PAC

[Poisson AC]

AC/RF Electric Field Analysis

Field Precision

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

PAC 5.0 Manual - Contents

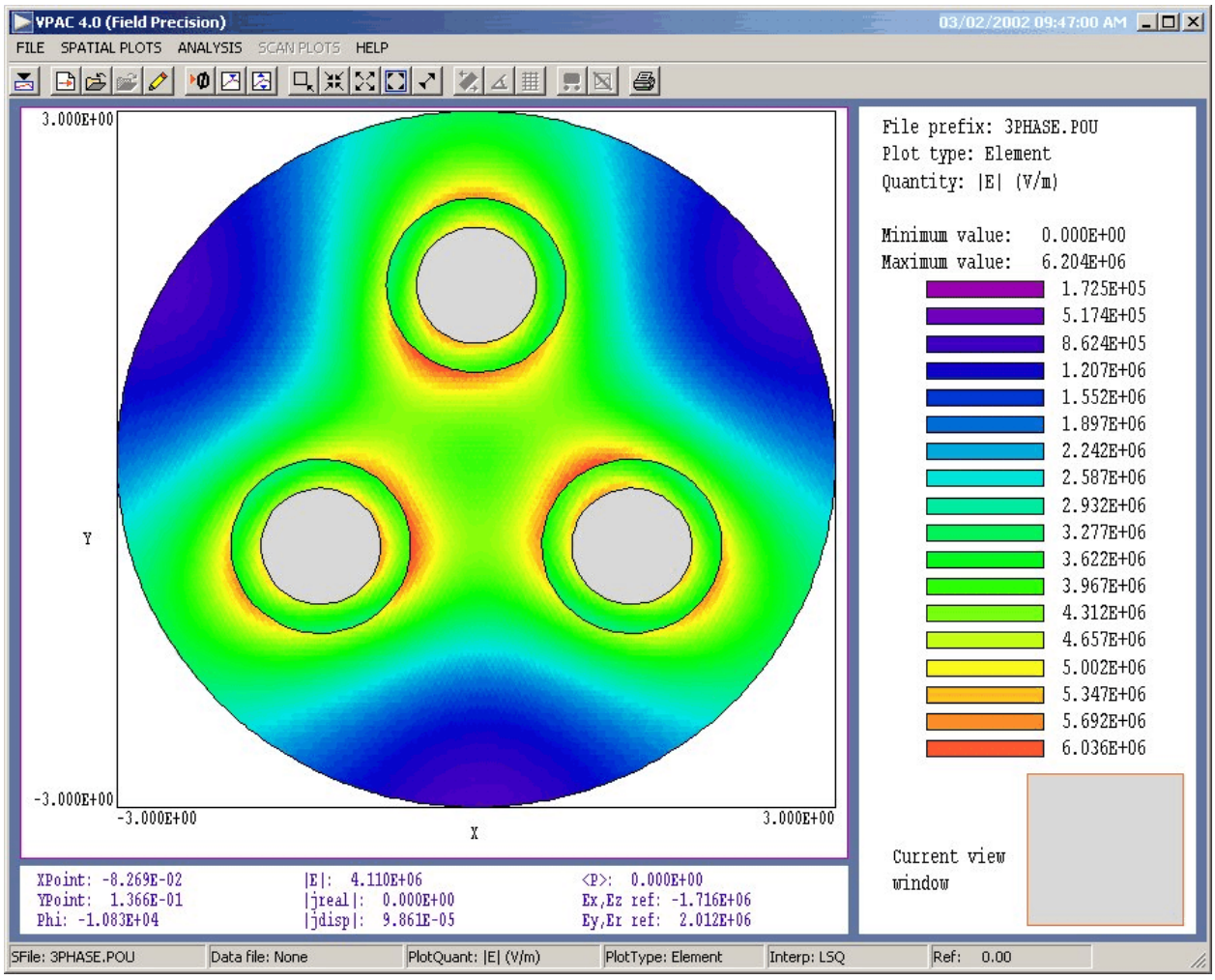

Figure 1.**VPAC** post-processor work environment

1. Program function

PAC is a versatile finite-element tool to calculate AC or RF electric fields generated by electrodes in conductive media. The program handles arbitrary cylindrical or rectangular geometries. You can define up to 127 electrical regions to represent electrodes or imperfect dielectric materials. The amplitude and phase of the harmonic potential on electrodes can be set individually. Dielectrics are characterized by values of the conductivity and relative dielectric constant. **VPAC**, an interactive graphics analysis program, provides full information on the amplitude and phase of electric field and current density throughout the solution regions. The post-processor automatically calculates power density and total current organized by regions. The program makes screen and hardcopy plots of potential, field amplitude current density and resistive energy deposition.

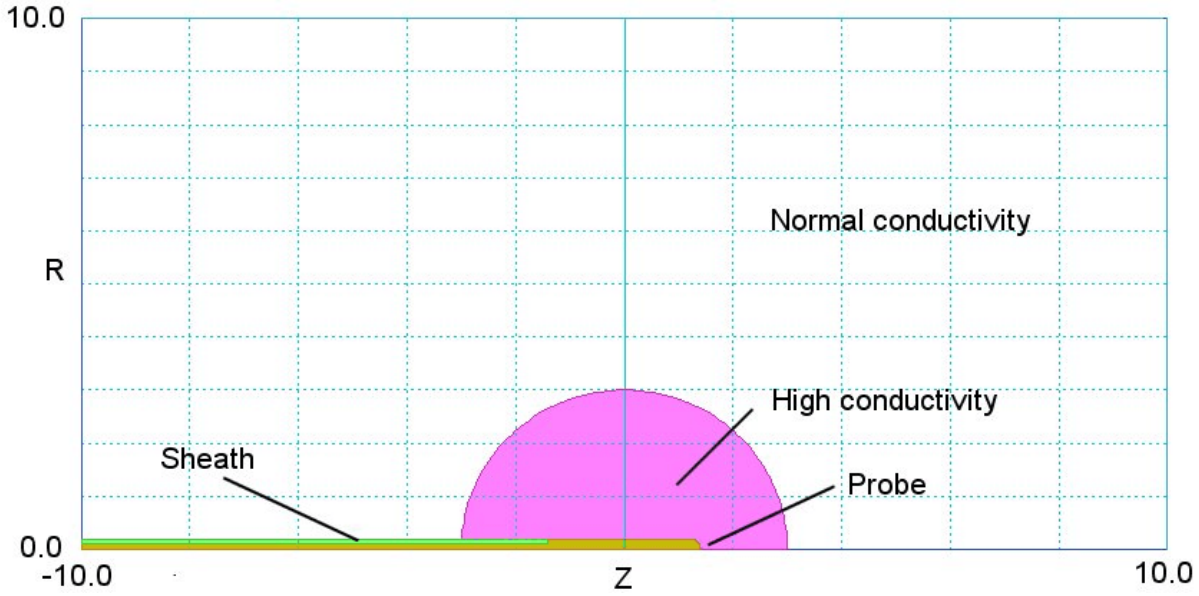

Figure 2. Geometry of the walkthrough example, *COND_LAYER*. Dimensions in cm.

2. Walkthrough example

To test the installation and to get acquainted with **PAC** procedures we shall follow an example that demonstrates many of the program features. In preparation, copy the files `COND_LAYER.DXF`, `COND_LAYER.MIN`, `COND_LAYER.PIN` and `COND_LAYER.SCR` to a working directory such as `\TRICOMP\BUFFER`. We want to calculate the electric field distribution and resistive power dissipation for a narrow probe inserted into a conductive medium. Figure 2 shows dimensions and region divisions. The probe has a diameter of 2 mm and an active length of 2.8 cm. The main difficulty in heating a medium with a narrow probe is that concentration of the electric field gives uneven power deposition. To alleviate the problem the probe is surrounded by a region of enhanced conductivity created by the injection of saline solution. The saline region has a radius of 1.5 cm. In principle, there is a grounded boundary at infinity. For a practical finite-element solution, we set the boundary at a large distance from the probe (10.0 cm in the axial and radial directions).

