



EMP

Finite-element Time-domain Electromagnetics

Field Precision

Copyright 2002

PO Box 13595

Albuquerque, New Mexico 87192 U.S.A.

Telephone: 505-220-3975

FAX: 505-294-0222

E Mail: techinfo@fieldp.com

Internet: <http://www.fieldp.com>

EMP 5.0 Manual - Contents

- Section 1. Program function
- Section 2. Walkthrough example
- Section 3. Two-dimensional electromagnetic waves
- Section 4. Courant stability condition
- Section 5. Current sources
- Section 6. Standard boundary conditions
- Section 7. Termination layer absorbing boundaries
- Section 8. Organizing runs
- Section 9. Structure of the command file
- Section 10. Program control commands
- Section 11. Region commands
- Section 12. Diagnostic commands
- Section 13. Tabular functions
- Section 14. Running EMP interactively
- Section 15. Running EMP from the command prompt
- Section 16. VEMP - file menu
- Section 17. VEMP - spatial plot menu
- Section 18. VEMP - analysis menu
- Section 19. VEMP - scan plots
- Section 20. VEMP - script operation
- Section 21. EMP output file format
- Section 22. Solution examples
- Section 23. Simulation techniques for transmission lines and pulsed-power switch assemblies
- Section 24. QuikPlot utility

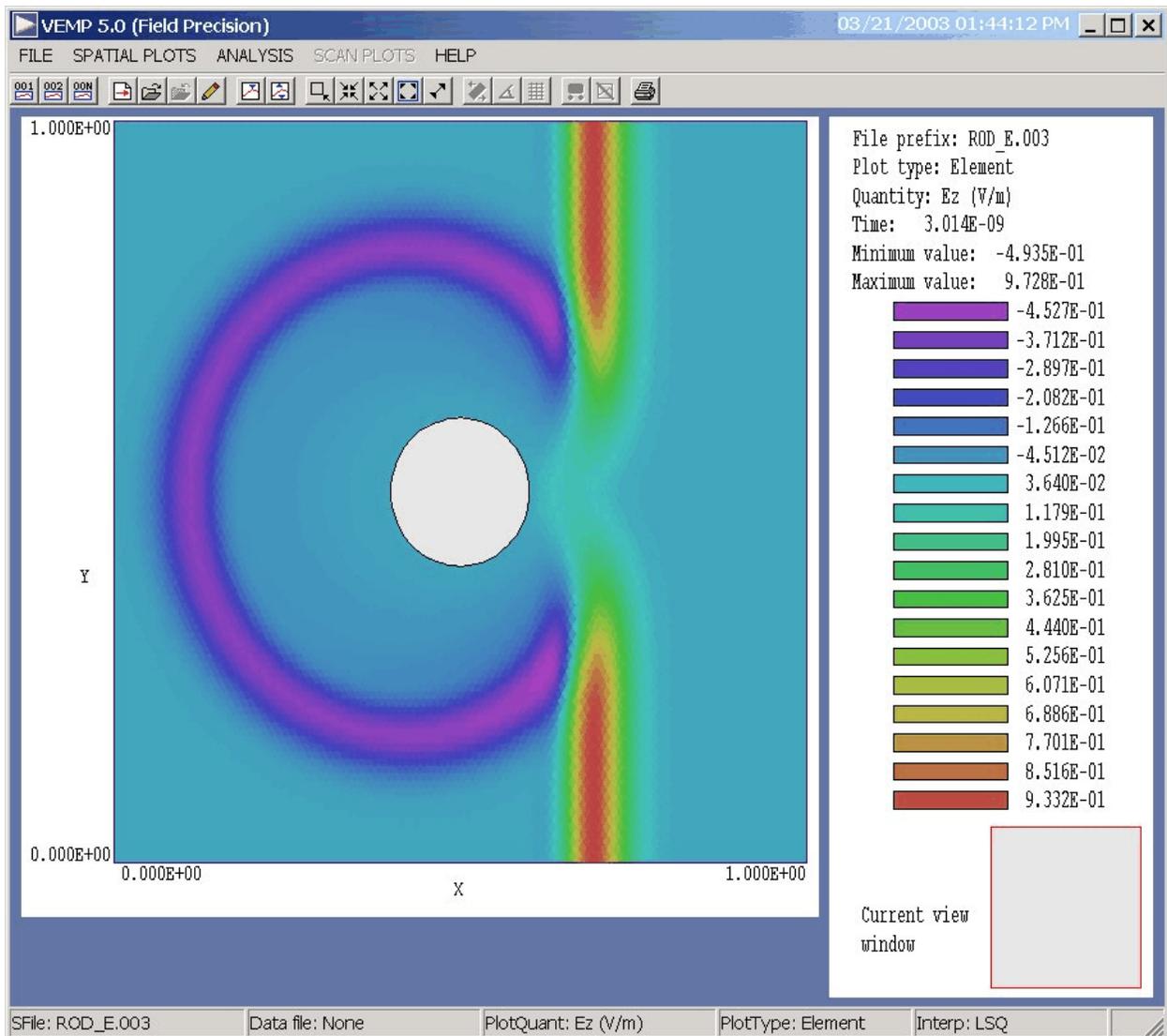


Figure 1. Screen shot of the VEMP post-processor.

1. Program function

EMP performs time-domain electromagnetic analyses on personal computers. The program follows propagating electromagnetic pulses in two-dimensional planar geometries and three-dimensional cylindrical systems. Applications include simulation of microwave devices, particle accelerators, pulsed-power generators and electromagnetic interference. **EMP** uses unique finite-element methods on conformal triangular meshes for accurate representations of curved and sloped material boundaries. You can generate pulses by specifying time-dependent fields on surfaces or by defining source current regions (equivalent to drive current loops or capacitive probes). The program can represent arbitrary time variations of

multiple sources through flexible tables with linear or cubic spline interpolations. **EMP** also has built-in harmonic sources for radiation scattering problems. Material types include vacuum, conductors, or lossy dielectrics and ferrites. In addition to the standard short circuit and open circuit boundary conditions, **EMP** supports matched termination absorbing boundaries to simulate open-space conditions. With this approach is also possible to assign general impedances (reactive and resistive) on boundaries.

EMP automatically picks time steps consistent with the Courant stability condition. Run times are short, typically a few seconds on a high-performance computer. The program records spatial distributions of fields at specified times or temporal variations at up to 20 probe locations. All data files are in structured ASCII format for easy transfer to spreadsheets or mathematical analysis programs. Spatial data can be analyzed quickly with the **VEMP** interactive post-processor (Fig. 1). This utility generates quantitative data and a wide variety of plots. The Field Precision utility program **PROBE** is supplied for the analysis of temporal information.

With care in preparation, **EMP** can quickly provide accurate results of high accuracy. To illustrate, Fig. 2 shows a comparison of **EMP** results for a pulsed dipole antenna to the published results of J.G. Maloney, G.S. Smith and W.R. Scott, IEEE Trans. Antennas Propagation **AP-38**, 1059 (1990).

2. Walkthrough example – radiation pulse from a conical antenna

It is useful to step through an example to understand the capabilities of **EMP**. Move the following files (supplied with the example library) to a working directory such as `\tricom\buffer: CONICAL.MIN`, `CONICAL.EIN` and `SQPULSE.SRC`. The first file, input to the **Mesh** program, describes the geometry. The second file, `CONICAL.EIN`, controls the **EMP** run. The file `SQPULSE.SRC` describes the pulse waveform. All files are in ASCII format so they can be inspected and modified with a text editor. Figure 3 shows a z - r plot of the cylindrically symmetric system. (Note that parts are figures-of-revolution about the z axis). The current source layer (bottom left-hand side) creates a TEM pulse in the 70Ω , air-filled coaxial transmission line ($r_o = 2.5$ cm, $r_i = 0.78$ cm). The line drives a conical antenna that produces a radiating pulse in the approximately spherical air volume. A matched absorbing layer on the periphery simulates the open-space condition.

The strategy for generating the mesh is influenced by the nature of the electromagnetic pulse solution. Section 3 reviews the theory of

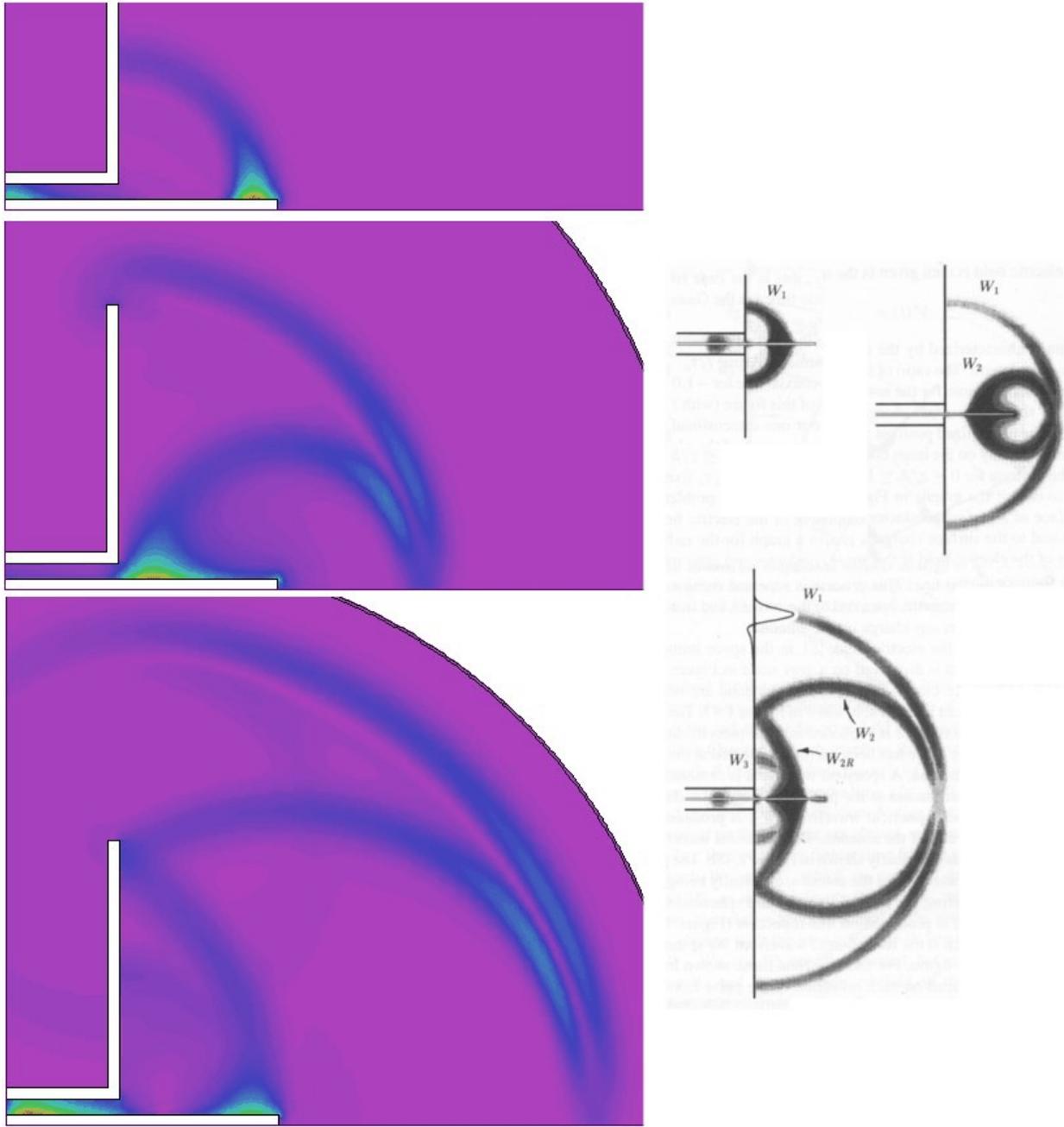


Figure 2. Benchmark test – radiation from a pulsed dipole antenna. Left-side: EMP results, r-z plots of $|\mathbf{E}|$ at 1 ns. Right-hand-side: cross-section plots of $|\mathbf{E}|$ at approximately the same times from J.G. Maloney, G.S. Smith and W.R. Scott, IEEE Trans. Antennas Propagation **AP-38**, 1059 (1990)

electromagnetic waves in two-dimensional systems. A TEM pulse in a

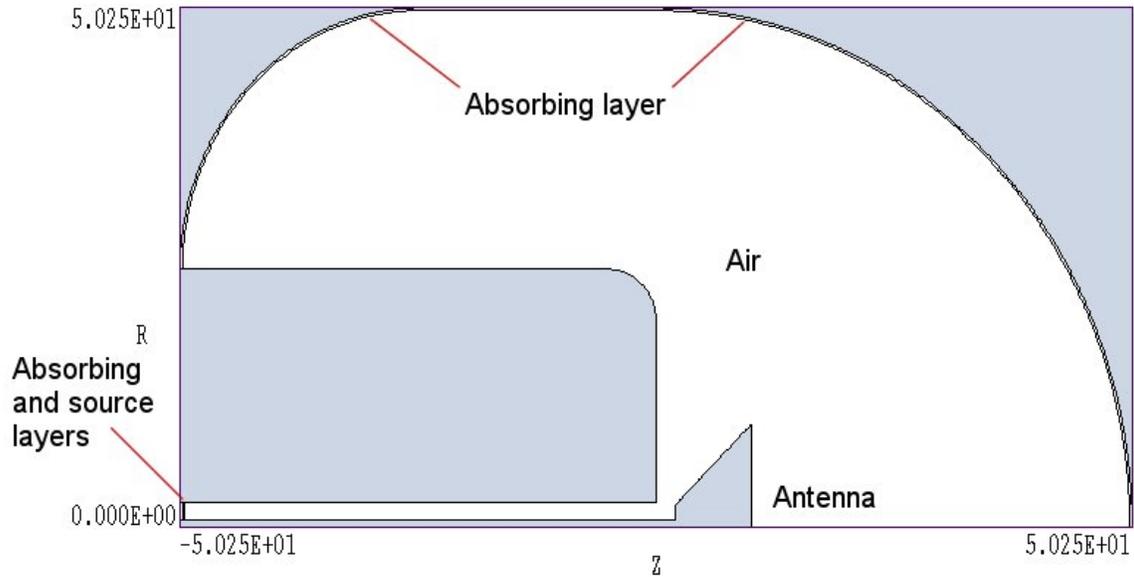


Figure 3. Geometry for the CONICAL example. A pulse on a transmission line in a finite-width ground plane drives a conical antenna. Dimensions in cm.

coaxial transmission line is an H type solution. **EMP** calculates the quantity rH_θ from a finite-element solution of the Helmholtz equation and then determines other field components (E_z , E_r) in terms of derivatives of the primary component. In particular the condition $\partial(rH_\theta)/\partial n = 0$ holds on the surface of a perfect conductor. The Neumann condition automatically applies to all unspecified solution boundaries. We therefore use the following strategy for mesh generation. We first fill in the entire solution volume with a resistive medium (Region 1) that will comprise the absorption layers in the transmission line and propagation volume. We exclude the inner and outer conductors of the transmission line from the solution volume. Note that the blue areas in Fig. 3 are not part of the solution volume. We next carve out the inside of the solution volume with Region 2 representing air. This process leaves a absorbing layer of uniform thickness 0.25 cm around the propagation volume and also at the end of the transmission line. The latter region absorbs any reflected energy traveling backward in the line. Run **Mesh** and follow the standard procedures to create the output file CONICAL.MOU. The geometry file will be used as input to **EMP**.

