



# **HiPhi**

## **Three-dimensional Electrostatics**

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

# HiPhi

## Three-dimensional Electrostatics

### Table of contents

#### **Chapter 1. Introduction to HiPhi**

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

#### **Chapter 2. Organizing AMaze simulations**

#### **Chapter 3. Script commands for run control**

- 3.1. Script conventions
- 3.2. Program control commands

#### **Chapter 4. Commands to set material properties**

#### **Chapter 5. Modeling small details in a large-scale solution**

#### **Chapter 6. Theoretical basics**

- 5.1. Dielectric and conductive solutions
- 5.2. Boundary conditions

#### **Chapter 7. Running HiPhi interactively**

#### **Chapter 8. Automatic runs under batch file control**

- 7.1. Command line operation
- 7.2. Batch files
- 7.3. Introduction to **GCon**
- 7.4. Running **GCon**

## **Chapter 9. PhiView - file menu**

## **Chapter 10. PhiView - plane plots**

## **Chapter 11. PhiView - slice plots**

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

## **Chapter 12. PhiView - surface plots**

## **Chapter 13. PhiView - analysis functions and script operation**

- 13.1. Point calculation - interpolation method
- 13.2. Line scans and matrix files
- 13.3. Automatic analyses
- 13.4. Script operation

## **Chapter 14. Structure of the HiPhi output files**

## **Chapter 15. Calculation of mutual capacitance**

# Chapter 1. Introduction to HiPhi

## 1.1 Installation

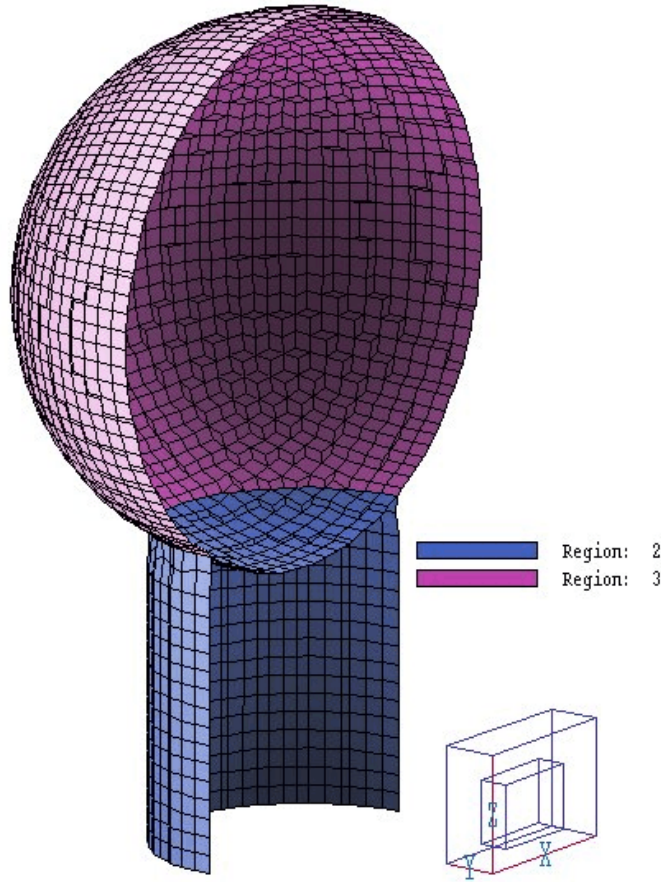
**HiPhi** calculates electrostatic fields in arbitrary three-dimensional systems. The program handles either dielectric or conductive solutions. In dielectric-type solutions regions in the simulation may represent electrodes, dielectrics and space-charge density. In conductive-type solutions the regions may represent electrodes or resistive media. Two programs are supplied with the package:

- **HIPHI . EXE** – determines one or more electrostatic solutions in a window or under batch file control
- **PHIVIEW . EXE** – generates plots and analyses of solutions from **HiPhi**.

**MetaMesh**, the **AMaze** conformal mesh generator, is required to create solutions. **HiPhi** 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 512 MB of memory can handle several million elements.

If you downloaded the programs electronically, unzip the files to a temporary directory. To install the programs, copy them from the temporary directory or the distribution disk to the directory `\AMAZE` on your hard disk. Set up a Windows shortcut to `AMAZE . EXE` on your Desktop or in your Start menu. You may also want to copy the sample input files in the directory `\EXAMPLES` of the distribution disk to the `\AMAZE\EXAMPLES` directory. If you downloaded the files electronically, unzip the file `EXAMPLES . ZIP` in the directory `\AMAZE\EXAMPLES`. Please review licensing regulations for the programs in the file `TERMS . TXT`.

The remainder of this chapter describes a step-by-step run that will test your setup and familiarize you with the programs. Chapter 2 covers the programs used for a complete electrostatic solution and the organization of input and output files. Chapter 3 covers the structure of the **HiPhi** control script. The script sets parameters for program operation and defines the material properties of regions. Chapters 4 and 5 review commands that may appear in the script. Chapter 6 briefly describes the equations that form the basis for **HiPhi** and reviews Dirichlet and Neumann boundary conditions. Chapter 7 describes features of the program when run as an



**Figure 1.1.** Geometry defined by the **MetaMesh** script WALKTHRU.MIN

interactive Windows utility. Chapter 8 describes how to run complex simulations automatically under batch file control. The chapter reviews features of the Field Precision utility program **GCon**. Chapters 9 through 13 describe the analysis and plotting capabilities of the **PhiView** postprocessor. Finally Chapter 14 summarizes the format of the **HiPhi** output file. This information is useful if you want to write your own analysis programs.

## 1.2. Test run

A quick way to understand the solution procedure is to step through an example. The calculation we shall consider describes a spherical high-voltage electrode on a nylon support at an offset position inside a rectangular grounded chamber. The calculation utilizes a symmetry boundary to minimize the run time. Figure 1.1 shows the geometry of the electrode and support. The **MetaMesh** script file is listed in Table 1.1.

**Table 1.1. MetaMesh script file for WALKTHRU.MIN**

<pre> * File WALKTHRU.MIN GLOBAL XMesh   0.00  9.00  N 90 End YMesh   0.00  3.00  N 30 End ZMesh   0.00  6.00  N 60 End RegName 1 Solution volume RegName 2 Dielectric support RegName 3 Spherical electrode RegName 4 Grounded boundary END  PART 1 * Solution region Type Box Region 1 Fab 18.0  6.00  12.00 END  PART 2 * Dielectric support Type Cylinder Region 2 Fab 0.80 4.00 Shift 4.00 0.00 0.00 Surface Region 1 END  PART 3 * Spherical electrode Type Sphere Region 3 Fab 1.5 Shift 4.00 0.00 3.00 Surface Region 1 Surface Part 2 END </pre>	<pre> PART 4 * Boundary, upper Y Type BoundYUp Region 4 END  PART 5 * Boundary, upper X Type BoundXUp Region 4 END  PART 6 * Boundary, lower X Type BoundXDn Region 4 END  PART 7 * Boundary, upper Z Type BoundZUp Region 4 END  PART 8 * Boundary, lower Z Type BoundZDn Region 4 END  ENDFILE </pre>
---	---

To prepare move the example files WALKTHRU .MIN, WALKTHRU .HIN and WALKTHRU .SCR to the working directory (*i.e.*, \AMAZE\BUFFER). Run AMAZE .EXE (the **AMaze** program launcher) and click on *Set data directory*. Move to \AMAZE\BUFFER and click *OK*. Then launch **MetaMesh**. Pick *Load MIN file* from the *File* menu. If **AMaze** is set up correctly, the program transfers to \AMAZE\BUFFER and the file

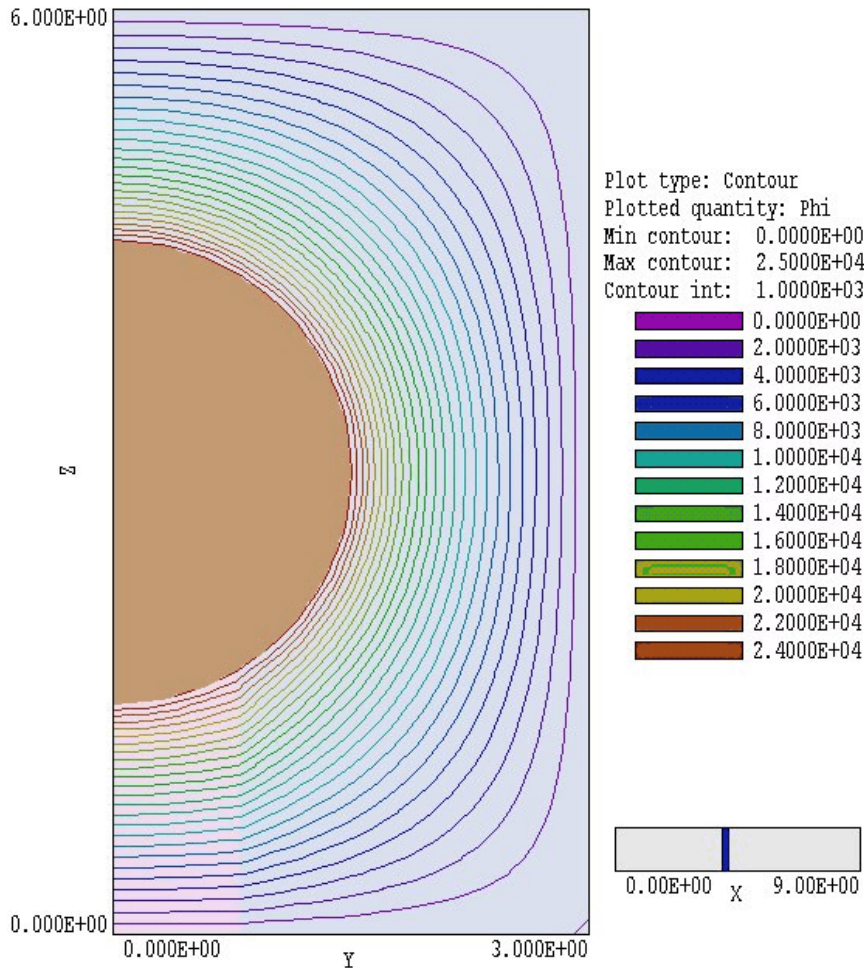
WALKTHRU.HIN appears in the dialog. Open the file.

Next, pick *Process mesh*. The program lists information on the conversion process on the screen and in the file WALKTHRU.MLS. Click the right mouse button to continue. Finally, pick *Save mesh* from the *File* menu. At this point, you can use the plot functions of **MetaMesh** to examine the geometry of the mesh or proceed to the electrostatic solution.

Run **HiPhi** from **AMaze**. After the program starts, pick *Run* from the main menu or use the run tool. In the dialog, pick the file WALKTHRU.EIN. Table 1.2 shows the contents of the file. **HiPhi** calculates the element matrices and coupling coefficient necessary for the finite-element solution and then proceeds with the relaxation process. The program creates the binary output file WALKTHRU.HOU and the ASCII listing file WALKTHRU.HLS. The run takes about 60 seconds on a 1.7 GHz Pentium 4 computer. You can use the *Edit listing file* command to inspect WALKTHRU.ELS.

**Table 2.1 HiPhi script WALKTHRU.HIN**

```
* File WALKTHRU.HIN
ResTarget = 1.0E-7
MaxCycle = 1000
DUnit = 100.0
NCheck = 25
* Solution volume
Epsi(1) = 1.0
* Dielectric support
Epsi(2) = 3.5
* Spherical electrode
Potential(3) = 25000.0
* Boundary
Potential(4) = 0.0
ENDFILE
```

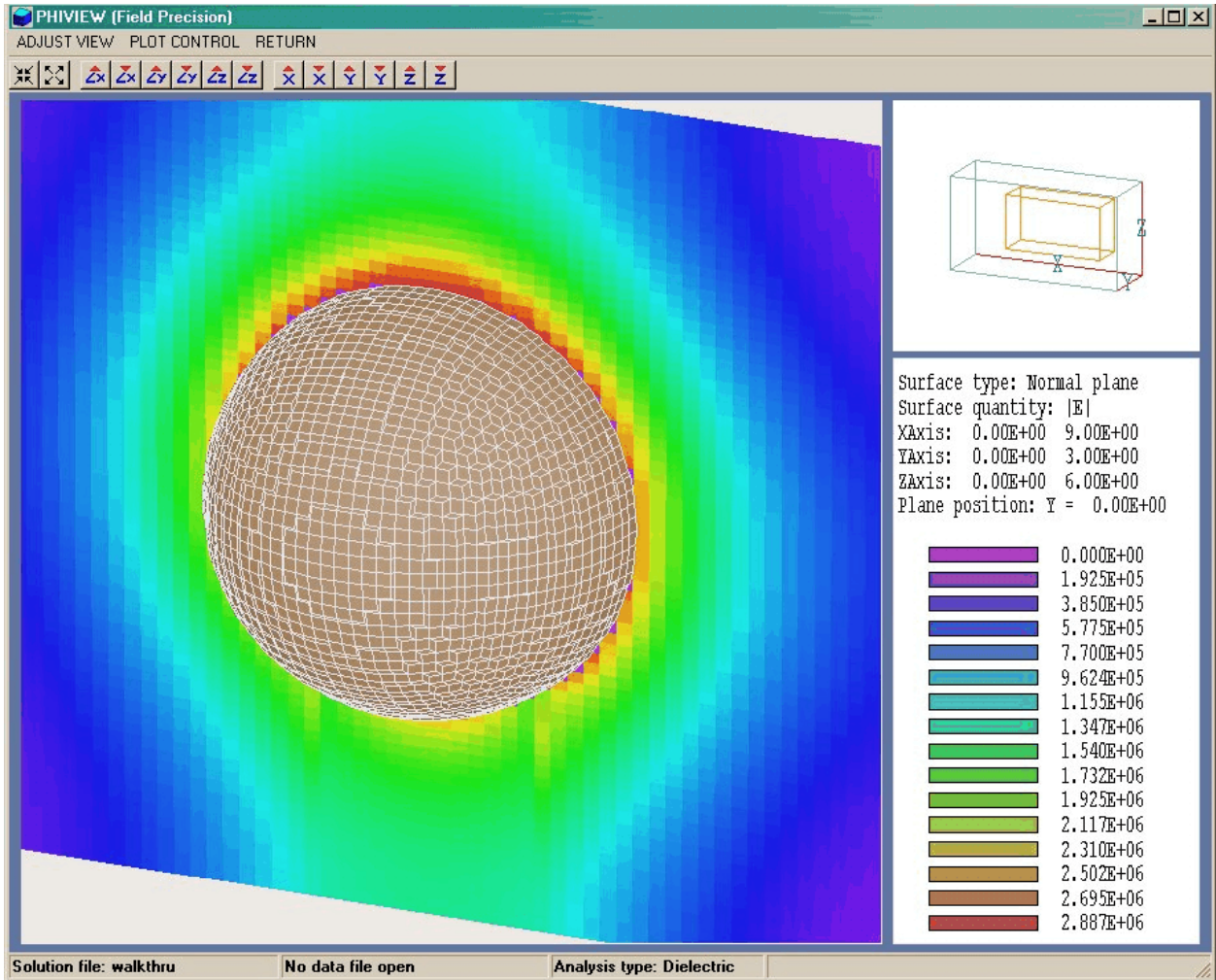


**Figure 1.2.** Slice plot normal to the  $x$  axis at position  $x = 4.05$

Following a successful solution, start the program **PhiView**. Pick *Load solution file* from the *File* menu or click on the tool. Pick the file WALKTHRU.HOU in the dialog. **PhiView** can generate a wide variety of 2D and 3D plots. Chapters 10, 11 and 12 give detailed information. Here, we will look at two examples. To create the 2D equipotential plot of Fig. 1.2, go to the *Slice plot* menu. On the *Change view* popup menu, pick the command *Set slice plane properties*. Click the *X Axis* button, move the slider to  $x = 4.05$ , and then click OK. You should see the equipotential plot of Fig. 1.2. Note the slope discontinuity of equipotential lines at the edge of the dielectric.