There are three tasks in a **PAC** solution that are handled by three programs:

- Create a conformal triangular mesh to represent the geometry (`mesh.exe`).

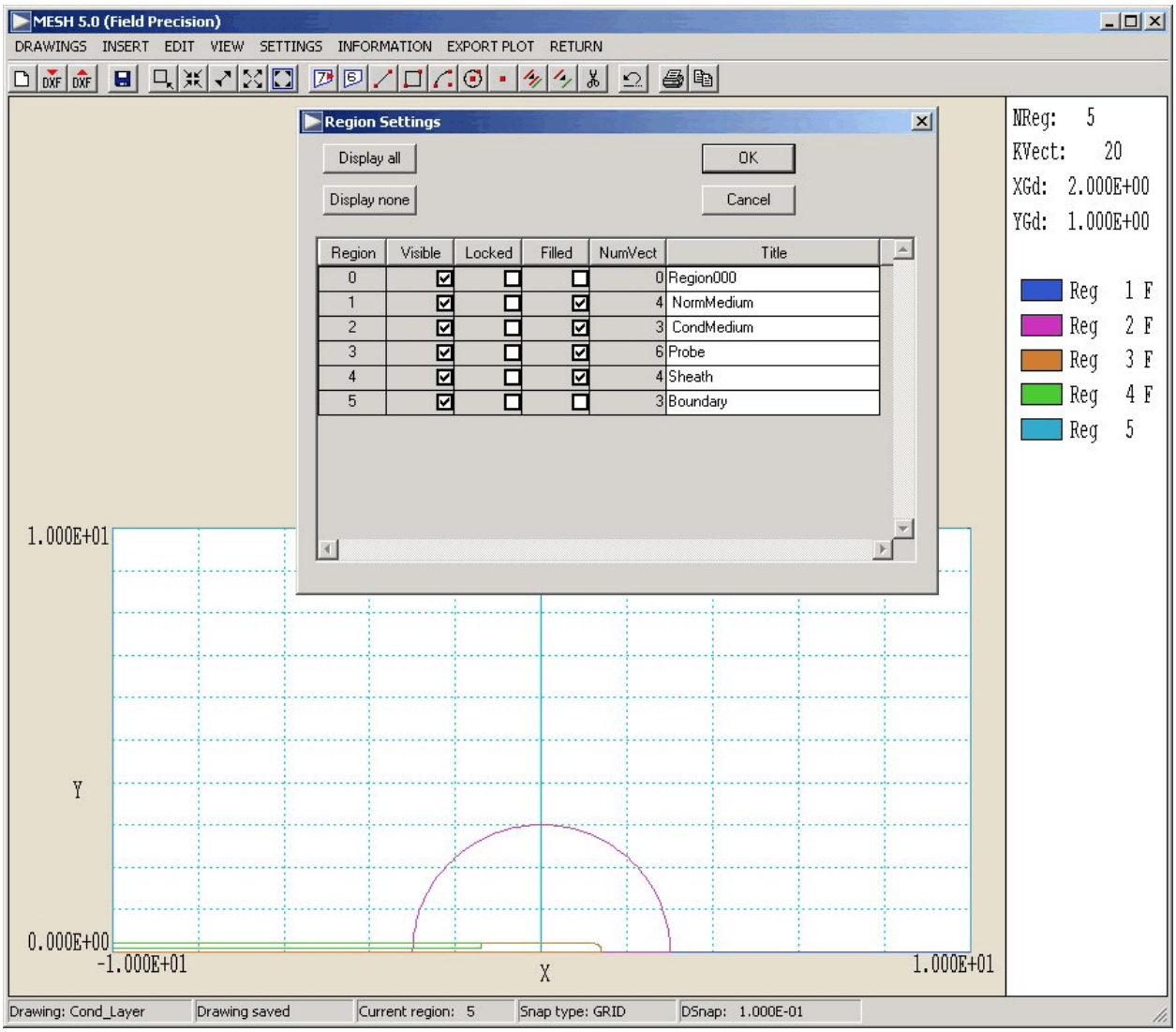

Figure 3. Working environment of the **Mesh** drawing editor.

- Solve for the complex potential on nodes of the mesh for defined physical properties of the elements (`pac.exe`).

- Create plots and perform analyses of the solution (`vpac.exe`).

Start the **TriComp** program launcher (`tc.exe`) and ensure that the data directory corresponds to the working directory. Run **Mesh** and click on *DRAWING* to go to the drawing editor. In the *DRAWINGS* menu click on *IMPORT DXF* and choose the file `COND_LAYER.DXF`. You should see a display similar to Figure 3 showing the boundaries of physical regions.

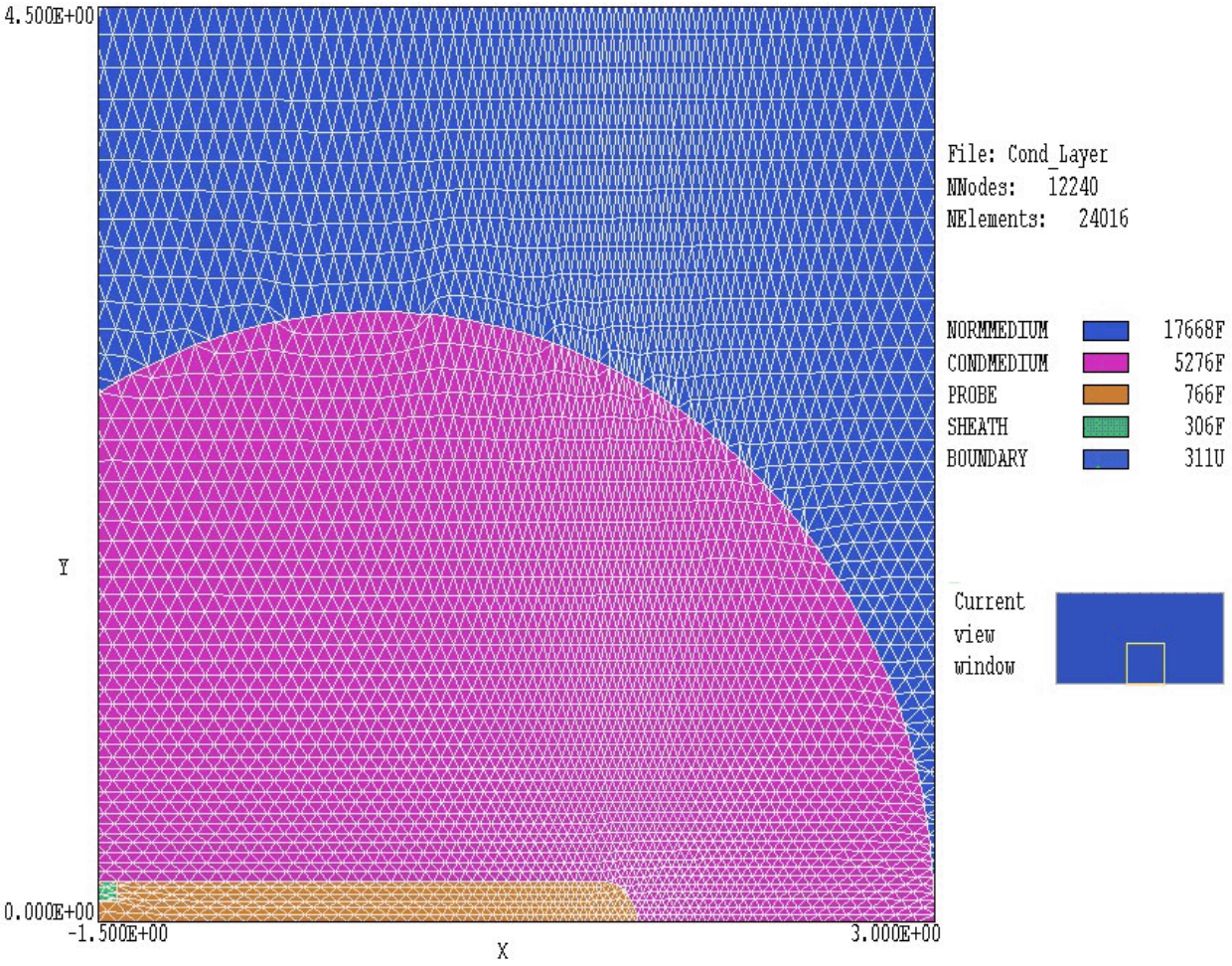

Figure 4. Detail of the mesh near the probe.

