TriComp 5.0

WaveSim
Frequency-domain electromagnetics

Field Precision
Copyright 2003

WaveSim 5.0 Manual - Contents

1. Program function

WaveSim calculates RF or microwave electromagnetic fields in the frequency-domain . The program determines two-dimensional solutions by finite-element methods applied on conformal triangular meshes. Planar solutions have field variations in *x* and *y* with no variation along *z*. In cylindrical solutions variations are in *r* and *z* with azimuthal symmetry. The symmetries apply to both the geometry and the field componehnts. **WaveSim** gives fast solutions using sparse matrix inversion techniques. The program covers the full range of electromagnetic calculations.

- Scans to investigate the frequency response of bounded systems

- Automatic searches to locate resonances of closed structures with the option for lossy materials

- Scattering solutions at given frequency for open and lossy systems, including free-space conditions

WaveSim applies an effective method of matched termination boundary layers in scattering calculations. These layers accurately represent open-space boundaries. They are more versatile and efficient than the look-back techniques used in finite-difference calculations. The method of distributed sources is another novel feature of **WaveSim**. It allows you to generate ideal incident plane waves in a bounded anechoic chamber for studies of electromagnetic wave scattering.

The material model of **WaveSim** is flexible. You can specify real and imaginary parts of the dielectric constant and magnetic permeability at the target frequency for up to 127 material regions. Ideal resistive materials can also be defined for frequency scans and absorbing layers in free-space solutions. An interactive graphical post-processor gives plots and scans of primary and secondary field components with automatic labeling of the mode options. The program calculates volume and surface integrals to determine material and wall losses in microwave devices.

This This manual describes features of the **WaveSim** and **WSV** programs. The following additional resources are supplied with the package:

- An library of examples illustrating **WaveSim** capabilities The file `example.txt` gives a brief description of the examples.

- The files `tutorial1.pdf` and `tutorial2.pdf` contain a detailed review of electromagnetic theory and the numerical methods used in **WaveSim**.
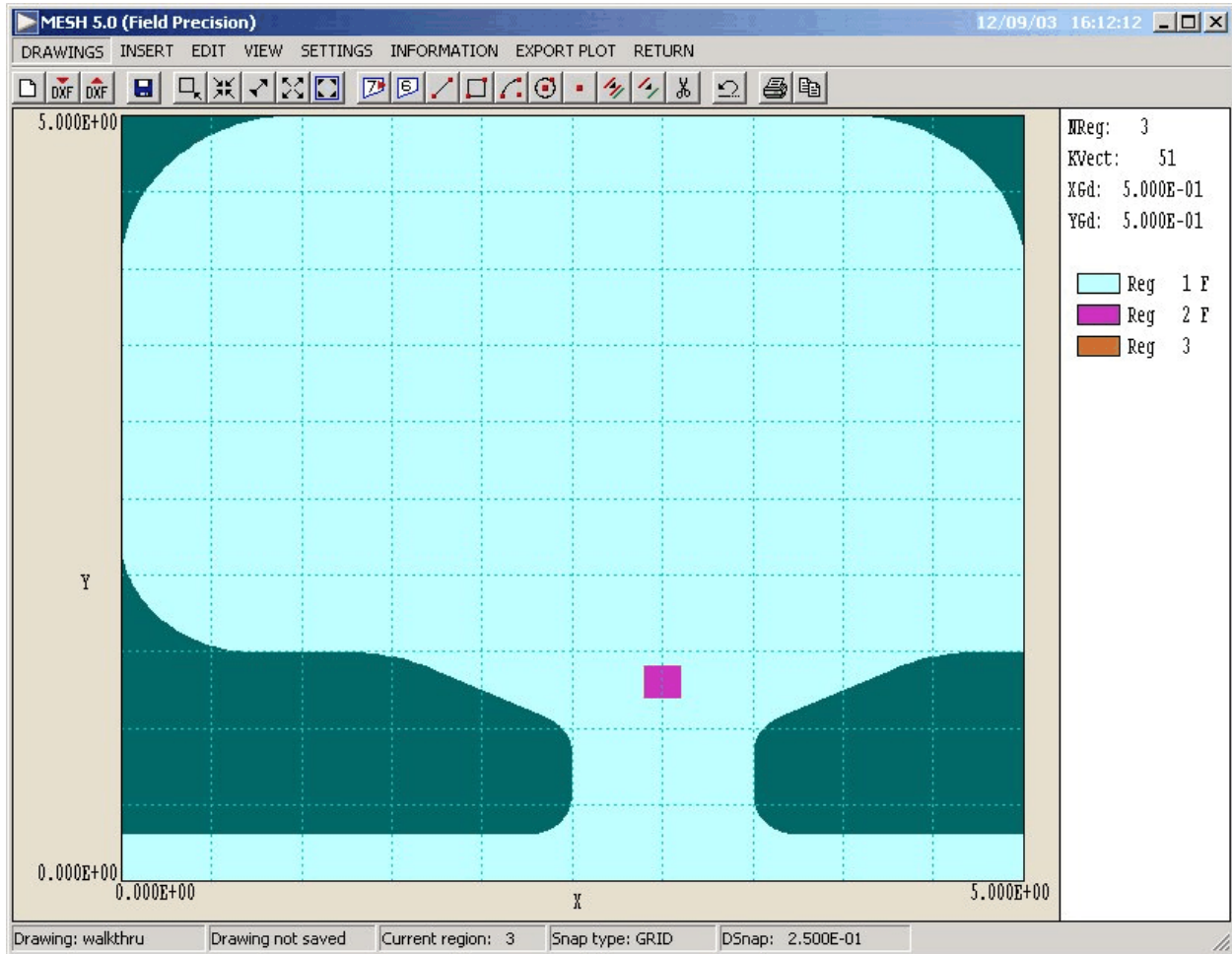
Figure 1. Definition of the cavity geometry for the WALKTHRU example. The cavity has cylindrical symmetry about the lower axis. Dimensions are in cm.

2. Walkthrough example

To test the installation and to get acquainted with **WaveSim** procedures, we shall follow through an example in detail. We want to calculate the resonant frequency and quality factor (Q) for the klystron cavity shown in Figure 1. Note that the cavity is a figure of revolution about the abscissa. In the **WaveSim** calculation, the horizontal axis will be associated with *z* and the vertical axis with *r*. To begin, move the files WALKTHRU.DXF, WALKTHRU.WIN and WALKTHRU.SCR to a working directory such as \TRICOMP\BUFFER. Run **TC** (the **TriComp** program launcher) and check that the program and data directories are set correctly. The standard values are \TRICOMP and \TRICOMP\BUFFER.
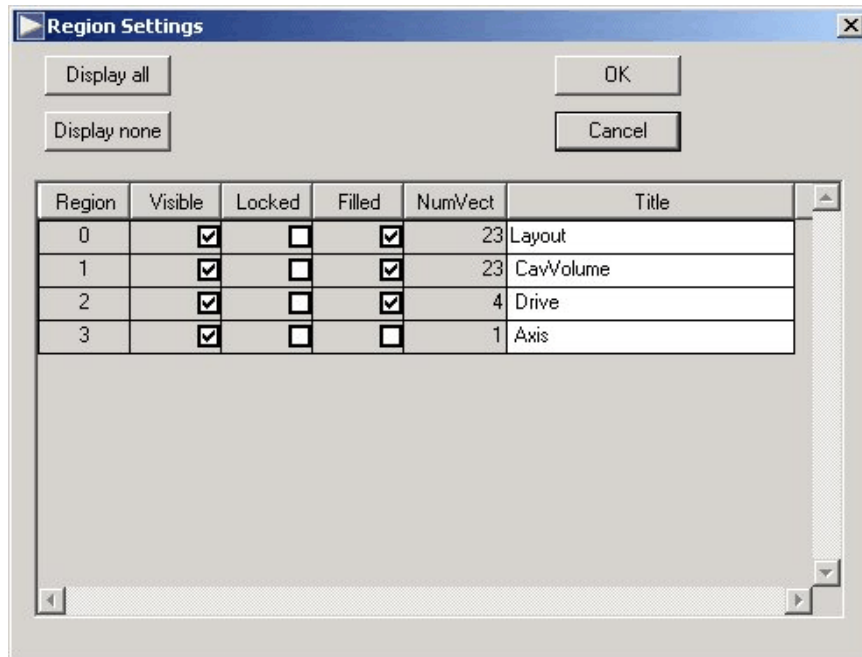
Figure 2. Region properties dialog.