The next step is to run EMP to carry out the electromagnetic solution and to create several data files. Table 1 shows the complete input script CONICAL.EIN with added line numbers. Lines 01-04 control the program operation. Lines 05-16 set material properties for regions of the solution space. Region 2 (air) has the properties $\epsilon_r = 1.0$, $\mu_r = 1.0$. Region 2 represents the absorbing layers adjacent to air. Following the discussion of Sect. 6, the conductivity of 1.060 S/m is given by $\sigma = 1/\eta\Delta z$, where η is the characteristic impedance of air (377.3Ω) and $\Delta z = 2.5 \times 10^{-3}$ m is the

Table 1. Control script CONICAL.EIN

```

01: SET TMax 8.01E-9
02: SET Geometry Cylin
03: SET DUnit 100.0
04: SET Solution H
05: * Resistive layers
06: REGION 1 Epsi 1.0
07: REGION 1 Mu 1.0
08: REGION 1 Cond 1.060
09: * Vacuum
10: REGION 2 Epsi 1.0
11: REGION 2 Mu 1.0
12: * Source
13: REGION 3 Epsi 1.0
14: REGION 3 Mu 1.0
15: REGION 3 Source SQPULSE.SRC 0.20E-9 127.3
16: REGION 3 Beta 90.0
17: * Axis
18: REGION 4 Fixed
19: DIAG DTime 2.0E-9
20: DIAG History -40.0 1.50
21: DIAG History 30.0 30.0
22: ENDFILE

```

thickness of the layer. Region 3 contains a current source to generate a 1 A pulse in the $70\ \Omega$ line. Line 15 specifies the time variation of the source. The entries in the file SQPULSE.SRC specify a normalized waveform that rises from 0.0 to 1.0 in a time 1.0, remains constant from 1.0 to 4.0, and returns to 0.0 at 5.0. The first real number parameter following the file name is a conversion factor for the time values to give a pulse length of 1 ns. The second value is a multiplication factor such that the quantity rj_r in the current layer corresponds to a total current of 2.0 A:

$$r j_r = \frac{2I_o}{2\pi\Delta z} = 127.3 \text{ A/m.} \quad (2.1)$$

In the equation j_r is the current density in the layer of thickness $\Delta z = 2.5 \times 10^{-3}$ m. We specify 2.0 A because half will travel down the transmission line and half is lost to the adjacent matched resistive layer. Finally Region 4.0, a line along the z-axis is set to the condition $rH_\theta = 0$ (a requirement in cylindrical H type solutions). The final commands (Lines 19-21) control

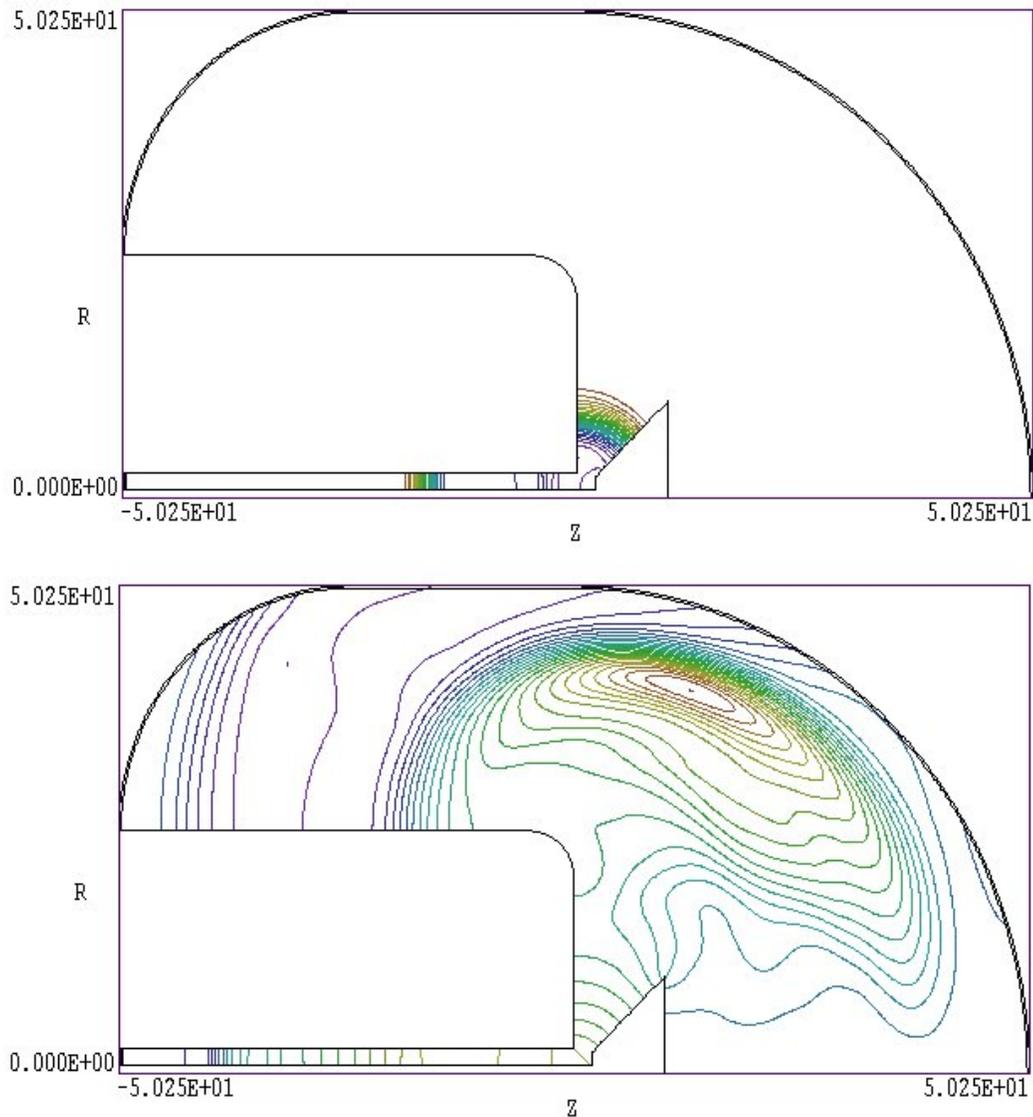


Figure 4. Electric field lines in the CONICAL example at 2 ns (top) and 4 ns (bottom).

the recording of data. Line 19 specifies that the program will create four data dump files (CONICAL.001, CONICAL.002, CONICAL.003 and CONICAL.004) at an interval of approximately 2 ns. A data dump is a complete record in space at a specified time (snapshot). Lines 20 and 21 set two probes that record complete temporal information at a specified position. The first probe is located near the upstream end of the transmission line ($z = -40.19$ cm) at a radius of $r = 1.515$ cm. The second probe is located in the propagation region 42 cm from the antenna ($z = 29.8$, $r = 30$.cm). Information from these probes is recorded in the files CONICAL.P01 and CONICAL.P02.

Start **EMP** and choose the *RUN/START RUN* command. Transfer to the working directory if necessary and pick the input file *CONICAL.EIN*. The program takes less than a minute to complete the solution. Next run **VEMP** and click on the command *FILE/LOAD FIRST SOLUTION FILE*. In the dialog pick *CONICAL.001*. After data loading you should see the plot shown on the top of Fig. 4. The plot shows electric field lines (contours of rH_0) at $t = 2.0$ ns. The leading edge of the pulse propagates through the antenna region. You can also see the trailing edge of the pulse in the transmission line and energy reflected from the antenna mismatch moving backward. The electric field distribution at $t = 4.0$ ns is shown on the bottom of Fig. 4. The launched electromagnetic pulse impinging on the absorbing layer is visible as well as energy moving upstream from the ground plane.

Signals from the two probes give quantitative data on field variations. Run the **Probe** program (supplied with the **EMP** package) and click on the command *LOAD PROBE FILE*. After you pick *CONICAL.PO1* the program displays information about the data file. Click *OK* to continue. Pick the command *PLOT PARAMETERS/PICK PLOT QUANTITY* and choose E_r . You should see the plot on the top of Fig. 5. For comparison, the theoretical electric field of a 1.0 A pulse at a radius of 1.51 cm in a 70Ω transmission line is about 4000 V/m. The plot shows the incident pulse, a small negative reflection, and then a positive reflection from energy that is not radiated at the open-circuit antenna termination. The initial negative reflected pulse occurs when the incident pulse meets the antenna which acts like a conical transmission line. The characteristic impedance of the conical line is:

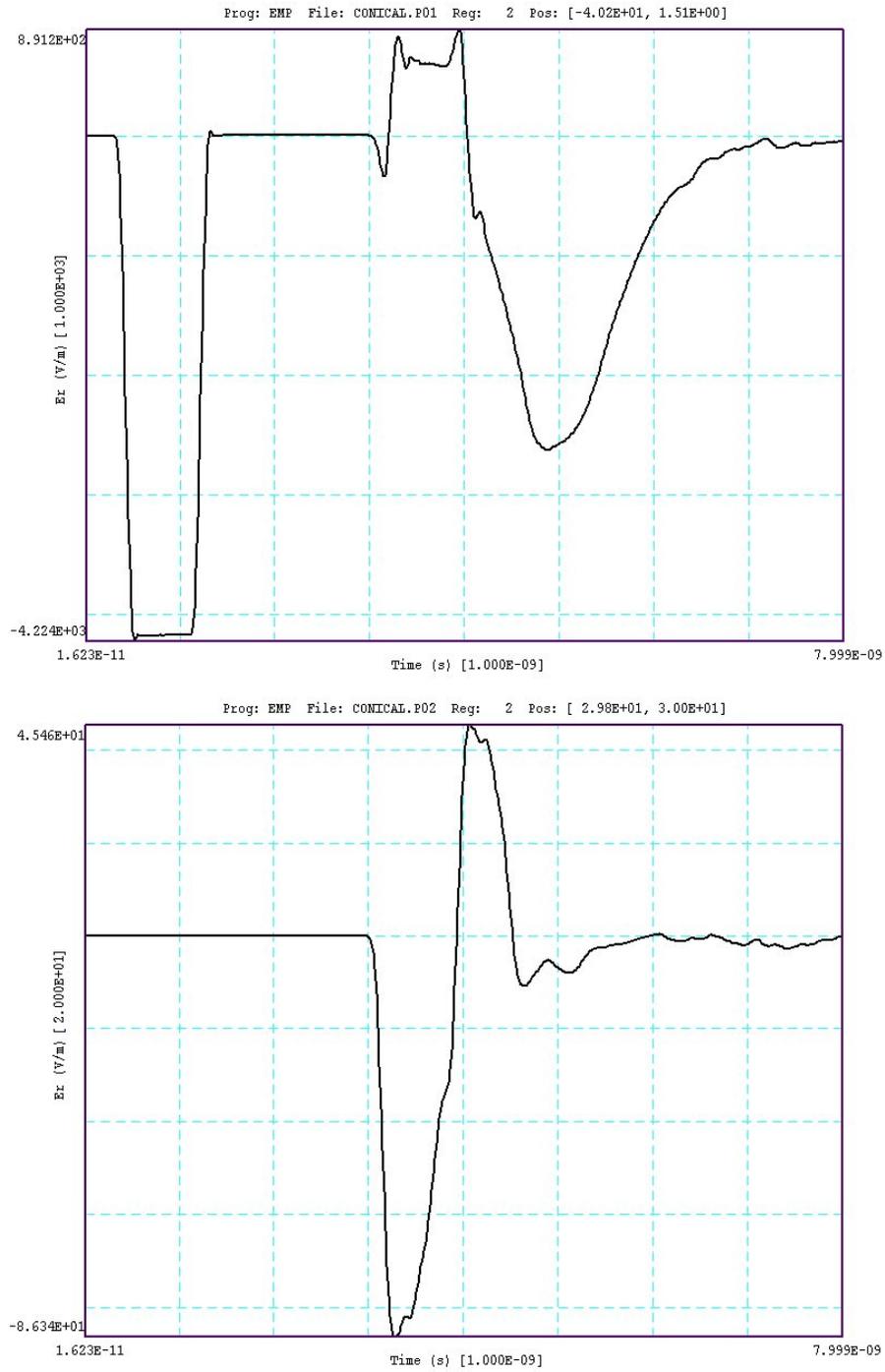


Figure 5. Radial electric field as a function of time for the CONICAL example in the transmission line (top) and in free space (bottom).

$$Z_o = \sqrt{\frac{\mu}{\epsilon} \frac{\tan(\alpha)}{2\pi}} . \quad (2.2)$$

Note that the impedance of the antenna in the example will be lower than this value because the apex is upstream from the ground plane. For $\alpha = 45^\circ$ we find $Z_o < 60 \Omega$ giving a reflection coefficient of $\rho < -0.08$. The bottom of Fig. 5 shows the radial electric field of the radiated pulse at probe 2.