The drawing editor is a graphical environment where you can import and modify existing DXF files (AutoCAD Drawing Exchange File format). You can also create drawings using advanced built-in CAD capabilities. The **Mesh** manual gives a detailed description of features. The function of the drawing editor is to generate geometric scripts in the standard **Mesh** format. You can also create and modify the scripts directly with an editor. In this case, we are goingto use a script that has already been prepared:

Exit the drawing editor by clicking *RETURN* in the menu at the top of the screen. Click on *EDIT FILE* in the *FILE* menu and choose `COND_LAYER.MIN`. The editor displays the contents shown in Table 1. The file was generated by **Mesh** using vectors in the DXF file. The commands of the *GLOBAL* section set the initial sizes of elements in the

Table 1. Contents of COND_LAYER.MIN

```
*  ------------------------------------------------------
GLOBAL
  XMESH
     -10.00  -4.00  0.25
      -4.00   1.00  0.10
       1.00   1.80  0.025
       1.80   4.00  0.10
       4.00  10.00  0.25
  END
  YMESH
       0.00   0.50  0.025
       0.50   4.00  0.10
       4.00  10.00  0.25
  END
  PRESMOOTH 5.0
END
*  ------------------------------------------------------
REGION  FILL  NormMedium
     L   -1.00E+01  0.00E+00  1.00E+01  0.00E+00
     L    1.00E+01  0.00E+00  1.00E+01  1.00E+01
     L    1.00E+01  1.00E+01 -1.00E+01  1.00E+01
     L   -1.00E+01  1.00E+01 -1.00E+01  0.00E+00
END
*  ------------------------------------------------------
REGION  FILL  CondMedium
     L   -3.00E+00  0.00E+00  3.00E+00  0.00E+00
     A    3.00E+00  0.00E+00  0.00E+00  3.00E+00  0.00E+00  0.00E+00
     A    0.00E+00  3.00E+00 -3.00E+00  0.00E+00  0.00E+00  0.00E+00
END
*  ------------------------------------------------------
REGION  FILL Probe
     L   -1.00E+01  0.00E+00  1.40E+00  0.00E+00
     A    1.40E+00  0.00E+00  1.20E+00  2.00E-01  1.20E+00  0.00E+00
     L    1.20E+00  2.00E-01 -1.40E+00  2.00E-01
     L   -1.40E+00  2.00E-01 -1.40E+00  1.00E-01
     L   -1.40E+00  1.00E-01 -1.00E+01  1.00E-01
     L   -1.00E+01  1.00E-01 -1.00E+01  0.00E+00
END
*  ------------------------------------------------------
REGION  FILL Sheath
     L   -1.00E+01  1.00E-01 -1.40E+00  1.00E-01
     L   -1.40E+00  1.00E-01 -1.40E+00  2.00E-01
     L   -1.40E+00  2.00E-01 -1.00E+01  2.00E-01
     L   -1.00E+01  2.00E-01 -1.00E+01  1.00E-01
END
*  ------------------------------------------------------
REGION Boundary
     L   -1.00E+01  1.00E+01  9.90E+00  1.00E+01
     L    1.00E+01  1.00E+01  1.00E+01  0.00E+00
END
*  ------------------------------------------------------
ENDFILE
```

foundation mesh. The foundation elements are then flexed to conform to the boundaries defined in the following *REGION* sections. Note that the file has been edited to modify the default element sizes set by the drawing editor in the horizontal (*XMESH*) and vertical (*YMESH*) directions. The values chosen define fine elements near the probe for an accurate calculation of current. Coarse elements are employed in the area between the conductive region and the boundary to reduce the solution time. The *REGION* sections contain line and arc vectors that outline the regions of Fig. 2. If a region has the *FILL* keyword, the current region number is assigned to all internal nodes and elements. Note that the order in which regions appear in the script is important. Regions over-write shared areas of previously-defined regions. In this way, the conductive medium replaces a part of the normal medium and the probe and sheath regions replace part of the conductive medium. Fixed potential regions must appear last.

Exit the editor by clicking *EXIT* in the *FILE* menu. In the main *FILE* menu, choose the command *LOAD SCRIPT (MIN)* and pick `COND_LAYER.MIN`. Click on *PROCESS*. In response, the program analyzes the script commands and creates the mesh. The screen display contains information that may be useful when problems occur. The information is also recorded in a listing file, `COND_LAYER.MLS`. Press any key to proceed.

Click on *PLOT-REPAIR* to see a picture of the mesh. In the *PLOT TYPE* menu choose *REGIONS*. Then use the *VIEW/ZOOM WINDOW* tool to narrow the view near the probe tip. Figure 4 shows the resulting mesh. Note the fine detail near the probe tip. The information window shows the current view and color-coding information for regions. The program lists the number of elements in filled regions and nodes in line regions. Click the *RETURN* command to return to the main menu. Then click *FILE/SAVE MESH (MOU)*. **Mesh** creates the file *COND_LAYER.MOU* that contains node coordinates and the region numbers assigned to the nodes and nearby elements. The output files of **TriComp** programs are in text format. You can inspect the mesh file with an editor or transfer information to your own programs.

You can close **Mesh** or leave it open if you want to try other examples later. Activate `tc.exe` and launch `pac.exe` (scroll downwards to find the program). Click on *FILE/EDIT INPUT FILES* and choose `COND_LAYER.PIN` (**P**ac **IN**put). This required text file (shown in Table 2) contains parameters to control the program operation and information on material properties associated with regions. The commands that begin with the keyword *SET* specify several conditions. For example coordinates

Table 2. Contents of COND_LAYER.PIN

```
* File: COND_LAYER.PIN
*  Region    Region
*  number    name
*  ====================
*       1    NORMMEDIUM
*       2    CONDMEDIUM
*       3    PROBE
*       4    SHEATH
*       5    BOUNDARY

SET DUnit = 100.0
SET Freq = 500.0E3
SET ResTarget = 5.0E-8
SET MaxCycle = 2500
SET Omega = 1.90
SET Geometry = Cylin

* Region 1: Normal medium
REGION(1) Epsi = 2000.0
REGION(1) Sigma 0.12

* Region 2: Conductive medium
REGION(2) Epsi = 2000.0
REGION(2) Sigma = 1.2

* Region 3: Probe
REGION(3) Potential = 30.0
REGION(3) Phase = 0.0

* Region 4: Sheath
REGION(4) Epsi = 2.8
REGION(4) Sigma = 0.0

* Region 5: Boundary
REGION(5) Potential = 0.0
REGION(5) Phase = 0.0

ENDFILE
```

units from **Mesh** will be converted from centimeters, the RF frequency is 500.0 kHz, and the solution has cylindrical symmetry. The conductive regions (1 and 2) have conductivities of 0.12 and 1.2 S/m and relative dielectric constant ϵ_r = 2000.0. The probe voltage (region 3) has an amplitude of 30 V. Close the editor to return to the **PAC** main menu. Click on *RUN/START RUN*. The program creates a set of coupled linear equations (one for each node) and solves them using an iterative matrix-inversion technique. The entire process takes about 6 seconds. **PAC** creates the output file `COND_LAYER.POU` that contains region numbers of nodes and elements, node coordinates, and real and imaginary values of the complex potential at node positions.