We will first learn how the **Mesh** drawing editor can be used to create an input script that describes the geometry. Don't worry if you make a mistake in the following sequence. You can always use the prepared script `WALKTHRU.MIN` included with the examples. The file `WALKTHRU.DXF` is a drawing file that was prepared with QCAD, a freeware 2D CAD program included with **WaveSim**. Advanced features of the program were employed to generate the fillets (arcs) on the cavity boundaries. We added some features such as region names in the **Mesh** drawing editor. Launch **Mesh** and choose the menu command *DRAWING*. The program enters the drawing editor, one of the options for preparing **Mesh** input. Click on *DRAWING/IMPORT DXF* and choose the file `WALKTHRU.DXF`. Click *OK* again when the program displays a dialog listing information on the drawing. Next, click on *SETTINGS/DISPLAY REGION FILL*. You should see a display like Fig. 1 (we have changed some of the colors for clarity). As loaded, the drawing contains a layout layer and three active regions. Click on *SETTINGS/REGION PROPERTIES* to display the dialog of Fig. 2. The vectors of Region 1 comprise the closed outline of the cavity. The region has the *FILLED* property so the region number is assigned to all enclosed nodes and elements (light blue area in Fig. 1). This region will have the properties of vacuum in the **WaveSim** calculation. Region 2 (a small rectangle) is a drive region required for resonant-cavity calculations. This filled vacuum region will have a RF current density of arbitrary
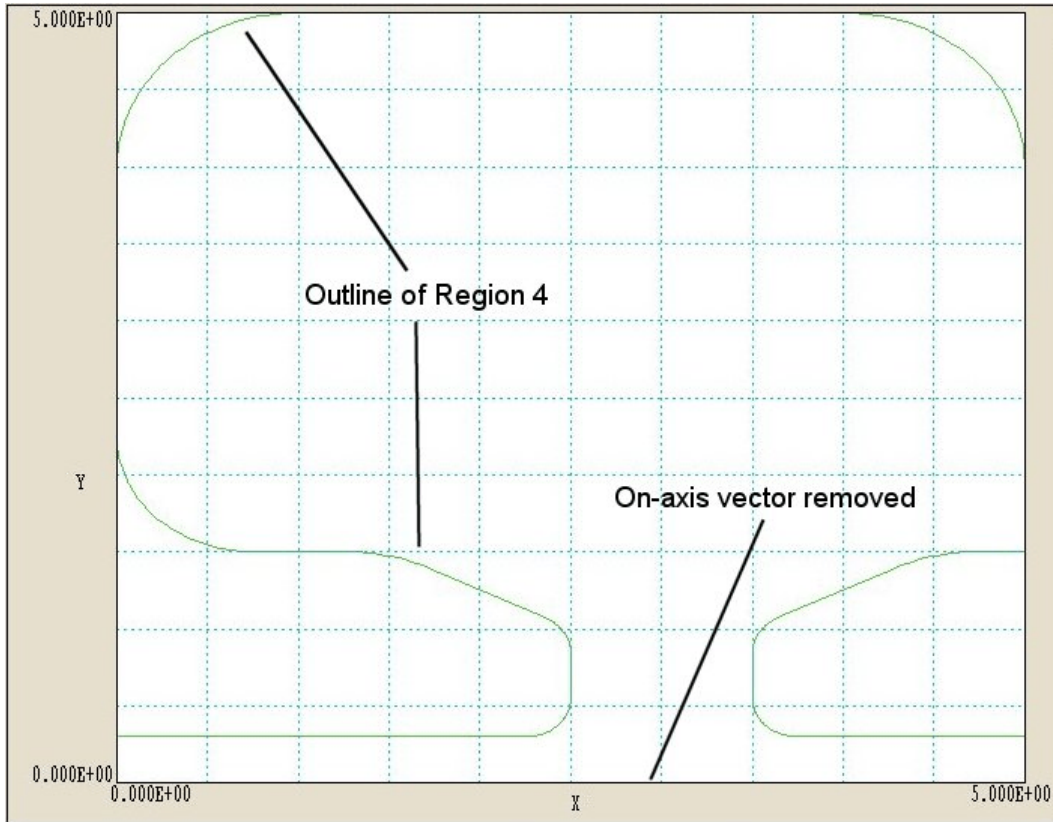
amplitude to initiate the mode. Region 3 consists of a single vector that extends along the axis. This region will be used to set nodes to the fixed condition $rH_\theta = 0.0$.

Before we create an input script for **Mesh** from the drawing, we need to create an additional region. We shall assign region number 4 to nodes on the cavity boundary with the exception of the axis. This identification will be used for a diagnostic in the **WSV** postprocessor. Click on *EDIT/COPY REGION*. To select Region 1, move the mouse inside the blue area close to the wall. Click the left button. The boundary of region 1 should be highlighted. The program next asks for two points that define a displacement for the copied region. We want no displacement, so click the left button twice with the mouse in the same position. The list on the right-hand side of the screen shows that the new region has been added. To view the region, return to *SETTINGS/REGION PROPERTIES*. Uncheck the *VISIBLE* attribute and check the *LOCKED* attribute for Regions 1, 2 and 3, then click *OK*. The display shows the green outline of Region 4 (Fig. 3). We need to delete the on axis vector. Choose

6
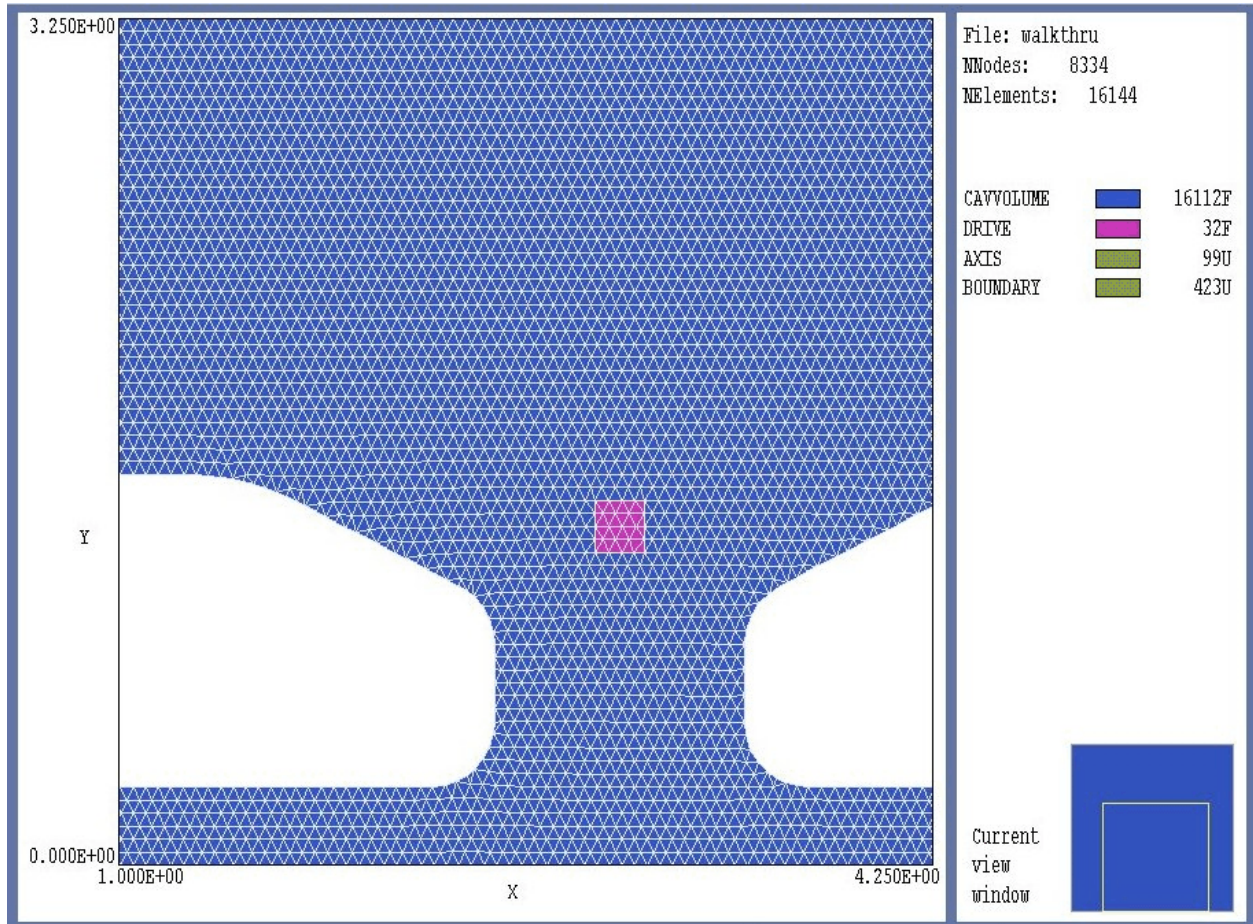
Figure 4. Mesh for the WALKTHRU example (detail near the axis)

EDIT/DELETE/VECTOR, move the mouse close to the axis and click the left button. You can check that the operation was successful by comparing the number of vectors in Regions 1 and 4 in *SETTINGS/REGION PROPERTIES*. While in the dialog, assign the name *BOUNDARY* to region 4. To save your work, choose *DRAWINGS/EXPORT DXF*.

The next step is to create a script in the **Mesh** input format. Click on *DRAWINGS/CREATE MIN FILE*. The program automatically translates the DXF vector information into a **Mesh** script and shows the result in the internal text editor. We shall make no changes, so simply close the text editor and exit the drawing editor by clicking on *RETURN*.

If you did not complete the above procedure, copy the file WALKTHRU.MIN (**M**esh **IN**put) from the example directory to continue the demonstration. In the main **Mesh** menu, click on *FILE/LOAD SCRIPT*

(MIN) and choose the file `WALKTHRU.MIN`. Next click on *PROCESS*. **Mesh** reads the boundary specifications, shapes elements to conform to the material divisions and assigns region numbers to nodes and elements. Click on `FILES/SAVE MESH (MOU)` to create an output file (**M**esh **OU**tput). This file will supply the geometric information for the **WaveSim** solution. At this point, you can experiment with some of the plot capabilities of Mesh. Figure 4 shows a detail of the conformal elements near the axis. The box to the right shows region names along the with number of elements (for filled regions) or the number of nodes (for open regions).

