



EStat

Finite-element Electrostatics

Field Precision
Copyright 2002

PO Box 13595
Albuquerque, New Mexico 87192 U.S.A.
Telephone: 505-220-3975
FAX: 505-294-0222
E Mail: techinfo@fieldp.com
Internet: <http://www.fieldp.com>

EStat 5.0 Manual - Contents

Section 1. Program function

Section 2. Walkthrough example

Section 3. Some electrostatic theory

Section 4. Organizing runs

Section 5. Structure of the command file

Section 6. Commands for general program control

Section 7. Commands for material properties

Section 8. Boundary and superposition commands

Section 9. Running EStat interactively

Section 10. Running EStat from the command prompt

Section 11. VESat - file operations

Section 12. VESat - spatial plots

Section 13. VESat - scan plots

Section 14. VESat - analysis functions

Section 15. VESat - script operation

Section 16. EStat output file format

Section 17. Treatment of anisotropic materials in VESat

Section 18. QuikPlot utility

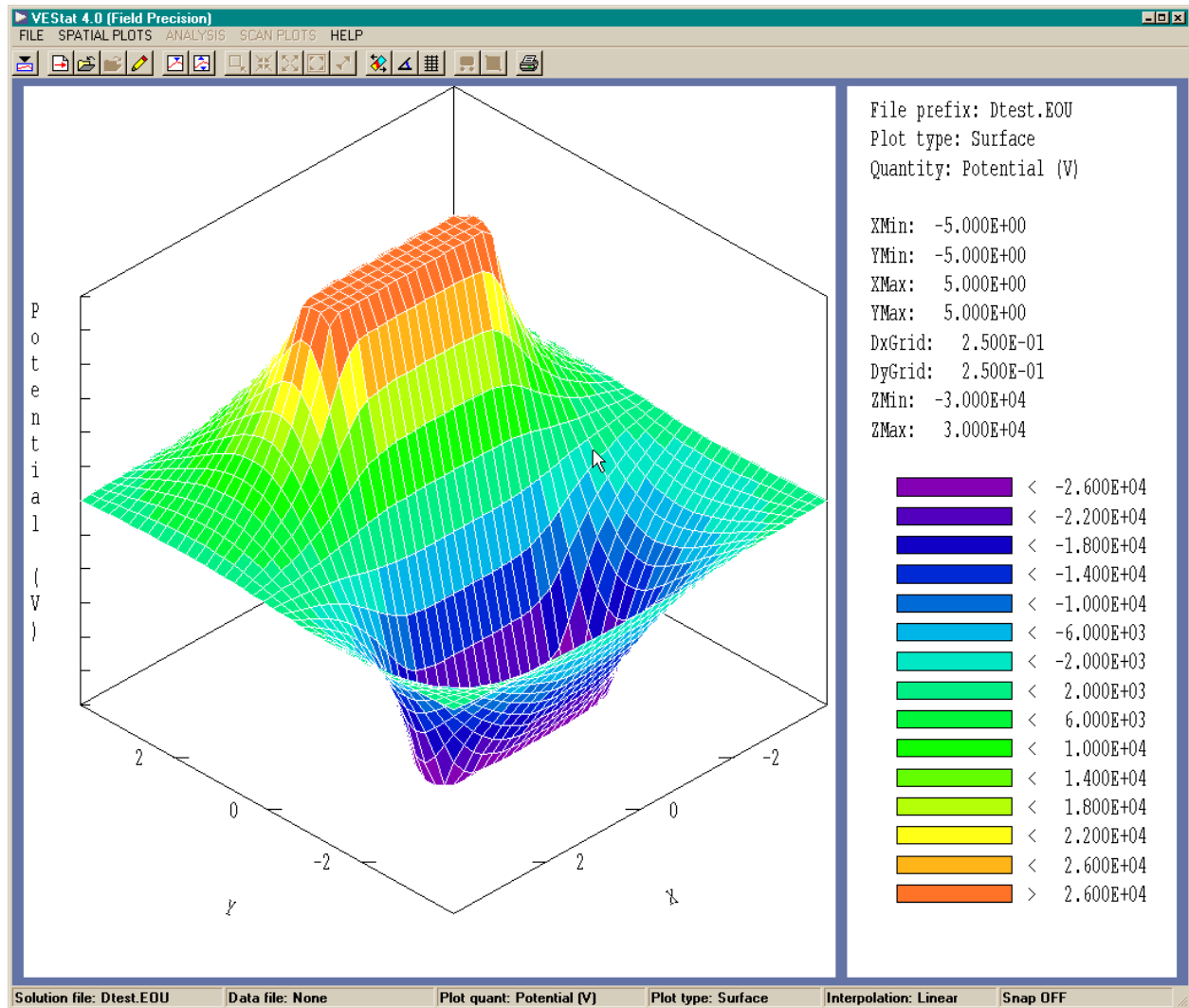


Figure 1. VESat screen display

1. Program function

EStat is a versatile numerical tool to find electrostatic fields in complex geometries with electrodes, conductors, dielectrics, and space-charge. The program handles electrostatic solutions in two limits:

- **Dielectric-type solutions.** All materials are insulators ($\sigma = 0$) with different values of relative dielectric constant ϵ_r and (optionally) space-charge density ρ .
- **Conductive-type solutions.** All materials are conductors with $\sigma > 0$. The dielectric constant ϵ_r does not affect the distribution of electric fields and there is no space-charge density ($\rho = 0$).

A *dielectric* solution applies to systems with perfect insulators or when short voltage pulses are applied to good insulators. An electroplating apparatus is an example of a *conductive* solution. In some cases, it is possible to model a mixed system of perfect insulators and conductors (for example, a resistive high-voltage probe). Section 3 discusses how to determine which solution type to use.

EStat applies finite-element methods on variable-resolution conformal triangular meshes. The mesh size is limited only by the installed memory of the computer. **EStat** handles three-dimensional cylindrical problems (azimuthal symmetry) and two-dimensional rectangular problems (variation in x and y with infinite extent in z). Using a simple command language you can define up to 127 electrical regions to represent electrodes or different materials. An interactive graphics analysis program, **VEStat**, provides complete information on the potential and electric fields. Other features include automatic Gaussian surface integrals and calculations of stored energy, induced charge and peak field values.

The walkthrough example in the following section gives an overview of the solution process. Section 3 reviews some elements of electrostatic theory that will help you set up effective solutions. Section 4 outlines typical steps in an **EStat** run while Section 5 covers the structure of the command script that controls the program. Sections 6, 7 and 8 give detailed descriptions of script commands to set control parameters and material properties. Section 9 covers operation of **EStat** as an interactive Windows program that you can run from the **TC** program launcher. Section 10 explains how to run **EStat** in the background from the Command Prompt or under the control of the **GCon** utility for extended data runs. Sections 11 through 15 address operation of the **VEStat** post-processor used to create plots and to perform quantitative analysis. Finally, Section 16 documents the format of the **EStat** output file.

2. Walkthrough example

We shall consider the *ElectronDiode* example from Section 5.2 of the **Mesh** manual to illustrate a complete electrostatic solution. Figure 2 shows the geometry of the cylindrical system with dimensions in inches. A voltage pulse from a coaxial water-filled transmission line is transferred through a vacuum insulator to an intense electron-beam load. The 100 ns voltage pulse is long enough to ensure that the electrostatic solution provides a good approximation but short enough so that the effects of conductivity in the water can be ignored. Three files (supplied in the example library) are required for the solution:

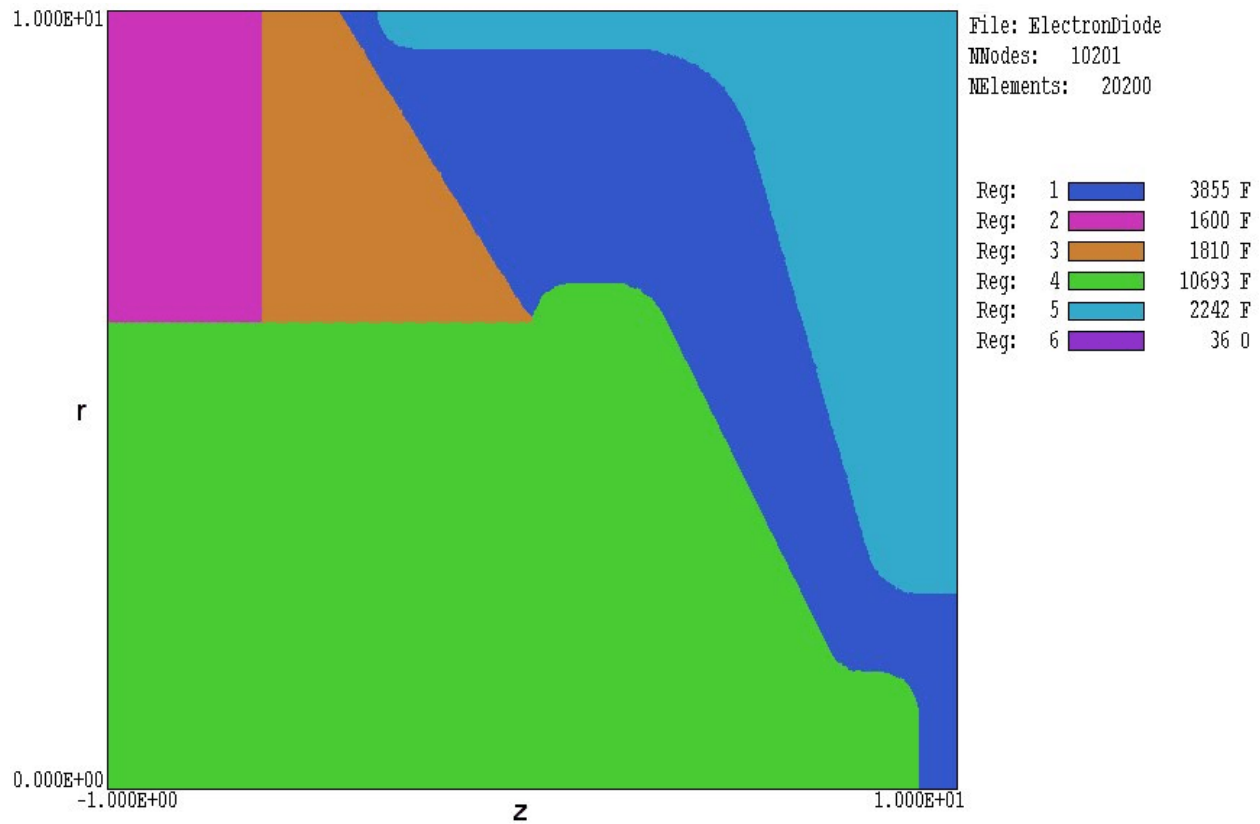


Figure 2. Geometry of the electron diode example. Region 1: Vacuum. Region 2: deionized water, Region 3: epoxy insulator. Region 4: high-voltage center-conductor. Region 5: shaped portion of vacuum chamber wall. Region 6 (not shown): other sections of upper and right-hand grounded boundaries.