**PhiView** also has 3D plot capabilities. Return to the main menu and click on *Surface plots*. On the *Plot control* popup menu click on *Surface plot style*. The default plot shows the variation of electric field in a plane





**Figure 1.3.** Working environment of **PhiView** showing a surface plot of  $|E|$  in a normal plane.

normal to  $z$ . Click the button for the  $y$  axis, move the slider to  $y = 0.0$  and click *OK*. Next, we shall add the spherical electrode to the plot for reference. On the *Plot control* menu click on *Displayed regions*. Activate Region 3 and click *OK*. To get a better view, we need to rotate the plot. On the *Adjust view* menu click on *Set surface view*. Set the view angles to approximately  $\theta_x = -80.0^\circ$ ,  $\theta_y = 0.0^\circ$  and  $\theta_z = -150.0^\circ$ . With some tuneup using the zoom and displacement tools, you should see the plot of Fig. 1.3. The figure shows the variation of  $|E|$  over the symmetry. The dielectric support is at the bottom.

You can experiment with commands to move through the solution space or to change the style of the plot. You can also check the *Hardcopy* or *Plot file* functions in the *Export plot* popup menu. Remember to set the correct

printer in Windows before using the *Default printer* command..

Plots are pleasing, but the ultimate aim of the program is to generate numbers. Return to the main menu and click on *Run script* in the *File* menu. In the dialog, pick the file WALKTHRU . SCR. There is a delay while **PhiView** performs an extensive analysis. To begin, we shall inspect the instructions in the data script. In the *File* menu click on *Edit script* and choose WALKTHRU . SCR. The internal program editor loads and shows the following content:

```
INPUT WalkThru.HOU
OUTPUT WallThru
FULLANALYSIS
REGION 3
ENDFILE
```

The first statement ensures that the proper solution file is loaded and the second statement opens a data file to write the results. The command *FullAnalysis* instructs the program to take volume integrals over all regions of the solution volume, while the command *Region 3* initiates volume and surface integrals over the electrode region. To inspect the new file WALKTHRU . DAT click on the command *Edit data file* in the *File* menu. Choose the file and click *OK*.

Table 1.3 summarizes results listed in the file. HiPhi has computed the volume of regions by taking Gaussian integrals over individual hexahedron elements. The computed volume of the spherical electrode (Region 3) is  $7.0657 \times 10^{-6} \text{ m}^3$ , close to the theoretical value of the half-sphere of radius 1.5 cm:  $7.0686 \times 10^{-6} \text{ m}^3$ . The computed total electrostatic field energy is  $4.8640 \times 10^{-4} \text{ J}$  at an applied potential of  $2.5 \times 10^4 \text{ V}$ . Using the expression  $E = CV^2/2$ , we find that the capacitance of the half-assembly is 1.5565 pF. Next consider the surface integral values determined in response to the command *Region 3*. The calculated surface area is  $1.4271 \times 10^{-3} \text{ m}^2$  while the theoretical prediction is  $1.4137 \times 10^{-3} \text{ m}^2$ . We can also double-check the capacitance calculation using the surface charge value determined by integrating the normal electric field over the surface of the electrode. The calculated result is  $3.9776 \times 10^{-8} \text{ coulombs}$ . The formula  $C = Q/V$  gives the capacitance as 1.5910 pF, within 2% of the volume integral result.

**Table 1.3. Information in the file WALKTHRU.DAT**

```
----- Global analysis of the solution volume -----  
  
      Fixed potential regions  
NReg  Potential      Volume  
      (V)            (m3)  
=====
```

3	2.50000E+04	7.06573E-06
4	0.00000E+00	0.00000E+00

```
=====
```

NReg	Dielectric Epsilon (relative)	regions Rho (c/m3)	E  (min) (V/m)	E  (max) (V/m)	Energy (J)
1	1.00000E+00	0.00000E+00	1.22847E+01	3.75397E+06	4.23005E-04
2	3.50000E+00	0.00000E+00	1.72222E+05	2.94126E+06	6.33973E-05

```
-----
```

Global field energy: 4.86402E-04 J  
Global volume: 1.61998E-04

```
----- Analysis of Region Number 3 -----  
  
Fixed potential: 2.50000E+04  
Volume integral quantities  
Volume: 7.06573E-06 (m3)  
Surface integral quantities  
Surface area: 1.42706E-03 (m2)  
Induced change: 3.97756E-08 (coulombs)
```

## Chapter 2. Organizing AMaze simulations

The **HiPhi** package consists of the program **HiPhi** that computes the physical solution and **PhiView**, a dedicated post-processor for analyses of the results. The solution program can run in two modes: interactively in a window or autonomously in the background under batch file control. Similarly, **PhiView** 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 **HiPhi** solution:

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

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

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

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

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

Table 2.1 summarizes the input and output file used in **HiPhi**.

<b>Table 2.1. HiPhi files</b>		
<b>Name form</b>	<b>Function</b>	<b>Status</b>
<code>FPREFIX.MIN</code>	Description of simulation geometry, input to <b>MetaMesh</b>	<i>Required</i>
<code>FPREFIX.MDF</code>	Output from <b>MetaMesh</b> , input to <b>HiPhi</b>	<i>Required</i>
<code>FPREFIX.HIN</code>	Description of material properties, input to <b>HiPhi</b>	<i>Required</i>
<code>FPREFIX.HOU</code>	Output from <b>HiPhi</b> , input to <b>PhiView</b>	<i>Required</i>
<code>ANYPREFIX.SCR</code>	Analysis control, input to <b>PhiView</b>	<i>Optional</i>
<code>ANYPREFIX.DAT</code>	Analysis data output from <b>PhiView</b>	<i>Optional</i>

## Chapter 3. Script commands for run control

### 3.1. Script conventions

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

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

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

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

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

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

have the same effect.

**HiPhi** 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.

**HiPhi** 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 **HiPhi**:

```
* File XFORM01.HIN
DUnit = 39.37
ResTarget = 5.0E-8
MaxCycle = 1000
NCheck = 1
Omega = 1.80
* Region assignments
* 1: Solution volume, oil
* 2: Casing, ground
* 3: Dielectric coil support
* 4: Oil penetration
* 5: High voltage electrode
Epsi(1) = 2.1
Potential(2) 0.0
Epsi(3) = 5.3
Epsi(4) = 2.1
Potential(5) = 380.0E3
EndFile
```

You can 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.

## 3.2. Program control commands

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

### **ResTarget: 5.0E-6**

The numerical calculation of electrostatic fields requires the solution of a large set of coupled linear equations, one for each active node in the solution volume. **HiPhi** uses an iterative technique based on corrections that reduce the error in electrostatic 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 HiPhi in the interactive mode by using the *Stop and save run* or *Abandon run* menu options. Default value: *ResTarget* =  $5.0 \times 10^{-7}$ .

### **MaxCycle 2500**

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

### **NCheck = 20**

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

### **Omega: 1.92**

The real-number parameter *Omega* is the over-relaxation factor used to correct potential errors during the solution. This quantity may have a significant effect on the run time. If the *Set omega* command does not appear, **HiPhi** automatically picks values that vary with the iteration cycle following the Chebyshev acceleration 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.

### **DUnit = 100.0**

Internally **HiPhi** uses SI units, with lengths in meters (m), potential in volts (V), electric field in volts/meter (V/m), conductivity in siemens per meter (S/m) and charge density in coulombs per cubic meter (coulomb/m<sup>3</sup>). Use the *Set DUnit* command to work in alternate length units. The real-number quantity is the number of working units per meter. For example, if you used dimensions of centimeters in the **MetaMesh** script, set *DUnit = 100.0*. Alternatively, if the mesh dimensions are in inches, use *DUnit = 39.37*. The quantity *DUnit* is recorded in the output file and is used in **PhiView** for the input and output of positions. Default: *DUnit = 1.0*.

### **Format Text**

By default, **HiPhi** creates output files *FPREFIX.HOU* in binary format. This format occupies less disk space. Binary output files are required if you want to use **PhiView** to analyze the solution. We have included an option to create output files in text format to make it easier to port results to your own analysis programs. The string parameter can assume the values *BINARY* or *TEXT*. Chapter 13 describes the file structure for both formats.

### **NGauss 3**

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

## Chapter 4. Commands to set material properties

Material properties in electrostatic solutions are relatively simple so only a few commands are required. HiPhi handles either dielectric-type solutions or conductive type solutions (Chapter 5). In a dielectric-type solution you can set both the relative dielectric constant and space-charge density of a filled region. In a conductive type solution you can specify the conductivity of a filled region. HiPhi issues an error message if you mix quantities in a script. Both symbolic and sample forms given for the commands.

**POTENTIAL RegNo Phi**  
**POTENTIAL(5) = 2500.0**

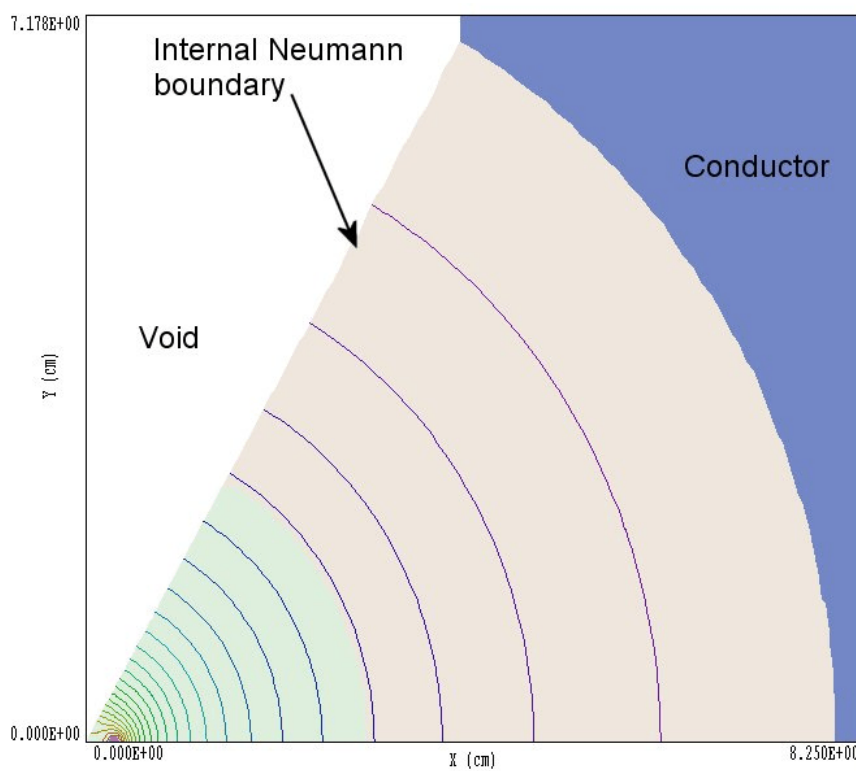
This command sets all elements or nodes associated with a region number to a fixed potential condition. The integer parameter is a region number defined in **MetaMesh**. The real number is the value of electrostatic potential in volts (V). The fixed-potential condition can be applied to both filled and open regions.

**EPSI RegNo EpsiRel**  
**EPSI(4) = 12.6**

This command sets a relative dielectric constant for all elements associated with a filled region. The integer parameter is the number of a region defined in **MetaMesh**. The second parameter is the value of relative dielectric constant  $\epsilon/\epsilon_0$  (dimension less). 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 excessive values of  $\epsilon_r$  (*i.e.*,  $10^8$ ) may slow the solution convergence. The default for all unspecified regions is  $\epsilon/\epsilon_0 = 1.0$ .

**RHO RegNo ChgDens**  
**RHO 3 4.5E-10**

This command assigns a space-charge density to elements within the specified filled region. The real number parameter is the value of  $\rho$  in coulombs/m<sup>3</sup>. A script may contain both *Rho* and *Epsi* commands for the same region. The default for all unspecified regions is  $\rho = 0.0$ .



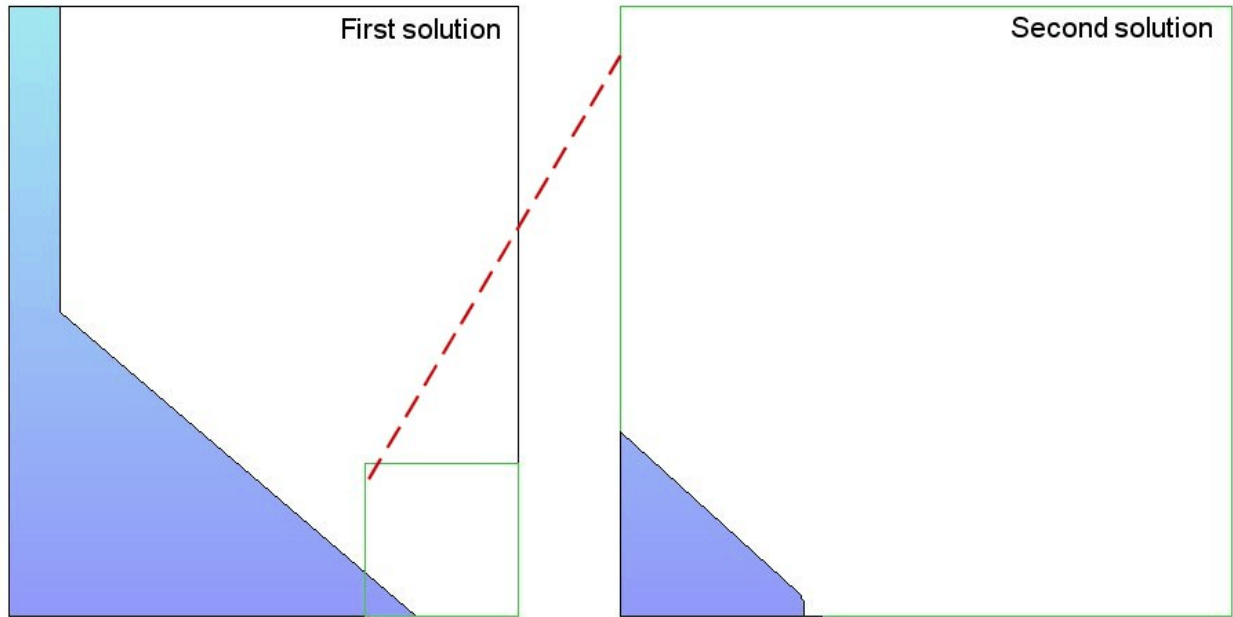
**Figure 4.1.** Use of a void to define a slanted Neumann boundary

**SIGMA RegNo ChgDens**  
**SIGMA(5) = 0.682**

This command assigns a conductivity to elements within the specified filled region. The real number parameter is the value of  $\sigma$  in siemens/m. To model a perfect insulator immersed in a conducting medium, assign a low but non-zero value of  $\sigma$  in the region (i.e.,  $1.0^{-4}$ ). To model a floating electrode, assign a relatively high value of conductivity. The default for all unspecified regions is  $\sigma = 1.0$  S/m.