3. Two-dimensional electro-magnetic waves

This section reviews the theory underlying **EMP**. The program applies the following form of the Maxwell equations:

$$\nabla \times \mathbf{H} = \epsilon \frac{\partial \mathbf{E}}{\partial t} + \mathbf{j}_o + \sigma \mathbf{E} , \quad (3.1)$$

$$\nabla \cdot \mu \mathbf{H} = 0 , \quad (3.2)$$

$$\nabla \times \mathbf{E} = - \mu \frac{\partial \mathbf{H}}{\partial t} , \quad (3.3)$$

$$\nabla \cdot \epsilon \mathbf{E} = 0 . \quad (3.4)$$

The magnetic field intensity is related to the magnetic flux density by $\mathbf{H} = \mathbf{B}/\mu$. We assume that the solution volume has no free space-charge. Time-dependent source current densities (\mathbf{j}_o) may represent drive structures like coupling loops and capacitive probes. Dielectrics and ferromagnetic materials are isotropic and linear. Furthermore, the magnetic permeability μ and dielectric constant ϵ are constant in time and uniform over the volume of small elements. The equations include the effects of currents driven by electric fields in materials with conductivity σ . This feature enables modeling of pulse attenuation and implementation of ideal absorbing boundaries. The conductivity is assumed independent of frequency over the range of interest.

Equations (3.1)-(3.4) can be combined into individual equations for \mathbf{E} and \mathbf{H} . The electric field equation is

$$-\nabla \times \frac{1}{\mu} \nabla \times \mathbf{E} = \frac{\partial}{\partial t} (\nabla \times \mathbf{H}) = \epsilon \frac{\partial^2 \mathbf{E}}{\partial t^2} + \frac{\partial \mathbf{j}_o}{\partial t} + \sigma \frac{\partial \mathbf{E}}{\partial t} . \quad (3.5)$$

Using Stoke's theorem, we can rewrite Eq. (3.5) as

$$-\oint \frac{1}{\mu} \nabla \times \mathbf{E} = \iint dS \left[\epsilon \frac{\partial^2 \mathbf{E}}{\partial t^2} + \frac{\partial \mathbf{j}_o}{\partial t} + \sigma \frac{\partial \mathbf{E}}{\partial t} \right] . \quad (3.6)$$

The surface integral on the right hand side extends over the region bounded by the circital integral.

A planar system has geometric variations in the x and y directions and is uniform along z . The pulse propagation vector lies in the x - y plane. We can divide pulses into two types.

- E type - electric field polarization in the z direction, E_z .
- H type - magnetic intensity polarization in the z direction, H_z .

Equation 3.6 becomes a scalar equation for E type pulses. Such pulses are created by time-varying current density in the axial direction, j_z .

A finite-element field solution requires the conversion of Eq. 3.6 to a large number of coupled linear difference equations. A linear equation can be generated for each node in the solution space by taking surface and line integrals in the x - y plane over the region shown in Fig. 6 (blue line). The right-hand side of Eq. 3.6 is converted using time-centered difference operators. The complete theory is described in S. Humphries, Jr., **Field Solutions on Computers** (CRC Press, Boca Raton, 1997), Chaps. 13 and 14. The solution procedure involves the simultaneous advance of all equations in small time steps Δt to determine new values of E_z at all nodes. It is important to recognize that E_z values are defined at nodes (vertices) while the quantities ϵ , μ , σ and j_z are characteristic of elements.

Knowing E_z at all positions and times, we can find the components of magnetic intensity within the elements by applying numerical differentiation in space and integration in time,

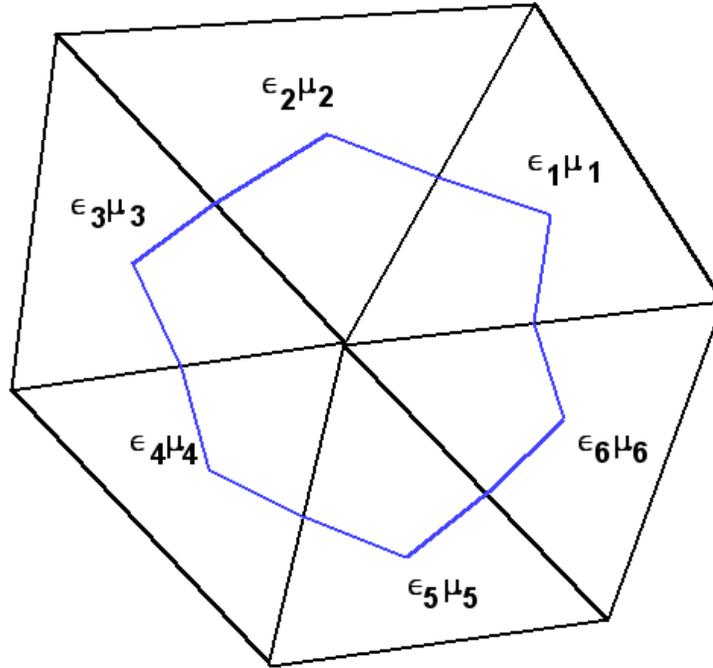


Figure 6. Line integral path through elements surrounding a node.

$$H_x = -\frac{1}{\mu} \int_0^t dt' \frac{\partial E_z}{\partial y},$$

$$H_y = \frac{1}{\mu} \int_0^t dt' \frac{\partial E_z}{\partial x}.$$

(3.7)

Equation 7 implies that lines of \mathbf{H} in the x - y plane lie on contours of E_z . This fact is used to create field line plots in **VEMP**. The equation for H type pulses in planar geometry is similar to Eq. 3.6. Here the numerical solution is carried out for H_z and the subsidiary quantities E_x and E_y are determined from the following equations:

$$\begin{aligned}
E_x &= \frac{1}{\epsilon} \left[\int_0^t dt' \frac{\partial H_z}{\partial y} + \int_0^t dt' j_{ox} + \sigma \int_0^t E_x \right], \\
E_y &= \frac{1}{\epsilon} \left[- \int_0^t dt' \frac{\partial H_z}{\partial x} + \int_0^t dt' j_{oy} + \sigma \int_0^t E_y \right].
\end{aligned}
\tag{3.8}$$

In this case the electric field lines follow contours of H_z only in regions with no source current and zero conductivity.

Table 2 lists the main and subsidiary field quantities for the four types of solutions in **EMP**. As an example, an H type solution in cylindrical geometry could describe a TEM wave propagating along a coaxial transmission line. Because of the nature of the curl operator, **EMP** solves for the quantities rE_θ and rH_θ in cylindrical quantities. It is easy to show that contours of these quantities are parallel to lines of \mathbf{H} and \mathbf{E} respectively in the r - z plane. For H type pulses, the correspondence holds only in regions with zero conductivity and source current.

From the above discussion, we can understand some limitations on the available data for H type solutions. A data dump at a specified time contains node values of H_z and $\int H_z dt$ at the specified time. Therefore, the post-processor can calculate E_x and E_y only in regions with no source current and zero conductivity. The limitation also applies to calculations of total field energy and resistive power dissipation. Because it is practical to maintain integrals of the components of current density at a limited number of locations, the probe files contain correct values of electric field components in source regions.

Table 2. Solution Types in EMP

Type	Geometry	Main field component	Subsidiary field components
<i>E</i>	Planar, variations in x and y	E_z	H_x, H_y
	Cylindrical, variations in r and z	rE_θ	H_r, H_z
<i>H</i>	Planar, variations in x and y	H_z	E_x, E_y
	Cylindrical, variations in r and z	rH_θ	E_r, E_z

4. Courant stability condition and numerical dispersion

The choice of Δt in a time-domain electromagnetic solution is important. For numerical stability, it must satisfy the *Courant condition*,

$$\Delta t < \min [\sqrt{\epsilon_i \mu_i} l_i] = \min \left[\frac{l_i}{v_i} \right]. \quad (4.1)$$

The quantity v_i is the speed of light in element i and l_i is the minimum dimension of the element. Equation 4.1 states that Δt must be shorter than the time for an electromagnetic disturbance to propagate across the smallest element dimension. If the Courant condition is violated, information can propagate through the mesh faster than the speed of light, violating the principle of causality and leading to a numerical instability. Before beginning a calculation, **EMP** scans all elements and takes l_i as the minimum distance from the element center-of-mass to a vertex. This gives a conservative value for the time step. You can over-ride the choice, but be aware that the choice could lead to an instability that crashes the program.

It is also important to recognize the possibility of *numerical dispersion*. Numerical simulations approximate continuous physical systems through discrete equations. We expect to find some imperfections in the model. As an example, Fig. 7a illustrates the long-distance propagation of a Gaussian pulse. A one-dimensional pulse with spatial width 0.067 m travels 9 m. Although energy conservation is almost ideal, the pulse shape is noticeably distorted. The problem arises because the Gaussian pulse contains a spectrum of traveling Fourier modes. The pulse shape will be preserved only if all modes have the same phase velocity. In the discrete approximation there are small shifts in phase velocity for modes with wavelength comparable to the element size. Therefore, we expect large distortions when the pulse width approaches the element size. Figure 7b illustrates declining fidelity with increasing element size for a pulse width of $w_p = 0.067$ m and 2.5 m propagation distance. The three cases correspond to different values of the ratio of pulse width to element width. Spatial displacements have been added for visibility. As a general rule, dispersion is small if the distance of propagation is less than or comparable to $2w_p^2/w_e$, where w_e is the element width.

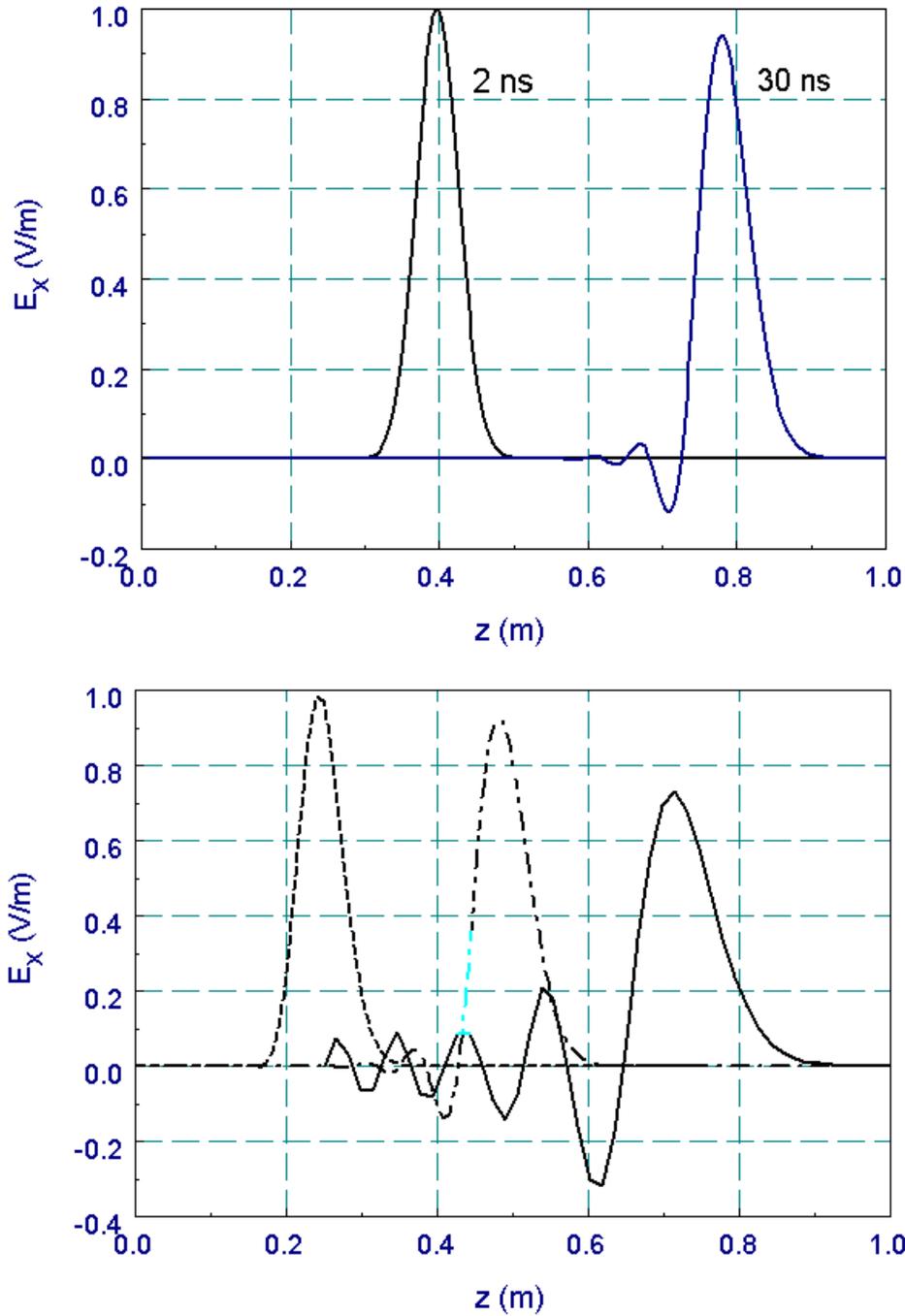


Figure 7. Numerical dispersion in time-domain electromagnetics. Gaussian pulse in vacuum, FWHM = 0.067 m. Top: Pulse shapes after 0.6 and 9.0 m propagation for an element width of 0.004 m. Bottom: Pulse shapes after 2.5 m of propagation as a function of element width. Ratio of pulse width to element width: 16.6 (dotted), 8.3 (dashed) and 4.2 (solid).