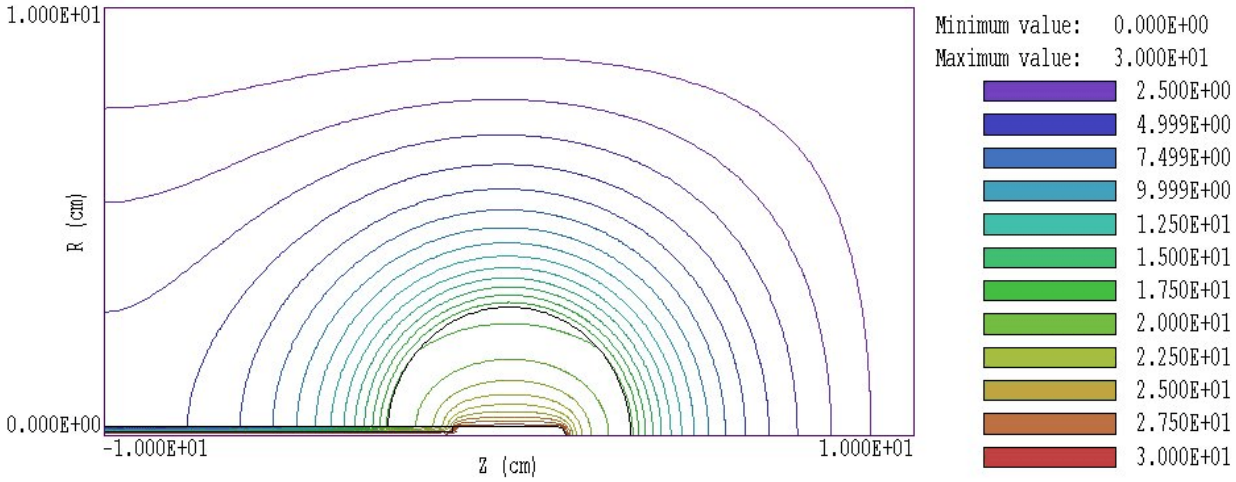

Figure 5. Lines of constant potential at phase = 0.0°.

To conclude, we shall analyze the solution with the **VPAC** post-processor. Launch `vpac.exe` from `tc.exe`. Click on *FILE/LOAD SOLUTION FILE* and choose `COND_LAYER.POU`. The program loads the solution file and creates the default plot shown in Fig. 5. The plot shows a snapshot of lines of constant ϕ at a phase of 0.0°. We shall construct a display of resistive power density in the region near the boundary between the two conductive regions. Click on *SPATIAL PLOTS/PLOT TYTPE* and choose the *ELEMENT* type. The plot type shows elements color-coded by values of a quantity. Click on *SPATIAL PLOTS/PLOT QUANTITY* and choose time-average power density <*P*>. Then use the *SPATIAL PLOTS/ZOOM WINDOW* tool to narrow the view near the probe. Most of the plot area has a uniform violet color because the power density has a very high value at the probe tip. We need to over-ride automatic selection of plot limits to show variations at larger radii. Click on *SPATIAL PLOTS/PLOT LIMITS*. In the dialog, uncheck the *AUTOSCALE* box. Enter the number 1.0E5 for the *MAXIMUM VALUE* and click *OK*. You should see the plot of Fig. 6 which displays the discontinuity in power density at the boundary. Elements where the quantity is out of range are omitted.

To display a plot of the variation of electric field amplitude with radius at the probe midplane, click *ANALYSIS/LINE SCAN*. Move the mouse cursor into the plot area. Note how the cursor symbol changes from an arrow to a cross-hair pattern to show that selection is active. The mouse coordinates are displayed at the bottom of the screen. The values change discontinuously in the snap mode. Move the cursor to a position where the coordinates are (0.00, 0.00). Click the left button and move the mouse

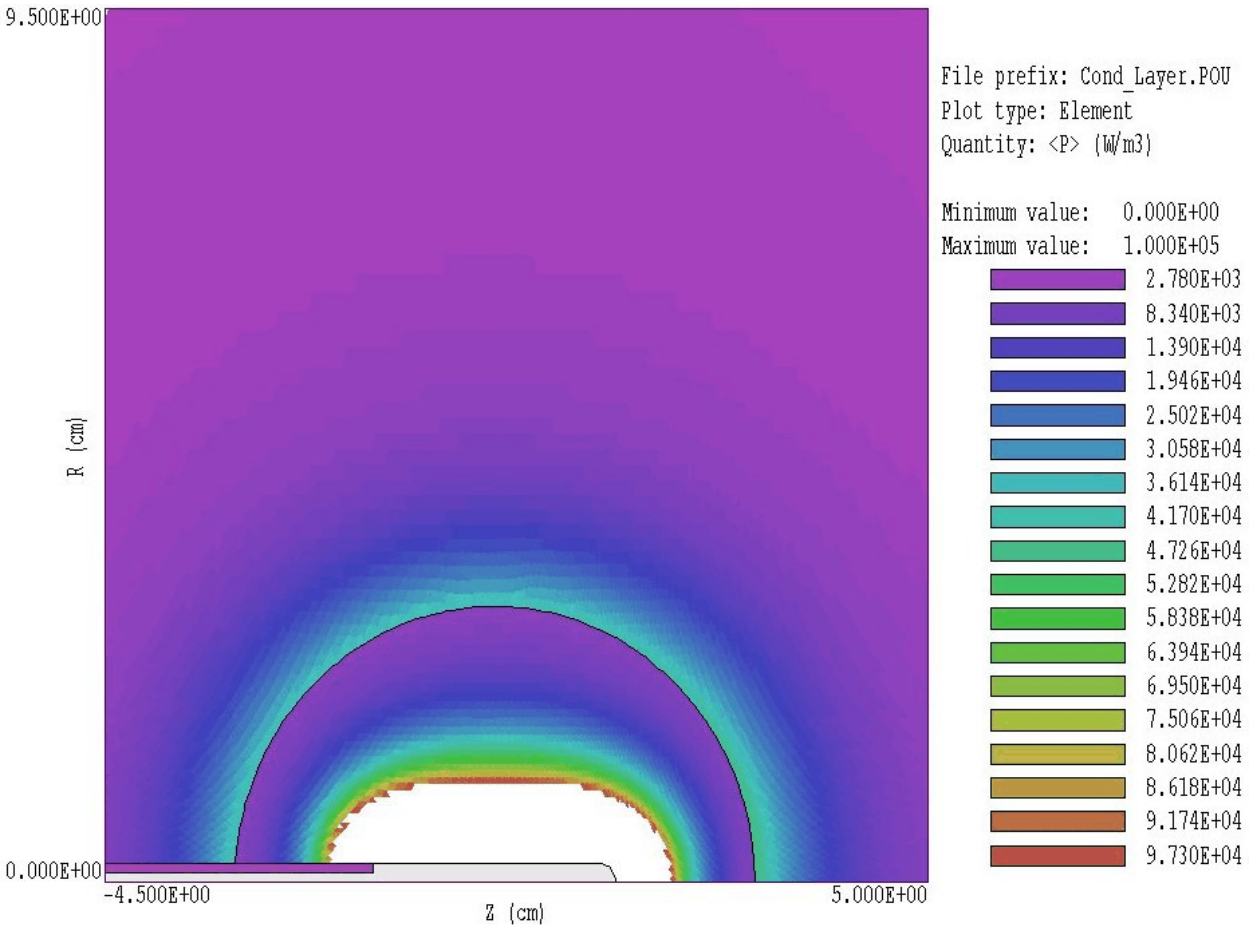

Figure 6. Element plot of <*P*> with adjusted plot limits.

radially outward to position (0.00, 4.50) and click the left button again. The program displays a plot of |**E**| along the line (Fig. 6). If a data file is open, **VPAC** also records an extended set of values in text format that you can transfer to other programs. The scan plot display also features a digital oscilloscope mode to measure points on the plot. Note that **VPAC** has extensive capabilities beyond those that we have covered in this introduction.