- `ElectronDiode.MIN` – the input file for **Mesh** that describes the geometry of system. The **Mesh** manual describes the file format and the use of the Drawing Editor to prepare the geometric specifications.
- `ElectronDiode.EIN` – the command script to control **EStat**.
- `ElectronDiode.SCR` – a script to control an automatic **VEStat** analysis session to find capacitance of the output region and the electric field distribution on the anode plate.

To begin, copy the three files to a working directory. If you are using **TC**, the working directory should correspond to the *Data folder* (i.e., `\Tricomp\Buffer`). Run **Mesh** then process and save the mesh following the instructions in the **Mesh** manual. The program creates the files `ElectronDiode.MLS` and `ElectronDiode.MOU`. The first file is a listing of run information that you can usually ignore if there are no

problems. The second file describes the mesh – it is a listing of node coordinates and region numbers associated with nodes and elements. Like all **TriComp** files, `ElectronDiode.MOU` is in ASCII format so you can inspect it with a text editor or transfer the information to your own programs.

Next, run **EStat** from **TC**. We shall inspect the command script before starting the solution. In the *File* menu pick *Edit input files*. In the dialog, pick `ElectronDiode.EIN` and click *OK*.. (If the file does not appear in the list, check that you have copied it correctly to the *Data folder* listed in **TC**.) The file has the following content:

```
* File ElectronDiode.EIN
Set DUnit 39.37
Set Geometry Cylin
Region 1 Epsi 1.0
Region 2 Epsi 81.0
Region 3 Epsi 2.7
Region 4 Potential -500.0E3
Region 5 Potential 0.0E3
Region 6 Potential 0.0E3
ENDFILE
```

The first command informs the program that the dimensions in `ElectronDiode.MOU` are in inches. The second command specifies that the geometry has cylindrical symmetry. The remaining commands set the properties of the dielectrics and electrodes in Fig. 2. The regions have the following associations: 1) high vacuum, 2) de-ionized water, 3) epoxy insulator 4) high-voltage center-conductor and 5) shaped portion of the grounded vacuum chamber wall. Region 6 is an open region (node assignment only) that does not appear in the element plot of Fig. 2. The region consists of the portions of the grounded upper and right-hand boundaries that are not part of Region 5.

In the *Run* menu, pick *Start run*. In the dialog, choose `ElectronDiode.EIN` and click *OK*. The electrostatic solution starts immediately and takes a second or two to complete. **EStat** creates two new files: `ElectronDiode.ELS` is a listing of run information that is useful if there are problems in the solution. The file `ElectronDiode.EOU` is the main output file that contains full information on the mesh as well as values of the electrostatic potential ϕ at each node. Section 16 describes the file format.

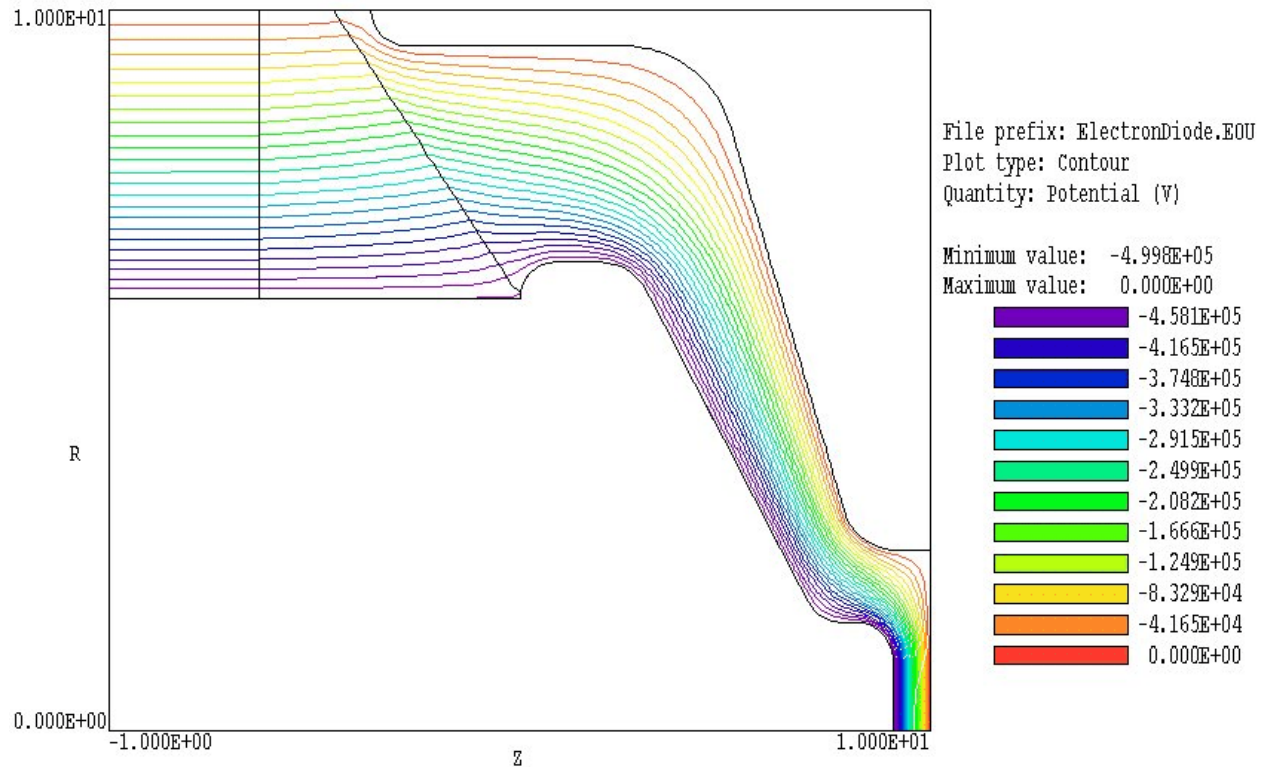


Figure 3. Equipotential contours in the full solution volume - *ElectronDiode* example.

The **VEStat** post-processor uses the information in `ElectronDiode.EOU` to create plots and to perform quantitative analyses. Run **VEStat** from **TC** and pick *Load solution file* in the *File* menu. Choose `ElectronDiode.EOU` in the dialog and click *OK*. The program loads the solution information and creates the default equipotential plot of Figure 3. You can experiment with the program capabilities to create a wide variety of plots. For example, Figure 4 shows variations of electric field amplitude $|\mathbf{E}|$ in the vacuum insulator region.

To complete the session, we shall run a script that performs automatic calculations. To inspect the file content, click on *Edit script* in the *File* menu and pick the file `ElectronDiode.SRC`. The file has the following content:

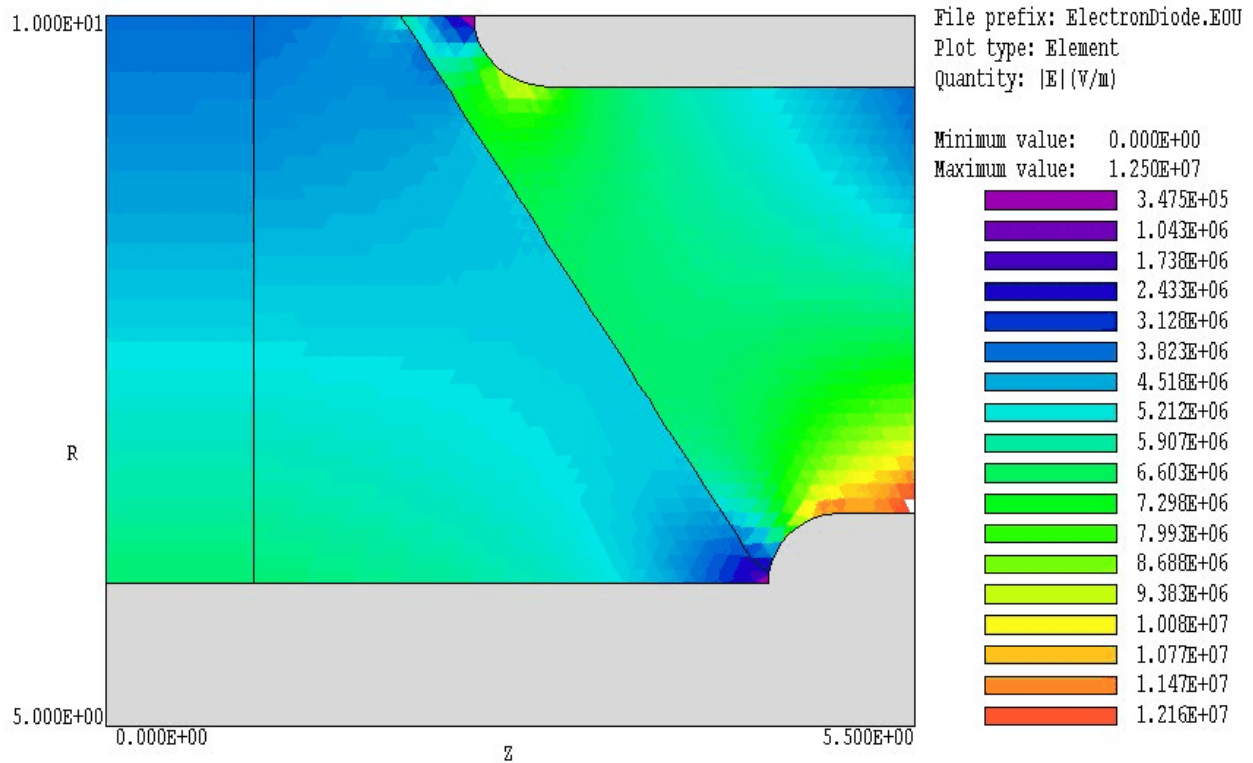


Figure 4. Variation of $|E|$ in the region of the vacuum insulator – *ElectronDiode* example.

```

INPUT ElectronDiode.EOU
OUTPUT ElectronDiode
VOLUMEINT
NSCAN 25
SCAN 9.9999 0.0000 9.9999 2.5000
ENDFILE

```

The first command ensures that the file `ElectronDiode.EOU` is loaded while the second command opens the data listing file `ElectronDiode.DAT`. The *VolumeInt* command initiates automatic volume integrals of electrostatic field energy. The *Scan* command writes a set of 25 data lines listing the potential and electric field components over the anode plane. The following is an excerpt from the listing:


```

--- Volume Integrals ---
Energy:  6.359E+01 J
Power:   7.182E+12 W
Epeak:   4.820E+07 V/m
ZPeak:   9.491E+00
RPeak:   1.242E+00

```

Integrals by region						
NReg	Volume (m3)	Energy (J)	Power (W)	PeakE (V/m)	PeakZ	PeakR
1	1.372E-02	5.545E+00	6.263E+11	4.820E+07	9.491E+00	1.242E+00
2	6.590E-03	5.602E+01	6.327E+12	6.365E+06	-9.381E-01	6.033E+00
3	7.073E-03	2.027E+00	2.289E+11	6.358E+06	1.022E+00	6.067E+00
4	1.583E-02	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
5	1.342E-02	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00

The field energy U_e in Region 1 (the vacuum diode) is 5.545 joules. Using the formula $U_e = CV^2/2$ and taking $V = 5.0 \times 10^5$, the capacitance of the diode region is approximately 44.36 pF.