5. Current sources

Source current density can be assigned to any material region in the solution space. **EMP** assumes uniform spatial distribution and a time dependence given by either a harmonic function or an arbitrary variation calculated from a numerical table. We have already seen that current densities of the form j_z in a planar geometry generate E type pulses. Here, lines of magnetic intensity in the x - y plane circulate around current carrying regions. The regions act like current loops that extend an infinite distance in z . Currents in the x - y plane generate H type pulses. They create field components E_x and E_y through displacement currents and act as capacitive drivers. For H type solutions we must specify an additional parameter for a source region, the direction of current density in the x - y plane. In **EMP** the quantity β is defined as the angle of the current density relative to the x -axis.

In cylindrical coordinates, an azimuthal current density j_θ drives E type pulses and current in the r - z plane drives H type pulses. In the latter case, the reference parameter β is the angle of the current density relative to the z -axis. Instead of taking a constant value of β in a region, the convention in **EMP** is to use a constant value of the quantity $\Gamma = r\mathbf{j}$. This choice ensures that 1) real current is conserved for radial flow in a region and 2) a source region generates an ideal TEM wave in a uniform coaxial geometry. As an example, suppose we have a source region for H pulses with axial length Δz and a current flux $\Gamma = \Gamma_o \mathbf{r}$. The total radial current in the region is $I_r = 2\pi\Gamma_o\Delta z$. Similarly, the axial current through an annular region with inner and outer radii r_i and r_o with an axial flux $\Gamma = \Gamma_o \mathbf{z}$ equals $I_z = 2\pi\Gamma_o(r_o - r_i)$.

6. Standard boundary conditions

In the finite-element treatment of **EMP** an unspecified boundary automatically satisfies the special Neumann condition. Here, the normal derivative of the calculated quantity (E_z , rE_θ , H_z or rH_θ) is zero. For E type pulses, the condition is equivalent to an *open-circuit* termination on the boundary. In this case, the magnetic intensity \mathbf{H} is normal to the surface and the surface current density equals zero. A pulse is reflected from the boundary with the same polarity and no change in amplitude. The condition is equivalent to a mirror condition for a symmetric system (equal values of E_z or rE_θ on both sides of the boundary). In contrast, a Neumann boundary is equivalent to a short-circuit condition (perfect conductor) for H type pulses. In this case, the electric field is normal to the surface. The outer boundaries of the solution volume are automatically set to the specialized Neumann condition. You can set internal Neumann boundaries using the *Region Void* command.

Alternatively, an internal region approximates the open-circuit property for E type pulses if it has a very high value of characteristic impedance:

$$Z/Z_o = \sqrt{\frac{\mu_r}{\epsilon_r}} \gg 1.0. \quad (6.1)$$

In setting open-circuit material properties, pick values of the relative dielectric constant and magnetic permeability such that

$$\sqrt{\epsilon_r \mu_r} \approx 1.0. \quad (6.2)$$

Equation (6.2) ensures that the speed of light in the medium is close to c . If regions have large differences in the electromagnetic propagation velocity it may be impossible to find a practical time step consistent with the Courant condition (Sect. 3). As an example, the values $\epsilon_r = 10^{-6}$ and $\mu_r = 10^6$ are a valid choice for a high-impedance material.

The special Dirichlet condition is a second option for external or internal boundaries. This type of boundary is defined by the *Region Fixed* command. In this case the calculated quantity (E_z , H_z , rE_θ or rH_θ) has the fixed value of 0.0. The special Dirichlet condition is equivalent to a *short-circuit* boundary (ideal conductor) for E type solutions. Here a pulse is reflected with inverted polarity. For H type pulses, a Dirichlet boundary is equivalent to an open-circuit condition (electric field vector parallel to the surface).

7. Termination layer absorbing boundaries

We can simulate free-space conditions in **EMP** by placing a resistive termination layer next to a boundary. The idea is similar to a matched termination at the end of a transmission line (equivalent to an infinite extension of the line). The region beyond the termination should satisfy the special Neumann condition for both E and H type of pulses. In most cases the termination layer is located on the outer surface of the solution volume so that the Neumann condition is automatically applied. As shown in Fig. 8a, a thin resistive layer with conductivity σ and width Δ is located between the boundary material and a material with electrical properties ϵ and μ . In the limit that Δ is much smaller than the spatial width of the

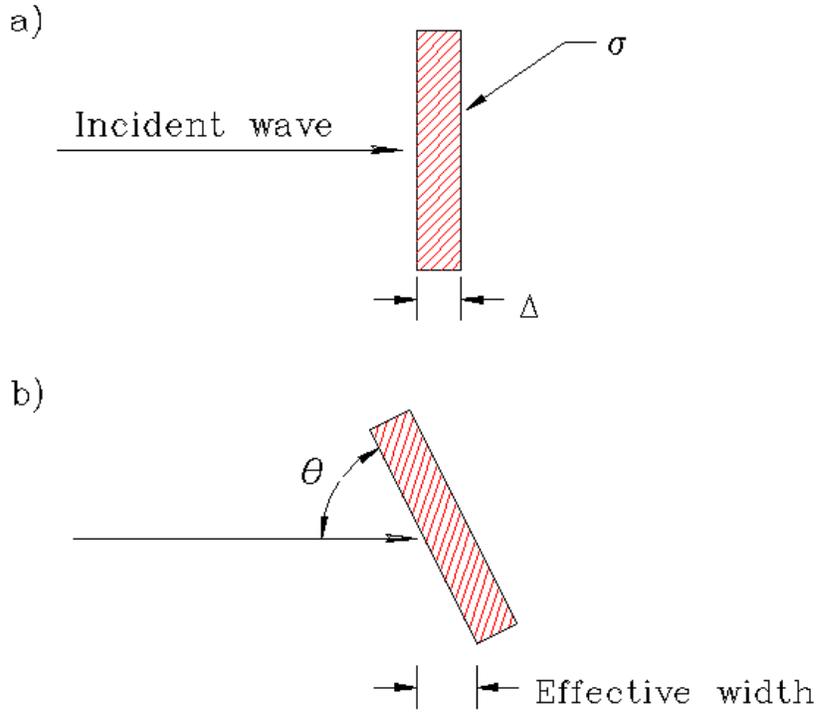


Figure 8. Absorbing boundary for two-dimensional solutions. *a)* Normal incidence. *b)* Incidence at an angle.

pulse, the electric field is approximately uniformly over the layer. If the resistor maintains the same conditions on \mathbf{E} and \mathbf{H} as an infinite extension of the material, there is no reflected wave from the boundary. We can derive the required value of σ by noting that a field E_x creates a linear current density equal to

$$J_x = \sigma \Delta E_x . \quad (7.1)$$

The quantity $1/\sigma\Delta$ has units of ohms. Assuming zero magnetic field on the right-hand side of the resistor, the value of magnetic intensity of the left-hand side is $H_y = J_x$. Noting that the ratio E_x/H_y should equal the characteristic impedance of the medium, the matching condition is

$$Z_o = \sqrt{\frac{\mu}{\epsilon}} = \frac{1}{\sigma \Delta} . \quad (7.2)$$

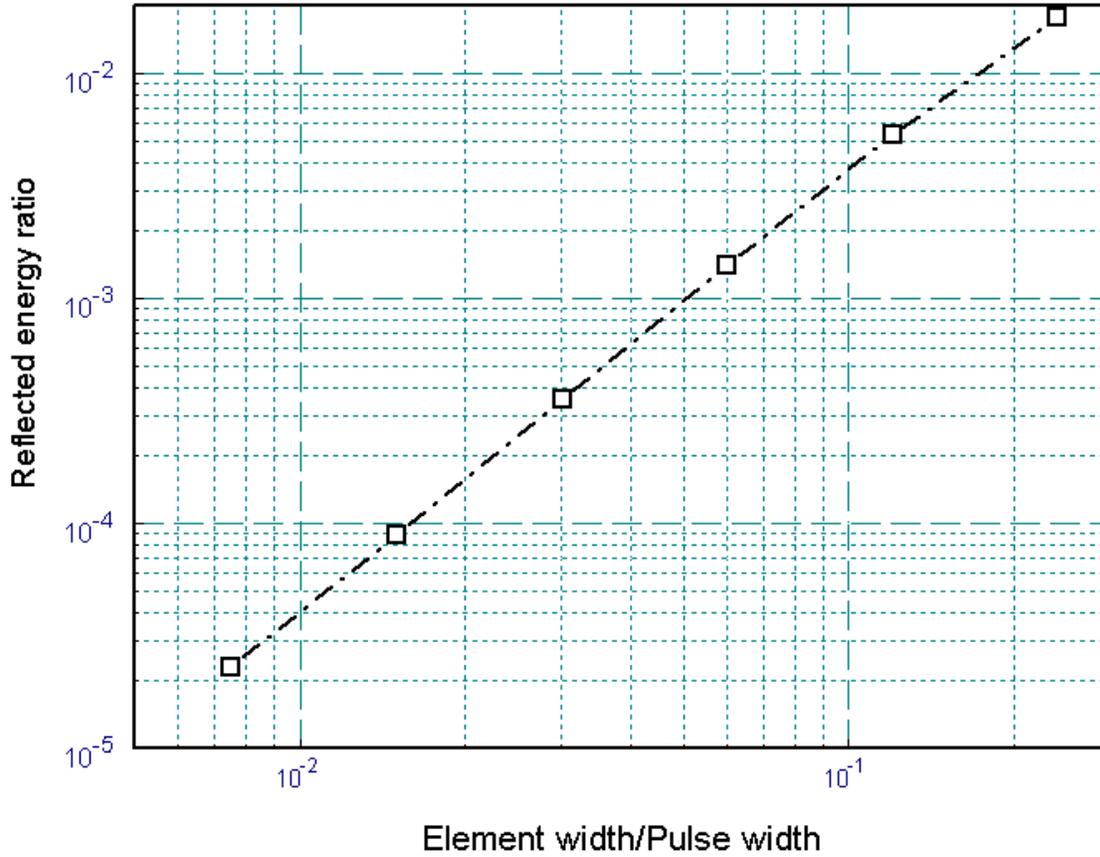


Figure 9. Performance of a matched termination boundary layer. Relative reflected energy as a function of the ratio of layer thickness to pulse spatial width (Gaussian pulse).

The conductivity for a matched termination is

$$\sigma = \frac{1}{\Delta \sqrt{\mu/\epsilon}}, \quad (7.3)$$

where ϵ and μ are the electrical properties of both the termination element and the adjacent material.

Figure 9 shows the performance of a termination boundary for normally incident pulses. A Gaussian pulse of width 0.067 m enters from a vacuum region. The figure plots the reflected pulse amplitude as a function of the termination element width Δ . Absorption is better for thin elements that better approximate a lumped element. On the other hand, there is a penalty to using very thin elements because the Courant condition depends on the

minimum width. As an example, for the baseline element width of 0.004 m, the termination element absorbs 99.86 per cent of the pulse energy.

The absorber has reduced effectiveness when pulses strike the boundaries at angles less than 90° (Fig. 8b). For an angle θ the effective layer thickness increases to $\Delta/\sin\theta$, reducing the surface resistance by a factor of $\sin\theta$. In this case, the termination is under-matched. The predicted reflection coefficient is

$$\frac{E_r}{E_o} = - \frac{1 - \sin\theta}{1 + \sin\theta} . \quad (7.4)$$

The degradation in performance is not catastrophic. A matched layer absorbs more than 90 per cent of the incident wave energy for $30^\circ \leq \theta \leq 90^\circ$.

Figure 10 shows a test of Eq. 12 using a Gaussian pulse with 0.067 m width incident from vacuum. Figure 10a shows results for a 45° corner reflector. The lines are contours of E_z separated by equal intervals. The absorbing layer on the bottom has $\Delta = 0.005$ m and conductivity $\sigma = 0.5301$ mhos/m. The top-left drive boundary creates the pulse, while the top left edge is an open symmetry boundary. The incident pulse with peak amplitude 1 V/cm is at the bottom-right. The reflected pulse traveling at a right angle has an amplitude of -0.17 consistent with the prediction of Eq. (7.4). Figure 10b shows an example with $\theta = 30^\circ$. The amplitude of the reflected pulse is -0.37 compared to a prediction of -0.33. A major advantage of the termination layer method is that we can improve the performance if we know the general topology of the pulses. For example, if pulses strike the surface at an angle θ , we can approach ideal absorption by setting the layer conductivity equal to

$$\sigma = \frac{\sin\theta}{\Delta \sqrt{\mu/\epsilon}} . \quad (7.5)$$

Figure 10c shows improvement of the solution of Fig. 10b with a layer conductivity of $\sigma = 0.2650$ mhos /m. In this case, the reflection coefficient is only -0.025.