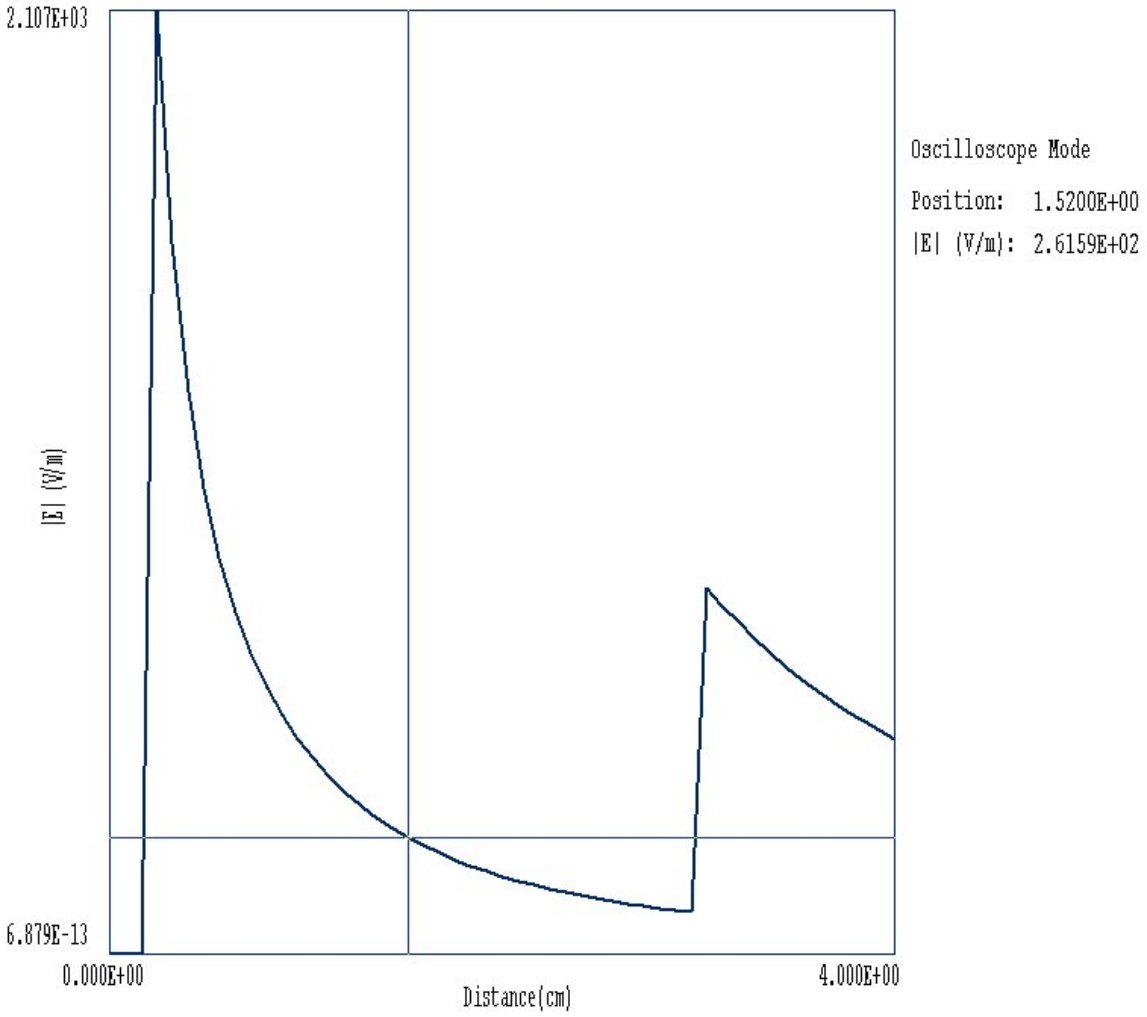

Figure 7. Scan plot of |**E**| with digital oscilloscope features active.

3. Electric fields in conductive media

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

$$\boldsymbol{E} = -\nabla\Phi. \tag{1}$$

Equation (1) is equivalent to the following condition on static fields:

$$\nabla\times\boldsymbol{E} = \mathbf{0}. \tag{2}$$

For a system of ideal dielectrics with zero conductivity, we can combine Eq. 1 with the divergence equation,

$$\nabla\cdot(\epsilon\bar{E}) = \rho/\epsilon_o \ , \tag{3}$$

to yield the Poisson equation,

$$\nabla\cdot\epsilon\nabla\Phi = -\rho/\epsilon_o. \tag{4}$$

The quantity ρ in Eq.(4) represents charge density excluding dielectric charge.

We can derive a similar equation for static fields in a conducting medium where there are no displacement currents. Because there is no time variation of the charge in the media, the divergence of current density **j** equals zero,

$$\nabla\cdot\boldsymbol{j} = 0. \tag{5}$$

The current density is related to the electric field by

$$\boldsymbol{j} = \sigma \boldsymbol{E} \ , \tag{6}$$

where σ is the conductivity in siemens/m. Combining Eqs. (1), (5) and (6) shows that the potential is determined by the Laplace equation

$$\nabla \cdot \left[\sigma \ \nabla\Phi\right] = 0. \tag{7}$$

Note that Eq. (7) is has the same form as Eq. (4) (in the absence of space charge) with the replacement of ϵ bt σ.

We can consolidate Eqs. (4) and (7) to describe RF electric fields in imperfect dielectrics at low frequency *f*. The theory applies when

$$L \ll \frac{1}{2\pi f \sqrt{\epsilon\mu}} \ . \tag{8}$$

In Eq. (8) L is the scale length of the system. The quantities ϵ and μ are values of the permittivity and magnetic permeability characteristic of the medium. The quantity $1/\sqrt{\epsilon\mu}$ equals the speed of light in the medium. Therefore, Eq. (8) implies that the system length L is small compared to the distance that light travels in one RF period. As an example, taking $\epsilon = 10\epsilon_o$, $\mu = \mu_o$ and and $L = 0.1$ m, the quasi-static approximation is valid for RF frequencies in the range $f \ll 151$ MHz.

We shall use complex-number notation to describe harmonically varying quantities. In **PAC** all electrode voltages (and hence all harmonic quantities) vary at the same angular frequency,

$$\omega = 2\pi f. \tag{9}$$

We use the following convention for harmonic functions:

$$A(x,t) = A(x) \cos[\omega t + \phi(x)]. \tag{10}$$

The phase angle ϕ is zero when the function varies as $\cos(\omega t)$. A quantity that varies as $\sin(\omega t)$ has a phase $\phi = -90°$. In Eq. (11) both the amplitude and phase of the quantity may vary with position. In complex notation, Eq. (11) has the form

$$A(x,t) = Re[\ \overline{A}(x) \exp(j\omega t)\ ] \ . \tag{11}$$

The complex number $\overline{A}(x)$ is called a *phasor* — it contains information on the spatial variations of the amplitude and phase. The parameters of the associated harmonic function can be determined from the phasor by applying the following equations:

$$A(x) = \sqrt{\bar{A}(x)\ \bar{A}(x)^*}\ , \tag{12}$$

$$\phi(x) = \tan^{-1}\left(\frac{Im[\bar{A}(x)]}{Re[\bar{A}(x)]}\right) \tag{13}$$

In Eq. (12) the quantity $\bar{A}(x)^*$ is the complex conjugate of $\bar{A}(x)$.