Table 1. File WALKTHRU.WIN

```
* File WALKTHRU.WIN

SET Geometry Cylin
SET DUnit 100.0
SET Solution H
SET Probe 2.5 4.8

* Commands to perform the initial scan
* SET FStep 30
SET Mode Scan
SET Range 1.2E9  2.0E9

* Commands to find the fundamental resonant mode
* SET Mode Search
* SET Range 1.45E9  1.55E9

REGION(1) Mu 1.0
REGION(1) Epsi 1.0
REGION(2) Drive 1.0 0.0 0.0
REGION(3) Reflect
REGION(4) Mu 1.0
REGION(4) Epsi 1.0

ENDFILE
```

A **WaveSim** calculation requires a **Mesh** output file to define the geometry and a script to control field solution. Table 1 shows the contents of the file `WALKTHRU.WIN` (**W**avesim **IN**put). Commands that begin with the keyword *SET* define values that control the program. Command that begin with *REGION* set physical properties for the regions. The *SET*

8

GEOMETRY command states that the system has cylindrical symmetry and the *SET DUNIT* command specifies that dimensions (including those in the **Mesh** file) are given in centimeters. The command *SET SOLUTION H* indicates that the primary quantity of the desired TM_{010} mode is rH_θ while the derived quantities are E_r and E_z. Section 5 discusses electromagnetic mode types. The command *SET PROBE* defines the position of a probe to sense the response of the cavity to the drive near the axis (Region 2). The probe is located near the anticpated position of maximum rH_θ.

Two next two groups of commands determine the type of solution that **WaveSim** will perform. The first group calls for a calculation in the *SCAN* mode, while the second group (deactivated by comment symbols) defines a high-accuracy resonance search. In a *SCAN* calculation, **WaveSim** carries out a series of solutions in a frequency interval and records field values at the probe location. Zero-crossings of the probe signal indicate resonances in the interval. Using the results of the scan, a high-accuracy search is performed over an interval that contains the desired resonance. In the example the cavity has an outer radius of $R_o = 0.05$ m. The maximum possible frequency for the TM_{010} mode is

$$f = \frac{2.405 \ c}{2\pi \ Ro} = 2.3 \quad (GHz).$$

The re-entrant beam tube structure adds extra capacitance near the axis and lowers the frequency. As an initial guess we picked the scan range 1.2 to 2.0 GHz.

The remaining commands assign physical properties to the regions:

■ Region 1. Vacuum with $\epsilon_r = 1$ and $\mu_r = 1$. The commands are included as reminders -- in the absence of any specification the default is vacuum.

■ Region 2. A vacuum region with a drive current density. The parameters of the *REGION DRIVE* command specify that the current density has zero phase points and points along the z axis with amplitude $rj_z = 1.0$ A/m. The drive amplitude is arbitrary in resonant-cavity calculations with no losses.

■ Region 3. The boundary condition $rH_\theta = 0$ on the axis.

■ Region 4. The line of nodes along the cavity wall for **WSV** diagnostics.
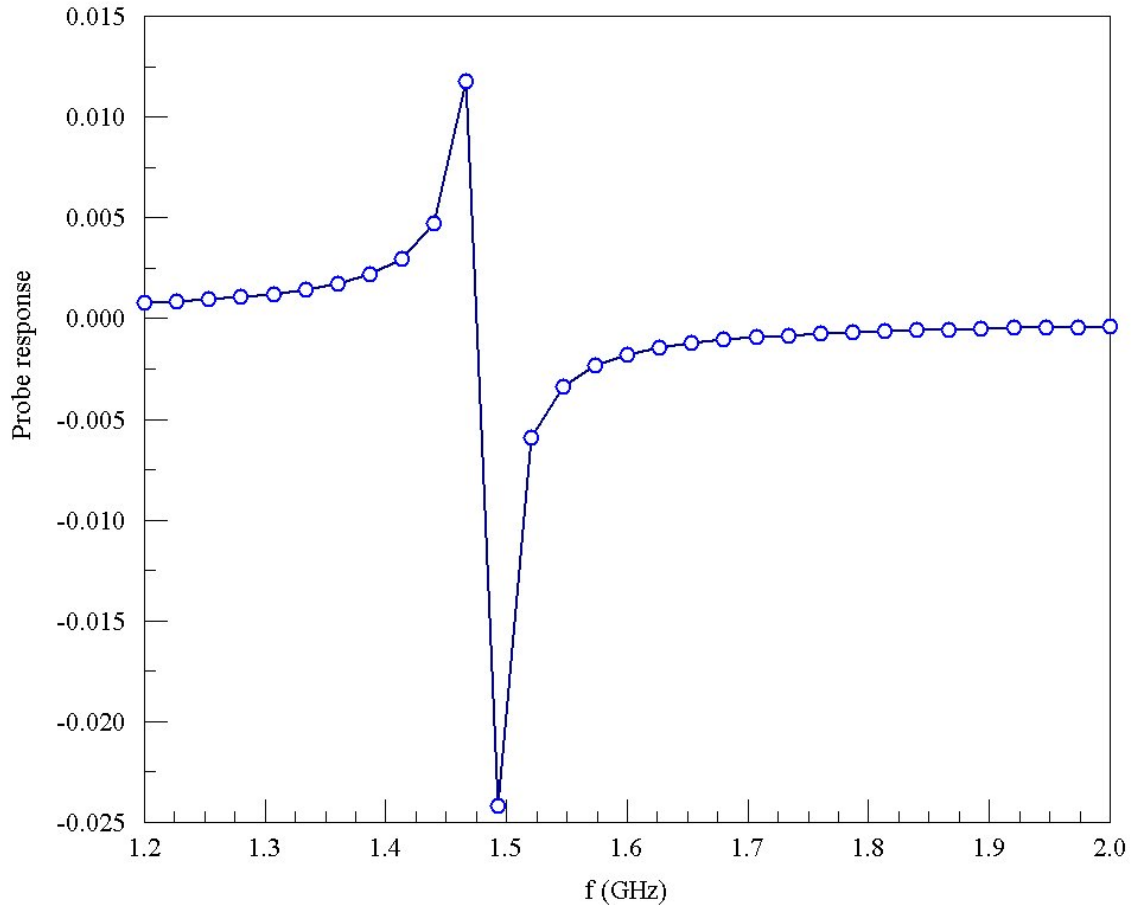
Figure 5. Probe response as a function of frequency for the `WALKTHRU` example.

Launch **WaveSim** from **TC.** Click on *RUN/START RUN* and choose the file `WALKTHRU.WIN`. The program loads the input data files, sets up node equations and carries out a series of sparse matrix inversions to find complex values of rH_θ at all nodes. The scan takes about 3-4 minutes. When the run is complete, click on *FILE/EDIT LISTING FILES* and choose `WALKTHRU.WLS`. The file (shown in Table 2) contains a listing of the probe response at 31 frequencies (30 uniform intervals). Figure 5 shows a plot of the real part of the probe response. The zero crossing near 1.5 GHz indicates the presence of a resonance. Based on this information, we choose the bracketing frequency interval of 1.45 to 1.55 GHz for the resonance search.

10

Table 2. Results of the frequency scan in the WALKTHRU example

```
--- Frequency Scan ---
    NStep:    30
  FreqLow:    1.200000E+09
 FreqHigh:    2.000000E+09
   ProbeX:    2.510869E+00
   ProbeY:    4.800000E+00

  NStep    Frequency     Probe(Real)    Probe(IMag)    Probe(Mag)
  ================================================================
     0  1.200000E+09   7.729098E-04   0.000000E+00   7.729098E-04
     1  1.226667E+09   8.485153E-04   0.000000E+00   8.485153E-04
     2  1.253333E+09   9.417076E-04   0.000000E+00   9.417076E-04
     3  1.280000E+09   1.059745E-03   0.000000E+00   1.059745E-03
     4  1.306667E+09   1.213002E-03   0.000000E+00   1.213002E-03
     5  1.333333E+09   1.421275E-03   0.000000E+00   1.421275E-03
     6  1.360000E+09   1.718083E-03   0.000000E+00   1.718083E-03
     7  1.386667E+09   2.178917E-03   0.000000E+00   2.178917E-03
     8  1.413333E+09   2.982848E-03   0.000000E+00   2.982848E-03
     9  1.440000E+09   4.748147E-03   0.000000E+00   4.748147E-03
    10  1.466667E+09   1.175621E-02   0.000000E+00   1.175621E-02
    11  1.493333E+09  -2.420435E-02   0.000000E+00   2.420435E-02
    12  1.520000E+09  -5.935348E-03   0.000000E+00   5.935348E-03
    13  1.546667E+09  -3.374799E-03   0.000000E+00   3.374799E-03
    14  1.573333E+09  -2.355437E-03   0.000000E+00   2.355437E-03
    15  1.600000E+09  -1.807685E-03   0.000000E+00   1.807685E-03
    16  1.626667E+09  -1.465374E-03   0.000000E+00   1.465374E-03
    17  1.653333E+09  -1.231654E-03   0.000000E+00   1.231654E-03
    18  1.680000E+09  -1.062099E-03   0.000000E+00   1.062099E-03
    19  1.706667E+09  -9.332518E-04   0.000000E+00   9.332518E-04
    20  1.733333E+09  -8.323748E-04   0.000000E+00   8.323748E-04
    21  1.760000E+09  -7.510412E-04   0.000000E+00   7.510412E-04
    22  1.786667E+09  -6.843203E-04   0.000000E+00   6.843203E-04
    23  1.813333E+09  -6.284751E-04   0.000000E+00   6.284751E-04
    24  1.840000E+09  -5.811739E-04   0.000000E+00   5.811739E-04
    25  1.866667E+09  -5.405628E-04   0.000000E+00   5.405628E-04
    26  1.893333E+09  -5.053715E-04   0.000000E+00   5.053715E-04
    27  1.920000E+09  -4.745641E-04   0.000000E+00   4.745641E-04
    28  1.946667E+09  -4.474191E-04   0.000000E+00   4.474191E-04
    29  1.973333E+09  -4.233323E-04   0.000000E+00   4.233323E-04
    30  2.000000E+09  -4.017980E-04   0.000000E+00   4.017980E-04
```

