program. In this case, an entry in the header of the solution file would set $DUNIT = 100.0. The following expression returns the distance from the origin in centimeters:

```
RDIST = $X 2 ^ $Y 2 ^ $Z ^ 2 + + @SQRT $DUNIT *
```

Unary operators have names that begin with '@'. Table 7 lists the available set. The following considerations apply with respect to the `@CHS` operator:

- You may include negative numbers (such as `-3.1456` and `-8.9E-09`) in expressions. The number `-22.56` is equivalent to `22.56 @CHS`.

- Expressions like `-$Epsi0` and `-&grady[3]` that mix the binary operator with a variable are invalid. Instead, use forms like `$Epsi0 @CHS`.

The following quantities represent values from the RF electric-field solution calculated at the current position:

```
&PhiR, &PhiI
&ExR, &ExI
&EyR, &ExR
&EzR, &EzI
&EMag
&Q[N]
```

Table 7: Unary operators

Name	Operation	Comments
@SIN	$\sin(X)$	Angle in radians
@COS	$\cos(X)$	Angle in radians
@TAN	$\tan(X)$	Angle in radians
@ASIN	$\sin^{-1}(X)$	Returns angle in radians
@ACOS	$\cos^{-1}(X)$	Returns angle in radians
@ATAN	$\tan^{-1}(X)$	Returns angle in radians
@EXP	e^X	
@LN	$\ln X$	
@LOG	$\log_{10}(X)$	
@SQRT	$\sqrt{X}$	
@ABS	$|X|$	
@EXCH	$X \rightleftharpoons Y$	
@OVERX	$1/X$	
@XSQ	X^2	
@ENTER	$X \Rightarrow X, Y$	Copy X and push on stack
@CHS	$X = -X$	

The names must begin with an ampersand. They are always followed by an integer in box brackets giving the number of the stored quantity. The quantity &Q[n] is the interpolated value of stored quantity n at the current position [$X,$Y,$Z]. For an RF electric field solution, &Q[1] is the same as &PhiR, &Q[2] corresponds to &PhiI, &Q[3] equals the relative dielectric constant ϵ_r, &Q[4] is the electrical conductivity in S/m and &Q[5] is the time-averaged power density in W/m^3. The program performs the interpolation and replaces the symbol with its numerical value before evaluating the expression. The interpolation method depends on whether the stored quantity is of type *Node* or *Element*. **RFE3View** performs detailed interpolations of *Node* quantities. In the LIN mode, the program interpolates within the target element using shape functions and values of the stored quantity at the eight nodes. In the LSQ mode, **RFE3View** collects data from surrounding elements of the same region type and makes a least-squares fit to a three-dimensional, second-order function.

Expressions may also include the RF frequency $Freq, transferred from **RFE3** in the PROGPARAM section. You may include parameters defined in the PROGRARAM section. The section in the file rfe3view_standard.cfg has this form:

```
PROGPARAM
  $Epsi0 = 8.854187E-12
  $TwoPi = 6.283185
END
```

The following expression for the amplitude of the total displacement current in the interpolation section uses both parameters:

```
JdMagAmp = &EMag &Q[3] * $Epsi0 * $Freq * $TwoPi *
```

You can define up to nine additional parameters in the `PROGRARAM` section. The name must begin with a dollar sign and may include up to 12 characters. A value in any valid real-number format follows the equal sign.

12.4   Setting up the calculated-quantity sections

The final sections of the configuration file define quantities for plots and calculations. There are four sections:

```
INTERPOLATION
  QuantityName = Definition
END
VECTOR
  QuantityName = Definition1;Definition2;Definition3
END
VOLUME
  QuantityName = Definition
END
SURFACE
  QuantityName = Definition1;Definition2;Definition3
END
```

The `INTERPOLATION` section may contain from zero to 50 quantity definitions. The `VECTOR`, `VOLUME` and `SURFACE` sections may contain a maximum of 20 quantity definitions each.

A definition line for a scalar quantity in the *Interpolation* section consists of a variable name, an equal sign and an RPN expression using the components discussed in the previous section. The following rules apply to quantity names in all sections:

- A name has a maximum length of 12 characters (the length is limited so that quantity names will fit in plots and formatted output lists).

- Names may include characters, numbers and the underscore symbol. Names may not include spaces and other punctuation marks.

The RPN expression is a string (maximum length of 500 characters) with entries separated by spaces. Here are some examples of quantity definitions:

```
  EPERP = &EY @XSQ &EY @XSQ + @SQRT
  LOGEXVCM = &EX 100.0 / @LOG
  DX = &EX $Epsi0 * &Q[2] *
  FIELDANGLE = &EY &EX / @ATAN $RADTODEG *
```

The last expression requires that the parameter

```
  $RADTODEG = 57.2958
```

was defined in the `PROGPARAM` section. Note that the result of the division operator in the expression is 0.0 if `&EX = 0.0`.