3. Some electrostatic theory

The reference S. Humphries, **Field Solutions on Computers** (CRC Press, Boca Raton, 1997) gives detailed information on the theory of electrostatics and the application of finite-element methods in the **TriComp** programs. This section summarizes some basic concepts that may help you in preparing solutions.

In a region of ideal dielectrics and space charge, the electrostatic potential ϕ is determined by the Poisson equation:

$$\nabla \cdot (\epsilon_r \nabla \phi) = -\rho/\epsilon_0 . \quad (1)$$

In Eq. (1), the quantity ϵ_r is the relative dielectric constant and ρ is the space-charge density (in coulombs/m³). In general, values of ϕ , ϵ_r and ρ vary with position. The essence of the finite-element method is to divide a solution volume into a large number of elements. In the two-dimensional solutions of **TriComp**, the elements have triangular cross-sections. In planar solutions, elements are triangular prisms of unit length while the elements in cylindrical solutions are figures-of-revolution about the z axis. The convention in **EStat** is that ϵ and ρ have uniform values over elements. Furthermore, all elements of a region have the same value of ϵ and ρ . Therefore a region corresponds to a particular type of material, and

the boundaries of regions should conform closely to the shape of physical objects. The finite-element method yields values of the electrostatic potential at nodes of the solution space. The term *node* refers to the intersection points of element facets (vertices of the triangles). Values of ϕ at other points can be determined by interpolation, and values of electric field can be computed from the equation

$$\mathbf{E} = -\nabla\phi. \quad (2)$$

by taking numerical derivatives.

Equation (1) holds when the distribution of charges on electrodes and dielectric surfaces is influenced mainly by the flow of displacement current. **EStat** can also be used to model steady-state flow of real current in conductive media. In the absence of space charge, Eq. (1) has the form of the Laplace equation:

$$\nabla \cdot (\epsilon_r \nabla \phi) = 0. \quad (3)$$

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

$$\mathbf{j}_r = \sigma \mathbf{E} = -\sigma \nabla\phi, \quad (4)$$

where σ is the conductivity in S/m. Conservation of charge in the medium implies that

$$\nabla \cdot \mathbf{j}_r = 0, \quad (5)$$

or

$$\nabla \cdot (\sigma \nabla \phi) = 0. \quad (6)$$

Equation (6) has the same form as Eq. (3) with the association:

$$\epsilon_r \Rightarrow \sigma. \quad (7)$$

The implication is that the same routines can be used in **EStat** to find either dielectric or conductive solutions.

We can derive a criterion for when to seek dielectric or conductive solutions. In the presence of an RF electric field oscillating at frequency f , the displacement current density in a dielectric medium is given by:

$$j_d = 2\pi f \epsilon_o \epsilon_r E. \quad (8)$$

The real current is given by Eq. (4). The dielectric limit applies when $j_d \gg j_r$, or

$$\sigma \ll 2\pi f \epsilon_r \epsilon_o \approx \frac{\epsilon_r \epsilon_o}{\Delta t}. \quad (9)$$

The quantity Δt in Eq. (9) represents the approximate time scale for non-harmonic field variations. For example, with a 100 ns voltage pulse we can treat de-ionized water as an ideal dielectric if the conductivity is $\sigma \ll 7.2 \times 10^{-3}$ S/m or if the volume resistivity is much higher than 140 Ω -m. The solution is conductive when

$$\sigma \gg 2\pi f \epsilon_r \epsilon_o \approx \frac{\epsilon_r \epsilon_o}{\Delta t}. \quad (10)$$

EStat does not apply in the intermediate regime where both real and displacement currents are significant. The **TriComp** program **PAC** addresses this problem.

There are some cases where **EStat** can be used to model systems with a mixture of materials where some satisfy Eq. (9) and others satisfy Eq. (10). For example, Fig. 5 shows a resistive voltage probe for a high-voltage pulse generator. The conductive solution inside the probe has been adjusted to satisfy Eq. 10. The plastic housing and vacuum space have $\sigma = 0.0$ and therefore satisfy Eq. (9). In the dielectric solution of Fig. 5 the plastic and vacuum regions have relative dielectric constants $\epsilon_r = 2.7$ and $\epsilon_r = 1.0$. The conductive solution was assigned a value $\epsilon_r = 1.0 \times 10^4$. The electrostatic solution in the conductive region is determined by the shape of the medium with little dependence on the surrounding dielectrics. Note that the electric field in the solution has uniform amplitude and is purely

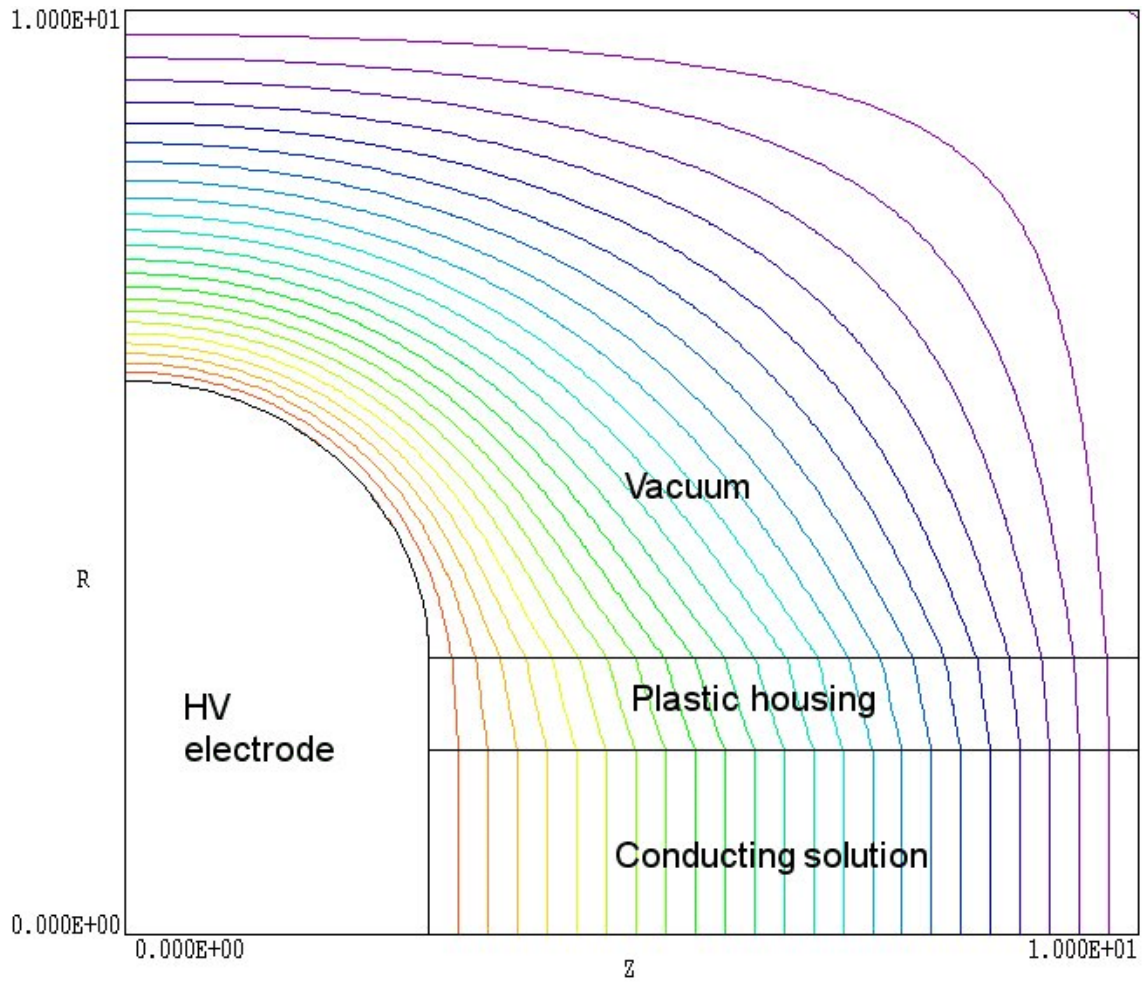


Figure 5. Illustration of a mixed dielectric-conductive solution - *ConductingProbe* example.

axial in direction. This solution, in turns, provides a boundary condition for the field solution in the surrounding insulators.

To conclude, we shall review boundary conditions in electrostatic solutions. The *boundary* is the outer edge of the solution volume. It may or may not correspond to the edges of the solution rectangle. Nodes on the boundary may assume one of two conditions.

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

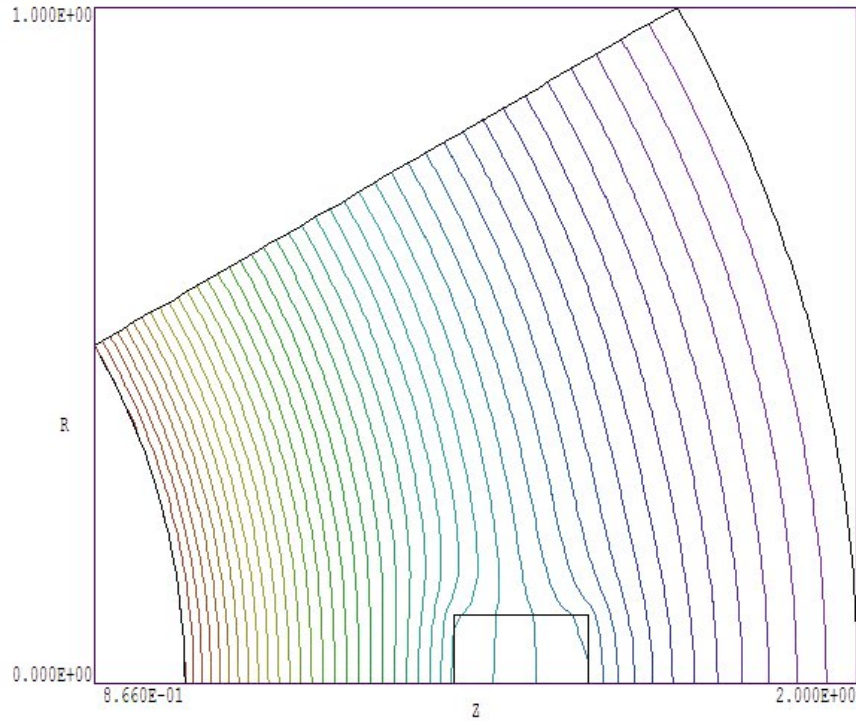


Figure 6. Application of Neumann boundaries to a symmetric system. The top and bottom surfaces assume the special Neumann condition.

■ A **Neumann** boundary is one where the normal derivative of the potential is specified. The boundaries in **EStat** are limited to the specific case $\partial\phi/\partial n = 0$. The special Neumann condition implies that the electric field is parallel to the boundary. One of the advantages of the finite-element method is that all boundaries that are not fixed automatically satisfy the special Neumann condition, even if they are slanted or curved.

Neumann boundaries are often used to reduce computation time for symmetric systems. For example, Figure 6 shows an equipotential plot for the *Gradient* example. The solution region represents a space between two spherical electrodes with a dielectric sample exposed to the electric field gradient. By utilizing automatic Neumann boundaries at the top and bottom, only part of the electrodes need be represented.