Exit the editor and click on *FILE/EDIT INPUT FILE*. Choose
WALKTHRU.WIN . The editor displays the contents of Table 1. Modify
the comment status of control lines to change the program mode for the
next calculation. The file should look like this:

```
* Commands to perform the initial scan
* SET FStep 30
* SET Mode Scan
* SET Range 1.2E9  2.0E9

* Commands to find the fundamental resonant mode
SET Mode Search
SET Range 1.45E9  1.55E9
```

Save the file and exit the editor. Start the second solution using the *RUN/START RUN* command. The resonance search takes about 30-40 seconds and produces the following entry in the listing file:

```
Iteration:   1   Frequency:   1.484390E+09   Response:   1.246726E+00
Iteration:   2   Frequency:   1.484653E+09   Response:  -1.332502E-01
Iteration:   3   Frequency:   1.484578E+09   Response:   3.498476E-01
Resonance search successful.
    Final frequency:   1.484653E+09
    Writing data to file WALKTHRU.WOU
```

In this run **WaveSim** identified the fundamental mode frequency as 1.4847 GHz and created an output file of field values at that frequency. We can now proceed to an analysis of the solution.

Run **WSV** from **TC** and click on *FILE/LOAD SOLUTION FILE*. Pick WALKTHRU.WOU and click *OK*. The program loads the data and displays the default plot of electric field lines shown in Fig. 6. Contours of rH_θ are separated by equal intervals of magnetic-field flux; therefore, the spacing is larger near the axis. **WSV** has extensive capabilities. We will perform two specific tasks:

  ■ Plot the magnitude and direction of the electric field near the interaction gap for a given amplitude of deceleration field

  ■ Determine the Q value for an ideal copper cavity.

Remember that the magnitude of field values for a resonant solution is arbitrary. First, we must renormalize the solution for a given peak deceleration field. Click on *ANALYSIS/SCAN PLOT QUANTITY* and choose $|E_z|$. Next click on *ANALYSIS/LINE SCAN* and move the mouse pointer into the plot area. Note that the cursor changes to a cross-hair pattern when the mouse is active inside the plot. Using the coordinates

12
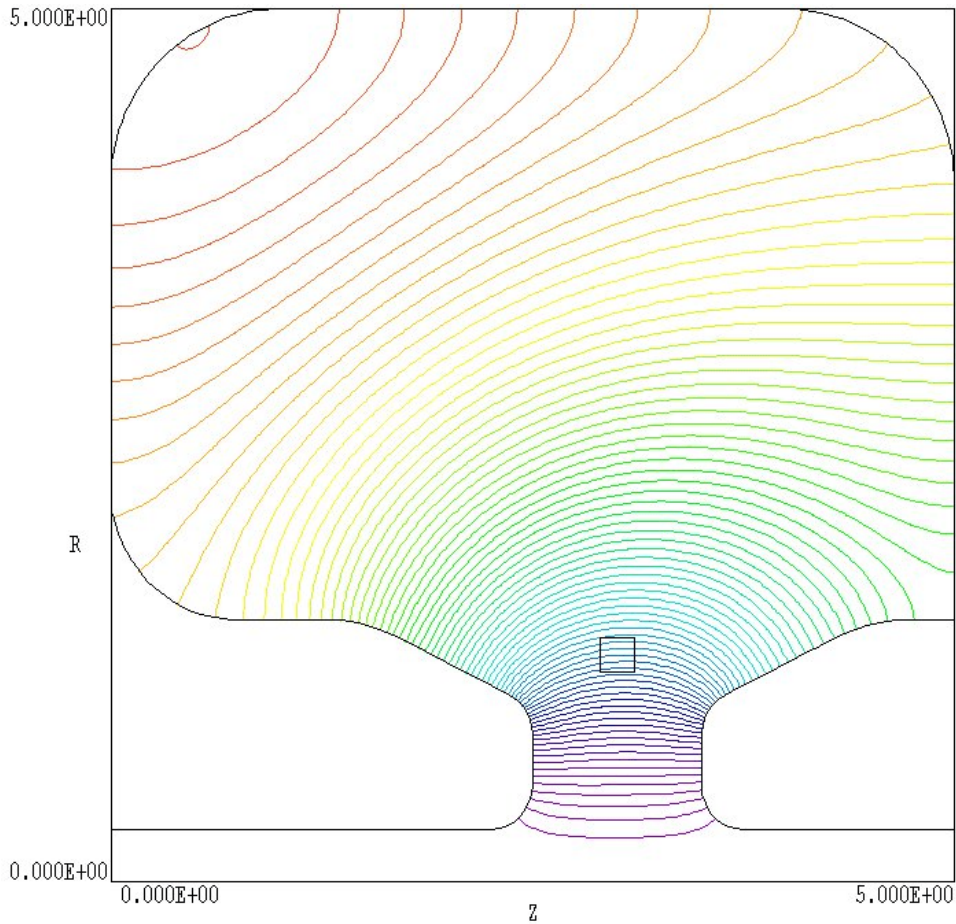
Figure 6. Electric field lines separated by equal intervals of magnetic field intensity flux, WALKTHRU example.

listed at the bottom of the screen, move the pointer to a psotion near the point [$x/z = 1.5$, $y/r = 0.0$] and click the left mouse button. Then move the mouse to the position [$x/z = -1.5$, $y/r = 0.0$] and click again. **WSV** calculates a sequence of |E_z| values and displays the results (Fig. 7). Note that the peak field is 1.524×10^5 V/m, while the target peak field is 7.5×10^6 V/m. We shall increase the field values by a factor of 49.21. Exit the scan plot menu by clicking *SCAN PLOTS/CLOSE SCAN PLOT* and choose the command *ANALYSIS/RENORMALIZE FIELD*. Type '49.21' in the box and click *OK*. **WSV** multiplies all values of rH_θ by the normalization factor.

To create the field plot, click on the command *SPATIAL PLOTS/PLOT TYPE*, choose the option *VECTOR* and click *OK*. The program issues an error message because the present quantity (rH_θ) is not supported in a vector plot. Click on *SPATIAL PLOTS/PLOTTED QUANTITY* and choose