The order in which quantities appear in the `INTERPOLATION` section affects some operations in **RFE3View**.

Table 8: Naming conventions in RPN expressions

Name form	Functions
`$NAME`	Program parameters
`$NAME`	Standard variables (*e.g.*, `$X`)
`&NAME`	Special quantities calculated from the solution (*e.g.*, `&Ex`)
`@NAME`	Unary operators in RPN expressions

- The first listed quantity is the default choice for *Slice* plots You can change the choice by entering a value in the *SliceDisplay* command in the `HEADER`. The integer is the order of the default plot quantity as listed in the `INTERPOLATION` section.

- In dialogs for picking a plot quantity, the options are listed in the same order that they appear in the configuration file. Therefore, you should put important quantities near the beginning and specialized quantities near the end.

- The same order is followed in listings created by the *Matrix file* command. If you are exporting tables for use in **OmniTrak**, be sure that the first three quantities are E_x, E_y and E_z.

Each quantity in the *Vector* section requires three expressions for the x, y and z components. The format for a definition line is

```
Name = (Expression x);(Expression y);(Expression z)
```

The line may contain a maximum of 500 characters. The component expressions are separated by semicolons. The following example defines the electric field vector if (and only if) the configuration `rfe3view_reference` has been loaded:

```
VECTOR
  EVect = $Phi0 $DegToRad * @ENTER @COS &ExR * @EXCH @SIN &ExI * -;
    $Phi0 $DegToRad * @ENTER @COS &EyR * @EXCH @SIN &EyI * -;
    $Phi0 $DegToRad * @ENTER @COS &EzR * @EXCH @SIN &EzI * -
END
```

Note that the single-line definition has been divided for display.

The volume integral of a quantity Q over a region N_R is given by

$$\int dx \int dy \int dz \ Q(x, y, z), \tag{40}$$

where only those elements with region number equal to N_R are included. You can define up to 20 quantities in the *Volume* section for integrals. The expressions should define quantities with dimensions of the form ($quant/m^3$). For example, the time-averaged power dissipated in units of W/cm^3 is given by

```
PDensAvg = &Q[5]
```

Volume integrals use internal program coordinates in units of meters. Note that the quantity $DUNIT is used in **RFE3View** only for display labels.

The surface integral of a vector quantity **S** over a region or a set of regions is given by

$$\int \int dA \ \mathbf{S} \cdot \mathbf{n}, \tag{41}$$

where dA is an element of area on the surface of the region set and **n** is a unit vector normal to the surface pointing out of the region set. You can define up to 20 quantities in the SURFACE section for integrals. The data line for each quantity should contain the name, an equal sign, and three RPN expressions separated by semicolons for the vector components. The expressions should define quantities with dimensions of the form ($quant/m^2$). As an example, a surface integral of the following quantity would give the real and imaginary parts of the total current out of a region set:

```
IcReal = &ExR &Q[4] *;&EyR &Q[4] *;&EzR &Q[4] *
IcImag = &ExI &Q[4] *;&EyI &Q[4] *;&EzI &Q[4] *
```

Note that quantity names must begin with special symbols if they are used in RPN expressions. Symbols for different quantity types are listed in Table 8. Therefore, quantity names in the PROGPARAM section must include the dollar sign. Quantity names that are defined only for display purposes need not include a special symbol. The includes quantities defined in the INTERPOLATION, VECTOR, VOLUME and SURFACE sections.

13   RFE3 file formats

RFE3 creates output files in binary or text format. The mode is controlled by the *Format* command (Sect. 4.1).

- The default binary mode should be used for most applications. It gives the minimum file size and preserves the full accuracy of the double precision numbers used in **RFE3**.

- Use the text mode to port information to your own programs or if you want to inspect results with an editor.

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. The FORTRAN code extract shown in Table 9 comprises the entire output algorithm. The header contains the following elements:

- The 6-character string *BINARY*.

- The number of regions in the solution space, $NReg$ (4-byte integer).

- A total of $NReg$ 20-character strings giving the names of regions obtained from the **MetaMesh** input file.

- A total of $NReg$ single-character strings, either I or E. The character I designates a standard material region that should be included in a RFE3View analysis and E indicates a fixed-potential that should be excluded.

- The mesh size parameters I_{max}, J_{max} and K_{max} (4-byte integers).

- The quantity $DUnit$, the unit conversion factor associated with the solution (double precision, 8-byte real).

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 4-byte integers. The coordinates X, Y and Z (in meters) are 8-byte real numbers as are the following stored quantities:

- The real and imaginary parts of the RF potential ϕ at the node in units of volts.

- The relative dielectric constant ϵ_r associated with the upper element.

- The electrical conductivity σ (in S/m) associated with the upper element.