**VOID RegNo**  
**VOID (4)**

A void material has the property  $\epsilon_r = 0.0$ . Electric field lines in the medium adjacent to a void are parallel to the boundary and equipotential lines are normal. Voids can be used to define symmetry boundaries that are not parallel to one of the axes. For example, Figure 4.1 shows a simulation of a 3-wire probe array in one sextant of the  $x$ - $y$  plane. Internal voids can also be used to represent ideal insulators of any shape in conductive solutions. As shown in the figure, **PhiView** does not calculate or plot electrical quantities inside a void.

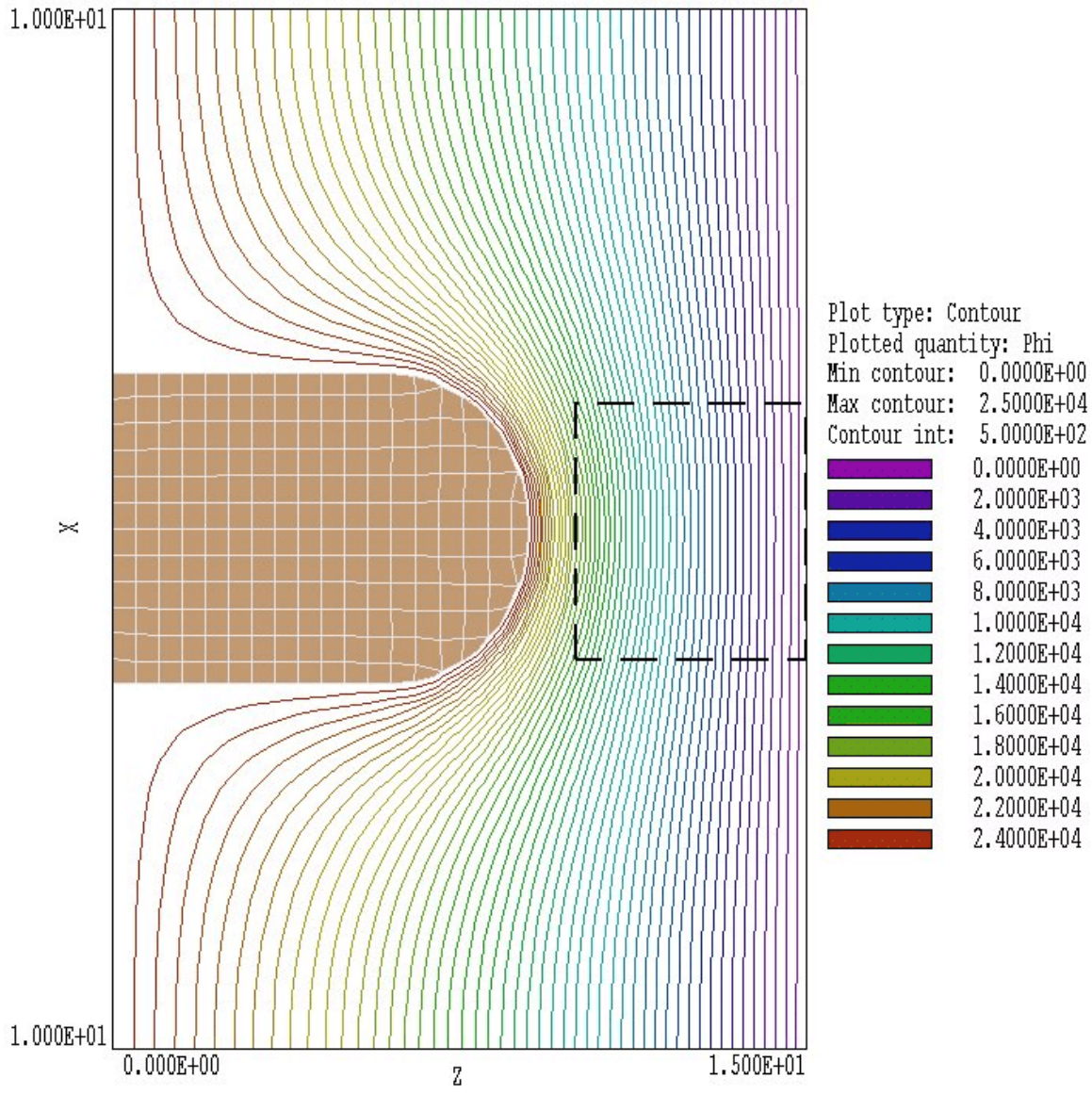


**Figure 5.1.** Creating a microscopic solution using the *BOUNDARY* command

## Chapter 5. Modeling small details in a large-scale solution

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

Figure 5.1 illustrates an alternate approach. We create a second solution that extends over a small subregion of the original solution (green outline). The microscopic solution contains any electrodes that are located within the subregion. The difference is that the electrode shapes may be resolved in much greater detail. The question is how to incorporate the macroscopic fields correctly into the microscopic solution. One resolution is to enclose the second solution in a variable-potential Dirichlet boundary. Values of  $\phi$



**Figure 5.2.** Equipotential lines and element resolution in the large-scale solution *BOUNDTEST01*. The dashed line shows the boundary of the small-scale solution.

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

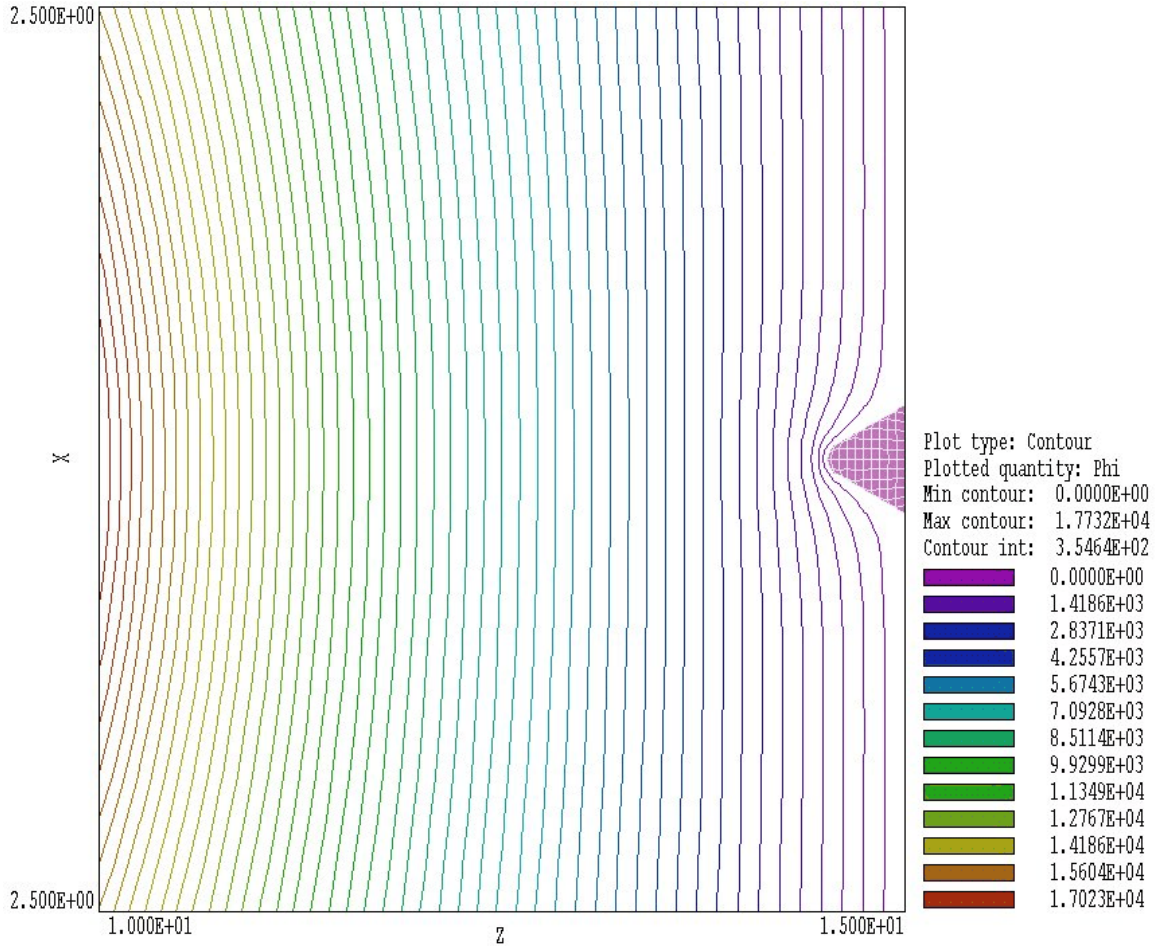
Definition of the variable Dirichlet boundary is controlled by the following command that appears in the control script of Solution 2:

**BOUNDARY FPrefix [BndScale]****BOUNDARY FEmitMacro**

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

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

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



**Figure 5.3.** Equipotential lines and element resolution in the small-scale solution *BOUNDTEST02*.

```

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

```

Figure 5.3 shows equipotential lines and the element division of the protrusion in the microscopic electrostatic solution, a subset of the original. The surface of the protrusion is well-resolved and electric field interpolations in the vicinity will be accurate. The process is valid because the presence of the small protrusion does not significantly influence values of potential at the boundary of Solution 2.

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

**SUPERPOSITION FPrefix [SuperScale]  
SUPERPOSITION UniField**

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

When this command is issued, **HiPhi** opens the file *FPrefix.HOU* after Solution 2 has been completed. The program performs an interpolation in the space of Solution 1 to determine the potential  $\phi_1$  at each node location in Solution 2 and adjusts potential values according to

$$\phi'_2 = \phi_2 + SuperScale \times \phi_1 , \quad (1)$$

before writing the output file.

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



## Chapter 6. Theoretical basics

### 6.1. Dielectric and conductive solutions

This section clarifies the two types of solutions encountered in electrostatics - dielectric and conductive. To begin, consider a volume with electrodes and ideal dielectrics (*i.e.*, zero conductivity). If we apply a pulsed voltage, the distribution of charges on electrodes and hence the fields in the volume are determined by capacitive coupling. The fields for the dielectric-type solution are given by the Poisson equation

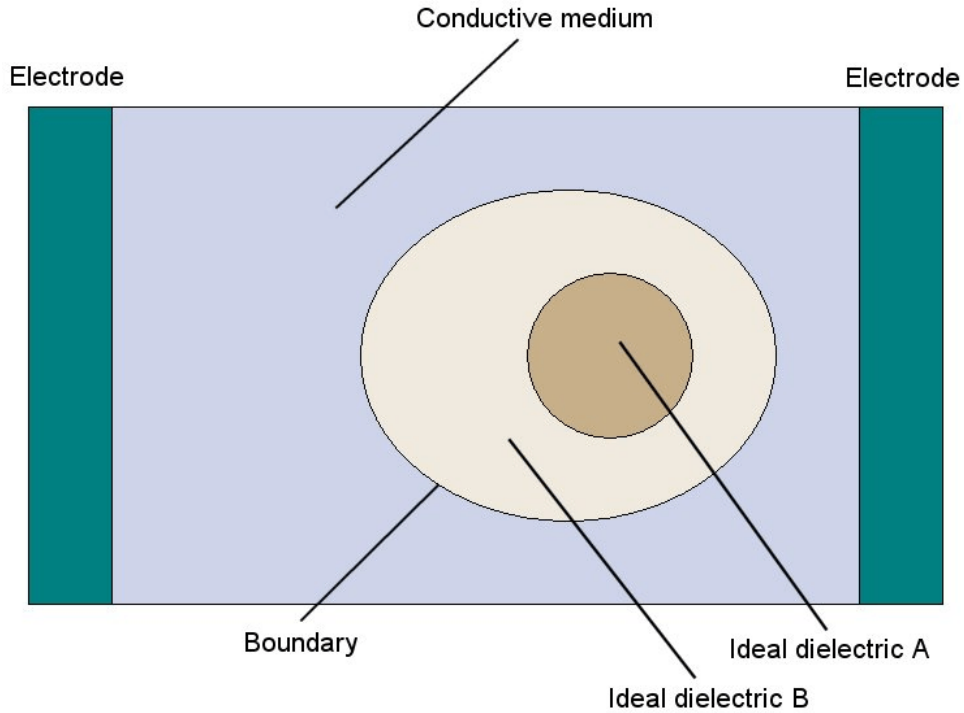
$$\nabla \cdot (\epsilon_r \nabla \phi) = -\rho / \epsilon_0 , \quad (6.1)$$

where  $\phi$  is the electrostatic potential,  $\epsilon_r$  is the relative dielectric constant,  $\rho$  is the space-charge density and  $\epsilon_0 = 8.854 \times 10^{-12}$  F/m. Next consider a system with imperfect dielectrics that non-zero conductivity. If we apply a rapidly-pulsed voltage, the charge distribution on electrodes early in time is determined by capacitive coupling. Later the initial charge relaxes by flow through the conductors and the distribution is determined entirely by the media conductivities. Therefore, we seek a dielectric-type solution when the media in the solution volume have zero conductivity or when the voltage pulse is short. The resistive relaxation time for a material with relative dielectric constant  $\epsilon_r$  and conductivity  $\sigma$  is given by

$$t_{RC} = \frac{\epsilon_r \epsilon_0}{\sigma} . \quad (6.2)$$

As an example, consider water with  $\epsilon_r = 81.0$  and volume resistivity of 1 M $\Omega$ -cm ( $\sigma = 10^{-4}$  S/m). Equation 6.2 implies a relaxation time of 7  $\mu$ s. Therefore the dielectric model would be justified if we apply a 200 ns voltage pulse to a small volume of water.

Conductive effects always dominate for long time scales and true static solutions. We need to qualify this statement for the special case of a conductive system with regions of multiple ideal dielectrics as in Fig. 6.1. We can determine the static field solution in the conductive regions by treating the dielectrics as perfect insulators. On the other hand, the field solution inside the dielectric region depends on the spatial variation of dielectric constant. Here the conductive solution determines a fixed



**Figure 6.1.** Example of a mixed dielectric-conductive electrostatic solution.

potential distribution over the dielectric boundary. We solve for the fields inside by applying Eq. (6.1) with a given potential distribution of the boundary (Dirichlet condition).

It is straightforward to use an electrostatic code for either dielectric or conductive solutions. For a dielectric-type solution with no space charge, **HiPhi** solves the Laplace equation

$$\nabla \cdot (\epsilon_r \nabla \Phi) = 0. \quad (6.3)$$

In a conductive medium, the current density is related to the gradient of potential by

$$\mathbf{j} = \sigma \nabla \Phi. \quad (6.4)$$

Conservation of charge in the medium implies that

$$\nabla \cdot \mathbf{j} = 0, \quad (6.5)$$

or

$$\nabla \cdot [\sigma \nabla \phi] = 0. \tag{6.6}$$

Equation (6.6) has the same form as Eq. (6.3) with the substitution

$$\epsilon_r \Rightarrow \sigma. \tag{6.7}$$

Table 6.1 reviews how to represent materials in the two solution types.