4. Organizing runs

The **EStat** package consists of a program that computes the physical solution (`EStat.EXE`) and a dedicated post-processor for analyses (`VEStat.EXE`). The solution program can run in two modes: interactively in a window or autonomously in the background with the option for batch file control. Similarly, **VEStat** 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.

EStat requires two input files.

- A **Mesh** output file that describes the conformal triangular mesh. The file contains node coordinates and the region numbers of elements and vertices.
- A command script file that sets control parameters for the solution program and describes the physical properties associated with region numbers.

The mesh output file always has a name of the form `FPREFIX.MOU`, where `FPREFIX` is a valid file prefix (1 to 20 characters). The command script must have a name of the form `FPREFIX.EIN` where `FPREFIX` is the same as that of the mesh file prefix. **EStat** issues an error message if both input files are not available in the working directory. To organize data, the resulting output files have names that begin with `FPREFIX`.

An **EStat** run consists of steps that involve three programs: mesh generation, field solution, and analysis. The programs communicate through data files. This arrangement is convenient because you can calculate several solutions from the same mesh by changing the material characteristics. Each run includes the following stages:

- Prepare a **Mesh** input script with a name of the form `FPREFIX.MIN` following the instructions in the **Mesh** manual. You can create the file directly with a text editor or graphically using the Drawing Editor of **Mesh**.
- Run **Mesh** either interactively from the **TC** program launcher or from the Windows Command Prompt to create the file `FPREFIX.MOU`.
- Prepare a command script (`FPREFIX.EIN`) using the *Create command file* menu entry in the solution program or your own text

editor. The allowed file commands are described in this manual.

- Run **EStat** to create the output file `FPREFIX.EOU`. The file is in ASCII format and contains information on the mesh geometry, the physical properties of regions, and values of computed quantities at the nodes. In electrostatic solutions the region properties are the relative dielectric constant (or conductivity) and space-charge density. The computed quantity is the electrostatic potential.

- Analyze the solution using **VEStat**. You can also transfer the information in `FPREFIX.EOU` to your own analysis programs.

5. Structure of the command file

The input script for all **TriComp** solution programs is a text file (ASCII format) with data lines containing commands and parameters. The file must end with the `EndFile` command. The entries on a line can be separated by the following delimiters

Space, Comma, Tab, Colon, Equal sign,
Left parenthesis, Right parenthesis

Any number of delimiters can be used in a line. Blank lines and comment lines are ignored. Comment lines begin with an asterisk (*). Most parameters are real numbers. The following formats are valid.

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

The bottom number is interpreted as 5.0.

The program will accept commands in any order. The following example illustrates a complete control file for **EStat**:

```
* File CYLPROBE.EIN
SET Geometry = Cylin
SET DUnit = 100
* Region 1: Set to EpsiR = 1.0
REGION(1) Epsi = 1.0
* Region 2: Dielectric sheath
```

```

REGION(2) Epsi = 1.0E-6
* Region 3: Probe
REGION(3) Potential = 1.0
* Region 4: Chamber wall
REGION(4) Potential = 0.0
ENDFILE

```

There are two types of commands that begin with different keywords:

```

SET
REGION

```

Commands that begin with the word SET control program settings. The keyword that follows determines the function of the command. One or more parameters follow the keyword. For example, the command

```

SET GEOMETRY CYLIN

```

instructs the program to use element weighting for cylindrical coordinates.

REGION commands set physical properties associated with elements and vertices. They all start with the word REGION and have same format.

Region	RegNo	Keyword	Value
--------	-------	---------	-------

Here, RegNo (an integer) is the region number defined in the **Mesh** input file. The string Keyword specifies the physical property. The keyword is followed by one or more numbers. As an example, the command

```

REGION 2 Potential -5500.0

```

sets vertices with region number 2 as fixed potential points with the value -5.5 kV.

6. Commands for general program control

EStat command files have the suffix **EIN**. Commands divide into two groups: those that control the program operation and those that set the physical properties of regions. We begin with the program commands. They begin with the keyword *Set*. Each command is written as it might appear in the **FPREFIX.EIN** file. The following two commands are used in all **TriComp** solution programs.

SET DUNIT 1.0E4

You can use any convenient distance units for the **Mesh** input file. The quantity **DUnit** tells how to convert coordinates supplied by **Mesh** to the standard distance units of meters used in **EStat**. The quantity *DUnit* equals the number of **Mesh** units per meter. For example, if the **Mesh** dimensions are entered in microns, set $DUnit = 1.0 \times 10^6$. The default value is 1.00.

Spatial quantities recorded in the output file **FPrefex.EOU** are always in meters. The value of *DUnit* is passed to **VEStat**. In post-processor analyses, spatial quantities in graphs and listing files are always scaled to the **Mesh** units. For example, if the **Mesh** dimensions are in cm and $DUnit = 100.0$ in the **EStat** command file, the output of spatial quantities in **VEStat** is in cm.

SET GEOMETRY CYLIN

EStat handles problems in rectangular or cylindrical geometries. Rectangular systems have variations in x and y with infinite length in z . Cylindrical systems have variations in r and z with azimuthal symmetry. The command options are *Rect* and *Cylin*. In cylindrical solutions the program takes the z -axis along the **Mesh** x direction and the r -axis along y . In this case the program issues an error message if any node has a y coordinate less than 0.0. **VEStat** automatically adjusts calculations and labels to represent the two geometries.

SET OMEGA 1.85 1.90

Controls the *over-relaxation factor* for the iterative matrix solution, a number in the range 1.0 to 2.0. The two parameters are *OmegaMin* and *OmegaMax*. The over-relaxation factor rises from *OmegaMin* to *OmegaMax* during an adjusted number of cycles. Generally, high values give faster convergence. Reduce the over-relaxation factor if the solution does not converge. If this command does not appear, the

program uses the Chebyshev acceleration method to pick optimal values of the over-relaxation factor.

SET MAXCYCLE 200

The maximum number cycles for the iterative matrix solution. The default value is 2500.

SET RESTARGET 5.0E-6

EStat keeps track of the relative error in the magnitude of the potential during the iterative matrix solution. The program stops when the error falls below *ResTarget*. Values of *ResTarget* less than about 5.0×10^{-6} give good accuracy. The default is 5.0×10^{-7} .

7. Commands for material properties

The **EStat** command set for defining the material properties of regions is quite simple. The four commands to set the physical properties of materials all start with the word *Region* and have same format.

Region	RegNo	Keyword	Value
---------------	--------------	----------------	--------------

Here, *RegNo* (an integer) is the region number defined in the **Mesh** input file. *Keyword* is one of the words listed below and *Value* is a real number.

REGION 4 POTENTIAL 3500.0

The keyword *Potential* designates that the potential of the region is fixed and will not change during the relaxation. The command sets the amplitude of the potential in volts. The default value for all fixed regions is 0.0 V. This command applies to both dielectric and conductive type solutions.

REGION 2 EPSI 5.8

Sets the relative dielectric constant, $\epsilon_r = \epsilon/\epsilon_0$, in dielectric regions. The default value in non-fixed regions is $\epsilon_r = 1.0$. This command applies only to dielectric type solutions.

REGION 5 EPSI 5.8 2.6 45.0

EStat can handle anisotropic dielectrics with different values of relative dielectric constant along two normal axes. The first real number (ϵ_1) following the keyword *EPSI* is the relative dielectric constant along Axis 1, and the second number (ϵ_2) is the dielectric constant along Axis 2. The default values are $\epsilon_1 = \epsilon_2 = 1.0$. The third number (θ_1) is the angle of Axis 1 relative to the x (or z) axis. The angle of Axis 2 is $\theta_2 = \theta_1 + 90^\circ$. It is important to note in cylindrical simulations that all structures must have cylindrical symmetry. Therefore, if $\theta_1 \neq 0.0^\circ$, Axis 1 has the shape of a cone. Because it is very unlikely to encounter such a material, practical cylindrical simulations are generally limited to the choice $\theta_1 = 0.0^\circ$ (*i.e.*, different relative dielectric constants along z and r).

REGION 7 RHO 3.6E-3

Sets the space-charge density in coulombs/m³. The quantity is not physically-meaningful in problems with conductive materials. This command applies only to dielectric type solutions. In this case both the *Epsi* and *Rho* commands can apply to the same non-fixed region.

REGION 7 SIGMA 0.145

Sets the conductivity in S/m. The conductivity is equal to the inverse of the volume resistivity ρ in units of Ω -m. This command applies only to conductive type solutions.

REGION 3 SIGMA 5.0 0.1 45.0

This form of the *SIGMA* command defines a region with anisotropic conductivity. The first real number (σ_1) following the keyword *SIGMA* is the electrical conductivity in S/m along Axis 1, and the second number (σ_2) is the conductivity along Axis 2. The third number (θ_1) is the angle of Axis 1 relative to the x (or z) axis. The angle of Axis 2 is $\theta_2 = \theta_1 + 90^\circ$.

In a single solution, you can use the *Sigma* command or the *Epsi/Rho* commands. **EStat** issues an error if dielectric and conductive quantities are mixed. The program inserts a flag in the output file to alert **VEStat** whether to calculate quantities appropriate to a dielectric or conductive solution.

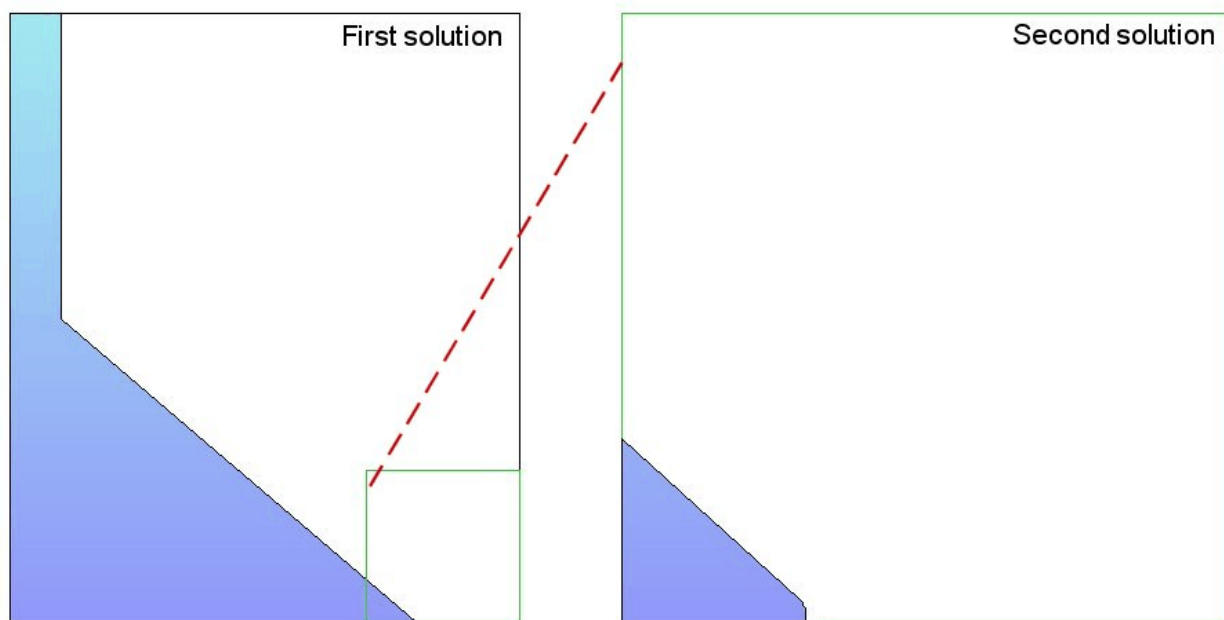


Figure 7. Interpolated Dirichlet boundary for a microscopic solution.