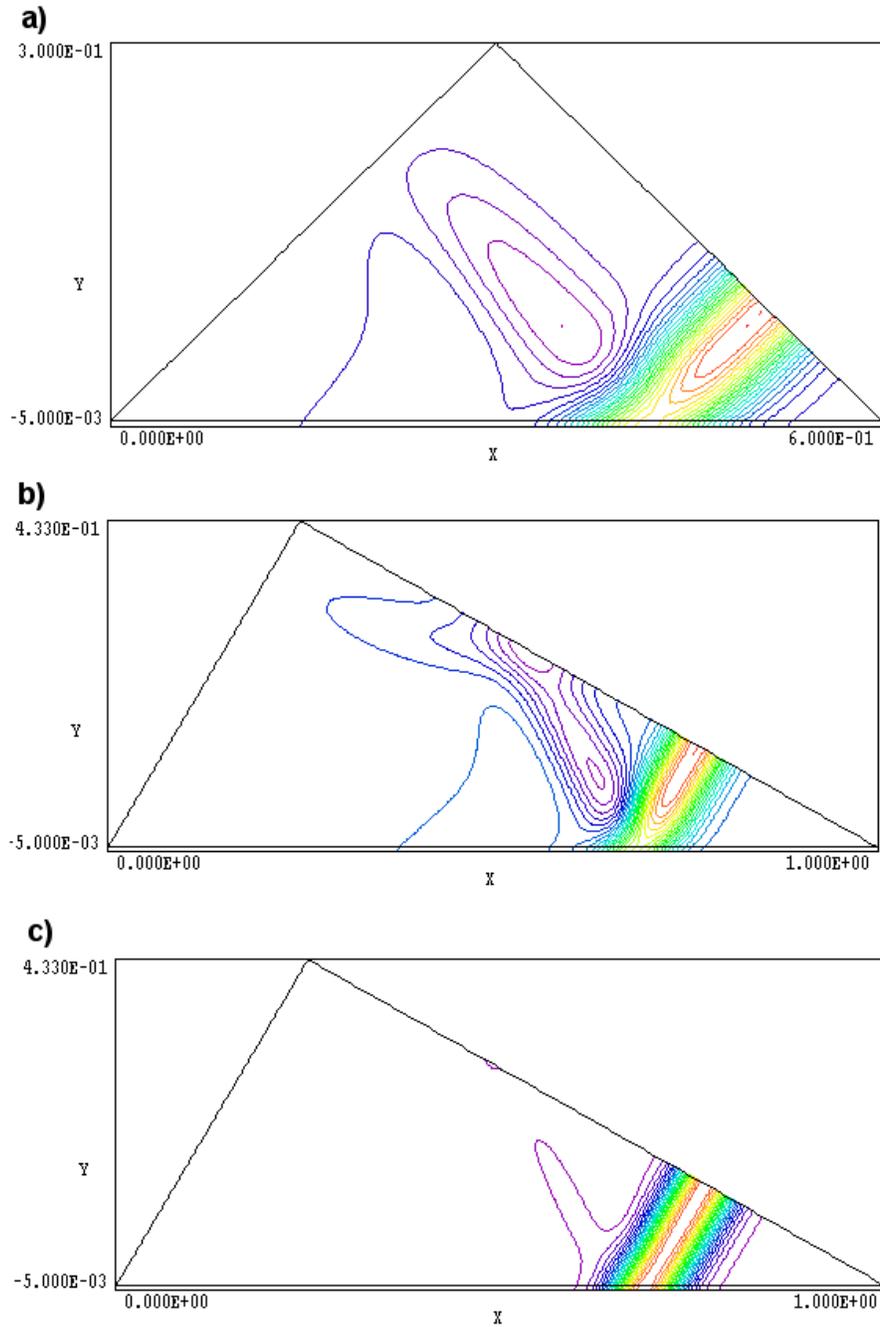


Figure 10. Performance of matched termination boundary as a function of incidence angle. Gaussian pulse in vacuum with spatial width of 0.067m. a) Contours of E_z showing incident and reflected pulses at a 45° corner reflector. Absorbing layer at bottom ($\Delta = 0.005$ m, $\sigma = 0.5301$ S/m) at bottom and symmetry boundary at top right. b) Pulse reflection at $\theta = 30^\circ$. c) Improvement of 30° pulse absorption by adjusting the layer conductivity to $\sigma = 0.2650$ S/m.

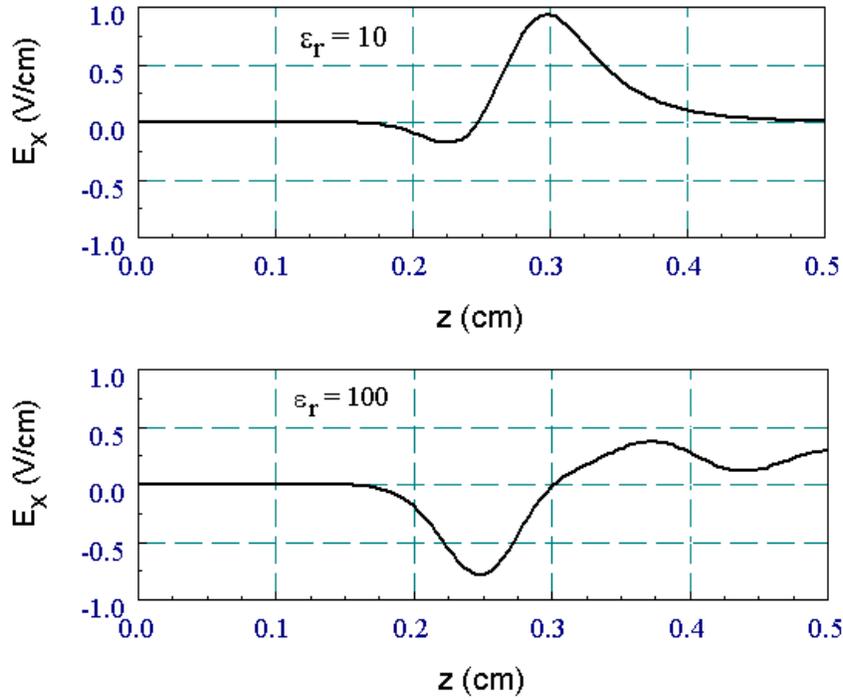


Figure 11. Demonstration of a reactive boundary layer - capacitive termination. Gaussian pulse with spatial width of 0.067 m. Layer thickness: 0.004 m. a) Low capacitance, $C = 0.35$ pF. b) High capacitance, $C = 35.4$ pF.

Termination layers can be used to construct generalized reactive boundaries. This is physically equivalent to transmission line terminations with resistive, capacitive and inductive components. To illustrate, consider the reflection of a pulse from the end of a vacuum transmission line terminated with a capacitor C . If the system extends an arbitrary distance L in the x and y directions, the capacitance of element of length Δz adjacent to an open boundary is

$$C = \frac{\epsilon L \Delta z}{L} = \epsilon \Delta z . \quad (7.6)$$

To represent an ideal capacitor, we set $\sigma = 0$ and $\epsilon = C/\Delta z$. The choice $\mu = 1/\epsilon v = \Delta z/Cv$, where v is speed of light in the adjacent material, maintains continuity of the pulse velocity. As an example, a capacitance has a strong effect on pulse of width Δt_p in a transmission line with impedance Z_0 if $C_0 = \Delta t_p/Z_0$. The value of C_0 is 6.0×10^{-13} for $\Delta t_p = 0.222$ ns and $Z_0 = 377.3$

Ω . When $C \ll C_0$, the termination acts like an open circuit, and approximates a short circuit when $C \gg C_0$. Figure 11 shows pulse reflection in the two regimes.

8. Organizing runs

The **EMP** package consists of a program that computes the physical solution (**EMP**.EXE) and a dedicated post-processor for analyses (**VEMP**.EXE). In addition, the utility **PROBE**.EXE can be used to display probe files. 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, **VEMP** 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.

EMP requires two input files.

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

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

An **EMP** run consists of several steps that involve four 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 command file with a name of the form **FPREFIX**.MIN following the instructions in the **Mesh** manual. You can create the file directly with a text editor or graphically using the Drawing Editor of **Mesh**.

- Run **Mesh** either interactively from the **TC** program launcher or from the Windows Command Prompt to create the file `FPREFIX.MOU`.
- Prepare a command script (`FPREFIX.EIN`) using the *Create command file* menu entry in the solution program or your own text editor. The allowed file commands are described in this manual and the manual for the specific solution program.
- Run **EMP** to create data dumps (`FPREFIX.001`, `FPREFIX.002`,...) or history files (`FPREFIX.P01`, `FPREFIX.P02`,...). All output files are in ASCII format. The data dumps contain information on the mesh geometry, the physical properties of regions, and values of the primary computed node quantity. In electromagnetic solutions the region properties are the relative magnetic permeability, the relative dielectric constant and the conductivity. The computed quantity depends on the solution type (see Table 2).
- Analyze data dumps using **VEMP**. You can also transfer the information in `FPREFIX.001` to your own analysis programs.
- Analyze history files using **Probe**.

9. Structure of the command file

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

Space, Comma, Forward slash,
Colon, Tab, Equal sign

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

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

The bottom number is interpreted as 5.0.

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

```
SET TMax 1.95E-9
SET Geometry Rect
SET DUnit 1.0
SET Solution E

REGION 1 Mu 1.0
REGION 1 Epsi 1.0
REGION 1 Cond 0.5301

REGION 2 Mu 1.0
REGION 2 Epsi 1.0

REGION 3 Drive GAUSS.SRC

DIAG SetTime 1.084E-9
DIAG History 0.30 0.125

ENDFILE
```

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

```
SET
REGION
DIAG
```

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 format.

```
Region RegNo Keyword Value
```

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

```
REGION 2 Mu 300.0
```

sets the relative magnetic permeability in elements with region number 2 to the value 300.0. Several **REGION** commands may refer to the same mesh volume and set different properties.

Finally **DIAG** commands control data recording. These commands appear in all TriComp programs for initial-value solutions. They are unnecessary in boundary-value problems where the code creates a single output file.

10. Program control commands

EMP commands divide into three groups: 1) control of the solution, 2) definition of the physical properties of regions, and 3) diagnostics. We begin with the program commands. They all begin with the keyword **Set** followed by the command type and one or more parameters. In the following, each command is written as it might appear in a command file.

```
Set TMax 100.0E-9
```

Function. Defines the maximum run.

Parameters. Run time in seconds, real number format.

Comments. Pick a run time slightly longer than the final target dump time.

```
Set Geometry Rect
```

Function. Defines whether the solution represents rectangular or cylindrical geometry.

Parameters. The string *Rect* or *Cylin*. The default is *Rect*.

```
Set NStep 4500
```

Function. An alternate way to limit the run time by setting the maximum number of computational steps.

Parameters. Maximum number of steps, integer format. The default is 100,000.

Set DUnit 1000.0

Function. Defines the interpretation of spatial coordinates from the mesh input file and the *Diag History* command.

Parameters. Spatial conversion factor equal to the number of spatial units per meter, real number format. The default is 1.0.

Comments. As an example, if coordinates in Mesh are given in mm, set $DUnit = 1000.0$. Be sure to enter the coordinates in the *Diag History* command in the same units.

Set Solution E

Function. Sets the solution type as *E* Pulse or *H* Pulse.

Parameters. The characters *E* or *H*. The default is *E*.

Comments. The solution type affects the interpretation of numerical quantities in the **VEMP** program.

Set Interpolation Spline

Function. Sets the interpolation method for input tables that define time variations of sources or boundary fields.

Parameters. The strings *Linear* or *Spline*. The default is *Linear*.

Comments. If you have a well-behaved table, the spline option gives better accuracy. On the other hand, a spline fit gives poor results if the table contains discontinuous or noisy data.

11. Region commands

Region commands set the physical properties of the regions defined in **MESH**. All commands begin with the keyword *Region* followed by the mesh region number, an integer. The first three commands assign properties to the elements of material regions (those with non-zero volume). **EMP** recognizes three properties: relative dielectric constant, relative magnetic permeability and electrical conductivity. The next three commands set boundary conditions. The final three commands add source current density to material regions.

Region 1 Mu 250.0

Function. Assigns a relative magnetic permeability to all elements with the corresponding region number.

Parameters. The relative magnetic permeability, μ/μ_0 , a real number. The default is 1.0.

Region 7 Epsi 7.3

Function. Assigns a relative dielectric constant to all elements with the corresponding region number.

Parameters. The relative dielectric constant, ϵ/ϵ_0 , a real number. The default is 1.0.

Region 6 Cond 6.4E5

Function. Assigns an electrical conductivity to all elements with the corresponding region number.

Parameters. The electrical conductivity in SI units of S/m (Siemens per meter), the inverse of the volume resistivity in Ω -m. Real number quantity. Default is 0.0.

Region 1 Fixed

Function. Assigns a special Dirichlet condition to the region.

Comments. A value of 0.0 is assigned to the quantity E_z , H_z , rE_θ or rH_θ for all vertices. The fixed field value holds throughout the run. *In cylindrical runs, you should always make the axis a special Dirichlet boundary.*

Region 9 Void

Function. Sets an internal region to the Void condition. In this case the special Neumann condition applies on the boundaries with standard material regions.