Table 6.1. Material types in dielectric and conductive solutions		
Object	Dielectric-type solution	Conductive-type solution
Metal electrode	Fixed potential region	Fixed potential region
Insulator		$\sigma \ll 1$
Floating conductor	$\epsilon_r \gg 1$	$\sigma \gg 1$

## 6.2. Boundary conditions

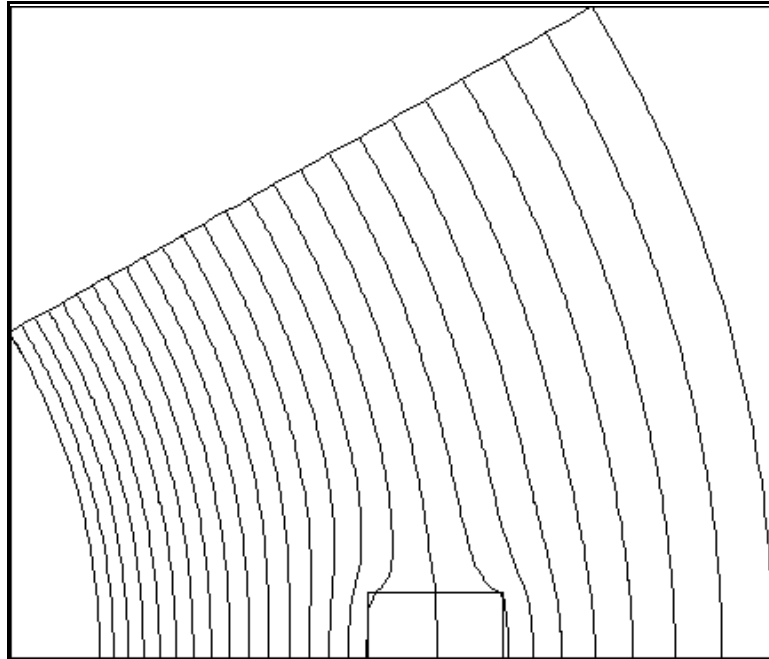
The *boundary* is the outer edge of the solution volume. For electrostatic problems, node potentials on the boundary can be set to one of two conditions.

### Dirichlet

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

### Neumann

A Neumann boundary is one where the normal derivative of the potential is specified. The boundaries in **HiPhi** 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



**Figure 6.2.** Special Neumann boundaries.

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. 6.2 shows an equipotential plot for a solution of the fields between two spherical electrodes with a dielectric inclusion. By utilizing automatic Neumann boundaries at the top and bottom, it is necessary to simulate on part of the interelectrode volume. A special Neumann boundary was also employed in the `WALKTHRU` example of Chapter 1

## Chapter 7. Running HiPhi interactively

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

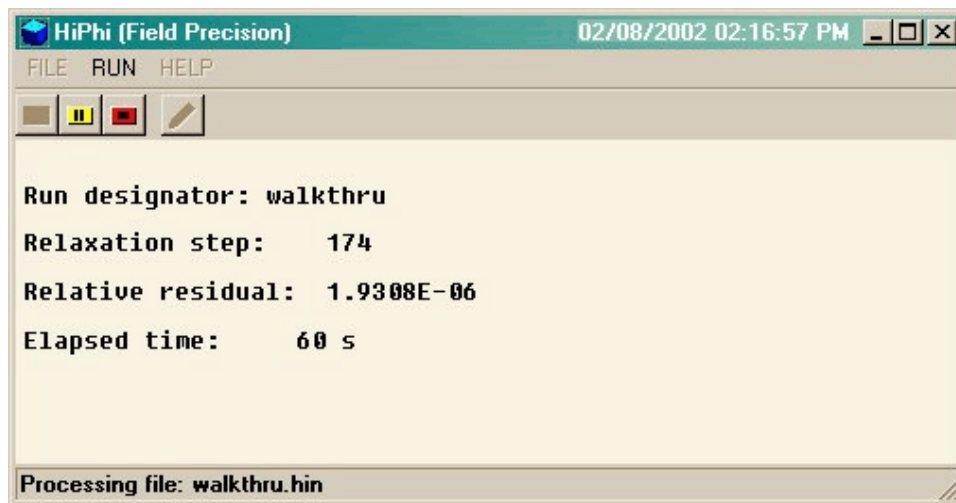


Figure 7.1. HiPhi display during a solution

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

### **Edit script file**

### **Edit listing file**

### **Edit file**

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

The *Run* menu has three commands.

**Start run**

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

**Pause run**

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

**Stop run**

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

The *Help* menu has a single command.

**Instructions**

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

## Chapter 8. Automatic runs under batch file control

### 8.1 Command line operation

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

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

```
..\HIPHI SWITCH <Enter>
```

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

### 8.2. Batch files

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

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

Type

```
SW_RUN <Enter>
```

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

### 8.3. Introduction to GCon

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

```
START /WAIT ProgName
```

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

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

#### **CD**

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

**Example:** `CD C:\AMaze\Buffer`

#### **COPY ... TO ...**

**Function:** Copies a file to another location.

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

#### **ERASE**

**Function:** Deletes a file.

**Example:** `ERASE diode.mls`

#### **REM**

**Function:** Displays the line, no action taken



### **MOVE ... TO ...**

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

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

### **RENAME**

**Function:** Renames a file or files.

**Example:** RENAME diode.mou diode\_compare.mou

### **RUN**

**Function:** Runs a specified program or command.

**Example:** RUN C:\tricom\mesh diode

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

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

## **8.4. Running GCon**

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

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

## Chapter 9. PhiView - file operations menu

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

### Load solution file

The *Load solution file* command displays a dialog (Fig. 9.1) with a list of solution files with names of the form `FPREFIX.HOU`. 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 displays information on the currently-loaded data file (Fig. 9.2)

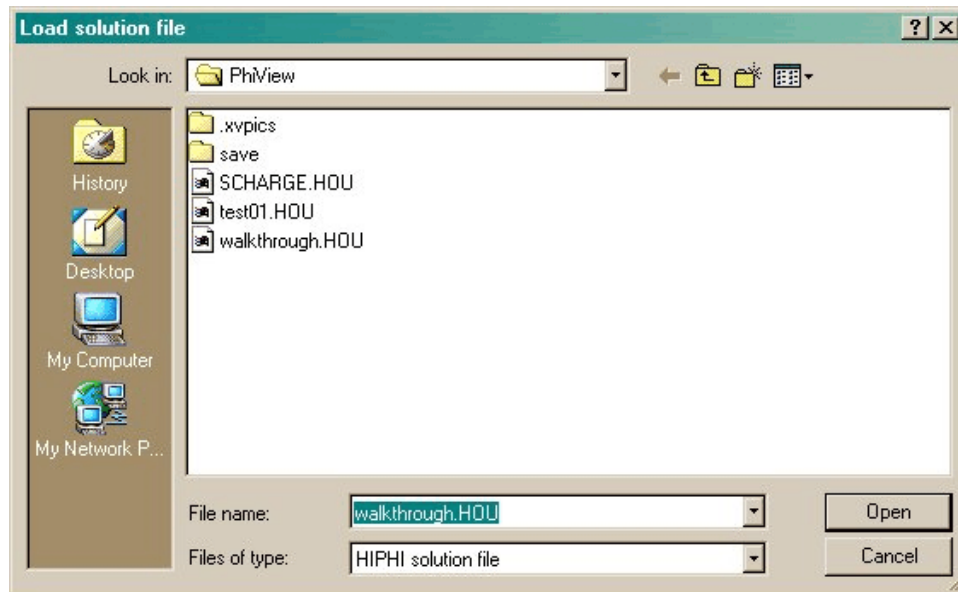
### Analysis type

Click on this command to toggle between dielectric and conductive type solutions. The choice affects the plotting options and listed quantities in the data file. For example, in a conductive analysis **PhiView** performs integrals of resistive power dissipation and total electrode current.

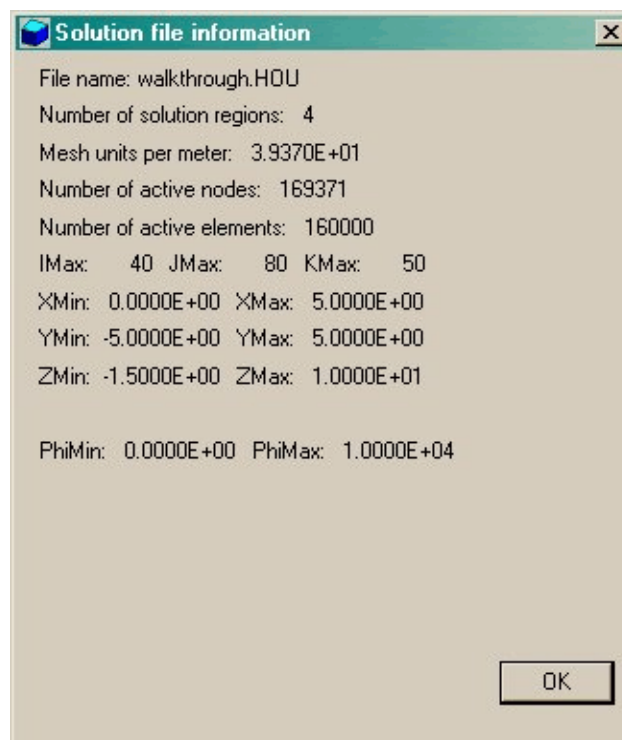
The status bar shows the current analysis type.

### Run script

Sometimes you may want to perform complex or repetitive analyses on a set of similar solutions. Script file operation is a powerful feature of **PhiView**. This command displays a dialog with a list of analysis files that you have created with the suffix `SCR`. Pick a file and click *OK*. The script file can load data files, open and close history files, and perform any of the quantitative analysis functions described in this



**Figure 9.1.** Load solution file dialog



**Figure 9.2.** Solution file information display

manual. Chapter 9 describes the uniform script command language used in all AMaze postprocessors. Note that the script file should be in the same directory as the data files.

### **Create script**

This command allows you to create script files using the internal editor. A dialog box requests a file prefix. The resulting script file will be saved as `FPREFIX.SCR`. Next, the program opens the file in the editor and writes a reference list of allowed commands. This list follows the *EndFile* command and will be ignored. Enter analysis commands above the *EndFile* command. After saving the file, you can run it using the *Run script* command.

### **Edit script**

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

### **Open data file**

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

### **Close data listing file**

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

### **Edit data file**

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

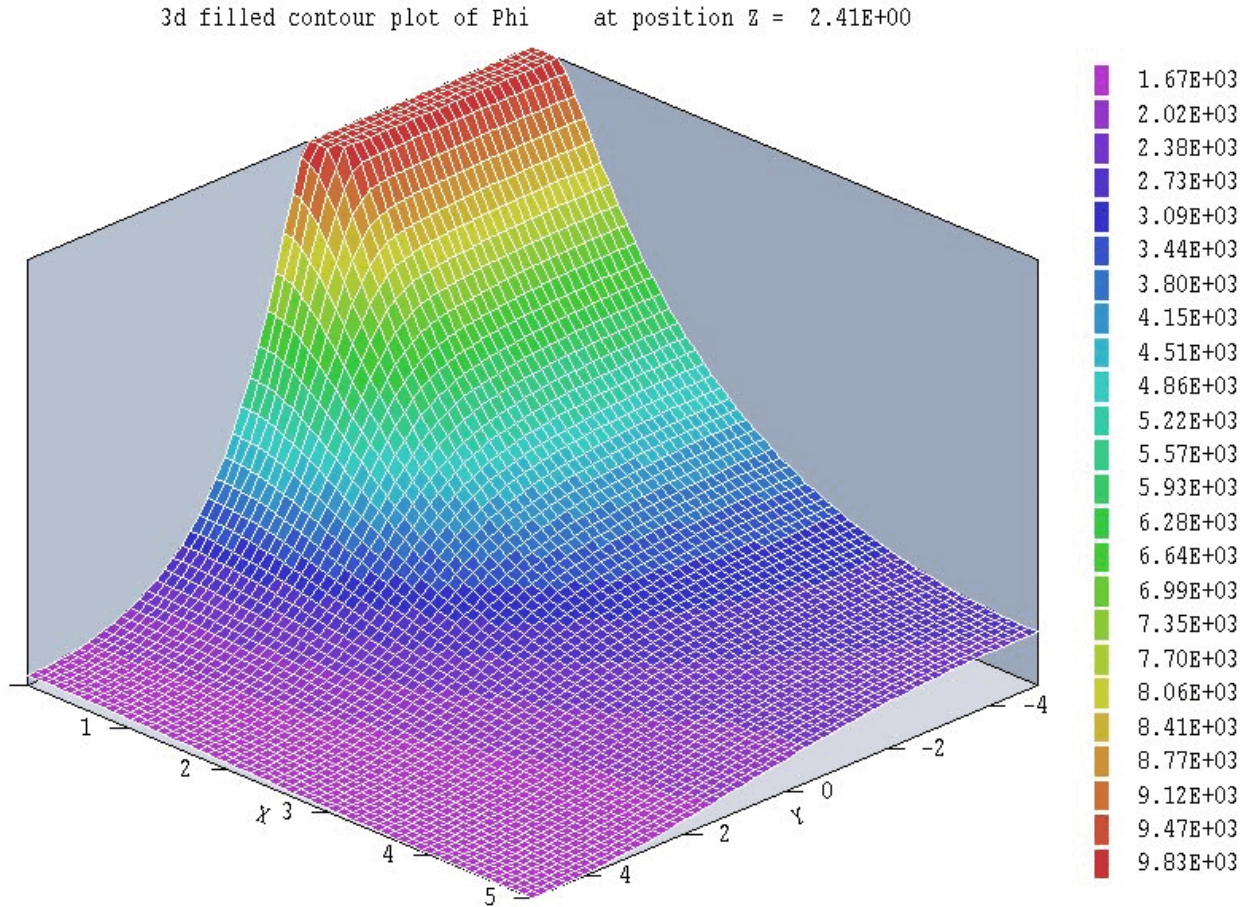
**Edit file**

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

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

**Instructions**

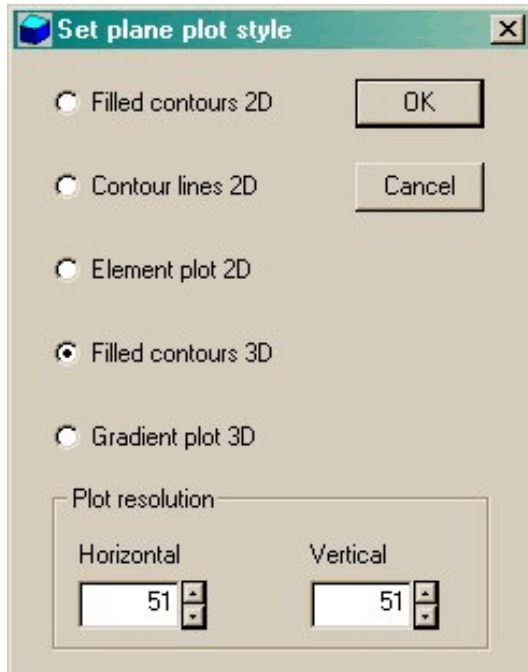
Displays the HTML help file using your default browser. In order for this command to operate correctly, the file `HIPHI.HTML` must be in the same directory as `PHIVIEW.EXE`.