8. Boundary and superposition commands

Version 5.0 of **EStat** incorporates a powerful new feature that enables accurate calculations of fields near small features in a large solution space. Figure 7 illustrates the procedure. 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 support 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 (**Mesh** manual, Section 6) to create small elements. A limitation to this technique arises from the structured mesh used in **TriComp** – the region of small elements must extend the full length of the solution volume.

Figure 7 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 may penetrate the subregion. The difference is that the electrode shapes may be resolved in much greater detail. The question is how to incorporate the macroscopic fields correctly into the microscopic solution. One resolution is to enclose the second solution in a variable-potential Dirichlet boundary. Values of ϕ on the boundary are calculated by interpolation at the corresponding point

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

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

SET BOUNDARY FPrefix [BndScale]

SET BOUNDARY FEmitMacro

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.

EStat issues an error message under the following conditions: 1) the output file for Solution 1 (*FPrefix.EOU*) is not available in the working directory, 2) Solution 2 does not fit completely inside Solution 1, 3) Solution 1 and Solution 2 have different symmetries, or 4) 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.

A node in Solution 2 is on the boundary under two conditions:

- It has indices $K = 1$, $K = K_{\max}$, $L = 1$ or $L = L_{\max}$. In solutions with cylindrical symmetry, points on the axis ($L = 1$ and $r = 0.0$) are *not* set to the Dirichlet condition.
- One of the adjacent elements has *RegNo* = 0. This condition means that you can use non-rectangular boundaries for Solution 2.

Figure 8 illustrates the boundary point criteria. Solution 1 (*BoundaryVal01*) represents electric fields between two spherical-section electrodes separated by a 3.00" gap. We want to find the fields on a small dielectric body inserted at the center of the gap. Solution 2 (*BoundaryVal02*) is a spherical region (radius 1.00") centered at the midpoint. The microscopic solution contains a dielectric sphere of radius 0.10". Note the assignment of potential values on the circular outer boundary of Solution 2. The lower boundary of the cylindrically-symmetric solution has not been set to the Dirichlet condition; therefore, the potential can adjust to the presence of the dielectric.

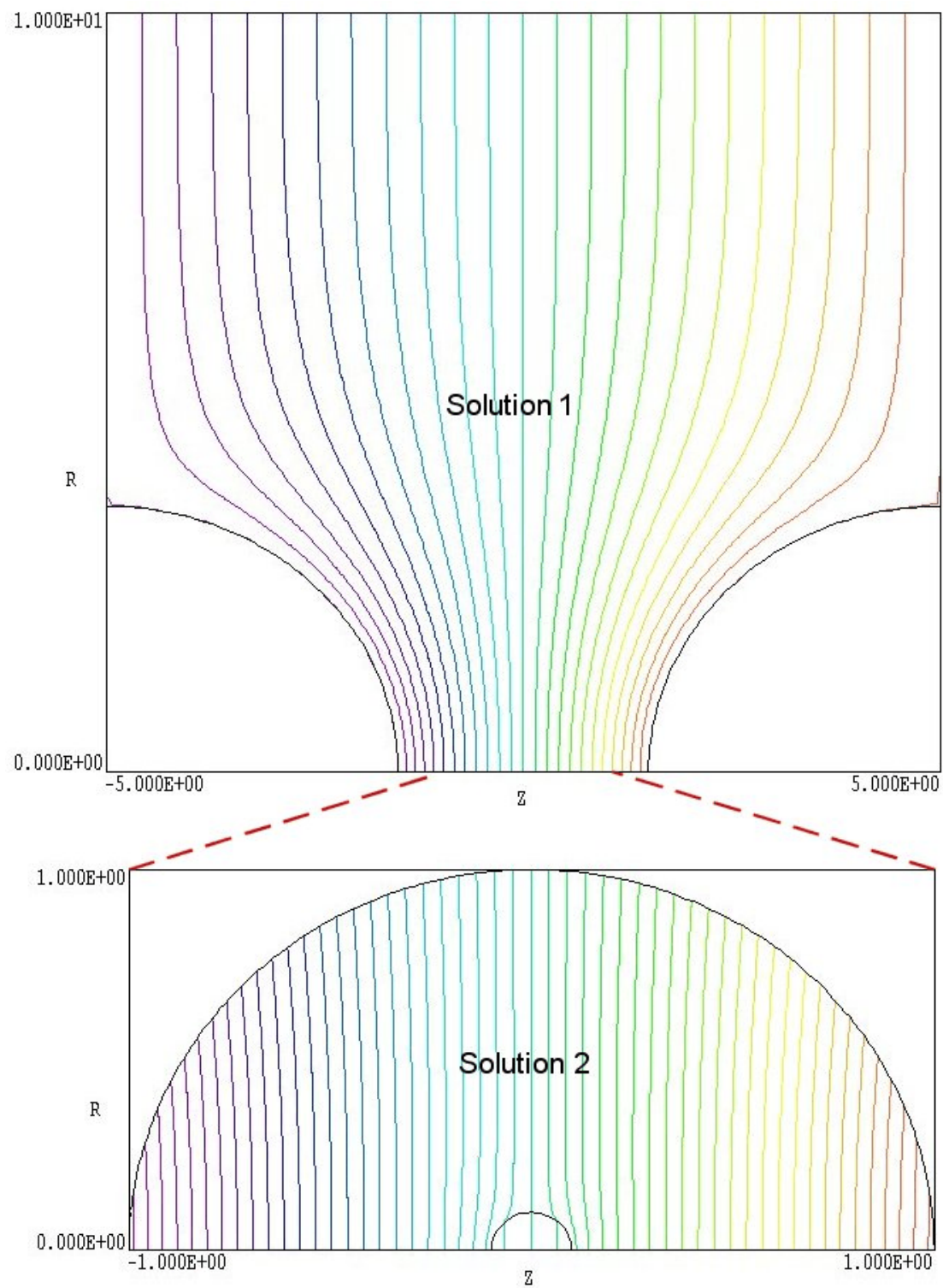


Figure 8. Example of a two-stage solution using the Boundary command.

In response to the *Superposition* command, **EStat** 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.

**SET SUPERPOSITION FPrefix [SuperScale]
SET 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.

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

$$\phi_2' = \phi_2 + \textit{SuperScale} \times \phi_1 , \quad (11)$$

before writing the output file.

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

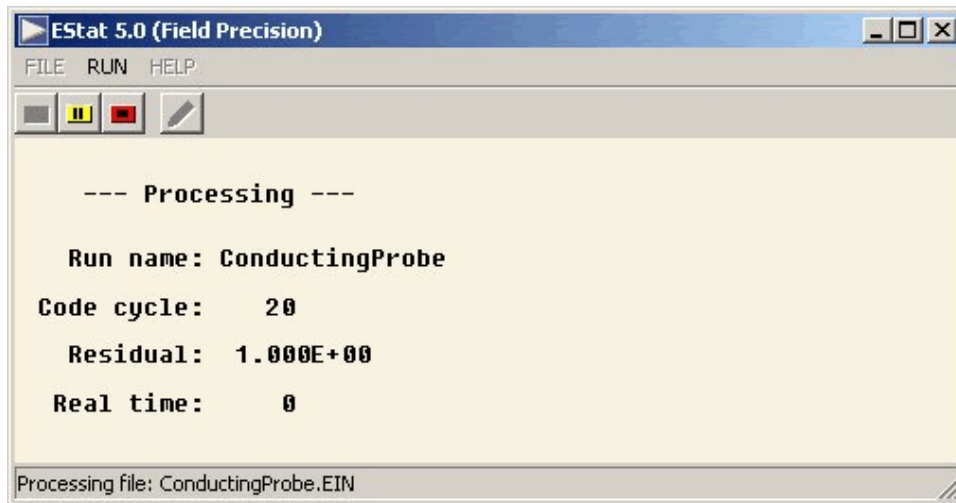


Figure 9. EStat in the interactive mode

9. Running EStat interactively

The program `ESTAT.EXE` can run interactively in a Window (Figure 9). In this mode you can carry out several solutions in a session or leave the program while you work in other Windows applications.

The most convenient way to run **EStat** and **VEStat** is from the **TC** program launcher. Operation and setup of the launcher is described in the *TC Manual*. You can also run the executable programs individually from Windows Explorer or add shortcuts to the desktop or the Windows Start Menu.

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

Create command file

This command calls up a full-featured internal Windows editor to create an input script. Supply a run prefix (1-20 characters). The allowed commands for the specific solution program appear as a set of comment lines. Save or abandon the file to return to the solution program.

Edit input file
Edit listing file
Edit file

The commands call the editor to inspect or to modify ASCII input and output files for **EStat**. Choosing a file from an alternate directory does not change the working directory. The *Edit input file* command shows a list of all files with names of the form `FPREFIX.EIN`. The *Edit listing file* command displays files with names `FPREFIX.ELS`.

The *Run* menu has four commands.

Start run

Pick an input file (such as `FPREFIX.EIN`) to start a solution. The working directory is changed if you pick a file from an alternate directory. The run begins if the file `FPREFIX.MOU` is present. The small text window displays information on the run status.

Pause run

The intensive calculations of the solution program 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 and reactivate it later without loss of data.

Stop run

This command terminates the program and saves the current state of the relaxation solution. For example, you may want to stop a relaxation solution at a moderate value of convergence to check whether the problem has been correctly defined.

10. Running EStat from the command prompt

To run **EStat** in the background, go to the Command Prompt from Windows and log to the data directory that contains the required MOU and EIN files.. For example, suppose the electrostatic input data files SWITCH.MOU and SWITCH.EIN are stored in \TRICOMP\BUFFER and that the program ESTAT.EXE is in the directory \TRICOMP. From \TRICOMP\BUFFER type

```
..\ESTAT SWITCH <Enter>
```

The program runs silently, writing detailed information in the listing file SWITCH.ELS. If the solution is successful, the program writes the output file SWITCH.EOU to the data directory. During lengthy runs you can perform other tasks in Windows. Note that the response of the computer may be slowed considerably.

The main function of the command mode is autonomous operation under batch file control. This feature is useful if you want to make an extended series of solutions overnight or in the background. As an example, suppose you have prepared the input files SWT01.MIN, . . . , SWT08.MIN and SWT01.EIN, . . . , SWT08.EIN in the directory \TRICOMP\BUFFER. Furthermore, you have written the following batch file SWRUN.BAT in the data directory using an ASCII text editor.

```
@ECHO OFF
ECHO Main switch data run
START ..\MESH.EXE SWT01
START ..\ESTAT.EXE SWT01
START ..\MESH.EXE SWT02
START ..\ESTAT.EXE SWT02
. . .
START ..\MESH.EXE SWT08
START ..\ESTAT.EXE SWT08
```

Type

```
SWRUN <Enter>
```

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

Microsoft has released over thirty versions of it's 32-bit operating system since Windows 95. There is considerable inconsistency in DOS emulation between versions. To ensure consistent batch file operation we supply the utility **GCon** with all our software. The program emulates many DOS

commands and has advanced features to organize and to analyze large data runs. To avoid problems, we advise running batch scripts from **GCon** rather than from the Command Prompt.