Comments. When the void condition is assigned, EMP treats all nodes and elements of the region as though they were not included in the solution volume. A void region acts as though it has characteristic impedance $\eta = \infty$ for E type pulses and $\eta = 0$ for H type pulses. The main use of a void is to represent internal conducting volumes for H type solutions. You could achieve a similar effect by setting $\epsilon = 10^6$ and $\mu = 10^{-6}$ manually. The difference is that **VEMP** will not try to plot field lines inside a void.

There are two important considerations for using interval voids:

- If you create a void region in a mesh by over-writing a portion of a material region, then the **Mesh** program will assign the void region number to all nodes including those on the boundary. For a correct

solution in **EMP** the boundary nodes should have the region number of the surrounding material. Therefore, you must insert a line region *after* the definition of the void to change the region number on the boundary. In **EMP** assign properties of the surrounding material to the additional region. The example ROD_H illustrates the setup technique.

- Use internal voids only if the region does not connect to the solution space boundary. Otherwise, define the boundary shape to exclude conducting regions as in the walkthrough example CONICAL.

Region RegNo Drive FileName [TMult DMult]

Region 5 Drive Gauss.DRV

Function. Defines a fixed field region with a specified variation in time. These regions are normally used to generate pulses.

Parameters. A string giving the full name of a tabular function file in the current directory that specifies the time variation of E_z , H_z , rE_0 or rH_0 . See Section 13 for a discussion of tabular functions.

Comments. The tabular function routines return values of zero when the time exceeds the largest values on the table. Therefore, a drive boundary assumes a short circuit condition after its active period. To define an open circuit or absorbing boundary, use the *TExpire* command for the region. When the optional real number parameters *TMult* and *DMult* are present, the program multiplies time and drive quantities as they are loaded from the table. The default is $TMult = 1.0$, $DMult = 1.0$.

Region RegNo Drive Sin Amp Freq Phase

Region 5 Drive Sin 1.0 6.0E8 0.0

Function. Defines a fixed-field region with a sinusoidal variation in time. These regions are normally used to generate pulses.

Parameters. There are three real number parameters after the *Sin* keyword. The first is the magnitude of the quantity of E_z , H_z , rE_0 or rH_0 . The second parameter is the frequency in Hz, and the third is the phase in degrees.

Region 7 TExpire 3.5E-9

Function. Defines an expiration time for a fixed field region condition. The boundary reverts to an unspecified (Neumann) condition after a time limit.

Parameters. The quantity t_{\max} , the expiration time in seconds. The

fixed field condition determined from a table holds in the interval $0.0 \leq t \leq t_{\max}$.

Comments. Pulses pass through the boundary unimpeded after the expiration time. The main use of this option is to create a pulse inside a volume and then to follow propagation with open space (E pulse) or conducting (H pulse) boundaries .

Region RegNo Source FileName [TMult SMult]

Region 3 Source Coupler.SRC 2.0E-9 150.0

Function. Adds spatially-uniform drive current sources with specified time variations to a material region. These regions represent current loops or capacitive probes that drive RF pulses..

Parameters. The string gives the full name of a tabular function file in the current directory that specifies the time variation of the source function. The two optional real-number parameters are multiplication factors for the time (*TMult*) and source (*SMult*) values. With this feature you can apply normalized waveforms to a variety of problems. The default values are *TMult* = 1.0 and *SMult* = 1.0.

Comments. The interpretation of the source function data in **EMP** depends on the geometry and solution type. Table 3 lists the options.

Table 3 Source Representations in Tabular Functions			
Pulse type	Geometry	Quantity	Units
E	Planar	dj_z/dt	A/m ² -s
E	Cylindrical	$d(rj_\theta)/dt$	A/m-s
H	Planar	j_x, j_y	A/m ²
H	Cylindrical	rj_z, rj_r	A/m

Region RegNo Source Sin Amp Freq Phase

Region 7 Source Sin 15.06 1.0E9 30.0

Function. Adds a uniform source with a harmonic time variation to a material region. The regions act as antennas for time-domain solutions of frequency-domain problems.

Parameters. There are three real number parameters after the *Sin* keyword. The first is the magnitude of the current density. In planar

geometry, the quantity is j_z (E type pulses) or $(j_x^2+j_y^2)^{1/2}$ (H type pulses). The units are A/m². In cylindrical geometry, the quantity is rj_θ or $r(j_r^2+j_z^2)^{1/2}$ in A/m. The second parameter is the frequency in Hz, and the third is the phase in degrees.

Comments. In contrast to the tabular function, **EMP** automatically uses the time derivative of current density for E type pulses. Table 4 summarizes the input parameters and the forms of current density variations for the different geometries and solution types.

Table 4					
EMP Conventions - Harmonic Sources					
Geometry	Solution	Param1	Param2	Param3	Program source term
Rect	E	j_z	f	ϕ	$2\pi f j_z \cos(2\pi f t + \phi)$
Cylin	E	rj_θ	f	ϕ	$2\pi f rj_\theta \cos(2\pi f t + \phi)$
Rect	H	$(j_x^2+j_y^2)^{1/2}$	f	ϕ	$(j_x^2+j_y^2)^{1/2} \sin(2\pi f t + \phi)$
Cylin	H	$r(j_z^2+j_r^2)^{1/2}$	f	ϕ	$r(j_z^2+j_r^2)^{1/2} \sin(2\pi f t + \phi)$

Region 7 Beta 22.5

Function. Sets the angle of region current density in the x - y or z - r plane for H type solutions.

Parameters. The angle in degrees of the current density relative to the x or z axes, equal to $\tan^{-1}(j_y/j_x)$ or $\tan^{-1}(j_r/j_z)$. The default is 0.0.

Region 7 TabShift 1.0 5.0 1.0E4 0.0

Function. Adjusts the interpretation of time-dependent functions for drive and source regions.

Parameters. There are four real number parameters after the *TabShift* keyword. The first, (T_{mult}) is a multiplication factor for values of time and the second (T_{off}) is an offset. The default values are 1.0 and 0.0. The third (V_{mult}) is a multiplication factor and the fourth (V_{off}) is an offset for the dependent variable.

Comments. This program function allows you to apply a normalized

tables to a variety of applications. If t is the program time, the value passed to the tabular function is

$$t_{\text{pass}} = (t - T_{\text{off}})/T_{\text{mult}}$$

If the interpolation gives a value ϕ , the value returned to the program is

$$\phi_{\text{pass}} = V_{\text{mult}} \phi + V_{\text{off}}$$

This command has no effect on regions that are not associated with a table. Note that this command supplements the *TMult*, *DMult* and *SMult* parameters in the *DRIVE* and *SOURCE* commands.

12. Diagnostic commands

Diagnostic commands that begin with the keyword *Diag* control data output from the program. **EMP** produces three types of data, all in ASCII format.

- From 1 to 999 data dump files with the names *FName*.001, *FName*.002, Each file contains a complete record of spatial information (the mesh geometry and distributions of vector potential, magnetic permeability and induced electric field) at a specified time. These files are usually analyzed with the **VEMP** program.
- A listing file with the name *FName*.ELS that can be inspected with a text editor. This file contains a variety of information about the **EMP** run used mainly to check the validity of input parameters and to diagnose problems.
- Optional history files with the names *Fname*.P01, *FName*.P02, . . . , *FName*.P10. Here, the magnetic and electric field components at one or more probe locations are recorded at each time step of the solution. The information can be inspected with a text editor or the utility program **Probe** (described in a separate manual).

Diag SetTime 2.5E-9

Function. Calls for a data dump at or immediately following a

specified time.

Parameters. The dump time in seconds, a real number.

Comments. When using automatic time step selection, it is usually uncertain that **EMP** will complete a cycle exactly at the specified time. The convention is that the code records a dump when the time is equal to or greater than the specified value. You can define up to 100 dump times with multiple *SetTime* commands. The only restriction is that the commands must be entered in chronological order.

Diag DTime 2.5E-9

Function. Calls for a set of data dumps at a uniform time interval.

Parameters. The interval in seconds, a real number.

Comments. Again, there is no way to guarantee that the target diagnostic times will fall exactly on cycle times. **EMP** records a data dump as soon as possible after the target time.

Diag NStep 250

Function. Calls for a set of data dumps separated by a uniform number of computational cycles.

Parameters. The number of cycles per data dump, an integer.

Comments. You can generate data at exact intervals by using this option and setting the time step with the *Set Dt* command.

Diag History 2.50 3.75

Function. Records all field values as a function of time at a specified location.

Parameters. The target coordinates for the probe location in distance units set by *DUnit*. The real number inputs are (x,y) for planar problems and (z,r) for cylindrical problems.

Comments. The probe coordinates in the history file usually will be slightly different from those specified. **EMP** replaces the target coordinates with the center-of-mass coordinates of the element that contains the target point. The spatial derivatives in Eqs. 7 are determined by a linear fit using main field values at the three vertices of the elements. The input file can contain up to 20 *Diag History* commands in any order.

Diag NHStep 10

Function. Sets the number of time steps per history record.

Parameters. Time steps per record, an integer.

Comments. This command allows you to reduce the number of entries in the history file for long runs and small time steps. The default value is 1.

13. Tabular functions

One of the most powerful features of **EMP** is the ability to represent any specified time dependence of driving fields and sources. The key to this flexibility is tabular function input. A tabular function is a text file containing up to 256 data entry lines. Each line contains two real numbers in any format: the time (independent variable) and a corresponding value of the field or current density (dependent variable). You can prepare tabular function files with a text editor or spreadsheet. It is easy to incorporate published data or digitized experimental traces. The example below shows a shortened version of a tabular function to define a Gaussian pulse variation with a full-width at half maximum of 0.22 ns. Note that the time interval need not be uniform. In fact, the entries can even be out of order. **EMP** arranges them chronologically when the table is loaded. The file syntax conforms to the same rules as the **EMP** command file. The free-form parser accepts real numbers in any format with a choice of delimiters. You can add documenting comment lines starting with asterisk-space. The end of the data is marked with the **EndFile** command. You can also add any text after **EndFile**.

* Gaussian pulse shape, 0.222 ns FWHM, amp 1.0

* Peak at 0.6667 ns

```
0.0000E+00 1.3886E-11
1.3333E-11 3.7370E-11
2.6667E-11 9.8581E-11
4.0000E-11 2.5490E-10
5.3333E-11 6.4605E-10
```

...

```
1.3067E-09 9.8609E-11
1.3200E-09 3.7381E-11
1.3333E-09 1.3890E-11
1.4000E-09 0.0000E+00
1.6000E-09 0.0000E+00
1.8000E-09 0.0000E+00
2.0000E-09 0.0000E+00
2.5000E-09 0.0000E+00
3.0000E-09 0.0000E+00
```

ENDFILE

The maximum number of tables is 16. The table interpolation method is controlled by the *Set Interpolation* command. Under the *Spline* option a table requires a minimum of 5 entries. The interpolation routines return 0.0 for values of the independent variable off the table. Table 5 shows how the *Region TabShift* command affects values of the independent variables passed to the interpolation routines and values of the dependent value return to the program. For example suppose we have a table with non-zero values over a normalized independent variable range from 0.0 to 1.0. With a choice of $T_{\text{mult}} = 1.0 \times 10^{-9}$ and $T_{\text{off}} = 5.0 \times 10^{-9}$, the pulse will be active from 5.0 to 6.0 ns.

Table 5		
Effect of the Region TabShift Command		
Time generated by program		Time used in interpolation
t	→	$T_{\text{mult}} t + T_{\text{off}}$
Value used by program		Value generated by interpolation
$V_{\text{mult}} \phi + V_{\text{off}}$	←	ϕ

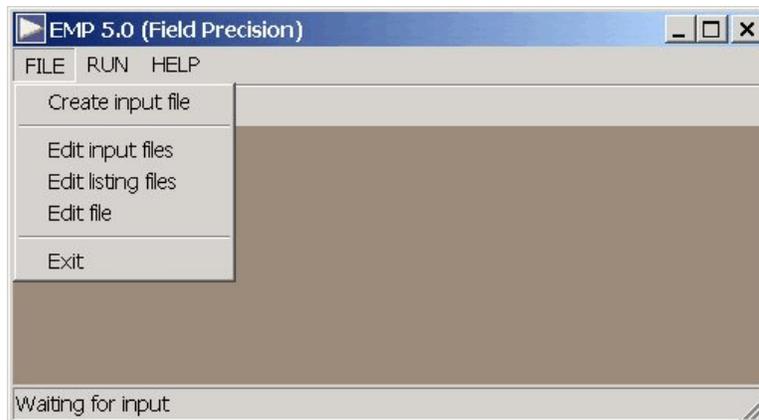


Figure 12. Screen shot of EMP in the interactive mode

14. Running EMP interactively

The program `BMP.EXE` can run interactively in a Window (Figure 12). 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 **EMP**. Choosing a file from an alternate directory does not change the working directory. The *Edit input file* command shows

a list of all files with names of the form `FPREFIX.EIN`. The *Edit listing file* command displays files with names `FPREFIX.ELS`.

The *Run* menu has four commands.

Start run

Pick an input file (such as `FPREFIX.EIN`) to start a solution. The working directory is changed if you pick a file from an alternate directory. The run begins if the file `FPREFIX.MOU` 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.

15. Running EMP from the command prompt

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

```
..\EMP TLINE <Enter>
```

The program runs silently, writing detailed information in the listing file `TLINE.ELS`. If the solution is successful, the program writes output files

to the data directory. During lengthy runs you can perform other tasks in Windows. Note that the response of the computer may be slowed considerably.