**Figure 10.1.** Plane plot of electrostatic potential - type 3D filled contour

## Chapter 10. PhiView - plane plots

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



**Figure 10.2.** Set plane dialog

## Set plane

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

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

## Plot style

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

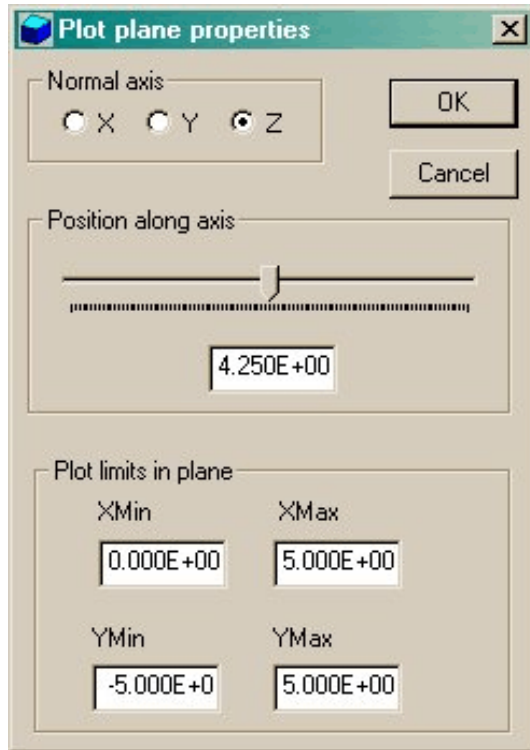


Figure 10.3. Plot style dialog

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

### Plot quantity

Set the quantity to be plotted. The choices are  $\phi$  (electrostatic potential) and  $|\mathbf{E}|$  (electric field) in dielectric-type analyses and  $\phi$ ,  $|\mathbf{E}|$  and  $|\mathbf{j}|$  (current density) in conductive analyses. Note that electric field and current density plots may appear jagged at discontinuities because of the limited resolution of the mesh.

### Plot limits

Set limits for the plotted quantity ( $\phi$ ,  $|\mathbf{E}|$  or  $|\mathbf{j}|$ ). When Autoscale is activated, **PhiView** 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* plot styles. You can rotate the plot in  $90^\circ$  increments for the best view.



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

### **Default printer**

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

### **Plot file (EPS)**

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

### **Plot file (BMP)**

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

### **Copy to clipboard**

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

The screen capture capability of PaintShop Pro (<http://www.jasc.com>) is a useful alternative to transfer screen plots to documents. We used this method to create the illustrations for this manual.

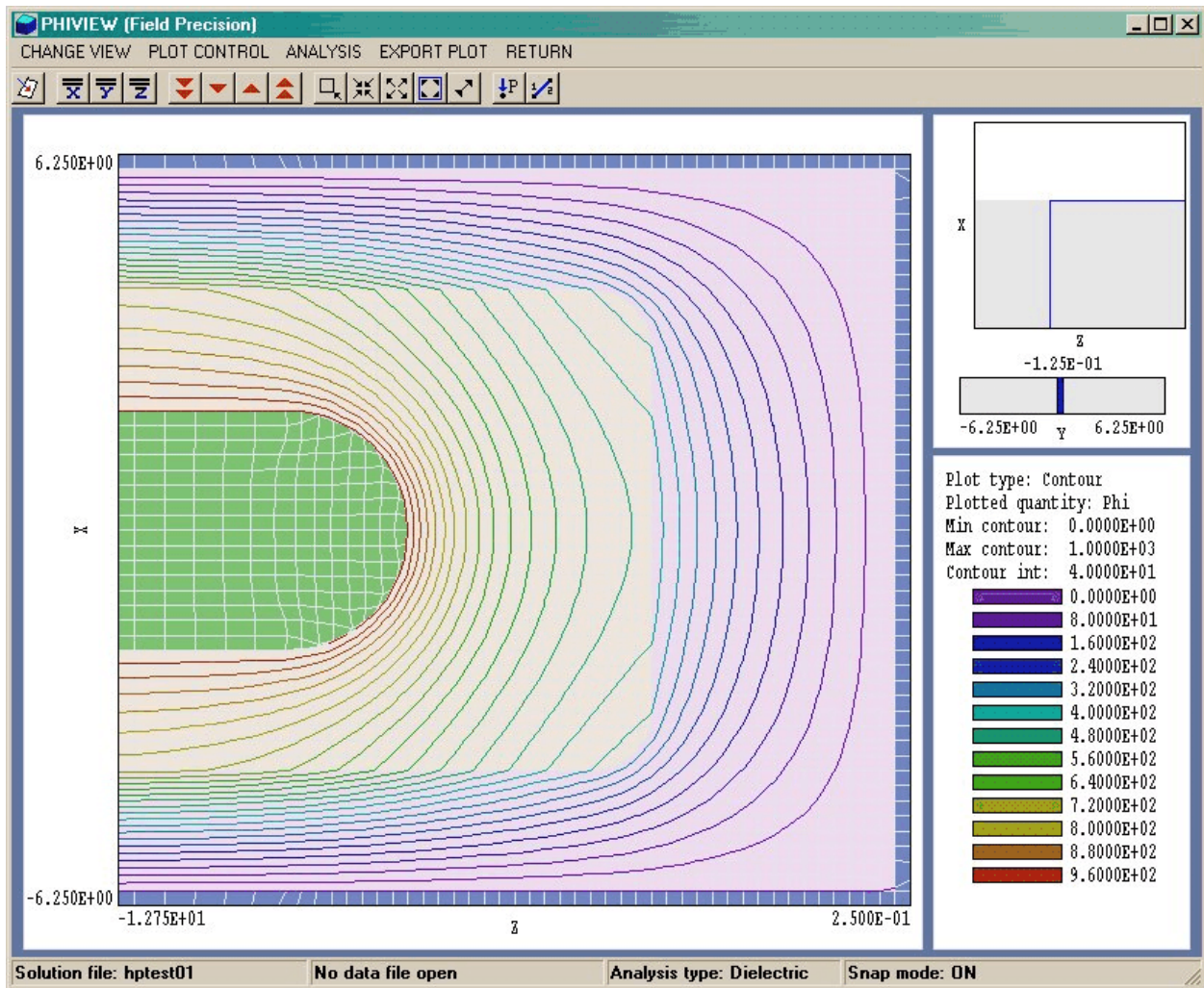


Figure 11.1. Slice plot - equipotential lines

## Chapter 11. PhiView - slice plots

### 11. 1. Setting the slice view

Slice plots are two-dimensional plots that show the variation of quantities over a plane normal to one of the Cartesian axes. In contrast to plane plots, slice plots are based on the structure of the mesh projected to a slice plane. This structure may be quite complex for a conformal mesh; therefore, slice plots require more computational effort. To facilitate the process, slices are constructed at discrete locations along the normal axis corresponding roughly to the planes of the foundation mesh. The precise rendering of spatial information allow 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. 9.2). You can change the normal axis, change the position along the normal axis, and set plot limits in the normal plane.

### **Slice normal to X**

### **Slice normal to Y**

### **Slice normal to Z**

Quick commands to change the normal axis.

### **Jump forward**

### **Step forward**

### **Step backward**

### **Jump backward**

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

### **Zoom window**

As an alternative to the entries in the *Set slice plane* dialog, you can interactively change plot limits in the normal plane using the mouse. Choose the command and move the mouse pointer into the plot area. The status bar enters coordinate mode. It shows the current mouse position in the plot and the snap mode. Use the left button to pick one corner and then move the mouse to create a view box. Click the left button again, and the plot regenerates. On any coordinate operation, press the *F1* key if want to enter values from the keyboard. Note that the normal plane box in the orientation area to the right of the plot (Fig. 11.1) 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 a relative starting and ending point for the shift.

## 11.2. Setting slice plot properties

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

**Set slice plot style**

This command brings up the dialog of Fig. 11.2. The *Region plot* style is a cross-section view of the mesh element divisions color-coded by region. In contrast to the logical plane plot of **MetaMesh**, **PhiView** attempts to resolve the exact mesh structure in the plane. The *Contour plot* style is an equipotential plot. The vector component of electric field in the normal plane is perpendicular to the equipotential lines and  $|\mathbf{E}|$  is inversely proportional to the distance between lines. The two options at the bottom of the dialog are active for contour plots. *Region shading* adds light background colors to show the boundaries between different dielectric or conductive regions. Only electrodes are colored when region shading is inactive. When *Element outline* is active the boundaries of elements are added in fixed-potential regions (Fig. 11.1).

**Plot limits**

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



**Figure 11.2.** Slice plot style dialog

### **Number of contour lines**

Change the number of lines for contour plots.

### **Toggle snap mode**

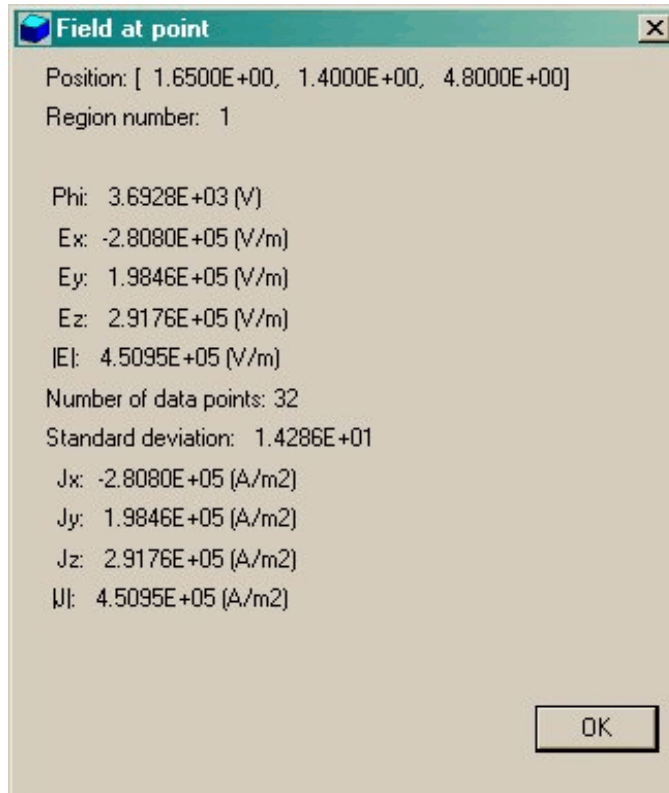
Mouse coordinates for commands such as *Zoom window*, *Pan*, and *Scan in slice* can be entered in two modes. In the normal mode, the returned position corresponds to the mouse position on the screen. In the snap mode, the program picks a point at an even interval close to the mouse position. The returned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

### **Set snap distance**

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

### **Toggle grid**

A set of dashed grid lines can be superimposed on slice plots. **PhiView** 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



**Figure 11.3.** Information display — point calculation

included in the axis labels. Grids corresponding to the normal plane axes are plotted as solid lines.

### 11.3. Analyses in a slice

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

#### Point calculation

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

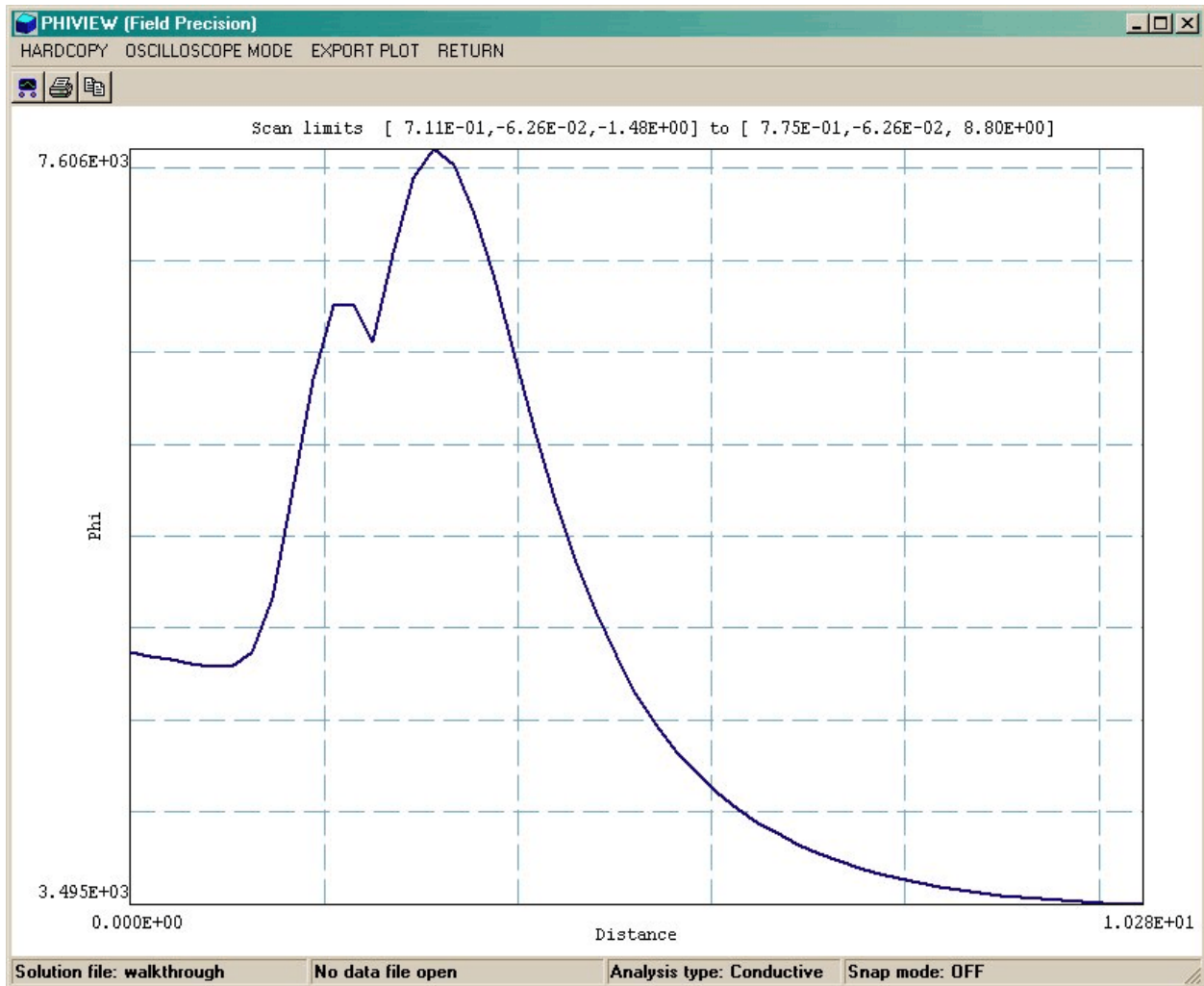


Figure 11.4. Scan plot display

following chapter describes the interpolation method. Figure 11.3 shows the information display. The included quantities depend on the solution type. The results are recorded if a data file is open.

### Line scan

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

recorded if a data file is open. The program also makes a screen plot of the currently-selected quantity versus distance along the scan and activates the *Scan plot* menu (Fig. 11.4). **PhiView** 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 (like 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 solution type. This setting has no effect on the data file listing which includes all 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 50 and the maximum number is 250.