Equation (2) holds approximately in the non-radiative limit of Eq. (8). The condition implies that the contribution of time-varying magnetic fields to the electric field is negligible. In this case the electric field can be written as the spatial gradient of a harmonically-varying scalar potential function [Eq. (1)]. The phaser $\bar{\Phi}(x,y,z)$ is a complex number with information on the amplitude and phase of the quasi-static potential. Suppose that the phasor $\bar{j}_r$ represents the resistive current density in a conductive media and $\bar{\rho}_r$ is the space-charge density resulting from this current. Conservation of charge implies that

$$\frac{\partial\bar{\rho}_r}{\partial t} = -\nabla\cdot\bar{j}_r = j\omega\bar{\rho}_r. \tag{13}$$

The divergence equation takes the form

$$\nabla\cdot\epsilon\bar{E} = \bar{\rho}_r = j\frac{\nabla\cdot\bar{j}_r}{\omega} = \nabla\cdot\left[\frac{j\sigma\bar{E}}{\omega}\right]. \tag{14}$$

The expression on the right-hand side proceeds from the relation $\bar{j}_r = \sigma\bar{E}$. Equation (14) can be written as

$$\nabla\cdot\left(\epsilon - \frac{j\sigma}{\omega}\right)\bar{E} = 0, \tag{15}$$

or

$$\nabla\cdot\left[\left(\epsilon - \frac{j\sigma}{\omega}\right)\nabla\overline{\Phi}\right] = 0. \qquad (16)$$

Equation (16) is identical to Eq. (7) (in the absence of space charge) if we interpret the dielectric constant as a complex number,

$$\epsilon \rightarrow \left(\epsilon - \frac{j\sigma}{\omega}\right). \qquad (17)$$

Equation (17) can be solved with the same solution methods used in **EStat** for static fields by replacing the potential and dielectric constant with appropriate complex numbers. This approach is the basis of **PAC.**

4. Organizing runs

The **PAC** package consists of a program that computes the physical solution (`PAC.EXE`) and a dedicated post-processor for analyses (`VPAC.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, **VPAC** 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.

PAC 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 `RUNNAME.MOU`, where `RUNNAME` is a valid file prefix (1 to 20 characters). The command script must have a name of the form `RUNNAME.PIN` where `RUNNAME` is the same prefix as that of the mesh file. **PAC** issues an error message if both input files are not available in the working directory. To organize

data, the resulting output file has the name RUNNAME.POU.

A **PAC** run consists of several steps that involve three different programs: mesh generation, field solution, and analysis. The programs communicate through data files. Sometimes, you may calculate several solutions from the same mesh by changing the characteristics of materials. Each run includes the following steps.

- Prepare a **Mesh** input script with a name of the form RUNNAME.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 RUNNAME.MOU.

- Prepare a command script (RUNNAME.PIN) using the *Create command file* menu entry in **PAC** or your own text editor. The allowed file commands are described in this manual and the manual for the specific solution program.

- Run **PAC** to create the output file RUNNAME.POU. It is in text format and contains information on the mesh geometry, the physical properties of regions and values of computed node quantities.

- Analyze the solution using **VPAC**. You can also transfer the information in RUNNAME.POU to your own analysis programs.

5. Structure of the command file

The input script for all **TriComp** solution programs is a text file 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, Forward slash,
Colon, Tab, Equal sign
```

Any number of delimiters may appear on 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.

```
1.250
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 **PAC**:

```
SET DUnit 100.0
SET ResTarget 5.0E-8
SET MaxCycle 200
SET Freq 60.0
SET Geometry CYLIN
*
* Region 1
REGION 1 Epsi 1.00
REGION 1 Sigma 1.399E-3
*
* Region 2
REGION 2 Potential 50.0
REGION 2 Phase 0.0
*
* Region 3
REGION 3 Potential 0.0
REGION 3 Phase 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. The available `SET` commands depend on the solution program.

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

```
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 VecPot 0.0
```

sets nodes with region number 2 as fixed vector-potential points with the value $A_z = 0.0$ or $rA_\theta = 0.0$. In some instances multiple `REGION` commands may refer to the same mesh volume and set different properties.

6. Commands for program control

PAC 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 that all begin with the keyword *SET*. Each command is written as it might appear in the `RunName.PIN` file.

SET FREQ 3.4E5

Sets the RF frequency of all quantities in the solution volume. Enter the value in Hz.

SET OMEGA 1.85

Controls the *over-relaxation factor* for the solution, a number in the range 1.0 to 2.0. If no values is supplied, **PAC** automatically sets *Omega* using the Chebyshev prescription. Generally, high values give faster convergence. Lower the over-relaxation factor if the solution does not converge.

SET MAXCYCLE 200

The maximum number of relaxation cycles. The default value is 2500.

SET RESTARG 5.0E-6

PAC keeps track of the relative error in the magnitude of the complex potential during the relaxation. The program stops when the error falls below *ResTarg*. Values of *ResTarg* less than about 1.0×10^{-6} give good accuracy. The default value is 2.0×10^{-7}.

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 in **PAC**. The quantity *DUnit* is the number of **Mesh** units per m. For example, if the Mesh dimensions are entered in microns, set $DUnit = 1.0 \times 10^6$. The default value is 1.00.

SET GEOMETRY CYLIN

PAC solves problems in rectangular systems (variations in x and y with infinite extent in z) or in cylindrical geometries (variations in r and z with azimuthal symmetry). The command options are *Rect* and *Cylin*. For cylindrical problems, the program takes the z axis along the x direction of **Mesh** and the r axis along y. In this case, the program issues an error message if y_{min} is less than 0.0. The **VPac** analysis program automatically adjusts calculations and labels to represent the two geometric options.

7. Commands for material properties

The **PAC** command set for defining the physical properties of regions is simple. There are four commands that divide into two classes.

- Fixed potential volumes or boundaries with a given potential amplitude and phase

- Conductive media that carry both real and displacement currents in response to the harmonic fields.

The definition of a conductive region requires two parameters

Relative dielectric constant ϵ_r (dimensionless)
Conductivity, σ (S/m)

The conductivity of a medium is the inverse of the volume resistivity ρ (given in units of Ω-m). Initially, all conductive regions are set to $\epsilon_r = 1.0$ and $\sigma = 0.0$.

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 script. The file `RunName.MLS` contains a formatted list. *Keyword* is one of the words listed below and *Value* is a real number. The first two commands apply to fixed potential regions.

REGION 4 POTENTIAL 3500.0

The keyword *Potential* designates that the complex potential of nodes in the region is fixed and will not change during relaxation solution. The command sets the amplitude of the complex potential in volts. The default value of the potential amplitude in all fixed regions is 0.0.

REGION 5 PHASE 45.0

The keyword *Phase* refers to a a fixed region potential. The command sets the phase of the complex potential. Enter the value in degrees (positive or negative). The default value of the phase in all fixed regions is 0.0^{o}.

The final two commands apply to conductive regions where the potential is determine by the relaxation solution.

REGION 2 EPSI 5.8

Sets the relative dielectric constant, $\epsilon_r = \epsilon/\epsilon_o$.

REGION 7 SIGMA 150.0

Sets the conductivity, σ. Enter the value in S/m.

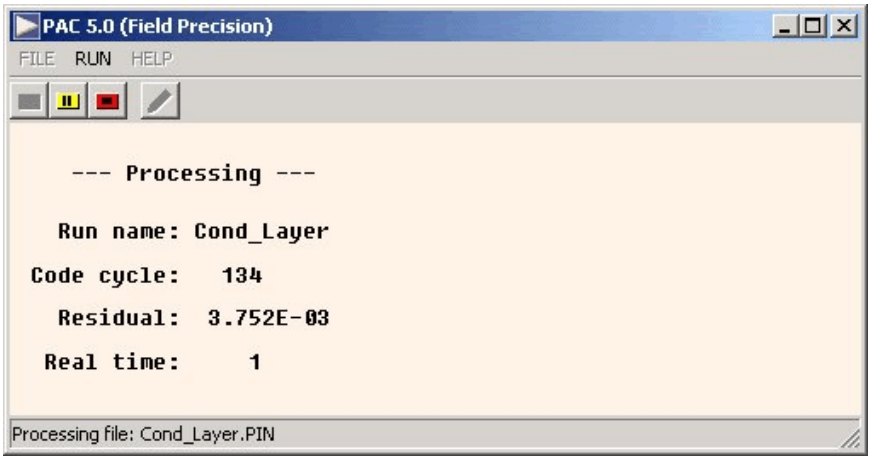

Figure 8. PAC running in a window.

8. Running PAC interactively

The program PAC.EXE can run interactively in a Window (Fig. 8). 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 solution programs and post-processors 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 input 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 **PAC**. 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.PIN. The *Edit listing file* command displays files with names FPREFIX.PLS.

The *Run* menu has four commands.

Start run

Pick an input file (such as FPREFIX.PIN) 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 and any data files specified in the command script are 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.

9. Running PAC from the command prompt

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