13
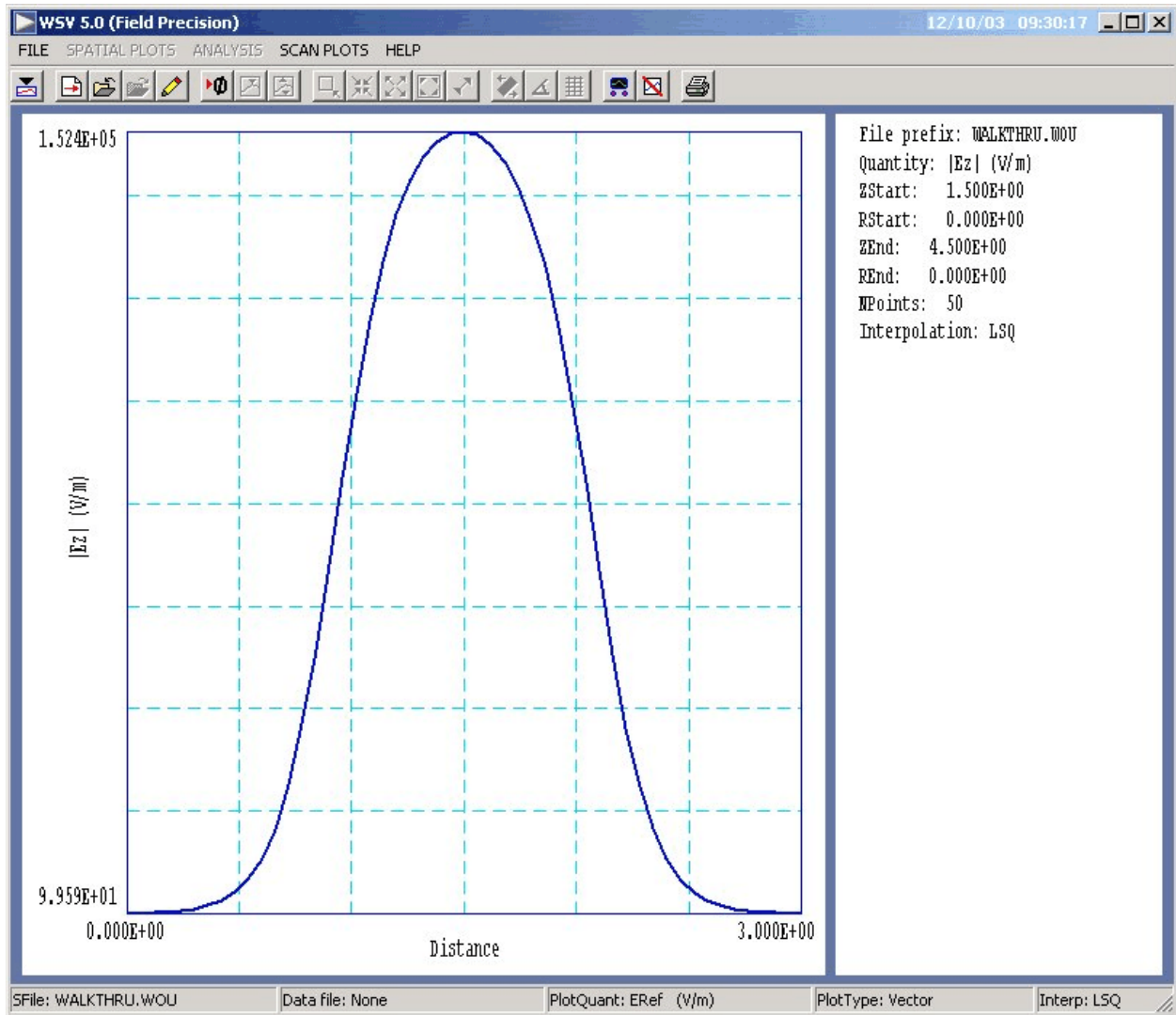
the single option |*ERef*|, the magnitude of the electric field at the reference phase. **WSV** creates a blank plot with no content. This is because the default value of reference phase is 0° and the electric field is 90° out of phase with the drive current (set to 0° in **WaveSim**). Click on *ANALYSIS/SET REFERENCE PHASE* and enter '90' in the box. Upon exiting the dialog, WSV creates a plot of electric field vectors with the background color-coded by the amplitude of the electric field. You can use the *ZOOM* tool to create the plot of Fig. 8.

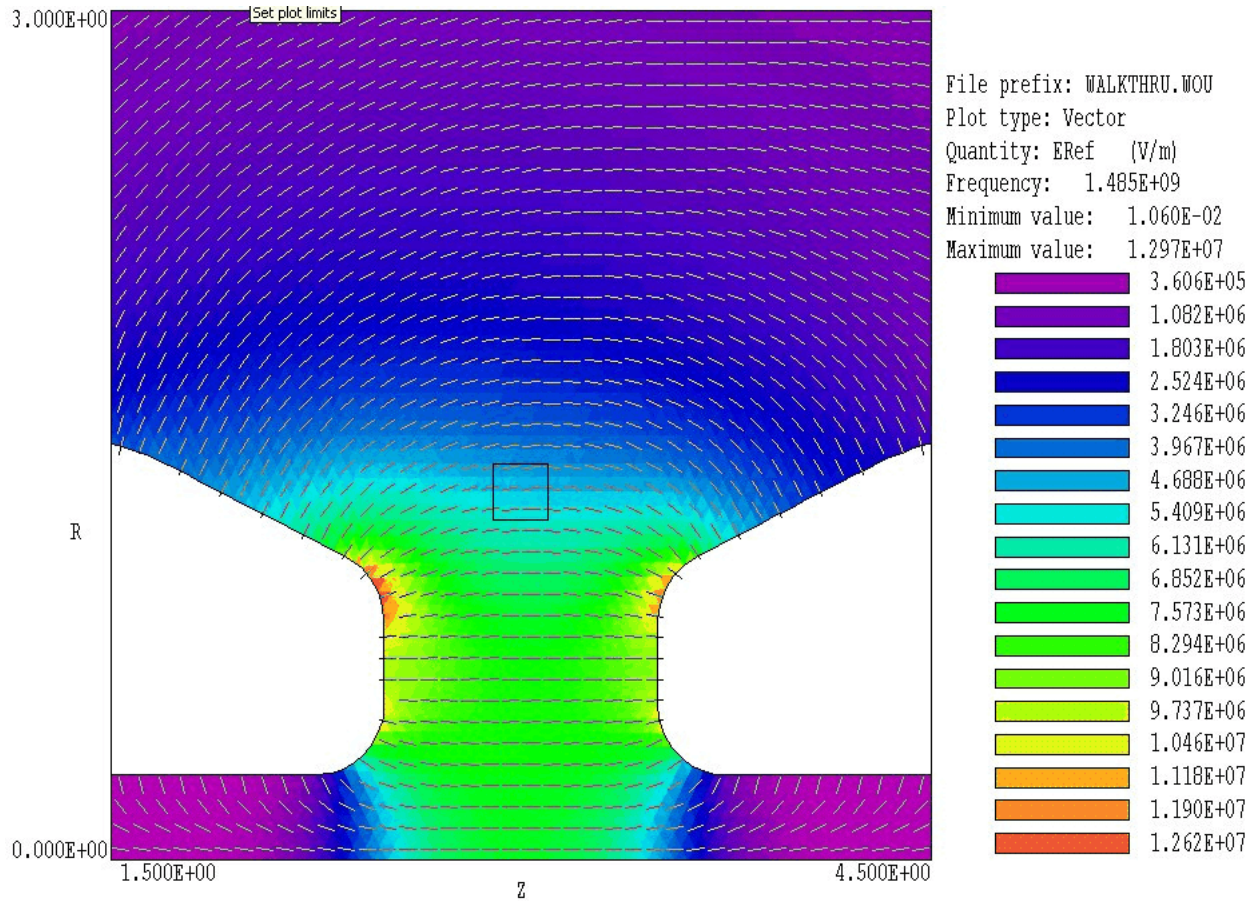**Figure 8**. Electric field magnitude and direction near the deceleration gap, WALKTHRU example.

Table 3. Contents of the file WALKTHRU.SCR

```
* File: WALKTHRU.SCR
* Script control demonstration
* Assume that the file WALKTHRU.WOU has been opened
* and that the field values have been normalized by
* a factor 49.21
INPUT WALKTHRU.WOU
OUTPUT WALKTHRU
VOLUMEINT
ENDFILE
```

To conclude, we shall calculate the quality factor for the cavity using the automatic-analysis feature of **WSV**. Click on *FILE/RUN SCRIPT* and choose WALKTHRU.SCR. This file has the contents shown in Table 3. The script opens an output file WALKTHRU.DAT and records the results of a volume analysis. To see the results, click on *FILE/EDIT DATA FILE* and choose WALKTHRU.DAT. Table 4 shows the results of the analysis. At the chosen field normalization, the electromagnetic energy inside the cavity is $U = 3.292 \times 10^{-3}$ joules. The other quantity of interest for the Q-factor calculation is the following integral taken over the surface of Region 4:

$$\iint_S dS \ H_p^2/2 \ .$$

In the equation H_p is the amplitude of the component of magnetic field intensity parallel to the wall. The calculated value is 2.444×10^5 A^2. Following S. Ramo, J. Whinnery and T. Van Duzer, **Field and Waves in Communication Electronics** (Wiley, New York, 1965), Sect. 5.15, the time-averaged power lost to walls is :

$$P = R_s \iint_S dS \ H_p^2/2 \ .$$

The quantity R_s is the surface resistivity. For copper, this quantity is given by:

$$Rs = 2.61 \times 10^{-7} \sqrt{f} \qquad \Omega \ ,$$

The surface resistivity is $R_s = 1.006 \times 10^{-2}$ at f = 1.485 GHz, so that average power loss is $P = 2.458 \times 10^3$ W. Using the formula

$$Q = \frac{2\pi f \ U}{P} \ ,$$

we find that the quality factor for ideal copper walls is $Q = 15,520$. In practice, the quality factor may be lower because of imperfect surface finish on the cavity walls.

Table 4. Volume integrals for the WALKTHRU example

```
 --- Volume Integrals ---
Volume:   3.558E-04 m3
Field energy:   3.292E-03 J
Volume power dissipation:   0.000E+00 W
EMax:   1.297E+07 V/m
    Z:   2.503E+00,  R:   9.629E-01
HMax:   8.232E+03 V/m
    Z:   5.881E-01,  R:   1.530E+00

    Integrals by region
  NReg    Volume        Energy       Power
          (m3)          (J)          (W)
  =====================================
     1  3.554E-04  3.273E-03  0.000E+00
     2  3.267E-07  1.930E-05  0.000E+00

 --- Line Integrals over Line Regions ---
  NReg     Int(Hp^2/2)
             (A^2)
  ====================
     3       0.000E+00
     4       2.444E+05
```

3. Organizing runs

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

WaveSim 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.WIN` where `RUNNAME` is the same prefix as that of the mesh file. **WaveSim** 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.WOU`.

A **WaveSim** 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.WIN`) 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 and the manual for the specific solution program.