11. VESat file menu

VEStat (for View EStat) is the interactive graphical analysis program for output files from **EStat**. The program automatically adjusts labels and calculated quantities depending on whether the solution is dielectric or conductive. In the following list, italicized quantities apply only to conductive solutions:

Program units

Spatial dimensions: meters or units set by *DUnit*.

Potential amplitude: V (volts)

Electric field: V/m (volts/meter)

Space-charge density: coulombs/m³

Current density: A/m² (amperes per square meter)

Energy density: J/m³ (joules per cubic meter)

Power density: W/m³ (watts per cubic meter)

VEStat has the following popup menus: *File*, *Spatial plots*, *Analysis*, *Scan 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* menu.

Load solution file

The *Load solution file* command displays a dialog with a list of solution files (suffix EOU). 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 *Spatial plots* and *Analysis* menus become active.

Open data listing 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.

Close data listing file

Use this command if you want to start a new file to record data. You must close the file before attempting to view or edit it using the built-in editor of the post-processor.

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 the **TriComp** postprocessing programs. This command displays a dialog with a list of analysis files that you have created with the suffix SCR. Pick a file and click *OK*. The script file can load data files, open and close history files, and perform any of the quantitative analysis functions described in this manual. The script command language is described in Sect. 15. 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 of the post-processor. 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 list of allowed commands for reference. This list follows the *EndFile* command and will be ignored by **VEStat**. 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.

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.

12. VESat spatial plot menu

Spatial plots show variations of quantities over the two-dimensional space of the simulation. **VESat** makes a default contour plot when a data file is loaded.

Plot type

VESat can create a variety of colored screen and hardcopy plots.

Mesh. Outline of the elements in the computational mesh.

Region. Computational mesh with elements color-coded by region number.

Contour. Lines that follow constant values of ϕ or $|\mathbf{E}|$. In electrostatic solutions constant values of potential ϕ are normal to electric field lines. The distance between lines is inversely proportional to the electric field amplitude. Note that contours of $|\mathbf{E}|$ may exhibit regions of compressed lines because is electrical field magnitude is not a continuous function of position.

Element. Elements of the solution space color-coded according to a computed quantity (such as the electric field magnitude). You can activate or deactivate element boundaries with the *Toggle element outline* command.

Vector. An element plot with orientation lines included in each element to show the local direction of a vector quantity.

Surface. A three-dimensional plot where a computed quantity is displayed along z over a region in the x - y or z - r plane. The spatial limits of the plot correspond to the current view window for *Mesh*, *Region*, *Contour*, *Element* or *Vector* plots.

A plot type may not support some plotted quantities. For example, a vector plot of electrostatic potential is undefined. If you receive a message when you switch plot types that the current quantity is not allowed, use the *Plotted quantity* command to pick a valid option.

Plotted quantity

A dialog shows a list of available quantities consistent with the current plot type. Note that the list will be empty for *Mesh* and *Region* plots.

VEStat supports the following plot quantities. Note that available quantities depend on the solution type (dielectric or conductive).

Contour plot quantities

Electrostatic potential, ϕ
Electric field amplitude, $|E|$

Element plot quantities

Electrostatic potential, ϕ
Electric field amplitude, $|E|$
Current density amplitude, $|j| = \sigma|E|$

Vector plot quantities

Electric field, E
Current density, $j = \sigma E$

Surface plot quantities

Electrostatic potential, ϕ
Electric field amplitude, $|E|$
Current density amplitude, $|j| = \sigma|E|$
Horizontal field, E_x or E_z
Vertical field, E_y or E_r

Plot limits

In the default *autoscale* mode the program picks limits in *Contour*, *Element*, *Vector* and *Surface* plots that span the full range of the current quantity. With this command you can set manual limits. In the dialog uncheck the *Autoscale* box and fill in minimum and maximum values. Note that the program does not check that the values you supply are physically reasonable. This operation will not affect scaling of other plot quantities. Check the box to return to *autoscale* mode.

Toggle grid

A set of dashed grid lines in x - y or z - r can be superimposed on *Mesh*, *Region*, *Contour*, *Element* and *Vector* plots. **VEStat** automatically chooses intervals and positions so that lines occur at convenient values of x or y (for example, 0.01 rather than 0.01153). Grids corresponding to the axes ($x = 0.0$ or $y = 0.0$) are plotted as solid lines.

Set contour plot style

This command is active only when the current plot type is *Contour*. There are four choices: monochrome, monochrome with labels, colored and colored with labels. In the colored mode, the lines are color-coded according to the value of the plotted quantity. A legend is included in the information window to the right of the plot. In the labeled modes, contour lines are numbered according to their values. Overlapping labels on closely-spaced lines may look better in a zoomed view.

Toggle element outline

This command determines whether the element boundaries are included in *Element* and *Vector* plots. It may be necessary to deactivate outlines to view large meshes.

Toggle mouse/keyboard

By default the program uses interactive mouse entry of coordinates for commands like *Line scan* and *Zoom*. This command switches the program between mouse and keyboard input. Enter keyboard coordinates in the distance units used in **Mesh**. In other words, if the solution program has $DUnit = 1.0 \times 10^6$, then enter dimensions in microns.

Toggle snap mode

The mouse snap mode is a convenient feature of **VEStat**. When snap mode is active, the mouse returns the coordinate values closest to an integer multiple of the quantity $DSnap$. In other words, if $DSnap = 0.5$ and the mouse position is [5.4331,-2.6253], the returned coordinates are [5.5,-2.5]. By default, snap mode is *ON*. Snap mode is automatically turned off for coordinate input to the commands *Point calculation* and *Element properties*. Otherwise, the program could pick a location closest to the snap point rather than the point at the tip of the mouse arrow, giving misleading information.

Set DSnap

Set the distance scale for the mouse snap mode.

The following commands change the view limits in *Mesh*, *Region*, *Contour*, *Element* and *Vector* plots. The current view limits of the two-

dimensional plots are used when creating three-dimensional *Surface* plots.

Zoom window

You can zoom in on any area of the plot by specifying the two corners of a view box with the mouse. The coordinates are displayed in the window below the plot. The returned coordinate values depend on whether snap mode is active. You can also enter view coordinates from the keyboard by issuing the *Toggle mouse/keyboard* command.

Zoom in

Magnifies the view about the center of the current plot.

Expand view

Expands the view area about the center of the current plot.

Global view

Returns the plot boundaries to the full solution area.

Pan

Moves the center of the plot. Enter two points to define a displacement vector.

The following commands control the appearance of *Surface* plots. The commands are active only when a *Surface* plot is displayed. You may notice that there is a delay if you choose the *Surface* option in the *Plot type* command or if you change the plotted quantity when a *Surface* plot is active. To create the plot, the post-processor must map the current quantity to a rectangular grid, performing a large number of interpolations.

Rotate 3D image

Rotate the *Surface* plot 90° in the spatial plane.

View angle 3D

Change the angle of view for the *Surface* plot.

Set grid 3D

To create the plot, a quantity is mapped to a rectangular grid with dimensions $NX \times NY$. These numbers also determine the total number of grid lines in the *Surface* plot. The default values are $NX = NY = 40$. You can change values with this command. Although higher numbers give plots with more detail, the regeneration time is longer and the screen display may be unattractive. The command may cause a program delay because values must be recalculated.

Default printer

Spatial and scan plots can be ported to any installed Windows printer (including network printers, postscript drivers,...). You can generate colored plots if you have a color printer. When you issue the *Default printer* command, the current screen plot is sent to the default Windows printer. If necessary, change the default using the *Settings* command of Windows *before* running **VEStat**.

Plot file (PostScript)

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

Plot file (BMP)

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

Plot file (PNG)

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

Copy to clipboard

Copy the current plot to the clipboard in Windows MetaFile format. The plot can then be pasted into a compatible graphics program.

13. VESat analysis menu

The commands in the *Analysis* menu generate numerical data. Most of the functions require coordinate input from the user, usually through the mouse. Therefore, the analysis menu is active only when a *Mesh*, *Region*, *Contour*, *Element* or *Vector* plots is displayed.

Point calculation

VEStat employs a sophisticated interpolation technique that preserves discontinuities at material boundaries. The program gives correct values for electric field on both sides of a dielectric boundary. Click on the command and then point to any position in a *Mesh*, *Region*, *Contour*, *Element* or *Vector* plot. Note that snap mode is turned *OFF* for coordinate input. The program writes a subset of interpolated quantities to the window below the plot and also records complete information if a data file is open. To enter point coordinates by keyboard, use the *Toggle mouse/keyboard* command.

Toggle interpolation

The default interpolation method for the *Point calculation* and *Line scan* commands is a second-order least-squares fit with intelligent collection of data points. For example, only points on the side of a dielectric boundary that contains the target point are included to give the correct field discontinuity at the boundary. The least-squares fit may fail in very small regions or enclosed areas if the program cannot identify enough data points. In this case, toggle to the linear mode. Here, field values are determined by a first order fit in the element that contains the target point.

Line scan

The line scan is one of the most useful functions of **VEStat**. After clicking on the command, supply two points with the mouse in a view of a *Mesh*, *Region*, *Contour*, *Element* or *Vector* plot to define a scan line. The snap mode is useful in this application (for example, you may want a 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 at equal intervals along the line. The information is recorded if a history file is open. The program also makes a screen plot of the currently selected quantity versus distance along the scan and activates the commands in the *Scan plot* menu (see Section 14). **VEStat** 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).

Scan plot quantity

With this command you can choose the quantity to display in screen and hardcopy plots of line scans. Pick the quantity from the list box and click *OK*. This setting has no effect on the history file listing which includes all field quantities. The available line scan quantities depend on the solution type. The choices are electrostatic potential (ϕ), electric field amplitude ($|\mathbf{E}|$), horizontal component of electric field (E_x or E_z), vertical component of electric field (E_y or E_r), current density amplitude ($|\mathbf{j}|$), horizontal component of current density (j_x or j_z), vertical component of current density (j_y or j_r) and power density (σE^2).

Set number of scan points

This command sets the number of line scan points used for the screen plot and recorded in the history file. The default value is 50 and the maximum number is 500.

Element properties

Pick a triangular element with the mouse (or keyboard) and the post-processor writes material and field properties of the element to the screen. The information is also recorded if a history file is open. It is usually best to start from a *Mesh*, *Region*, *Element* or *Vector* plot to identify the element.

Region properties

To see the physical properties associated with a region of the solution space, click the mouse close to any arc or line vector of the region. Partial results are shown on the screen and a complete analysis is included in the data file. **VEStat** calculates volume integrals of field energy density and power density over the specified region. The following line integrals are evaluated around the region boundary.