Table 9: Code to create the **RFE3** binary output file

```
! File type
    Character6 = 'BINARY'
    WRITE (OutField) Character6
! NReg
    WRITE (OutField) NRegMax
! Region names
    DO N=1,NRegMax
      WRITE (OutField) Reg(N).RName
    END DO
! Region status
    DO N=1,NRegMax
      IF (Reg(N).Fixed) THEN
        Character1 = 'E'
      ELSE
        Character1 = 'I'
      ENDIF
      WRITE (OutField) Character1
    END DO
! Mesh size
    WRITE (OutField) IMax,JMax,KMax
! DUnit
    WRITE (OutField) DUnit
! Node quantities
    DO K=0,KMax
      DO J=0,JMax
        DO I=0,IMax
          MC = M(I,J,K)
          NReg = C(MC).RegUp
          EpsiR = ElementEpsiR(I+1,J+1,K+1,NReg)
          Sigma = ElementSigma(I+1,J+1,K+1,NReg)
          PDens = PowerDens(I+1,J+1,K+1)
          WRITE (OutField) &
            C(MC).RegNo,C(MC).RegUp,C(MC).x,C(MC).y,C(MC).z, &
            DBLE(C(MC).Phi),DIMAG(C(MC).Phi),EpsiR,Sigma,PDens
        END DO
      END DO
    END DO
  ENDIF
```

- The time-averaged resistive power density p (in W/m^3) associated with the upper element. This information may be passed to **HeatWave**.

Table 10 shows the header and initial node data lines from an output file in text format (note that the node data lines have been broken for readability). Three region names are shown in the example. The names are strings up to 20 characters in length with no spaces. Each name is followed by the character I (included) or E (excluded). The order of nodes is the same as that of Table tab:outputcode. Real numbers are recorded in format (1P,E14.6).

Files created in response to the **RFE3View** *Matrix file* and `RF matrix file` commands are in text format and have a structure similar to that of Table 10. The files are useful to transfer information to your own analysis programs and they are compatible with **FEVision**. There are two differences from the `ROU` file in text format. First, calculations are performed over a regular mesh at locations

$$X(I) = X_{min} + I\Delta x, \tag{42}$$
$$Y(J) = Y_{min} + J\Delta y, \tag{43}$$
$$Z(K) = Z_{min} + K\Delta z. \tag{44}$$

where

$$\Delta x = (X_{max} - X_{min})/I_{max}, \tag{45}$$
$$\Delta y = (Y_{max} - Y_{min})/J_{max}, \tag{46}$$
$$\Delta z = (Z_{max} - Z_{min})/K_{max}. \tag{47}$$

Second, the set of calculated quantities differs. For the standard matrix file, all quantities defined in the `INTERPOLATION` section of the configuration file are included. The RF matrix file includes the following quantities: ϕ(real), ϕ(imag), E_x(real), E_x(imag), E_y(real), E_y(imag), E_z(real) and E_z(imag).

Table 10: Header and data extract showing the form of the **RFE3** `ROU` file under the `Text` option and the **RFE3View** matrix file

```
TEXT
  1.000000E+07
NReg:   4
CHAMBER E
CONDUCTIVEMEDIUM I
BUSHING I
HEATER E
IMax:    52
JMax:    52
KMax:    77
DUnit:  1.000000E+02


 RegNo RegUp      X               Y               Z            PhiReal
 ====================================================================
     1     1  2.000000E-03   3.000000E-02   5.600000E-02   7.524430E-01
     1     1  4.000000E-03   3.000000E-02   5.600000E-02   7.500872E-01
     1     1  6.000000E-03   3.000000E-02   5.600000E-02   7.433185E-01
     1     1  8.000000E-03   3.000000E-02   5.600000E-02   7.316038E-01
     1     1  1.000000E-02   3.000000E-02   5.600000E-02   7.145547E-01
     1     1  1.200000E-02   3.000000E-02   5.600000E-02   6.918953E-01
     1     1  1.400000E-02   3.000000E-02   5.600000E-02   6.632492E-01
     1     1  1.600000E-02   3.000000E-02   5.600000E-02   6.287581E-01
      ...


        PhiImag        EpsiR         Sigma         PDens
     ========================================================
      5.974013E-08   8.100000E+01   1.000000E-03   8.056705E-01
      5.723061E-08   8.100000E+01   1.000000E-03   8.571188E-01
      5.927211E-08   8.100000E+01   1.000000E-03   9.118180E-01
      5.980883E-08   8.100000E+01   1.000000E-03   9.734815E-01
      6.427057E-08   8.100000E+01   1.000000E-03   1.028880E+00
      6.965484E-08   8.100000E+01   1.000000E-03   1.066633E+00
      7.152705E-08   8.100000E+01   1.000000E-03   1.083529E+00
      7.416678E-08   8.100000E+01   1.000000E-03   1.081640E+00
         ...
```

Index