■ Run **WaveSim** to create listing output of the output file `RUNNAME.WOU` in the *SEARCH* and *SCATTER* modes. The output file 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 **WSV**. You can also transfer the information in `RUNNAME.WOU` to your own analysis programs.

4. 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 **WaveSim**:

```
    * File BENCH02.WIN
    SET Geometry Rect
    SET DUnit 1.0
    SET Solution E
    SET Mode Scatter
    SET Freq 30.0E6
    SET PlaneWave -0.6283  0.00  1.00  T
    * Absorber
    REGION 1 Mu 1.0
    REGION 1 Epsi 1.0 -7.944
    * Vacuum region
    REGION 2 Mu 1.0
    REGION 2 Epsi 1.0
    * Perfect reflector
    REGION 3 Mu 1.0
    REGION 3 Epsi 1.0E12
    REGION 3 NReplace 2
    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.

5. Commands for general program control

Like all **TriComp** solution programs, **WaveSim** is controlled by a script file consisting of a set of standard commands. The file has a name of the form `RunName.WIN`, where *RunName* is the prefix of the mesh file `RunName.MOU` prepared with Mesh 4.5. The suffix *WIN* stands for **WaveSim** input. **WaveSim** 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 `RunName.WIN` file.

SET SOLUTION H
The keyword *Solution* is used to set the primary field quantity following the choices for two-dimensional solutions listed in Table 5. The parameters are the characters *E* or *H* in either upper or lower case. In *E* wave solutions the *primary field quantity* is either E_z or rE_θ. In *H* wave solutions the primary field quantity is either H_z or rH_θ.

SET GEOMETRY CYLIN

The keyword *Geometry* sets the solution symmetry according to the choices of Table 5. The parameter options are the strings *RECT* or *CYLIN*. All units and interpolation methods are automatically adjusted in *WaveSim* and *WSV*.

SET MODE SEARCH

The keyword *MODE* sets the nature of the calculation according to the options of Table 6. The parameter options are the strings *Scatter*, *Scan* or *Search*. *Scan* and *Search* calculations involve several solutions and may lead to long run rimes for large meshes.

SET DUNIT 100.0

The keyword *DUnit* signals the specification of length parameters in units other than meters. The real number parameter is the number of input units per meter. For example, use $D_{unit} = 100.0$ if dimensions in **Mesh** were entered in centimeters.

SET FREQ 100.0E6

The keyword *FREQ* sets a value for the wave frequency (*f*) in Hz for *Scatter* type calculations. This command has no effect in the *Search* and *Scan* modes.

SET RANGE 120.0E6 150.0E6

The keyword *RANGE* defines a range of frequencies for *Scan* or *Search* calculations. The parameters are f_{min} and f_{max} in Hz. In the *Scan* mode, **WaveSim** performs a series of calculations from f_{min} to f_{max} in uniform frequency steps of $(f_{min} - f_{max})/f_{step}$ (see the *FStep* command below). In the *Search* mode, f_{min} and f_{max} define a frequency interval for the search that should bracket a single resonance. The program will not proceed unless the probe response changes sign over the interval. The command has no effect in the *Scatter* mode.

SET FSTEP 25

The keyword *FSTEP* sets the number of frequency intervals in the *Scan* mode or the maximum of cycles in the *Search* mode. The integer parameter is the number of steps. The defaults values are 20 in the *Scan* mode and 7 in the *Search* mode. The command has no effect in the *Scatter* mode.

SET TOLERANCE 5.0E-5

The keyword *TOLERANCE* sets the target accuracy for a resonance search in the *Search* mode. The real number parameter is the target accuracy ϵ. **WaveSim** exits the search when the relative change in frequency between two cycles satisfies $\Delta f/f < \epsilon$. The default value is 1.0×10^{-4}. The command has no effect in the *Scatter* and *Scan* modes.

SET PROBE 0.01 0.02

The keyword *PROBE* sets a location for a virtual probe that senses the real and imaginary parts of the primary field component for the *Scan* and *Search* modes. The real number parameters are the approximate coordinates of the position, either (x,y) or (r,z). **WaveSim** places the probe at the nearest node. The positions should be entered in meters or alternate units if the *DUnit* command appears in the input file. This command has no effect in the scatter mode.

SET CONDMODE

The standard input method for lossy materials is to give relative values for the real and imaginary parts of the dielectric constant (ϵ'/ϵ_o, ϵ''/ϵ_o) and the magnetic permeability (μ'/μ_o, μ''/μ_o). In the *Scan* and *Search* modes it is sometimes better to keep the effective material conductivity constant over the frequency range. This is true when we want to maintain an ideal absorbing layer. This command signals *WaveSim* to expect the quantities $\omega\epsilon''/\epsilon_o$ or $\omega\mu''/\mu_o$ as the imaginary material properties. In this case, the program calculates ϵ''/ϵ_o and μ''/μ_o for each frequency by dividing the quantities by ω. For an ideal resistive material, the dielectric quantity equals $\omega\epsilon''/\epsilon_o = \sigma/\epsilon_o$, where σ is the conductivity is S/m.

SET LOWQ

For a resonance search in an ideal or high-Q structure, *WaveSim* looks for a zero in the reciprocal of either the real or imaginary part of the probe response. In low Q structures it is sometimes better to search for a zero of the direct probe signal. This command activates the low Q search option.

SET PLANEWAVE 13.6 0.0 5000.0 T

This command sets up an incident plane electromagnetic wave for scattering solutions in rectangular geometry using the method of virtual sources. There are three real number parameters.

k_x: The wave number in m^{-1} along the x direction

k_y: The wave number in m^{-1} along the y direction

A: The wave amplitude for the primary field component

Depending on the mode, the quantity A may have units of V/m, V, A/m or A. As an example, the primary field for an E wave solution in planar geometry field has the variation

$$E_z(x,y,t) = A \exp[j(k_x x + k_y y - \omega t)]$$

Values of $k_x > 0$ and $k_y = 0$ give a plane wave propagating in the positive x direction. The optional single-character parameter *T* controls the field solution output. If it appears, WaveSim makes a data file of the total field - incident plus scattered. Otherwise, the programs records the scattered field only. For more information on free-space scattering calculations, see the *NREPLACE* command.

Table 5 Options for two-dimensional solutions in WaveSim			
Type	**Geometry**	**Primary field component**	**Subsidiary field components**
E	Rect	E_z	H_x, H_y
E	Cylin	rE_θ	H_z, H_r
H	Rect	H_z	E_x, E_y
H	Cylin	rH_θ	E_z, E_r

Table 6 Solution modes in WaveSim		
Mode	**Function**	**Comments**
Scatter	Simulation of open or closed systems with specified frequency and drives	Single solution at the given frequency output to a data file `FName.WOU`.
Scan	Searches for resonances in closed structures or peak output from open structures	A sequence of solutions with a probe listing of the primary field component at a given location. No data file.
Search	Search for a resonant mode within a frequency range for ideal or lossy structures	Several solutions to search for maximum probe response followed by output to a data file `FName.WOU` at the final frequency.

6. Commands for material properties

Because frequency-domain solutions are meaningful only for linear materials, the set of *WaveSim* commands to define region material properties is simple. The commands to set the physical properties of materials all start with the word *Region* and have similar format:

```
Region    RegNo    Keyword    Value[s]
```

Here, the integer *RegNo* is the region number defined in the **Mesh** input file. *Keyword* is one of the options listed below and *Value[s]* represents one or more parameters.

REGION 4 REFLECT
The keyword *REFLECT* designates that primary field has the fixed value [0.0,0.0] in the region. The reflection condition represents a metal surface (short-circuit boundary) for *E* type waves and an ideal open non-radiating boundary (open-circuit) for *H* type waves.

REGION 7 VOID

A *VOID* boundary is the inverse of a *REFLECT* boundary. It represents a metal wall for *H* type waves and an open-circuit for *E* type waves. *WaveSim* implements internal void regions by setting $\epsilon = \epsilon_o$ and $\mu = 10^{12}\mu_o$ for *E* type waves and $\epsilon = 10^{12}\epsilon_o$ and $\mu = \mu_o$ for *H* type waves. The *WSV* program suppresses field line plotting in void regions. Note that all unspecified external boundaries automatically assume *VOID* conditions.

REGION 2 EPSI 5.8 -56.03

This command sets the dielectric constant for elements of a region. The first real number parameter is the real part of the relative dielectric constant, ϵ'/ϵ_o. The optional second parameter equals the imaginary part of the dielectric constant, ϵ''/ϵ_o, if *CondMode* is not set. Otherwise, enter the quantity $\omega\epsilon''/\epsilon_o$. (See the *SET CONDMODE* command.) The code takes the imaginary part of the dielectric constant as zero if there is no second parameter. The default value for all regions is [1.0,0.0].

REGION 2 MU 5.8 -56.03

This command sets the magnetic permeability for elements of a region. The first real number parameter is the real part of the relative magnetic permeability, μ'/μ_o. The optional second parameter equals the imaginary part of the magnetic permeability, μ''/μ_o, if *CondMode* is not set. Otherwise, enter the quantity $\omega\mu''/\mu_o$. (See the *SET CONDMODE* command.) The code takes the imaginary part of the magnetic permeability as zero if there is no second parameter. The default value for all regions is [1.0,0.0].

REGION 7 FIXED 2500.0 45.0

The keyword *FIXED* indicates a fixed field region that acts as a wave source. The two real parameters are the amplitude and phase of the primary field at all points in the region. Specify the amplitude in the standard units of the primary field and the phase in degrees.

REGION 3 DRIVE   4500.0 -90.0   0.0

The keyword *DRIVE* indicates that the region has a drive current density. The first real number is the amplitude of the drive. The interpretation of the number and units depends on the geometry and solution type. Table 7 shows the options. The second real number

parameter is the phase in degrees. The third optional parameter gives the direction of the current for *H* wave initiation. The quantity is the angle of the current density vector (in degrees) in the *x-y* plane relative to the *x* axis for planar problems. For cylindrical problems, the quantity gives the angle in the *r-z* plane relative to the *z* axis. The parameter has no meaning for *E* wave solutions where the current density points along the direction of the primary field (**z** or **θ**). Note that you can specify values of relative dielectric constant and magnetic permeability in a drive region using the *EPSI* and *MU* commands.

REGION 5 NREPLACE 2

This command is used to implement the distributed source method for free-space scattering solutions. The source terms are computed as though the scattering object were removed from a homogenous region of space. In this command, the first integer specifies the number of a region that constitutes a part of the scattering object. The second integer is the region number of the bulk of the solution volume. This volume carries a plane wave in the absence of the scattering object. Each region of the scattering object must have an *NREPLACE* command. (See also the *SET PLANEWAVE* command.)

<table>
<tr><td colspan="4">**Table 7**
Drive currents in WaveSim</td></tr>
<tr><td>**Type**</td><td>**Geometry**</td><td>**Specified quantity**</td><td>**Units**</td></tr>
<tr><td>E</td><td>Rect</td><td>j_z</td><td>A/m^2</td></tr>
<tr><td>E</td><td>Cylin</td><td>rj_θ</td><td>A/m</td></tr>
<tr><td>H</td><td>Rect</td><td>j_x, j_y</td><td>A/m^2</td></tr>
<tr><td>H</td><td>Cylin</td><td>rj_z, rj_r</td><td>A/m</td></tr>
</table>

7. Absorbing boundaries

Boundaries that absorb almost 100 per cent of the incident wave energy are used to terminate waveguides or the output ports of microwave devices. They are essential to simulate free-space conditions in scattering solutions. The method used in *WaveSim* is physically similar to resistive terminations on transmission lines. We define a thin layer and assign a resistive impedance matched to the characteristic impedance of the adjacent medium. The theory is described in Sects. 13.3 and 14.2 of the Appendices. This section gives a step-by-step summary of how to construct a termination layer boundary.

- During mesh generation, be sure to include an extra material layers of uniform thickness Δ along absorbing boundaries. The layer should be only a single-element thick as the absorption approaches 100 per cent as Δ approaches zero. The outer surface of the termination layer should have an unspecified (open-circuit) boundary.

- Calculate the characteristic impedance of the non-absorbing medium adjacent to the termination layer, $Z_1 = 377.3 \, (\mu_1'/\epsilon_1')^{\frac{1}{2}}$, where μ_1' and ϵ_1' are the real parts of the relative magnetic permeability and dielectric constant.

- Assign the values shown in Table for the relative dielectric constant and magnetic permeability in the termination layer.

Note that the method is very flexible. You can also create boundaries with capacitive or inductive properties.

<table>
<tr><th colspan="6">Table 8
Termination layer material properties for ideal absorption</th></tr>
<tr><th>Type</th><th>CondMode</th><th>ϵ_2'</th><th>ϵ_2''</th><th>μ_2'</th><th>μ_2''</th></tr>
<tr><td>E</td><td>Off</td><td>ϵ_1'</td><td>$-1/\omega\Delta z Z_1 \epsilon_o$</td><td>$\mu_1'$</td><td>0</td></tr>
<tr><td>H</td><td>Off</td><td>ϵ_1'</td><td>0</td><td>μ_1'</td><td>$-Z_1/\omega\Delta z \mu_o$</td></tr>
<tr><td>E</td><td>On</td><td>ϵ_1'</td><td>$-1/\Delta z Z_1 \epsilon_o$</td><td>$\mu_1'$</td><td>0</td></tr>
<tr><td>H</td><td>On</td><td>ϵ_1'</td><td>0</td><td>μ_1'</td><td>$-Z_1/\Delta z \mu_o$</td></tr>
</table>

8. Resonance searches

WaveSim will automatically locate resonances of high Q structures in the *Search* mode. When defining boundaries in **Mesh**, include a small region for a drive current. For *E* type solutions, the drive should be near the anticipated location of maximum **H**. For *H* type solutions, the drive should be near the location of maximum **E** and point in the direction of the electric field. The drive current may have any amplitude, but must have zero phase. Place a probe at the anticipated location of the maximum primary field closest to the drive point. Enter a range of frequencies that brackets the resonance of interest using the *Range* command. If you are unsure of the approximate frequency or if there are several nearby resonances, run a *Scan* over the frequency range. For *H* type waves, a resonance occurs when the real part of the probe response goes from a positive to negative value. For *E* type waves, the imaginary part of the probe signal makes a zero crossing at resonance. During the search, *WaveSim* seeks the zero crossing of the reciprocal of the appropriate component of the probe signal. Sometimes it may be preferable to use the signal values directly. For this case enter the *Set LowQ* command.

9. Running WaveSim interactively

The program WaveSim.EXE can run interactively in a Window 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 **WaveSim** and **WSV** 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 **WaveSim**. 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.WIN`. The *Edit listing file* command displays files with names `FPREFIX.WLS`.

The *Run* menu has four commands.

Start run

> Pick an input file (such as `FPREFIX.WIN`) 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 WaveSim from the command prompt

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

```
..\WaveSim TWT <Enter>
```

The program runs silently, writing detailed information in the listing file
TWT.WLS. If a *SEARCH* or *SCATTER* solution is successful, the program
writes the output file TWT.WOU 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
KLYSTRON01.MIN,...,KLSTRON08.MIN and
KLYSTRON01.WIN,...,KLYSTRON08.WIN in the directory
\TRICOMP\BUFFER. Furthermore, you have written the following batch
file KRUN.BAT in the data directory using a text editor.

```
@ECHO OFF
ECHO Main switch data run
START ..\MESH.EXE KLYSTRON01
START ..\WaveSim.EXE KLYSTRON01
START ..\MESH.EXE KLYSTRON02
START ..\WaveSim.EXE KLYSTRON02
 ...
START ..\MESH.EXE KLYSTRON08
START ..\WaveSim.EXE KLYSTRON08
```

Type

```
KRUN <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.

11. WSV file menu

WSV (for **W**ave**S**im **V**iew) is the interactive graphical analysis program for output files from **WaveSim**. The program automatically adjusts labels and calculated quantities depending on the solution symmetry and RF mode type.

Program units

Spatial dimensions: meters or units set by *DUnit*.
Frequency: Hz
Electric field (**E**): volts/m
Magnetic intensity (**H**): amp/m
Energy density (time averaged): J/m^3
Power density (time averaged): W/m^3

WSV 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 WOU). 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 text format. You can use an 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 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. 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 text file.

12. WSV spatial plot menu

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

Plot type

WSV 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. 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. 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.

Table 9 shows available quantities in WSV. Note that the plot depends on the symmetry and electromagnetic mode type of the solution.

Table 9. WSV Plot Quantities				
Solution type	**E planar**	**E cylindrical**	**H planar**	**H cylindrical**
Contour plots	E_z (**H** lines)	rE_θ (**H** lines)	H_z (**E** lines)	rH_θ (**E** lines)
	E_z (ref)	E_θ (ref)	H_z (ref)	H_θ (ref)
	$|E_z|$	$|E_\theta|$	$|H_z|$	$|H_\theta|$
Element plots	$|E_z|$	$|E_\theta|$	$|H_z|$	$|H_\theta|$
	$|H_x|$	$|H_z|$	$|E_x|$	$|E_z|$
	$|H_y|$	$|H_r|$	$|E_y|$	$|E_r|$
	$|\mathbf{H}|$	$|\mathbf{H}|$	$|\mathbf{E}|$	$|\mathbf{E}|$
	U	U	U	U
	P	P	P	P
Vector plots	$\mathbf{H}$ (ref)	$\mathbf{H}$ (ref)	$\mathbf{E}$ (ref)	$\mathbf{E}$ (ref)
Surface plots or line scans	E_z (ref)	E_θ (ref)	H_z (ref)	H_θ (ref)
	$|E_z|$	$|E_\theta|$	$|H_z|$	$|H_\theta|$
	H_x (ref)	H_z (ref)	E_x (ref)	E_z (ref)
	$|H_x|$	$|H_z|$	$|E_x|$	$|E_z|$
	H_y (ref)	H_r (ref)	E_y (ref)	E_r (ref)
	U	U	U	U
	P	P	P	P
	E_z (real)	E_θ (real)	H_z (real)	H_θ (real)
	E_z (imag)	E_θ (imag)	H_z (imag)	H_θ (imag)
	H_x (real)	H_z(real)	E_x (real)	E_z (real)
	H_x (imag)	H_z (imag)	E_x (imag)	E_z (imag)
	H_y (real)	H_r (real)	E_y (real)	E_r (real)
	H_y (imag)	H_r (imag)	E_y (imag)	E_r (imag)
	$\mathbf{H}$ (ref)	$\mathbf{H}$ (ref)	$\mathbf{E}$ (ref)	$\mathbf{E}$ (ref)

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* $= 1.0 \times 10^6$, then enter dimensions in microns.