```
..\PAC FURNACE <Enter>
```

The program runs silently, writing detailed information in the listing file FURNACE.PLS. If the solution is successful, the program writes the

output file FURNACE.POU 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.PIN,...,SWT08.PIN in the directory \TRICOMP\BUFFER. Furthermore, you have written the following batch file SWRUN.BAT in the data directory using a text editor.

```
@ECHO OFF
ECHO Main switch data run
START ..\MESH.EXE SWT01
START ..\PAC.EXE SWT01
START ..\MESH.EXE SWT02
START ..\PAC.EXE SWT02
 ...
START ..\MESH.EXE SWT08
START ..\PAC.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 programs, we advise running batch scripts from **GCon** rather than from the Command Prompt.

10. VPAC file menu

VPAC (for View PAC) is the interactive graphical analysis program for output files from **PAC**. The program automatically adjusts labels and calculated quantities depending on whether the solution is dielectric or conductive:

Program units

Spatial dimensions: meters
Potential amplitude: volts
Potential phase: degrees
Electric fields: V/m
Conductivity: S/m
Resistive power dissipation: W/m^3
Current density: A/m^2

VPAC 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 `POU`). 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 Section 14. 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. 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.

11. VPAC spatial plot menu

Spatial plots show variations of quantities over the two-dimensional space of the simulation. **VPAC** makes a default plot when a data file is loaded.. The main feature of calculated quantities (compared to a program like **VEStat)** is that **VPAC** deals with complex values to represent the amplitude and phase of harmonic quantities. Several conventions are used to display information on the harmonically-varying quantities:

- A snapshot of the time-dependent fields at a reference phase ϕ. For example, if the electric field in a region varies as $\cos(\omega t + 45°)$, then a setting of $\phi = 45°$ would show the field at its peak value. You can set value of ϕ with the *Reference phase* command.

- The peak amplitude designated as $|\mathbf{E}|$. The value equals the maximum amplitude of the quantity that occurs during the harmonic oscillation.

- Time-averages of quantities, such as the resistive power density.

- The amplitude and phase such as the surface integral of resistive current over a region boundary. The integral is calculated by summing the complex values of $\overline{\boldsymbol{I}_{res}} \cdot \boldsymbol{n}\ dA$ and then computing the amplitude and phase from Eqs. (12) and (13).

Plot type

VPAC 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 a calculated quantity.

Element. Elements of the solution space color-coded according to a computed quantity (such as the electric field amplitude). You can add or remove 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 magnetic permeability 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.

VPAC supports the following plot quantities.

Contour plot quantities

Electrostatic potential Φ (reference phase)
Electric field magnitude, $|\mathbf{E}|$

Electric field lines are perpendicular to lines of constant Φ. Furthermore, the field magnitude is inversely proportional to the distance between lines. The equipotential plot therefore summarizes the electric field distribution at the point in time determined by the reference phase. In a cylindrical simulation the quantity equals $|\mathbf{E}| = |E_r^2 + E_z^2|$. The plot shows the strength of the peak electric field but does not give information on the direction.

Element plot quantities

Electrostatic potential Φ (reference phase)
Electric field magnitude, $|\mathbf{E}|$
Resistive current density amplitude, $|\mathbf{j}_{res}|$
Displacement current density amplitude, $|\mathbf{j}_{disp}|$
Time-averaged resistive power dissipation, *P*

In a rectangular simulation, the amplitude of the resistive current density is given by $|\mathbf{j}_{res}| = \sigma |E_x^2 + E_y^2|$. The displacement current amplitude equals $|\mathbf{j}_{disp}| = j\,\omega\epsilon\, |E_x^2 + E_y^2|$.

Vector plot quantities

Electric field at the reference phase, **E**.

Surface plot quantities

Electrostatic potential Φ at the reference phase
Electric field amplitude, $|\mathbf{E}|$
Resistive current density amplitude, $|\mathbf{j}_{res}|$
Displacement current density amplitude, $|\mathbf{j}_{disp}|$
Time-averaged resistive power dissipation, P
E_x or E_z at the reference phase
E_y or E_r at the reference phase
Magnitude of **E** at the reference phase

The final quantity is equal to $(E_x^2 + E_y^2)^{1/2}$ in rectangular simulations and $(E_z^2 + E_r^2)^{1/2}$ in cylindrical simulations.

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. The program 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 for 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 between mouse and keyboard input. Enter keyboard coordinates in the distance units used in **Mesh**. In other words, if the solution program has *DUnit* = 1.0×10^6, then enter dimensions in microns.

Toggle snap mode

The mouse snap mode is a convenient feature of **VPAC**. When snap mode is active, the mouse returns the coordinate values closest to an integral 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 would 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, **VPAC** must map the current quantity to a rectangular grid, a process that requires 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 causes 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.

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. 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.

17. VPAC 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

VPAC employs a sophisticated interpolation technique that preserves discontinuities at material boundaries. The program gives correct values for electric magnetic field on both sides of a material 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 **VPAC**. 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 15). The program 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 are 1) electrostatic potential Φ at the reference phase, 2) electric field amplitude, $|\mathbf{E}|$, 3) resistive current density amplitude, $|\mathbf{j}_{res}|$, 4) displacement current density amplitude, $|\mathbf{j}_{displ}|$, 5) time-averaged resistive power dissipation, P, 6) E_x or E_z at the reference phase, 7) E_y or E_r at the reference phase and 8) magnitude of **E** at the reference phase.

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

After you pick a triangular element with the mouse (or keyboard), **VPAC** 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.

Matrix file

VPAC can make matrix files of field values to help you create your own analysis routines. Although information is also available in the output file of the solution program, it is difficult to deal with the conformal triangular mesh. The *Matrix file* command uses the interpolation capabilities of the program to create an text 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.

Line integrals

In response to this command, **VPAC** calculates the amplitude and phase of the resistive, displacement and total current (I_{res}, I_{disp} and I_{tot}) through the surface defined by the line. If the line is represented by the vector **L**, then positive current flows in the direction **L** × **z** for rectangular simulations and **L** × θ for cylindrical simulations. You can find the total current emitted by an object by circumnavigating it with lines in the direction of positive rotation.

Volume integrals

In response to the *Volume integrals* command, **VPAC** calculates the resistive power dissipation for the global solution volume and for each region with non-zero volume.

Region properties

In response to this command, **VPAC** identifies element facets on the border of a specified regions and takes integrals along the boundary surface. Facets on the periphery of the active solution volume are not included. The program calculates the amplitude and phase of the resistive, displacement and total current (I_{res}, I_{disp} and I_{tot}) If a facet is represented by the vector **dL**, then positive current flows in the direction **dL** × **z** for rectangular simulations and **L** × θ for cylindrical simulations. As an example, suppose we have current flow between fixed-potential Regions 1 and 3 through a medium (Region 2). Current flow in to Region 3 is the negative of the current leaving Region 1 (*i.e.*, 180° phase shift) while the current for Region 2 is numerically zero because equal currents flow in to and out of the region. It is important to note that the current integral may be inaccurate in cases where the field calculation is ambiguous (*i.e.*, electrodes with sharp edges or curved surfaces represented by only a few facets.