The main function of the command mode is autonomous operation under batch file control. This feature is useful if you want to make an extended series of solutions overnight or in the background. As an example, suppose you have prepared the input files

SWT01.MIN, . . . , SWT08.MIN and SWT01.EIN, . . . , SWT08.EIN in the directory \TRICOMP\BUFFER. Furthermore, you have written the following batch file SWRUN.BAT in the data directory using an ASCII text editor.

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

Type

```
SWRUN <Enter>
```

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

Microsoft has released over thirty versions of its 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.

16. VEMP file menu

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

Program units

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

Electric fields, **E**: V/m

Magnetic intensity, **H**: A/m

VEMP 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 first solution file

The *Load first solution file* command displays a dialog with a list of initial data dumps (suffix 001) . Changing the directory in the dialog changes the program working directory. Pick an available file and click *OK*. The program loads the first solution in the series and updates the status bar. If data retrieval is successful, the *Spatial plots* and *Analysis* menus become active.

Load next solution file

Load the next date dump in the series (*i.e.*, *FPrefix.002*). **VEMP** issues an error message if no more solution files.

Load solution file number

Specify the numbered solution file in the series. For example, if you enter "3" **VEMP** loads *FPrefix.003* if it is available.

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 16. 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.

17. VEMP spatial plot menu

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

Plot type

VEMP 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. In two-dimensional magnetostatic solutions constant values of A_z or rA_θ are parallel to magnetic field lines. Contours are separated by equal increments of magnetic flux. Therefore, in cylindrical coordinates the line spacing increases near the axis even if the field magnitude $|\mathbf{B}|$ is uniform.

Element. Elements of the solution space color-coded according to a computed quantity (such as the magnetic field magnitude). 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.

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

Contour plot quantities

Main field component (E_z , rE_θ , H_z or rH_θ), equivalent to lines of \mathbf{H} for E type solutions or lines of \mathbf{E} for H type solutions. (Note: For H type solutions contours of H_z or rH_θ follow electric field lines only in regions with zero source current and zero conductivity).

Element plot quantities

Main field component (E_z , rE_θ , H_z or rH_θ)
Magnitude of the transverse field ($|\mathbf{H}|$ or $|\mathbf{E}|$)

Vector plot quantities

Transverse field, \mathbf{H} or \mathbf{E}

Surface plot quantities

Main field component (E_z , rE_θ , H_z or rH_θ)
Magnitude of the transverse field, $|\mathbf{E}|$ or $|\mathbf{H}|$
Transverse field components: H_x , H_y , H_z or H_r for E type solutions, E_x , E_y , E_z or E_r for H type solutions

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 **VEMP**. When snap mode is active, the mouse returns the coordinate values closest to an integer multiple of the quantity $DSnap$. In other words, if $DSnap = 0.5$ and the mouse position is [5.4331,-2.6253], the returned coordinates are [5.5,-2.5]. By default, snap mode is *ON*. Snap mode is automatically turned off for coordinate input to the commands *Point calculation* and *Element properties*. Otherwise, the program 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, **VEMP** 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.

18. VEMP 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

VEMP employs a sophisticated interpolation technique that preserves discontinuities at material boundaries. The program gives correct values for electric and magnetic fields 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 **VEMP**. 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 depend on the solution type. Main field component (E_z , rE_θ , H_z or rH_θ), magnitude of the transverse field, $|\mathbf{E}|$ or $|\mathbf{H}|$, transverse field components: H_x , H_y , H_z or H_r for *E* type solutions, E_x , E_y , E_z or E_r for *H* type solutions

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), **VEMP** writes material and field properties of the element to the screen. The information is also recorded if a history file is open. It is usually best to start from a *Mesh*, *Region*, *Element* or *Vector* plot to identify the element.

Region properties

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

Volume integrals

No input is needed for this command. **VEMP** performs the following analyses:

- 1) Integrals of instantaneous magnetic field energy density ($B^2/2\mu_o\mu_r$) and electric field energy density ($\epsilon_o\epsilon_r E^2/2$) over the solution space and region volumes. The output units are J/m in rectangular solutions and J in cylindrical solutions.

2) Integral of instantaneous power dissipation σE^2 over the solution space and region volumes. Output units are W/m in rectangular solutions and W in cylindrical solutions.

Matrix file

VEMP 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 may be difficult to deal with the conformal triangular mesh. The *Matrix file* command uses the interpolation capabilities of the program to create an ASCII data file of field quantities on a rectangular grid in x - y or z - r . The command displays a dialog box where you set the matrix file prefix, the dimensions of the box and the number of intervals along x and y (or z and r). The program creates the file `FPREFIX.MTX` in the current directory.

19. VEMP 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 **VEMP**. This command closes the scan plot and returns the program to the previous spatial plot.

20. VEMP script file commands

Script files to control **VEMP** have a name of the form `FPREFIX.SCR`. They should be in the same directory as the data files. Scripts are ASCII files that follow the **TriComp** syntax conventions. The program ignores blank lines and indentations. Data lines use the standard delimiters and comment lines begin with ‘*’ (asterisk). Processing ends when the *EndFile* command is encountered.

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

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

```
..\VEMP 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 CONICAL.003

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 `CONICAL.003`. 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.

Volumelnt

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 SWITCH.001
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH.002
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH.003
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH.004
SCAN 0.00 -50.00 0.00 50.00
ENDFILE
```

17. Format of the EMP output file

The **EMP** data dump files `FPrefix.001`, `FPrefix.002`, ... are in ASCII format. It can be inspected with any text editor. Each file has three sections:

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

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

```

--- Run parameters ---
XMin: -1.270003E-01
XMax:  1.270003E-01
KMax:  101
YMin:  0.000000E+00
YMax:  2.540005E-01
LMax:  101
DUnit:  3.937000E+01
NReg:   5
ICylin: 1
Time:   2.002203E-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 specifies the time of the data dump and Line 12 gives the solution 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	A	IntA
1	1	4	0	0	-5.025000E-01	0.000000E+00	0.000000E+00	0.000000E+00
2	1	4	0	0	-4.999241E-01	0.000000E+00	0.000000E+00	0.000000E+00
3	1	4	0	0	-4.973531E-01	0.000000E+00	0.000000E+00	0.000000E+00
4	1	4	0	0	-4.947889E-01	0.000000E+00	0.000000E+00	0.000000E+00
5	1	4	0	0	-4.922323E-01	0.000000E+00	0.000000E+00	0.000000E+00
6	1	4	0	0	-4.896838E-01	0.000000E+00	0.000000E+00	0.000000E+00
7	1	4	0	0	-4.871431E-01	0.000000E+00	0.000000E+00	0.000000E+00
8	1	4	0	0	-4.846098E-01	0.000000E+00	0.000000E+00	0.000000E+00
9	1	4	0	0	-4.820830E-01	0.000000E+00	0.000000E+00	0.000000E+00
10	1	4	0	0	-4.795620E-01	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 primary field quantity, E_z (V/m), rE_θ (V), H_z (A/m) or rH_θ (A).
- The time-integral of the primary field quantity used to determine the transverse field components: (H_x, H_y) , (H_z, H_r) , (E_x, E_y) or (E_z, E_r) .

The region section consists of four title lines following by $NReg$ data lines, one for each region. For EMP solutions, the section contains the following information for each region:

RegNo: Region number
 Fix: 1 for fixed-fixed (Dirichlet) regions, 0 otherwise
 Void: 1 if the region is a void, 0 otherwise
 Epsi: Relative dielectric constant
 Mu: Relative magnetic permeability
 Cond: Conductivity in S/m
 Area: Mesh area occupied by the region in m²
 Beta: Angle of the drive current

22. Solution examples

This section gives a detailed overview of two of the examples in the **EMP** solution library to demonstrate program implementation. The **Parabola** example illustrates an *E* type pulse in planar geometry with pulse initiation by a drive region. Figure 13 shows the system with numbered solution regions. The **Mesh** file input for the reflector surface is a good example of the using of multiple line vectors to define a complex shape. The input control file is listed below. Note the specification for Region 2, the thin absorbing layer on the top and sides. The layer with thickness 0.01 m is adjacent to a vacuum region with characteristic impedance $Z_o = 377.3 \Omega$. Equation 11 gives the matched conductivity as 0.26504 mhos/m. The axial electric field applied to the rod (Region 3) has the baseline Gaussian shape with a pulselength of 0.22 ns. The *TabShift* command is used to set a peak field value of 10,000 V/m. Note the use of the expiration time to remove the effect of the rod on the reflected pulse. The value $T_{Expire} = 0.65$ ns is longer than the pulse but shorter than the transit time to the reflector and back. The function of Region 4 is to define the reflective coating on the bottom boundary.

```
* Parabolic reflector
SET TMax 12.01E-9
SET Geometry Rect
SET DUnit 1.0
* Vacuum
REGION 1 Mu 1.0
REGION 1 Epsi 1.0
* Resistor, thickness 0.01 m
REGION 2 Mu 1.0
REGION 2 Epsi 1.0
REGION 2 Cond 0.26504
* Drive rod
REGION 3 Drive Gauss.SRC
REGION 3 TabShift 1.0 0.0 10000.0 0.0
REGION 3 TExpire 0.650E-9
* Reflector
REGION 4 Dirichlet
DIAG DTime 4.0E-9
DIAG History 0.00 0.20
ENDFILE
```

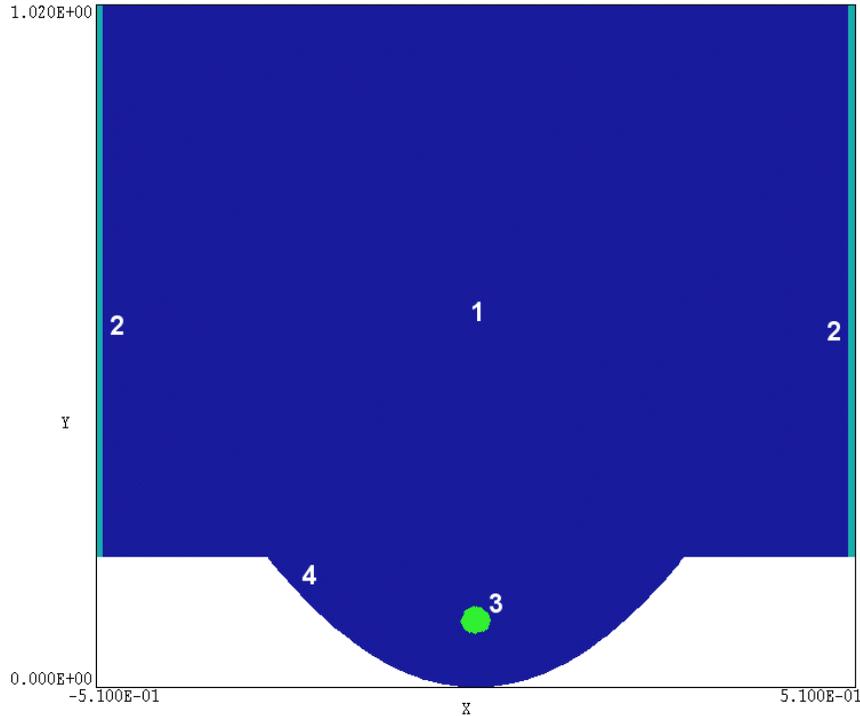


Figure 13. Region assignments in the Parabola example

The calculation exercises the full power of **EMP**. The mesh has over 250,000 elements. The run time for a 12 ns solution is 15 seconds on a 1.7 GHz computer. Figure 14 shows an element plot of E_z at 8 ns. An expanding cylindrical pulse with positive polarity propagates directly from the rod. The reflected plane pulse has negative polarity and an amplitude about 2.7 times higher. Figure 15 shows the variation of electric field close to the source (0.0,0.2).

The second example, illustrated in Fig. 16a, is a study for a two beam particle accelerator. Here, an annular drive electron beam at large radius excites converging H type pulses in a series of cylindrical cavities to generate a large electric field on axis. The input control file is listed below. The metal walls of the structure are represented by unspecified external boundaries and internal Neumann regions. Most of the structure is defined by the complex outer boundary of Region 1. The three internal plates (Regions 2,3 and 4) are Neumann regions. The relativistic drive beam is simulated with four phased source regions (Regions 5,6,7 and 8). The *TabShift* command is used to increase the pulselength of the baseline Gaussian waveform to 0.4 ns and to introduce time shifts in the regions

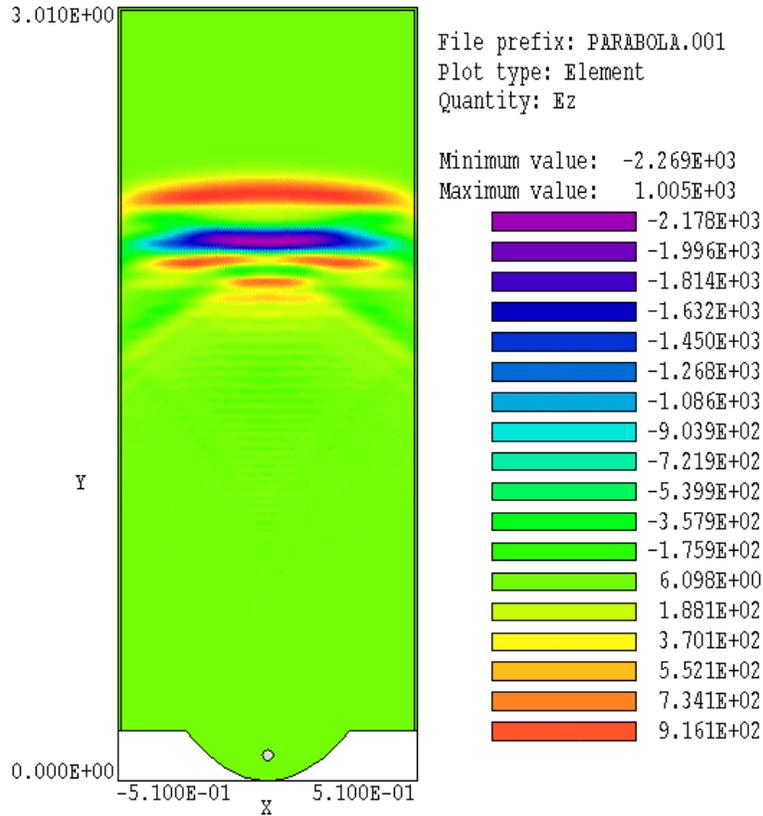


Figure 14. Electric field distribution at 8 ns, Parabola example

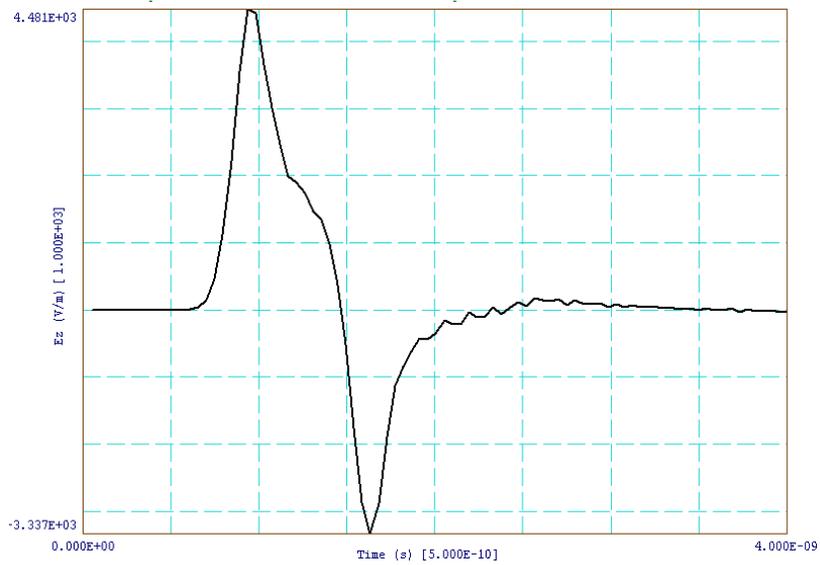


Figure 15. Parabola example, variation of electric field at $x = 0.0$ m, $y = 0.2$ m

consistent with a beam traveling near the speed of light. The choice of $V_{\text{mult}} = 7.958 \times 10^5$ gives a peak beam current of 50 kA. Figure 16b shows electric field lines at 3 ns.