Toggle snap mode

The mouse snap mode is a convenient feature of **WSV**. 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, **WSV** 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.

13. WSV 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

WSV employs a sophisticated interpolation technique that preserves discontinuities at material boundaries. The program gives correct values for electromagnetic field components 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 material 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 **WSV**. 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). 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 listed in Table ##.

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), **WSV** 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

WSV 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 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.

Volume integrals

The following analyses are performed in response to the *Volume integrals* command.

1) Time-averaged electromagnetic field energy density,
$U = (\epsilon_r\, E^2 + \mu_r\, H^2\,)\, /\, 4$ integrated over the solution space and region volumes. The output units are J/m in rectangular solutions and J in cylindrical solutions. The quantities ϵr and μr are the real parts of dielectric constant and magnetic permeability.

2) Time-averaged power dissipation integrated over the over the solution space and region volumes, $P = (\omega/2)\,(\epsilon_i\, E^2 + \mu_i\, H^2)$. The results are non-zero only for lossy materials. The output units are W/m in planar solutions and W in cylindrical solutions. The quantities ϵi and μi are the imaginary parts of dielectric constant and magnetic permeability.

3) The location and value of the peak electric field (**E**) and the magnetic intensity (**H**) within the current view.

Line integrals

WSV also performs integrals of the quantity $\int\int dA\, H_{\parallel}^2/2$ over the surfaces represented by any line regions. Here, $H_{\parallel}$ is the component of the magnetic intensity parallel to the boundary. Multiplication of this quantity by a surface resistivity yields an estimate of time-averaged power dissipation in a metal wall because of skin depth effects [see, for example, S. Ramo, J. Whinnery and T. Van Duzer, **Fields and Waves in Communication Electronics** (Wiley, New York, 1965),

Sect. 5.15]. To control the output, you can define special dummy line regions to represent metal walls during mesh generation. These lines do not affect the solution if they are not assigned physical properties in **WaveSim**.

Region properties

In response to the *Region properties* command, **WSV** calculates volume integrals of field energy density and power density over the chosen region.

14. WSV 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 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 **WSV**. This command closes the scan plot and returns the program to the previous spatial plot.

15. WSV script file commands

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

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

`..\WSV 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 TM11.WOU

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 `TM11.POU`. You can load several files for sequential analysis.

Output TLINE

Close the current history file and open an output file `TLINE.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 DIFFRACT01.WOU
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT DIFFRACT02.WOU
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT DIFFRACT03.WOU
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT DIFFRACT04.WOU
SCAN 0.00 -50.00 0.00 50.00
ENDFILE
```

16. Format of the WaveSim output file

The **WaveSim** output file `FPrefix.WOU` 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 11 data lines:

```
--- Run parameters ---
XMin:  0.000000E+00
```

```
          XMax:   5.000000E-02
          KMax:   101
          YMin:   0.000000E+00
          YMax:   5.000000E-02
          LMax:   101
          DUnit:   1.000000E+02
          NReg:      4
          ICylin:   1
          Frequency:   1.484642E+09
          Type:   H
```

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). Line 11 lists the frequency in Hz and line 12 gives the mode type (*E* or *H*).

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            FReal         FImag
==========================================================================
   1    1    4    1    0   0.000000E+00  0.000000E+00  1.465509E-11  0.000000E+00
   2    1    3    1    0   6.093723E-04  0.000000E+00  0.000000E+00  0.000000E+00
   3    1    3    1    0   1.107950E-03  0.000000E+00  0.000000E+00  0.000000E+00
   4    1    3    1    0   1.606621E-03  0.000000E+00  0.000000E+00  0.000000E+00
   5    1    3    1    0   2.105418E-03  0.000000E+00  0.000000E+00  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 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 real and imaginary parts of the primary field component: E_z (V/m), rE_θ (V), H_z (A/m) or rH_θ (A).

The region section consists of four title lines following by *NReg* data

45

lines, one for each region. The region section has the following appearance:

```
--- Regions ---
RegNo Fix Void                          EpsiReal       EpsiImag

    1    0    0    0    0    0  8.854188E-12  0.000000E+00
    2    0    0    0    0    0  8.854188E-12  0.000000E+00
...

        MuReal          MuImag          AReal           AImag

    1.256637E-06  0.000000E+00  0.000000E+00  0.000000E+00
    1.256637E-06  0.000000E+00  0.000000E+00  0.000000E+00
...
        JReal           JImag          JAngle           Area

    0.000000E+00  0.000000E+00  0.000000E+00  2.015530E-03
    1.000000E+00  0.000000E+00  0.000000E+00  3.999995E-06
```

An entry of unity in the second column designates a fixed-field region. The ten real number parameters represent the following quantities:

- Real and imaginary parts of the permittivity (F/m)

- Real and imaginary parts of the permeability (H/m)

- Real and imaginary parts of primary field component in a fixed region

- Real and imaginary parts of the current density magnitude in a drive region (A/m^2 or A/m)

- Current density direction for a drive region, H-type solution (radians)
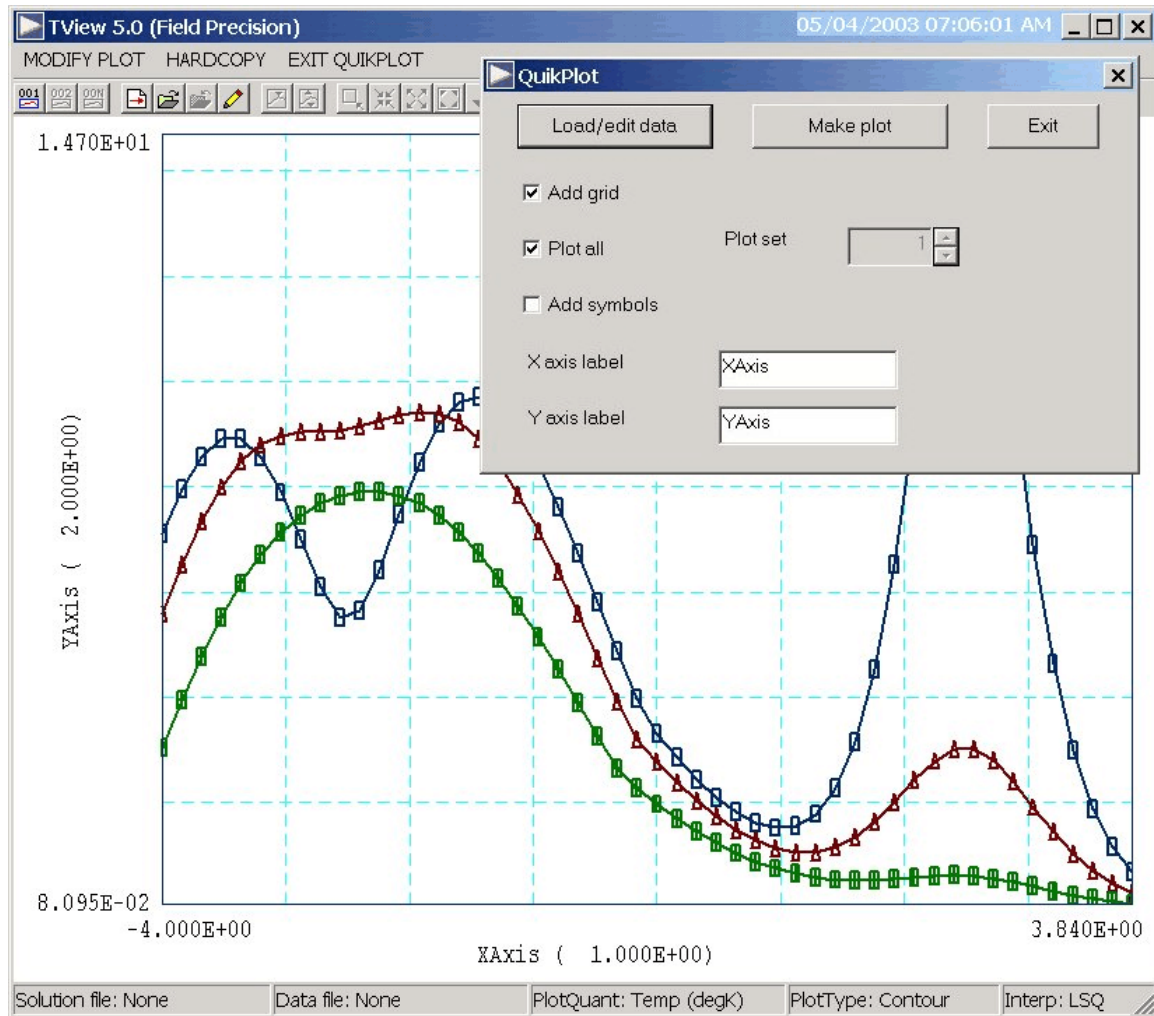
- Cross-section area of the region (m^2)

Figure 10. Control dialog superimposed on **QuikPlot** screen.

17. 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 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 **WSV**.