13. VPAC 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 11. This section describes unique commands of the Scan plot menu.

Oscilloscope mode

In the oscilloscope mode, a scan plot assumes many of the 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 **VPAC**. This command closes the scan plot and returns the program to the previous spatial plot.

14. VPAC script file commands

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

VPAC 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 `VPAC.EXE` is in the directory `\TRICOMP.` From `\TRICOMP\BUFFER`, type

```
..\VPAC 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 FURNACE.POU

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 `FURNACE.POU`. 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_{start} y_{start} x_{end} y_{end}] or [z_{start} r_{start} z_{end} r_{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_{start} y_{start} x_{end} y_{end}] or [z_{start} r_{start} z_{end} r_{end}].

VolumeInt

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`.

```
OPEN OUTPUT COMP
OPEN INPUT SWITCH01.POU
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH02.POU
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH03.POU
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH04.POU
SCAN 0.00 -50.00 0.00 50.00
ENDFILE
```

15. Format of the PAC output file

The **PAC** output file `FPrefix.POU` is in text format. You can inspect it 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 10 data lines:

```
--- Run parameters ---
XMin: -1.000000E-01
XMax:  1.000000E-01
KMax:  153
YMin:  0.000000E+00
YMax:  1.000000E-01
LMax:   80
DUnit:  1.000000E+02
NReg:    5
ICylin:  1
Freq:  5.000000E+05
```

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). Finally, Line 11 specifies the RF frequency in Hz.

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

```
--- Vertices ---
  k    l  RgNo RgUp RgDn       x             y           PhiReal       PhiImag
================================================================================
   1    1    3    3    0 -1.000000E-01  0.000000E+00  3.000000E+01  0.000000E+00
   2    1    3    3    0 -9.747066E-02  0.000000E+00  3.000000E+01  0.000000E+00
   3    1    3    3    0 -9.495030E-02  0.000000E+00  3.000000E+01  0.000000E+00
   4    1    3    3    0 -9.244276E-02  0.000000E+00  3.000000E+01  0.000000E+00
   5    1    3    3    0 -8.995134E-02  0.000000E+00  3.000000E+01  0.000000E+00
   6    1    3    3    0 -8.747877E-02  0.000000E+00  3.000000E+01  0.000000E+00
   7    1    3    3    0 -8.502740E-02  0.000000E+00  3.000000E+01  0.000000E+00
   8    1    3    3    0 -8.259930E-02  0.000000E+00  3.000000E+01  0.000000E+00
...
```

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 above 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 real and imaginary parts of the complex potential in volts (ϕ).

The region section consists of four title lines following by *NReg* data lines, one for each region. The region section has the following appearance:

```
--- Regions ---
RegNo Fix                        Conductivity    Epsilon      Potential       Phase

   1    0    0    0    0    0  1.200000E-01  1.770838E-08  0.000000E+00  0.000000E+00
   2    0    0    0    0    0  1.200000E+00  1.770838E-08  0.000000E+00  0.000000E+00
   3    1    0    0    0    0  0.000000E+00  8.854187E-12  3.000000E+01  0.000000E+00
   4    0    0    0    0    0  0.000000E+00  2.479172E-11  0.000000E+00  0.000000E+00
   5    1    0    0    0    0  0.000000E+00  8.854187E-12  0.000000E+00  0.000000E+00
```

An entry of unity in the second column designates a fixed-potential region such as a Dirichlet boundary or electrode. The real number parameters have the following values: electrical conductivity in S/m, absolute dielectric constant (F/m) and the amplitude and phase of the potential for fixed regions (V). Note that all values may not be used for a particular region.

16. 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 magnetic permeability. Click on *QUIKPLOT* in the *TOOLS* menu to bring up a dialog (Fig. 10). 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

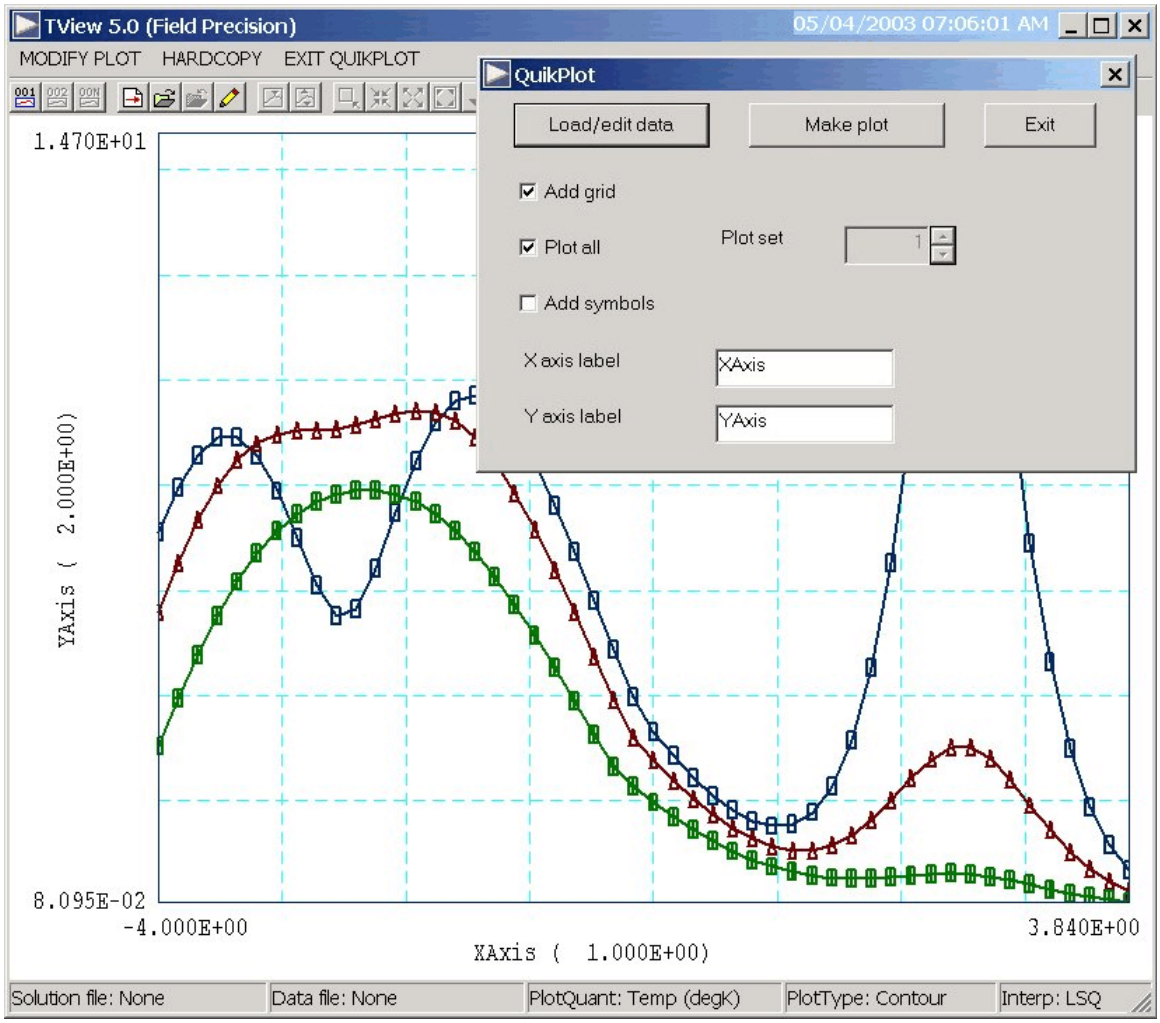

Figure 9. Control dialog superimposed on **QuikPlot** screen.

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 QUIKPLOT* to return to the normal functions of **VPAC**.