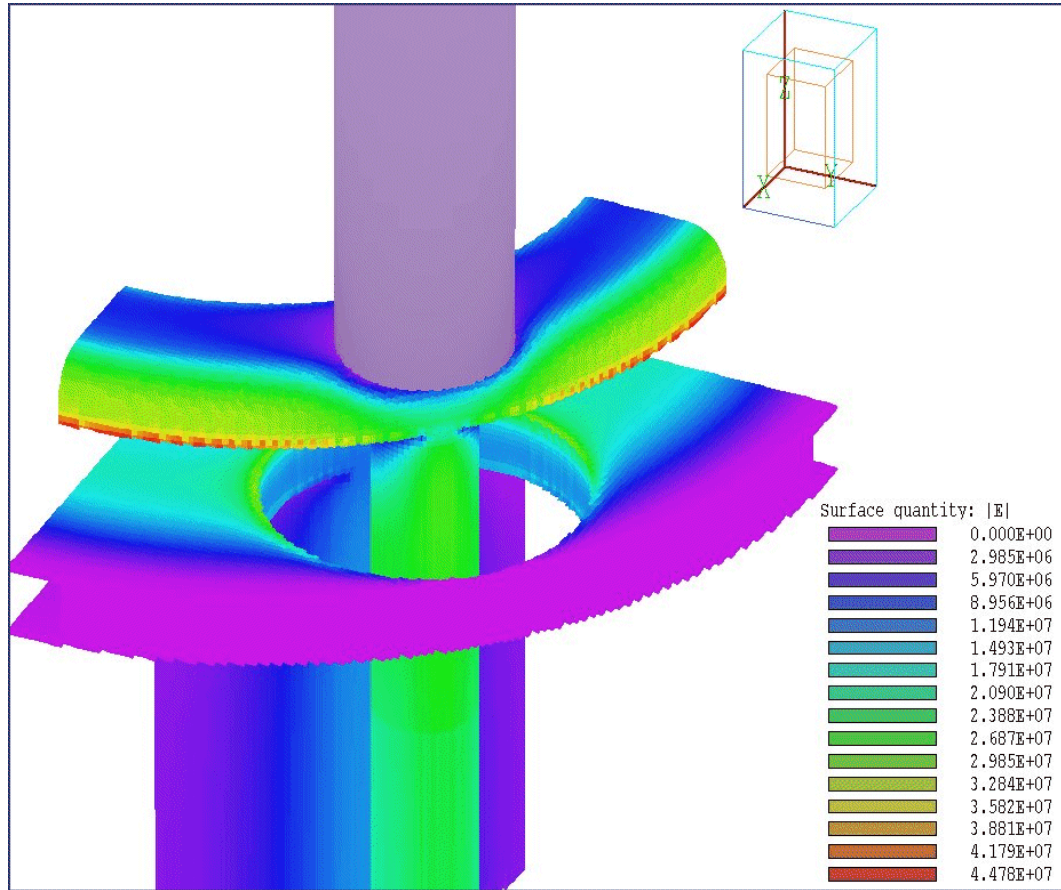
In addition to the standard *Export plot* options, the *Scan plot* menu contains the following command:

### **Oscilloscope mode**

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

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



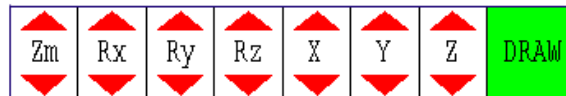
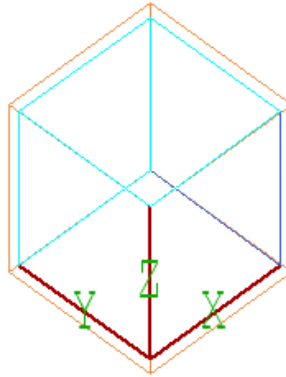


**Figure 12.1.** Three-dimensional surface view of region boundaries with color coding by electric field magnitude.

## Chapter 12. PhiView - surface plots

Surface plots are three-dimensional views of the solution space. Three types of information can be displayed: 1) boundaries of regions, 2) computed quantities in a slice plane normal to one of the Cartesian axes and 3) electric field magnitude on region surfaces. Region boundaries can be superimposed on either of the data plots. Surface plots are created from the conformal mesh and preserve true spatial scaling.

You can control the three-dimensional display with the view toolbar on the right-hand side of the screen (Figure 12,2). Move the mouse over one of the red arrows and hold down the left button. The orientation box rotates or moves to shows the effect. To update the main drawing area,



**Figure 12.2.** Tool area to adjust the 3D view.

click the right mouse button or move the mouse to the DRAW area and click the left button. The arrows control the view magnification and rotations and translations about the x, y or z axes.

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

### Set view dialog

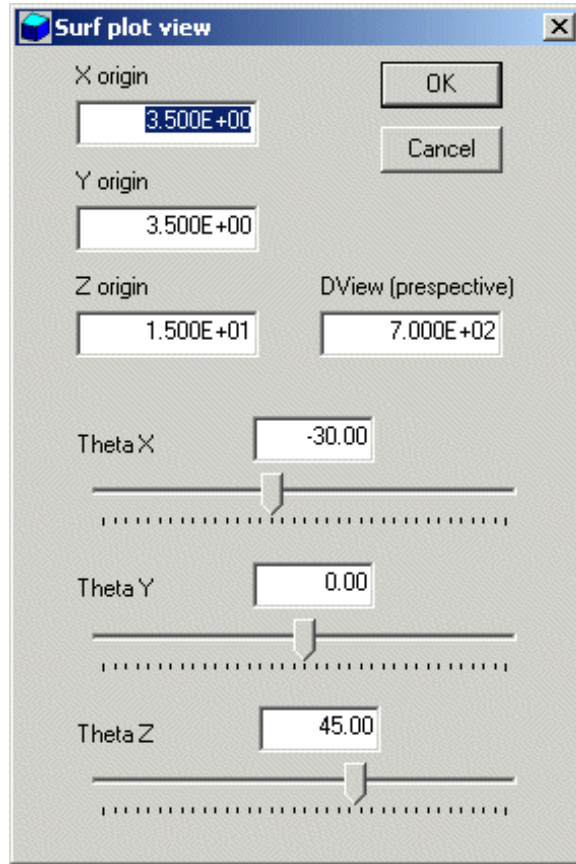
This command brings up a dialog (Fig. 12.3) where you can set specific view angles, displacements and the distance to the viewpoint, *DView*. The parameter *DView* controls perspective. Set it to a large value for an orthographic view. A value smaller than the limits of the system may result in an incoherent 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:  $\theta_x = -30^\circ$ ,  $\theta_y = 0^\circ$  and  $\theta_z = 45^\circ$  with the origin at the center.

### +X view, +Y view, +Z view

Rotate to views from the positive x, y or z directions. Origin shifts are not affected



**Figure 12.3.** Dialog to set the three-dimensional view.

**Center view**

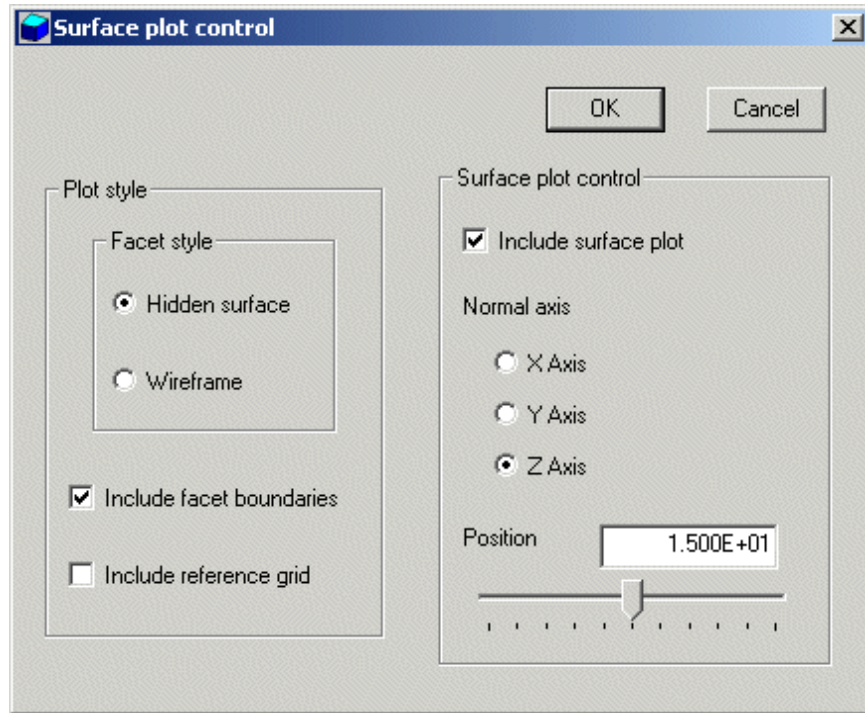
Remove shifts by setting the origin to (0.0,0.0,0.0).

**Save current view**

Use this command to save the rotations, translations, displayed regions and plot style of the current view. The view is saved until the program is closed and is preserved if you load another solution.

**Restore saved view**

Restore the saved view.

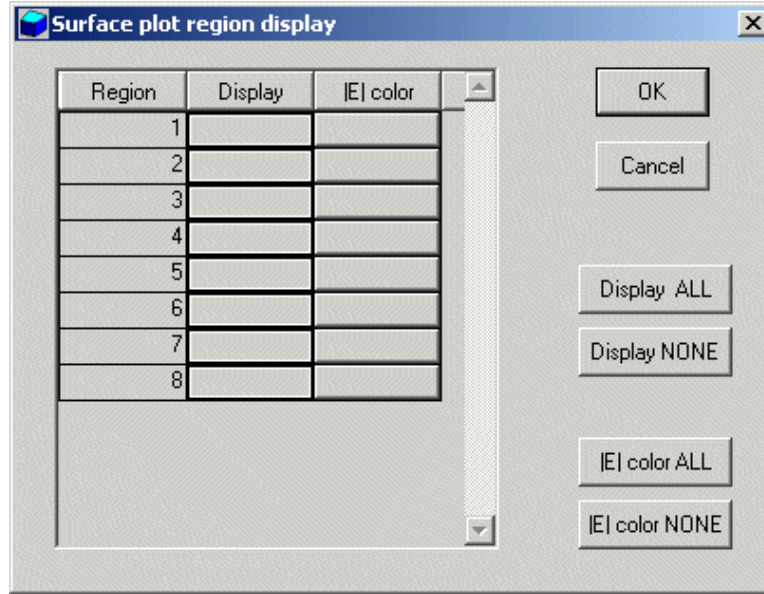


**Figure 12.4.** Surface plot control dialog.

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. 12.4. 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 number or electric field magnitude  $|\mathbf{E}|$  and 2) rectangular facets comprising a slice plane with color-coding by the values of a selected field quantity. Slice plane and region boundary information can be superimposed. The *Facet style* radio buttons control whether the facets of region boundaries are plotted as solid plates (hidden surface) or wireframe outlines. The *Include facet boundaries* 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). **PhiView** plots a reference grid along the boundaries of the solution volume when the *Include reference grid* box is checked. The group of commands on the right-hand side controls the slice plane. The *Include slice plane* box determines whether a slice plot is included. (Note:



**Figure 12.5.** Region display dialog.

activating the slice plane automatically deactivates color-coding by  $|E|$  on region facets). The radio buttons determine the axis normal to the slice plane. You can move the plane along the chosen axis by entering values in the box or moving the slider.

### Displayed regions

The command brings up the dialog of Fig. 12.5 where you can pick region boundaries to include in the plot. Depress a button in the *Display* column to activate a region. Initially, no regions are displayed when you enter the surface plot mode. The buttons in the column marked *|E| color* determine the presentation style for boundary facets. If no buttons are depressed, **PhiView** colors boundaries by the region number. In this case, a slice plane may also be displayed. If there is no slice plane plot, the information window shows the color key for regions. Otherwise, the window shows the correspondence between field values and colors in the slice plane. If one or more buttons are depressed in the *|E| color* column, the slice plane plot is deactivated and the information window shows the correspondence between color and  $|E|$ . The surfaces of the activated regions are color-coded according to the value of  $|E|$  at the surface (Fig. 12.1). There are two conventions for  $|E|$  plots on region boundaries: 1) if the chosen region is a fixed-potential electrode, field calculations are performed on the

common facets referenced to the adjacent dielectric elements, and 2) if the region is a dielectric, the field calculations are referenced to the target region. In this case, you could observe different values of  $|\mathbf{E}|$  on a common dielectric boundary, depending on which region is active.

### **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.

**PhiView** 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 slice plot quantity**

Set the quantity to be plotted if a slice plane plot is active. The choices are  $|\mathbf{E}|$ ,  $E_x$ ,  $E_y$ ,  $E_z$ ,  $|\mathbf{j}|$ ,  $j_x$ ,  $j_y$  or  $j_z$ . The latter quantities are physically meaning only for conductive type solutions.

### **Set plot limits**

By default the limits of  $|\mathbf{E}|$  in region surface plots or the limits of the plotted quantity in a slice plane are automatically computed by **PhiView**. With this command you can set limits manually. Facets where  $|\mathbf{E}|$  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 certain value). The command is essential on occasions when a distorted element on a region surface leads to an inaccurate field interpolation, skewing the plot range.

## Chapter 13. Analysis

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

### 13.1. Point calculation - interpolation method

The *Point calculation* command brings up a dialog where you enter the  $x$ ,  $y$  and  $z$  coordinates of a general point in the solution space. In contrast to the *Point calculation* command of the *Slice plots* menu, the point is not limited to a slice plane. When you click *OK*, **PhiView** displays the information box of Fig. 10.3 and also writes the results to the data file in the following format:

```
--- Point Field Analysis ---
Position: [ 0.0000E+00, 2.0000E+00,
3.0000E+00]
Region number: 1
Phi: 3.0572E+03 (V)
Ex: 9.5810E+01 (V/m)
Ey: 3.4628E+04 (V/m)
Ez: -5.3599E+03 (V/m)
|E|: 3.5040E+04 (V/m)
```

**PhiView** must perform a complex sequence of operations to make the calculation

- Estimate the indices of the element that contains the target point.
- Check this element and its neighbors using a normal coordinate inversion to determine the precise element that contains the target point.
- Collect the nodes of the target element as well as neighboring nodes that border on at least one element with the same region number as the target element
- Fit a three-dimensional, second-order function to the electrostatic potential values of the data set using a least-squares method.

- Find the potential at the target point using the interpolation function.
- Find the electric field values by taking derivatives of the interpolation function.