1) Integrals of the surface charge density to find the induced charge on electrodes. The calculation is carried out only on the surfaces of fixed-potential regions that have non-zero volume (*Filled* regions). The density of free charge on the surface is $\sigma_{free} = \epsilon_o \epsilon_r E_{\perp}$ (coulombs/m²)

where E_{\perp} is the normal component of electric field just outside the electrode. The density of total charge (free plus dielectric) at the surface is $\sigma_{tot} = \epsilon_o E_{\perp}$. **VEStat** identifies all element boundaries on the region surface. For each boundary segment the program calculates the normal field component in the adjacent element and multiplies the result by the surface area. The results are in coulombs/m for rectangular solutions and coulombs in cylindrical geometries. Note that this process may be inaccurate for electrodes with sharp edges. In this case, it is better to use line scans to take a Gaussian integral around the electrode.

2) Integrals of current density to find the total current to electrodes in resistive simulations. The calculation uses the same method as that for the induced charge except the calculated quantity is $j_{\perp} = \sigma E_{\perp}$. The results are in A/m in planar geometry and A for cylindrical simulations.

If a data file is open, **VEStat** also writes an ordered list of electric field values on facets that constitute the region surface. Such a listing may be useful for the design of high-voltage equipment. The following is an example of a surface list:

X	Y	D	E	Ex	Ey	NBorder
=====	=====	=====	=====	=====	=====	=====
-1.7709E+00	1.0009E+00	0.0000E+00	3.8431E+06	-1.6070E+05	-3.8397E+06	1
-1.8150E+00	1.0048E+00	4.4245E-02	4.0071E+06	-5.2130E+05	-3.9730E+06	1
-1.8616E+00	1.0132E+00	9.1649E-02	4.1001E+06	-9.1630E+05	-3.9964E+06	1
-1.9025E+00	1.0242E+00	1.3400E-01	4.3106E+06	-1.3157E+06	-4.1049E+06	1
-1.9415E+00	1.0387E+00	1.7565E-01	4.2288E+06	-1.6218E+06	-3.9055E+06	1
-1.9826E+00	1.0580E+00	2.2100E-01	4.2118E+06	-1.9614E+06	-3.7272E+06	1
...						

Entries include the coordinates of the midpoint of the facet and the corresponding values of $|E|$, E_x and E_y (or E_z and E_r) calculated in the element adjacent to the target region. In other words, if the target region is an electrode the calculation is performed in the adjacent dielectric. The parameter *NBorder* is the region number of the adjacent element. The routine also works on borders between dielectrics. In fact, you can get find field values on both sides of a dielectric interface by doing region analyses of the two materials. This feature is useful for checking vacuum insulator stress.

Note that **VEStat** has placed the facets in the order of a continuous surface. The start facet is the one with a midpoint closest to the x axis (planar) or z axis (cylindrical). If the region surface is closed and does

not lie on the boundary of the solution area, the list goes around and meets at the start point. The quantity D is the surface distance from the start point. All coordinates and distances are given in units set by $DUnit$. If the region surface is not closed or if the region intersects a boundary, then the ordering may not be perfect.

Line integrals

Input for this command is similar to that for the *Line scan* command. Enter two points to define a line. **VEStat** calculates line integrals that may be useful for calculations of capacitance and resistance. In a planar dielectric simulation, Gauss' law has the form

$$\oint dl \epsilon_o \epsilon_r E_{\perp} = q_{free},$$

$$\oint dl \epsilon_o E_{\perp} = q_{tot}.$$

The integral is taken around a closed path in the x - y plane. The quantity E_{\perp} is the electric field component normal to the path pointing out of the closed volume. The quantity q_{free} is the free charge per meter enclosed and q_{tot} is the total enclosed charge (free plus dielectric). In a cylindrical simulation, Gauss' law is given by

$$\oint dl 2\pi r \epsilon_o \epsilon_r E_{\perp} = Q_{free},$$

$$\oint dl 2\pi r \epsilon_o E_{\perp} = Q_{tot}.$$

The closed path lies in the r - z plane and Q_{free} is the enclosed charge.

VEStat computes the following line integrals for a dielectric solution:

$$\int dl \epsilon_o \epsilon_r E_{\perp},$$

$$\int dl \epsilon_o E_{\perp},$$

$$\int dl 2\pi r \epsilon_o \epsilon_r E_{\perp},$$

$$\int dl 2\pi r \epsilon_o E_{\perp},$$

depending on whether the geometry is planar or cylindrical. You can find the total charge on an electrode by enclosing it with multiple line integrals or by using symmetry properties of the solution. If you use multiple integrals, be sure that they enclose the volume in a clockwise direction.

For a conductive simulation, the following integrals give the current/meter (planar) or current (cylindrical) through the surface defined by the line.

$$\int dl \sigma E_{\perp},$$

$$\int dl 2\pi r \sigma E_{\perp}.$$

Volume integrals

No input is needed for this command. **VEStat** automatically computes integrals of quantities over the full solution volume and over individual regions. Information is recorded on the screen or in a history file. The following analyses are performed:

- 1) Volume integrals of electrostatic field energy density,
 $u = \epsilon_o \epsilon_r E^2/2$ over the full volume and region volumes of dielectric solutions. The output units are J/m in rectangular solutions and J in cylindrical solutions.
- 2) Volume integrals of power dissipation $p = \sigma E^2/2$ over the full volume and region volumes of conductive solutions. The output units are W/m in rectangular solutions and W in cylindrical solutions.
- 3) The location and value of the maximum electric field in the solution space and in each region.

Matrix file

VEStat can make matrix files of field values to help you create your own analysis routines. Although information is available in the output file of the solution program, it may be difficult to deal with the conformal triangular mesh. The *Matrix file* command uses the interpolation capabilities of the program to create an ASCII data file of field quantities on a rectangular grid in x-y or z-r. The command

displays a dialog box where you set the matrix file prefix, the dimensions of the box and the number of intervals along x and y (or z and r). The program creates the file `FPREFIX.MTX` in the current directory.

14. VESat scan plot menu

The commands of the *Scan plot* menu become active when a plot is created following the line scan command. The commands to export plots were described in Section 12. This section describes unique commands of the Scan plot menu.

Oscilloscope mode

In the oscilloscope mode, a scan plot assumes characteristics of a digital oscilloscope. The program 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 displays the plot values along with the numerical derivative and integral of the curve. The definite integral is taken from the left-hand side of the plot to the current point. Values are displayed on the screen and written to the data file if open. Press the right mouse button to exit the oscilloscope mode.

Toggle scan symbols

The setting determines whether plot symbols are added to the scan plot showing calculated points.

Toggle grid

The setting determines whether grid lines are added to the screen and hardcopy scan plots.

Close scan plot

The scan plot must be closed before you can use the *File* and *Analysis* functions of **VESat**. This command closes the scan plot and returns the program to the previous spatial plot.

15. VESat script file commands

Script files to control **VESat** have a name of the form `FPREFIX.SCR`. They should be in the same directory as the data files. Scripts are ASCII

files that follow the **TriComp** syntax conventions. The program ignores blank lines and indentations. Data lines use the standard delimiters and comment lines begin with '*' (asterisk). Processing ends when the *EndFile* command is encountered.

To run a script, click on the *Run script* command in the *File* menu. The program shows a list of available script files. Pick a file and click *OK*. The script operates on the presently loaded data file or you can load other files from within the script. You can also sequentially open one or more data files.

VEStat can run autonomously under script file control from the Windows Command Prompt. Suppose you have a file *GTest .SCR* in the directory `\TRICOMP\BUFFER` and that *VEStat .EXE* is in the directory `\TRICOMP`. From `\TRICOMP\BUFFER`, type

```
..\VESTAT GTEST <Enter>
```

The main application of the Command Prompt mode is to generate data files and to perform extensive analyses under batch file control.

The following commands can appear in a script:

Input Switch1.EOU

Close the current data file and load a file for analysis. The parameter is the full name of the data file. For the command illustrated, the post-processor would load the file *SWITCH1 .EOU*. You can load several files for sequential analysis.

Output SW02

Close the current history file and open an output file *SW02 .DAT*.

Point 5.65 10.68

Perform interpolations at the specified point and write the results to the data file. The two real number parameters are the coordinates of the point (*x-y* or *z-r*) in **Mesh** units.

Scan 0.00 0.00 10.00 0.00

Write the results of a line scan between the specified points to the data file. The four real number parameters are the starting and end

coordinates in **Mesh** units: $[x_{\text{start}} y_{\text{start}} x_{\text{end}} y_{\text{end}}]$ or $[z_{\text{start}} r_{\text{start}} z_{\text{end}} r_{\text{end}}]$.

Interpolation LSQ

Set the interpolation method for subsequent *Point*, *Line scan* and *Matrix* commands. The options are *LSQ* (least-squares fit) and *Linear*.

Element 5.65 10.68

Write the properties of the element at the specified point to the data file. The two real number parameters are the coordinates of the point (x - y or z - r) in **Mesh** units.

NScan 150

Set the number of points in a line scan. The default is 50 and the maximum number is 500.

Region 5

Write volume and surface integrals for a region to the data file. The integer parameter is the region number.

LineInt 0.00 0.00 10.00 0.00

Write line integrals along a scan line to the data file. The four real number parameters are the starting and end coordinates in **Mesh** units: $[x_{\text{start}} y_{\text{start}} x_{\text{end}} y_{\text{end}}]$ or $[z_{\text{start}} r_{\text{start}} z_{\text{end}} r_{\text{end}}]$.

VolumInt

Write volume integrals for the full solution and regions to the data file.

Matrix Switch1 10 20 0.00 0.00 5.00 10.00

Create a matrix file and record values. This command must have the following seven parameters.

Param 1: The prefix of the matrix file `FPREFIX.MTX` (string).

Param 2: Number of intervals along the x (or z) direction (integer).

Param 3: Number of intervals along the y (or r) direction (integer).

Param 4-7: The coordinates of the corners of a box in the solution volume, $[x1,y1,x2,y2]$ or $[z1,r1,z2,r2]$ (real).

Endfile

Terminate execution of the script file. You can add descriptive text in any format after this command.

The following is an example of a script file to compare field values along the axes of four different solutions and to write the results in a file COMP.DAT.

```
NSCAN 200
OUTPUT COMP
INPUT SWITCH01.EOU
SCAN 0.00 -50.00 0.00 50.00
INPUT SWITCH02.EOU
SCAN 0.00 -50.00 0.00 50.00
INPUT SWITCH03.EOU
SCAN 0.00 -50.00 0.00 50.00
INPUT SWITCH04.EOU
SCAN 0.00 -50.00 0.00 50.00
ENDFILE
```

16. Format of the EStat output file

The **EStat** output file `FPrefix.EOU` is in ASCII format. It can be inspected with any text editor. The file has three sections:

- Header with general information on the run
- Node and element information
- Region information

The header section consists of a title line and 9 data lines:

```

--- Run parameters ---
XMin: -1.270003E-01
XMax:  1.270003E-01
KMax:  101
YMin:  0.000000E+00
YMax:  2.540005E-01
LMax:  101
DUnit:  3.937000E+01
NReg:    5
ICylin:  1

```

Lines 2 and 3 list x_{\min} and x_{\max} , the limits along horizontal axis (x or z) of the solution volume. Dimensions are given in meters. The quantity K_{\max} in Line 4 is the number of nodes along the horizontal direction. Lines 5-7 describe the vertical axis (y or r). Line 8 contains the quantity $DUnit$, the conversion factor from dimensions used in **Mesh** to meters. Line 9 gives the number of regions in the solution, while Line 10 specifies the symmetry (0: planar, 1: cylindrical).

The node section consists of 4 title lines and $K_{\max} \times L_{\max}$ data lines, one for each node of the solution space.

```

--- Nodes ---
k      l  RgNo RgUp RgDn      x      y      Phi
=====
  1     1   -2   2    0 -1.270003E-01  0.000000E+00 -1.000000E+03
  2     1   -2   2    0 -1.243827E-01  0.000000E+00 -1.000000E+03
  3     1   -2   2    0 -1.217702E-01  0.000000E+00 -1.000000E+03
  4     1   -2   2    0 -1.191645E-01  0.000000E+00 -1.000000E+03
  5     1   -2   2    0 -1.165665E-01  0.000000E+00 -1.000000E+03
...
 98    101   -1   0    1  1.198864E-01  2.540005E-01  9.449078E+02
 99    101   -1   0    1  1.224252E-01  2.540005E-01  9.645715E+02
100    101   -1   0    1  1.249641E-01  2.540005E-01  9.842325E+02
101    101   -5   0    0  1.270003E-01  2.540005E-01  9.999997E+02

```

Each data line contains the following quantities:

- The indices of the node (K, L)
- The region number of the node ($RgNo$) and region numbers for two associated elements ($RgUp$ and $RgDn$). The upper element lies about the line between nodes (K, L) and ($K+1, L$) and the lower element lies below the line.
- The coordinates of the element in meters (x, y)
- The electrostatic potential at the element in volts (ϕ)

The region section consists of four title lines following by $NReg$ data lines, one for each region. For a dielectric solution, the region section has the following appearance:

```

--- Regions ---
RegNo Fix                                     Epsilon      Space charge      Potential
=====
   1    0    0    0    0    0  1.000000E+00  0.000000E+00  0.000000E+00
   2    0    0    0    0    0  2.700000E+00  0.000000E+00  0.000000E+00

```

The region section for a conductive solution is as follows:

```

--- Regions ---
RegNo Fix                                     Sigma                                     Potential
=====
   1    0    0    0    0    0  1.000000E+00  0.000000E+00  0.000000E+00
   2    0    0    0    0    0  2.700000E+00  0.000000E+00  0.000000E+00

```

A one in the second column designates a fixed-potential region (electrode).

17. Treatment of anisotropic materials in VEStat

VEStat includes effects of anisotropic materials in calculations of field energy, resistive power dissipation, induced charge and current density. This section summarizes the formulas used in the calculations. The discussion covers planar solutions. The relationships also apply to cylindrical simulations with the substitution $x \Rightarrow z$, $y \Rightarrow r$.

Anisotropic conductive materials have different values of electrical conductivity along two normal axis. The value is σ_1 along Axis 1 and σ_2 along Axis 2. The quantity θ is the angle between the x (or z) axis and Axis 1. The angle between Axis 2 and the x axis is $\theta + 90^\circ$. The local components of current density are related to the electric field by

$$\begin{bmatrix} j_x \\ j_y \end{bmatrix} = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} \\ \sigma_{yx} & \sigma_{yy} \end{bmatrix} \begin{bmatrix} E_x \\ E_y \end{bmatrix}.$$

where

$$\sigma_{xx} = \sigma_1 \cos^2\theta + \sigma_2 \sin^2\theta ,$$

$$\sigma_{xy} = \sigma_{yx} = (\sigma_1 - \sigma_2) \cos\theta \sin\theta ,$$

$$\sigma_{yy} = \sigma_1 \sin^2\theta + \sigma_2 \cos^2\theta .$$

Given the electric field and current density, the resistive power density is

$$p = \frac{1}{2} \left[j_x E_x + j_y E_y \right] .$$

Anisotropic dielectrics have different values of relative dielectric constant (ϵ_1 and ϵ_2) along two normal axis. We shall write equations in terms of the applied field \mathbf{E}_0 (the total electric field minus contributions from dielectric materials). The applied field is related to the displacement vector by

$$\mathbf{D} = \epsilon_0 \mathbf{E}_0 .$$

The applied field is related to the total electric field by

$$\begin{bmatrix} E_{0x} \\ E_{0y} \end{bmatrix} = \begin{bmatrix} \epsilon_{xx} & \epsilon_{xy} \\ \epsilon_{yx} & \epsilon_{yy} \end{bmatrix} \begin{bmatrix} E_x \\ E_y \end{bmatrix}.$$

where

$$\epsilon_{xx} = \epsilon_1 \cos^2\theta + \epsilon_2 \sin^2\theta ,$$

$$\epsilon_{xy} = \epsilon_{yx} = (\epsilon_1 - \epsilon_2) \cos\theta \sin\theta ,$$

$$\epsilon_{yy} = \epsilon_1 \sin^2\theta + \epsilon_2 \cos^2\theta .$$

The electrostatic field energy density is given by

$$u = \frac{\epsilon_0}{2} [E_{0x} E_x + E_{0y} E_y] .$$

The induced charge density on the surface of a conductor can be expressed in terms of the electric field component normal to the surface. The *total* surface charge density (in coulomb/m²) is given by

$$\rho = \epsilon_0 E_{\perp} .$$

The total charge includes contributions from free charge on the electrode and displaced charge in the adjacent dielectric. The *free* charge density is given in terms of the applied field by

$$\rho = \epsilon_0 E_{0\perp} .$$

VEStat calculates both dielectric and resistive quantities. The user decides which ones are relevant to a particular calculation. The operations in **EStat** and **VEStat** are identical for both types of solutions. Therefore, in a dielectric solution the quantity reported by **VEStat** as current density is equal to the applied field.

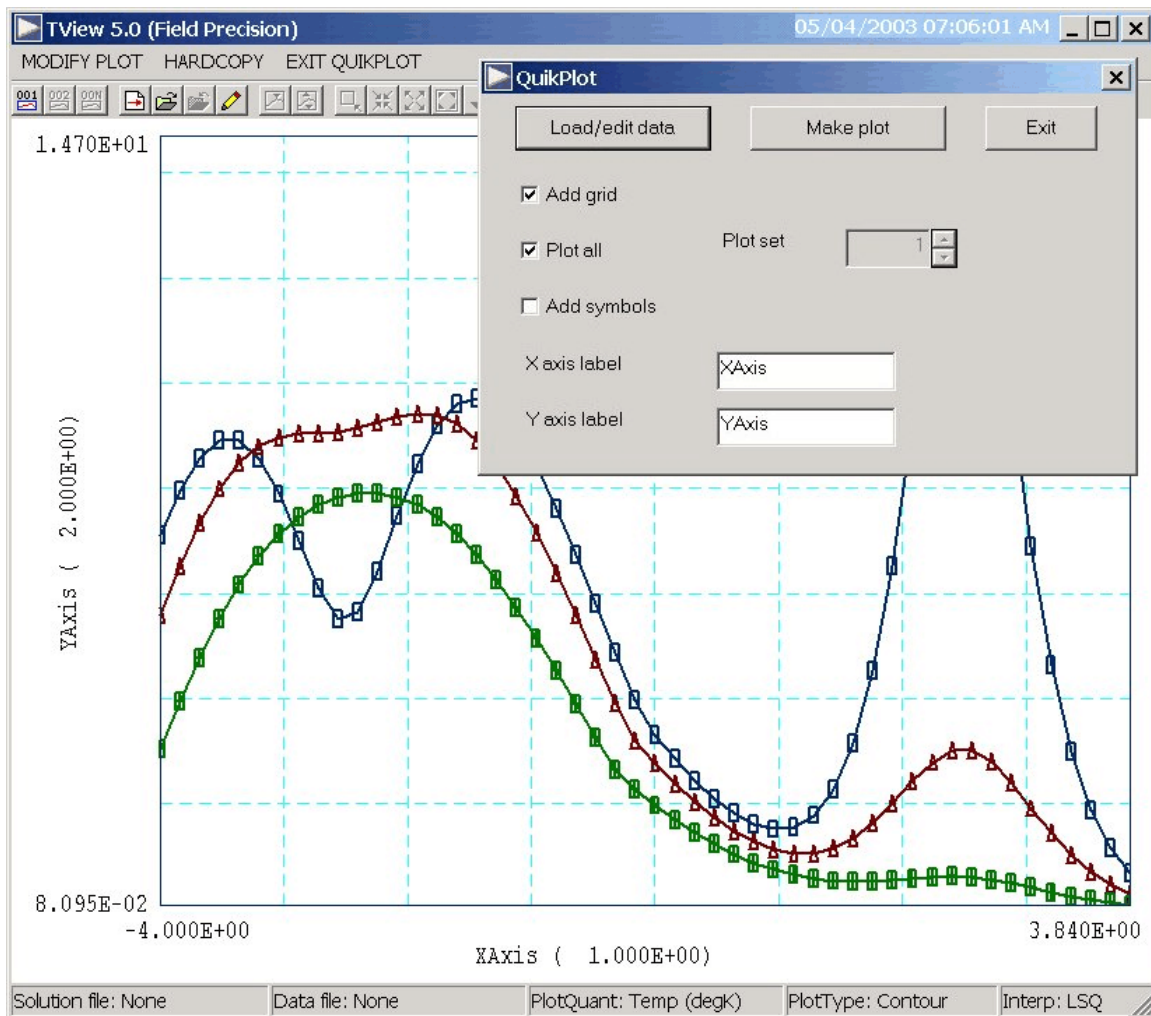


Figure 9. Control dialog superimposed on QuikPlot screen.

18. QuikPlot utility

Postprocessors for **TriComp** and **Amaze** programs contain a utility for making fast, simple plots of numerical data. The feature may be useful, for example, to check a table that defines non-linear thermal conductivity. Click on *QUIK PLOT* in the *TOOLS* menu to bring up a dialog (Fig. 9). Initially, all commands are deactivated except *LOAD/EDIT DATA*. Clicking this command brings up an editor window. You can paste numerical data that you have copied from text files or spreadsheets into the window. You can also use the editor function to make changes or to save the data to a file.

The data should consist of columns of numbers separated by any of the standard delimiters (space, tab, comma, equal sign, parentheses). The first

column contains x -values. Additional columns contain from 1 to 5 sets of y -values. All columns must have the same number of rows. Although there is no limit to the number of rows, the program will pick a maximum of 250 evenly-spaced values to make the plots. Click *SAVE DATA* when you are through.

When you return to the main dialog all commands should be active. Click *MAKE PLOT* to see a plot of the data. The program adjusts the ranges and grid intervals automatically. When the plot is active you can export it in various formats or copy it to the clipboard. Click *MODIFY PLOT* to return to the main dialog to make changes. Here you can change the display of grids and plot symbols or add text labels to the x and y axes. Note that the x - y labels also show the grid intervals. By default, **QuikPlot** shows all data sets (columns of y -values) with different line colors and plot symbols.

You can plot a particular set by unclicking the *PLOT ALL* box and specifying the set number (1-5). The plot range is automatically adjusted to the set. In the main dialog, click on *EXIT QUIK PLOT* to return to the normal functions of **VEStat**.