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

A simple example will serve to demonstrate the accuracy of **HiPhi** and **PhiView**. Consider a grounded sphere of radius  $R_o = 0.05$  m filled with a uniform space charge density  $\rho_o = 1.0 \times 10^{-5}$  coulombs/m<sup>3</sup>. Solving the spherical Poisson equation

$$\frac{1}{R^2} \frac{\partial}{\partial R} \left( R^2 \frac{\partial \phi}{\partial R} \right) = -\frac{\rho}{\epsilon_o}, \quad (13.1)$$

we find that the potential at the center of the sphere is

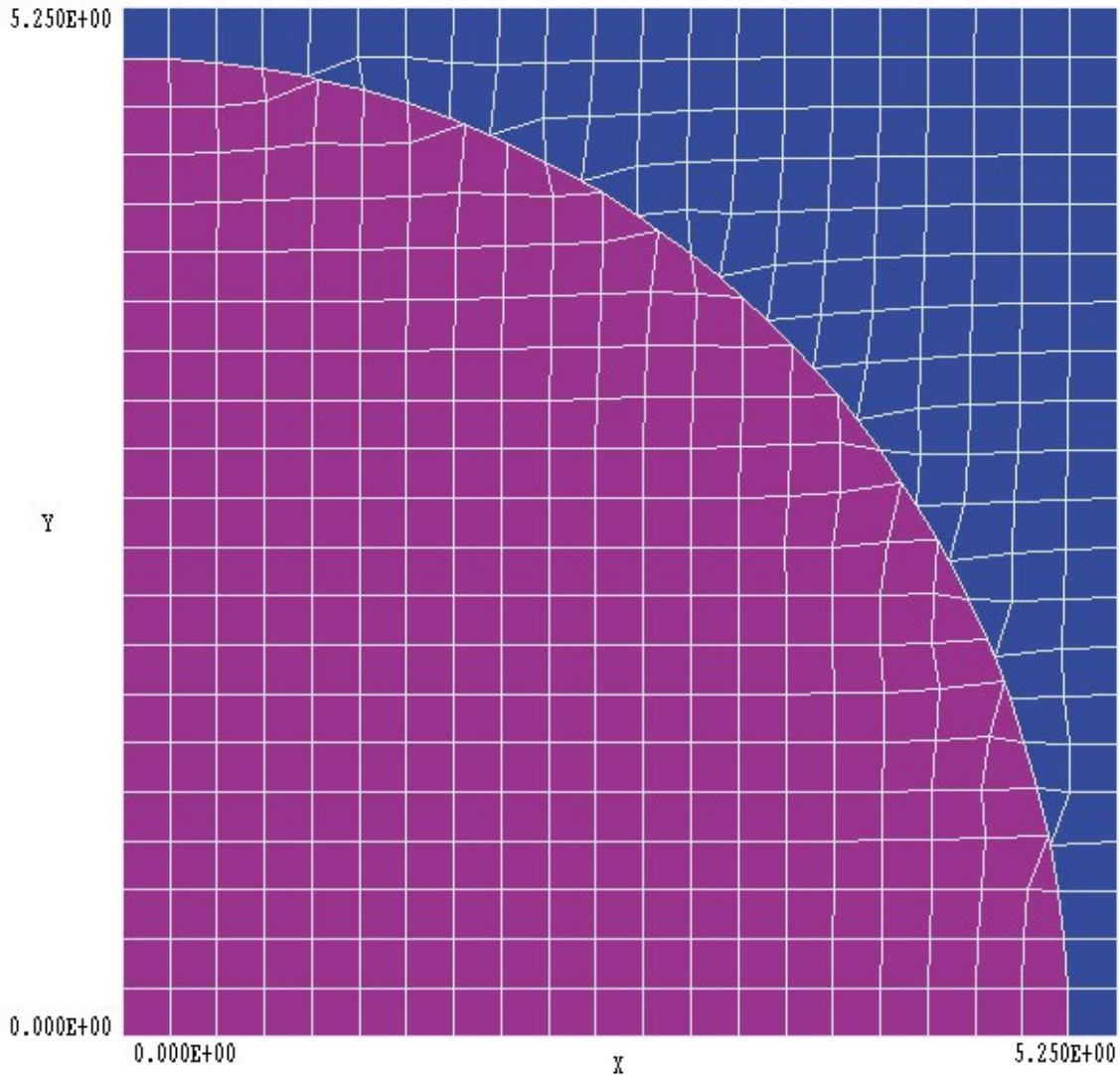
$$\phi(0) = \frac{\rho_o R_o^2}{6\epsilon_o}, \quad (13.2)$$

and the electric field at the wall is

$$E_R(R_o) = \frac{\rho_o R_o}{3\epsilon_o}. \quad (13.3)$$

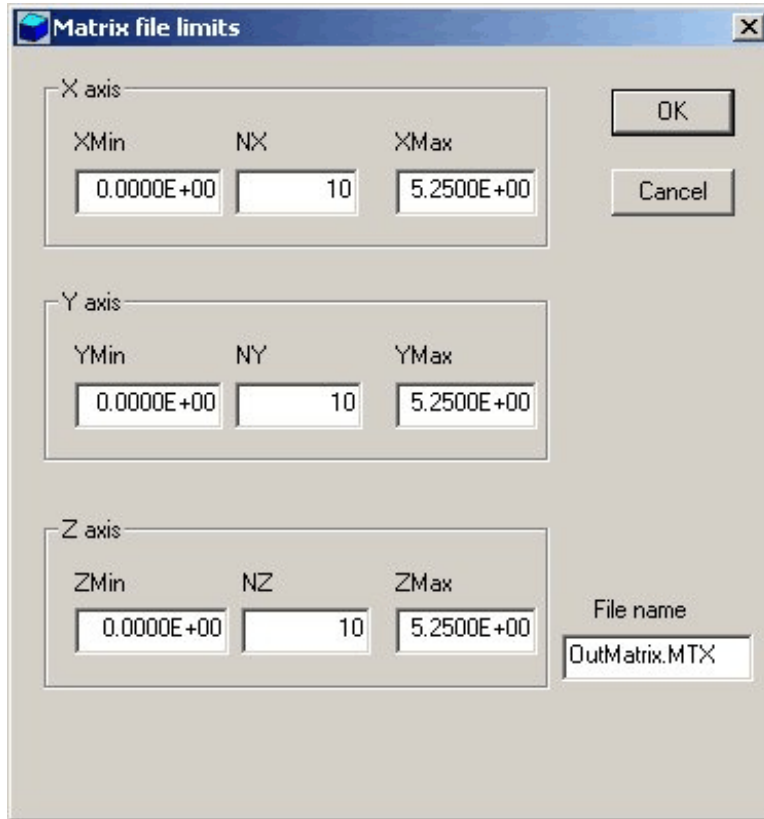
Inserting numerical values gives  $\phi_o = 470.587$  V and  $E_R(R_o) = 1.88235 \times 10^4$  V/m. We simulate one octant of the sphere with the moderately-coarse mesh shown in Fig. 13.1. There are about 20 elements over the radius of the sphere. The numerically calculated volume of the faceted sphere is 65.3843 cm<sup>3</sup> compared to the theoretical value of 65.4498 cm<sup>3</sup>.





**Figure 13.1.** Orthographic view of the mesh used for the Space\_Charge example ( $z = 0.0$ )

The **PhiView** interpolation of the **HiPhi** solution gives a potential of 470.68 V at the center of the sphere ( $x = y = z = 0.0$  cm), an error of only 0.02%. The calculated electric field near the wall ( $x = y = z = 2.8867$  cm) is  $E_R = 1.9046 \times 10^4$  V/m, in error by about 1%. The larger error reflects the fact that wall is faceted and there is some ambiguity making a comparison with a perfect sphere. In contrast, the calculated field away from the wall at  $R = 3.0$  cm is  $1.1293 \times 10^4$  V/m, within 0.01% of the theoretical value.



**Figure 13.2.** Create matrix file dialog

## 13.2. Line scans and matrix files

In the *Analysis* menu scans can be performed along arbitrary straight lines between any two points in the solution volume. When you click the *Line scan* command a dialog appears where you can specify the start and end points. Enter values in units set by *DUnit*. In other words, if you used dimensions of cm in **MetaMesh** and the **HiPhi** script contains the command  $DUnit = 100.0$ , then enter the dimensions in cm. In the interactive mode, **PhiView** creates a screen plot of a selected quantity determined by the *Set scan quantity* command. The command has no effect on the data file listing which contains all calculated quantities. The following is an example of a listing.

```

----- Field scan between points -----
XStart:  0.0000E+00  YStart:  0.0000E+00  ZStart:  0.0000E+00
XEnd:    2.8867E+00  YEnd:    2.8867E+00  ZEnd:    2.8867E+00

```

X	Y	Z	Phi	Ex	Ey	Ez	E	NReg
0.0000E+00	0.0000E+00	0.0000E+00	4.7068E+02	9.9926E-03	4.7400E-03	-1.4593E-03	1.1156E-02	2
5.7734E-02	5.7734E-02	5.7734E-02	4.7049E+02	2.1736E+02	2.1735E+02	2.1735E+02	3.7647E+02	2
1.1547E-01	1.1547E-01	1.1547E-01	4.6992E+02	4.3470E+02	4.3470E+02	4.3470E+02	7.5292E+02	2
1.7320E-01	1.7320E-01	1.7320E-01	4.6898E+02	6.5205E+02	6.5205E+02	6.5205E+02	1.1294E+03	2
2.3094E-01	2.3094E-01	2.3094E-01	4.6766E+02	8.6940E+02	8.6940E+02	8.6940E+02	1.5058E+03	2
2.8867E-01	2.8867E-01	2.8867E-01	4.6597E+02	1.0868E+03	1.0867E+03	1.0868E+03	1.8823E+03	2
3.4640E-01	3.4640E-01	3.4640E-01	4.6390E+02	1.3041E+03	1.3041E+03	1.3041E+03	2.2588E+03	2
4.0414E-01	4.0414E-01	4.0414E-01	4.6145E+02	1.5214E+03	1.5214E+03	1.5215E+03	2.6352E+03	2
4.6187E-01	4.6187E-01	4.6187E-01	4.5863E+02	1.7388E+03	1.7388E+03	1.7388E+03	3.0117E+03	2

The *Create matrix file* command controls a **PhiView** feature that is useful if you want to write your own analysis routines or port results to mathematical software. In response to the command, **PhiView** 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. Clicking on *Create matrix file* calls up the dialog of Fig. 13.2. Specify the dimensions of the box along each axis (in units set by DUnit) and the number of calculation intervals. To illustrate, calculations are performed at positions with x coordinates given by

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

where  $n = 0, 1, 2, \dots, n_x$ . For example, if you set  $x_{\min} = 0.5$ ,  $x_{\max} = 1.5$  and  $n_x = 10$ , the calculations are performed at points with  $x = 0.5, 0.6, \dots, 1.4, 1.5$ . You can also specify an output file name. The text file is created in the standard AMaze Field Exchange format (Chap. 14). The stored quantities depend on the solution type. You can inspect the file with an editor.

### 13.3. Automatic analyses

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

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

```

      Fixed potential regions
NReg  Potential      Volume
      (V)            (m3)
=====
   1  0.00000E+00  7.93184E-05

      Dielectric regions
NReg  Epsilon      Rho      |E| (min)  |E| (max)  Energy      Volume
      (relative)  (c/m3)   (V/m)     (V/m)     (J)        (m3)
=====
   2  1.00000E+00  1.00000E-05  8.15066E+02  2.92211E+04  6.04156E-08  6.53844E-05

Global field energy:  6.04156E-08 J
Global volume:      1.44703E-04

```

You can use the energy values to determine capacitance in a dielectric solution with a single electrode with voltage  $V_o$  from the equation

$$U_o = \frac{C V_o^2}{2} . \tag{13.5}$$

In Eq. (13.5) the quantity  $U_o$  is the global field energy. Similarly, you use the equation

$$P_o = \frac{V_o^2}{2R} , \tag{13.6}$$

to find the resistance (and hence the current ) between the electrode and ground in a conductive solution. Here the quantity  $P_o$  is the global power.

The volume integral method is not applicable to systems where there are several electrodes. In this case, you can use the *Region* command to perform surface integrals over selected regions. In response to the *Region* command with a specified region number, **PhiView** performs the following operations:

- Carries out volume integrals over the region using the same routines as the *Full analysis* command.
- Determines all element facets on the region surface, excluding those on the outer surface of the solution volume.
- Performs a surface integral over each facet. At an integration point representing a differential surface area  $dA$ , the program determines the

unit vector  $\mathbf{n}$  normal to the surface. **PhiView** also calculates the electric field  $\mathbf{E}$  at a nearby position in the adjacent element with relative dielectric constant  $\epsilon_r$ .

- Forms the sum of  $\epsilon_r \mathbf{E} \cdot \mathbf{n} dA$ .

The end result is a calculation of the free surface charge on the region.

With information on surface charge (or current), we can calculate the matrix of mutual capacitances or resistances. To illustrate, suppose we have two electrodes and a ground plane and we want to find the capacitance between Electrode 1 and Electrode 2 as well as the capacitance between Electrode 1 and ground. We apply a voltage  $V_0 = 1\text{V}$  to Electrode 1 and determine the surface charge on Electrode 2 ( $Q_2$ ) and the ground electrodes ( $Q_g$ ). The mutual capacitance between Electrode 1 and 2 is  $C_{12} = Q_2$  and the capacitance to ground is  $C_{1g} = Q_g$ . We could then do a second solution with a voltage on Electrode 2 to find  $C_{2g}$ .

To illustrate the accuracy of the technique, consider again the space-charge example of Fig. 13.1. The theoretical total charge in the octant is  $6.54498 \times 10^{-10}$  coulombs. A numerical surface integral over the inside of the grounded wall gives  $6.56073 \times 10^{-10}$  coulombs, an error of only 0.25%. Despite the good agreement in this case, you must be cautious in applying surface integrals. The accuracy depends on accurate field calculations and will degrade severely if the surface has few facets. Results are meaningless if the surface has a non-physical sharp edge.

## 13.4. Script operation

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

```
START \AMAZE\PHIVIEW SCRIPTNAME
```

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

**Table 13.1. Create script - default content**

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

ENDFILE

    --- Script command summary ----

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

**Symbolic command:** INPUT FPrefix.HOU

**Example:** INPUT WGUIDE1.HOU

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

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

**Symbolic command:** OUTPUT FPrefix

**Example:** OUTPUT WGUIDE1

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

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

**Symbolic command:** NSCAN *NScan*

**Example:** NSCAN 100

**Function:** Set the number of intervals for line scans

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

**Symbolic command:** POINT *xp yp zp*

**Example:** POINT 0.00 0.05 4.67

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

**Symbolic command:** LINE *xp1 yp1 zp1 xp2 yp2 zp2*

**Example:** LINE 0.00 0.00 15.00 12.00 0.00 15.00

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

**Command:** FULLANALYSIS

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

**Symbolic command:** REGION *RegNo*

**Example:** REGION 7

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

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

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

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

**Command:** ENDFILE

**Function:** Terminate the analysis

## Chapter 14. Structure of the HiPhi output file

**HiPhi** creates output files in binary or ASCII format. The mode is controlled by the *FORMAT* command (Sect. 3.2). The *BINARY* mode must be used to communicate results to **PhiView**. The ASCII mode is useful to transfer results to your own analysis programs.

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

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

The next step is to record information for each node. The quantities *RegNo* (the region number of the node) and *RegUp* (the region number of the adjacent element in the direction of increasing  $I$ ,  $J$  and  $K$ ) are 1-byte integers. The coordinates ( $X, Y$  and  $Z$ ) and the stored quantity are 4-byte real numbers. The final step is to record region information. The quantity *NRQuant*, the number of physical quantities stored per region, is equal to 3 in **HiPhi**. The quantities are the relative dielectric constant  $\epsilon_r$ , the space-charge density  $\rho$  in coulombs/m<sup>3</sup>, and the value of  $\phi$  for fixed-potential regions. The region quantity names (*RQuantName*) are strings of length 12. The variable *NFix* is a 1-byte integer and the region quantities are 4-byte real numbers.



**Table 14.1. Code to create the HiPhi binary output file**

```
! --- Mesh size parameters
WRITE (OutField) IMax,JMax,KMax,DUnit,NCQuant
DO N=1,NCQuant
  WRITE (OutField) CQuantName(N)
END DO
! --- Node and element properties
DO K=0,KMax
  DO J=0,JMax
    DO I=0,IMax
      MC = M(I,J,K)
      WRITE (OutField) &
        C(MC).RegNo,C(MC).RegUp,C(MC).x,C(MC).y,C(MC).z, &
        REAL(C(MC).Phi)
    END DO
  END DO
END DO
! --- Region properties
WRITE (OutField) NRQuant
DO N=1,NRQuant
  WRITE (OutField) RQuantName(N)
END DO
WRITE (OutField) NRegMax
DO N=1,NRegMax
  IF (Reg(N).Fixed) THEN
    NFix = 1
  ELSE
    NFix = 0
  ENDIF
  WRITE (OutField) NFix,Reg(N).Epsi,Reg(N).Rho,Reg(N).Pot
END DO
```

**Table 14.2. Code to create the HiPhi text file**

```
! --- Mesh size parameters
WRITE (OutField,5000)
WRITE (OutField,5100) OutFieldName,IMax,JMax,KMax,DUnit,NCQuant
WRITE (OutField,5200) (CQuantName(N),N=1,NCQuant)
! --- Node and element properties
DO K=0,KMax
  DO J=0,JMax
    DO I=0,IMax
      MC = M(I,J,K)
      WRITE (OutField,5300) &
        C(MC).RegNo,C(MC).RegUp,C(MC).x,C(MC).y, &
        C(MC).z,REAL(C(MC).Phi)
    END DO
  END DO
END DO
! --- Region properties
WRITE (OutField,5400) (RQuantName(N),N=1,NRQuant)
DO N=1,NRegMax
  IF (Reg(N).Fixed) THEN
    NFix = 1
  ELSE
    NFix = 0
  ENDIF
  WRITE (OutField,5500) N,NFix,Reg(N).Epsi,Reg(N).Rho, &
    Reg(N).Pot
END DO
```

The **PhiView** program can create text files in the standard AMaze Field Exchange format. The file information can be ported to **OmniTrak** or to your own applications. A sample is shown below. The header contains any number of comment lines starting with "\*" (asterisk) and two data lines:

```
IMax JMax KMax
XMin YMin ZMin XMax YMax ZMax
```

The header is followed by  $(IMax+1)(JMax+1)(KMax+1)$  data lines that contain the information

```
Ex Ey Ez Phi EpsiR NReg
```

Data lines are recorded in the order

```

DO K=0,KMax
  DO J=0,JMax
    DO I=0,IMax
      . . .
    END DO
  END DO
END DO

```

The field value  $E_x(I,J,K)$  is calculated at the position:

```

X(I) = XMin + I*Dx,
Y(J) = XMin + I*Dy,
Z(K) = XMin + I*Dz.

```

where

```

Dx = (XMax-XMin)/IMax,
Dy = (YMax-YMin)/JMax,
Dz = (ZMax-ZMin)/KMax.

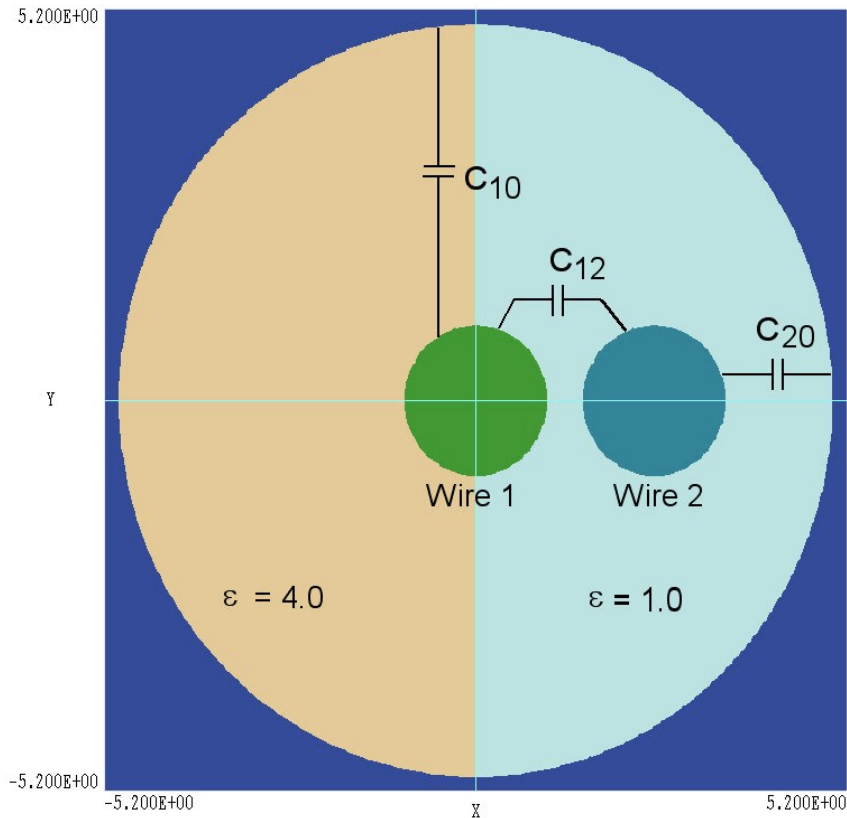
```

The following in an extract from a **PhiView** matrix file:

```

* HiPhi matrix file: OMNITESTE.DAT
      20      20      20
-2.0000E+00 -2.0000E+00 -1.0000E+00  2.0000E+00  2.0000E+00
5.0000E+00
 1.1803E+04  1.1803E+04 -1.3097E+03 -5.8684E+01  1.0000E+00  1
 1.1898E+04  1.6661E+04 -1.8386E+03 -8.2466E+01  1.0000E+00  1
 1.1941E+04  2.1697E+04 -2.3602E+03 -1.0635E+02  1.0000E+00  1
 1.1821E+04  2.6736E+04 -2.8792E+03 -1.3021E+02  1.0000E+00  1
 1.1446E+04  3.1778E+04 -3.3818E+03 -1.5361E+02  1.0000E+00  1
 1.0706E+04  3.6696E+04 -3.8544E+03 -1.7593E+02  1.0000E+00  1

```



**Figure 15.1.** Benchmark example -- geometry and definition of quantities (system is uniform in  $z$  with a length  $L_z = 0.02$  m).

## Chapter 15. Calculation of mutual capacitance

The calculation of mutual capacitance in a multi-electrode system is a common application of finite-element programs for electrostatics. **PhiView** has several automatic calculation routines to facilitate the process. Because the user must exercise some care to ensure accuracy and physical validity, we shall review available methods in this chapter and work through an example. Figure 15.1 shows a benchmark example that gives good accuracy and runs quickly (input files `MUTUAL_CAP.MIN` and `MUTUAL_CAP.HIN`). The geometry consists of two circular wires inside a grounded outer conductor with a circular inner radius. The enclosed volume is divided into two dielectric regions with  $\epsilon_r = 4.0$  and  $\epsilon_r = 1.0$ . In this case, there are three unknown capacitance quantities:  $C_{10}$  (capacitance between Wire 1 and ground),  $C_{20}$  (capacitance between Wire 2 and ground) and  $C_{12}$  (mutual capacitance between Wires 1 and 2). The assembly is uniform along the  $z$ -direction with length  $L_z = 0.02$  m.

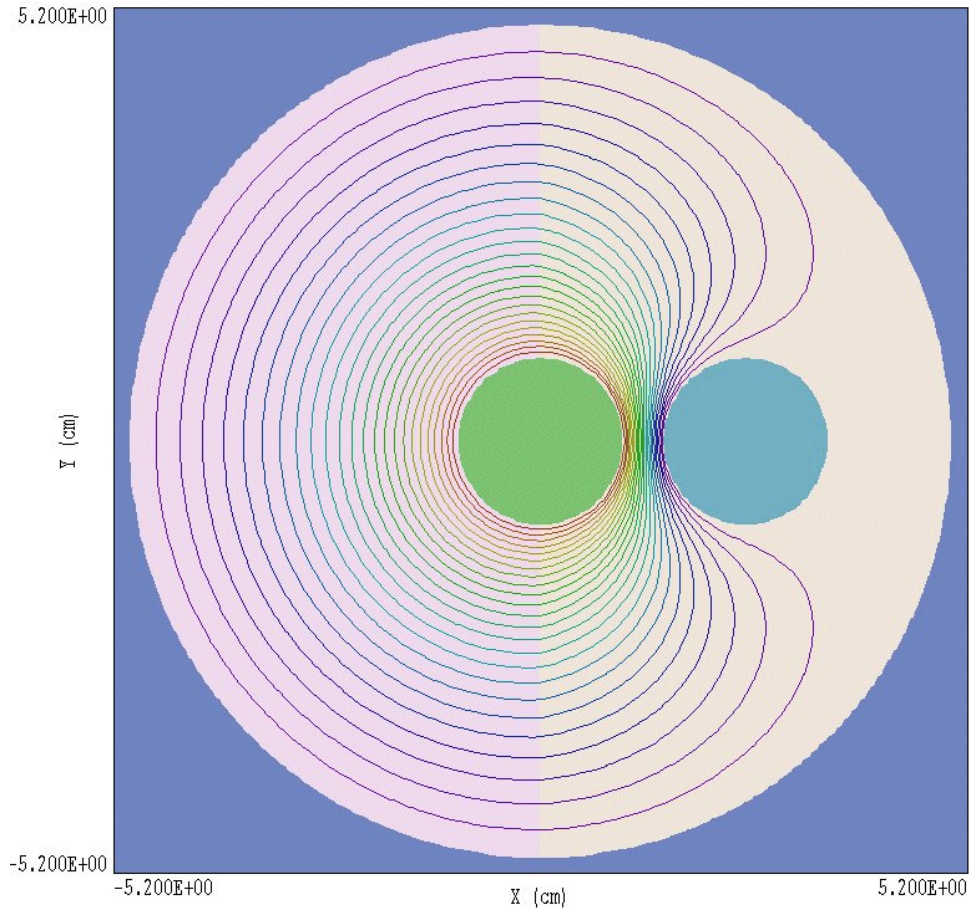
Table 15.1. Calculated quantities for example MUTUAL_CAP						
Solution	V <sub>1</sub>	V <sub>2</sub>	Q <sub>1</sub>	Q <sub>2</sub>	Q <sub>0</sub>	U
1	1.0	0.0	2.132 × 10 <sup>-12</sup>	6.086 × 10 <sup>-13</sup>	1.533 × 10 <sup>-12</sup>	1.053 × 10 <sup>-12</sup>
2	0.0	1.0	6.085 × 10 <sup>-13</sup>	1.273 × 10 <sup>-12</sup>	6.665 × 10 <sup>-13</sup>	6.272 × 10 <sup>-13</sup>
3	1.0	1.0	1.524 × 10 <sup>-12</sup>	6.640 × 10 <sup>-13</sup>	2.199 × 10 <sup>-12</sup>	1.082 × 10 <sup>-12</sup>

One approach is to employ the automatic surface integral routines in **PhiView** to calculate induced charge. To perform an integral, open a data file in the main menu and then go to the *ANALYSIS* menu. Click on the command *REGION PROPERTIES* and specify the number of the region over which integrals will be performed. **PhiView** lists results on the screen and writes a record in the data file.

To illustrate, consider the capacitance quantities in the example of Fig. 15.1. We create Solution 1 with Wire 1 set to V<sub>1</sub> = 1.0 V and Wire 2 to V<sub>2</sub> = 0.0 V (Fig. 15.2). In PhiView we issue the *REGION PROPERTIES* command for Regions 1 (ground electrode), 4 (Wire 1) and 5 (Wire 2). Table 15.1 shows the calculated surface charge values in coulombs. In the table, Q<sub>1</sub> is the surface charge on Wire 1, Q<sub>2</sub> is the charge on Wire 2, and Q<sub>0</sub> is the charge on the inside of the ground electrode. Because we applied a 1.0 V drive, the numbers also equal the capacitance in farads. From the table, we can determine that C<sub>12</sub> = 6.086 × 10<sup>-13</sup> F and C<sub>10</sub> = 1.053 × 10<sup>-12</sup> F. To check accuracy, we can verify that Q<sub>1</sub> ≈ Q<sub>2</sub> + Q<sub>0</sub>. We next generate Solution 02 with V<sub>1</sub> = 0.0 V and V<sub>2</sub> = 1.0 V. The results (shown in the second data row of Table 15.1) imply that C<sub>12</sub> = 6.085 × 10<sup>-13</sup> F and C<sub>20</sub> = 6.65 × 10<sup>-13</sup> F.

The surface integral method works well in the example because we have used a fine mesh for good resolution of the curved electrode surfaces. The accuracy of the method declines significantly when electrodes have sharp corners where the electric field is effectively undefined. For these situations we can get better results with volume integral methods that are less sensitive to local field interpolation errors. For a simple two-electrode system, the volume integral of field energy is related to the capacitance by

$$U_e = \frac{1}{2} CV^2. \quad (1)$$



**Figure 15.2.** Calculated equipotential lines for Solution 1.

We can use the *FULL ANALYSIS* command in the *ANALYSIS* menu to perform a field energy integral. In response **PhiView** lists the global energy and components associated with different regions. It is not difficult to extend this concept to multiple electrodes. For an applied voltage of 1.0 V, the global electrostatic energy  $U_1$  for Solution 01 is related to the capacitance values by

$$U_1 = \frac{1}{2} ( C_{12} + C_{01} ) . \quad (2)$$

Similarly, the field energy for Solution 02 is

$$U_2 = \frac{1}{2} ( C_{12} + C_{02} ) . \quad (3)$$

We perform a third solution where a 1.0 V potential is applied to both

Wires 1 and 2. Here, the field energy is related to capacitance values by

$$U_3 = \frac{1}{2} ( C_{10} + C_{20} ) . \quad (4)$$

Using the three equations, we can solve for the uniform capacitance values in terms of the known calculated values of field energy:

$$\begin{aligned} C_{12} &= U_1 + U_2 - U_3 , \\ C_{10} &= U_1 - U_2 + U_3 , \\ C_{20} &= -U_1 + U_2 + U_3 . \end{aligned} \quad (5)$$

Table 15.1 shows field energy values for the three solutions. The calculated values of capacitance are  $C_{12} = 5.982 \times 10^{-13}$  F,  $C_{10} = 1.506 \times 10^{-12}$  F and  $C_{20} = 6.652 \times 10^{-13}$  F. These values are quite close to those determined by the surface integral method.

There are six unknown quantities for a system with three electrodes and a ground reference:  $C_{12}$ ,  $C_{23}$ ,  $C_{31}$ ,  $C_{10}$ ,  $C_{20}$  and  $C_{30}$ . We can generate a set of six equations similar to Eqs. (5) with the set of voltages shown in Table 15.2.

<b>Table 15.2. Applied voltages for a three-electrode system with ground</b>			
<b>Solution</b>	<b>V<sub>1</sub></b>	<b>V<sub>2</sub></b>	<b>V<sub>3</sub></b>
1	1.0	0.0	0.0
2	0.0	1.0	0.0
3	0.0	0.0	1.0
4	1.0	1.0	0.0
5	1.0	0.0	1.0
6	0.0	1.0	1.0