```
* File: TwoBeam.EIN
* Adjustment factors for beams
*   FWHM: 0.4 ns   TMult: 1.818
*   Cavity spacing: 0.05 m   Add offsets 0.1667 ns
*   50.0 kA beam at 0.50 m, thickness 0.02 m
*   Current density: 2500.0/0.06283 = 3.98E4 A/m2
*   Gamma = 0.5*3.98E4 = 7.958E5 A/m
SET TMax 4.1E-9
SET Geometry Cylin
SET DUnit 100.0
SET SOLUTION H
* Main solution volume
REGION 1 Mu 1.0
REGION 1 Epsi 1.0
* Plate 1
REGION 2 Void
* Plate 2
REGION 3 Void
* Plate 3
REGION 4 Void
* Beam 1
REGION 5 Source Gauss.SRC
REGION 5 Beta 0.0
REGION 5 TabShift 1.818 0.0 7.958E5 0.0
* Beam 2
REGION 6 Source Gauss.SRC
REGION 6 Beta 0.0
REGION 6 TabShift 1.818 0.1667E-9 7.958E5 0.0
* Beam 3
REGION 7 Source Gauss.SRC
REGION 7 Beta 0.0
REGION 7 TabShift 1.818 0.3333E-9 7.958E5 0.0
* Beam 4
REGION 8 Source Gauss.SRC
REGION 8 Beta 0.0
REGION 8 TabShift 1.818 0.5000E-9 7.958E5 0.0
* Axis
REGION 9 Fixed
DIAG DTime 1.0E-9
DIAG History 7.5 0.0
DIAG History 12.5 0.0
DIAG History 15.5 0.0
DIAG History 17.5 0.0
ENDFILE
```

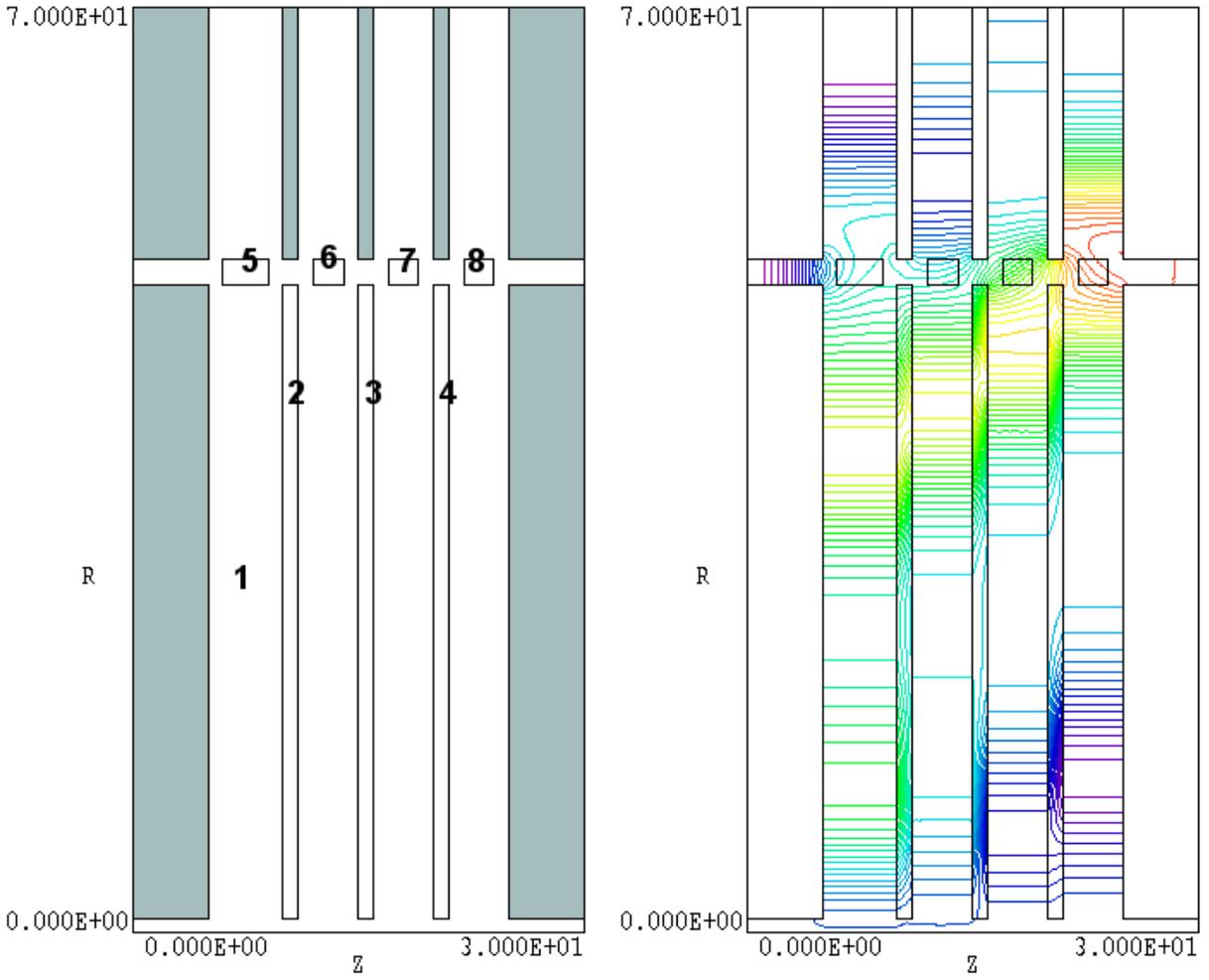


Figure 16. Study for a two-beam accelerator, four cylindrical cavities shown. a) Geometry showing region designations. b) Contours of rH_0 (electric field lines) at 3.0 ns.

23. Simulation techniques for transmission lines and pulsed-power switch assemblies

This section reviews useful **EMP** techniques to generate TEM waves in coaxial transmission lines. As an example, we shall use the program to investigate switching time and transients for a high-voltage spark gap in a Blumlein line assembly. To begin, consider the simple problem of generating a square-pulse in an ideal coaxial line. The line has inner radius $R_i = 5.0''$, outer radius $R_o = 8.0''$ and a fill medium of transformer oil with a relative dielectric constant $\epsilon_r = 2.8$. The line has characteristic impedance

$$Z_o = \frac{1}{2\pi} \sqrt{\frac{\mu}{\epsilon}} \ln \left(\frac{R_o}{R_i} \right) = \frac{1}{2\pi} \frac{377.3}{\sqrt{\mu_r \epsilon_r}} \ln \left(\frac{R_o}{R_i} \right) \quad (\Omega) . \quad (23.1)$$

and a capacitance per length of

$$c = \frac{2\pi\epsilon}{\ln (R_o/R_i)} \quad (F/m) . \quad (23.2)$$

The system has cylindrical symmetry. A TEM pulse has a radial electric field and an azimuthal magnetic field. The calculation is therefore an H type solution and the primary quantity is rH_θ .

Figure 17 shows the geometry and assignment of regions. The active solution volume has a length of 16.0" in the z direction and extends between R_i and R_o in the radial direction. Thin absorbing layers matched to the line impedance are located upstream and downstream. A radial current source layer drives the pulse. The three layers have thickness $\Delta z = 0.1''$. The current layer creates a TEM pulse that propagates both upstream and downstream — the upstream pulse is absorbed by the adjacent resistive layer. The downstream resistive layer absorbs the pulse after propagation through the solution volume. There are no special boundary conditions. The default Neumann boundary acts as a perfect conductor for H type pulses.

The **EMP** input file COAX01.EIN is listed below. We shall review the motivations for choosing the parameters in the file. The speed of light in the oil is 1.793×10^8 m/s; therefore, the single pass electromagnetic transit time is 2.27 ns. The run time of 4.01 ns is sufficient to follow the initial pulse and to check reflected waves resulting from non-ideal performance

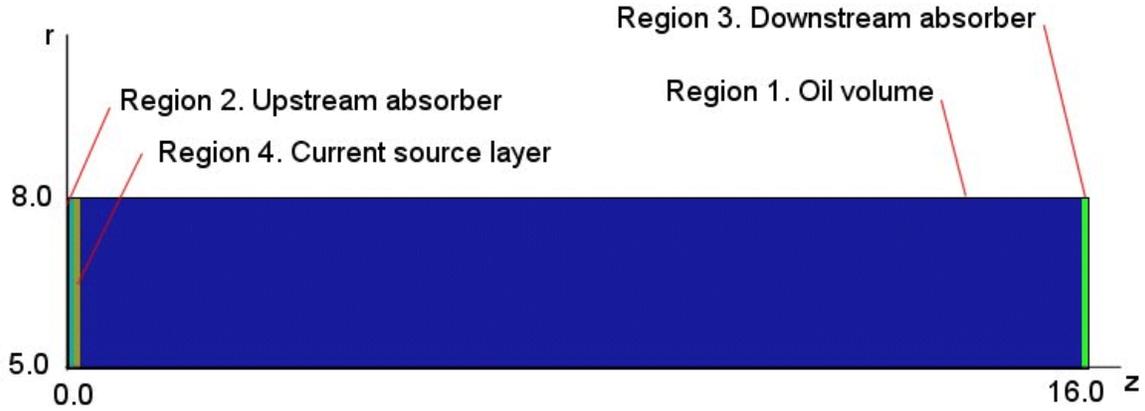


Figure 17. Geometry for the example COAX01

of the absorbing layer. The value $DUnit = 39.37$ converts dimensions in the **Mesh** input file from inches to meters. The solution volume is an ideal dielectric with $\epsilon_r = 2.8$ and $\mu_r = 1.0$. The corresponding impedance for the transmission line is $Z_o = 16.87 \Omega$. The absorbing layers have the same dielectric properties and a non-zero conductivity. The matched conductivity for total absorption is given by

$$\sigma = \frac{1}{\Delta z \sqrt{\mu/\epsilon}} = \frac{1}{\Delta z \eta} . \quad (23.3)$$

The quantity η is the characteristic impedance of the medium. For the given dielectric constant, $\eta = 377.3/\sqrt{\epsilon_r} = 225.5 \Omega$. With $\Delta z = 2.54 \times 10^{-2}$ m the matched conductivity is $\sigma = 1.746$ S/m. The resistance of a coaxial resistor is given by

$$R = \frac{\ln(R_o/R_i)}{2\pi\Delta z\sigma} . \quad (23.4)$$

Inserting values in the equation we find that $R = Z_o = 16.87 \Omega$.

Next we consider choosing the magnitude of the radial current in the source layer (Region 4). We want to generate a downstream pulse with a peak current of 1.0 A. Because the layer must drive both upstream and downstream pulses, the current should be 2.0 A. The following lines in the EMP input file define the current source:

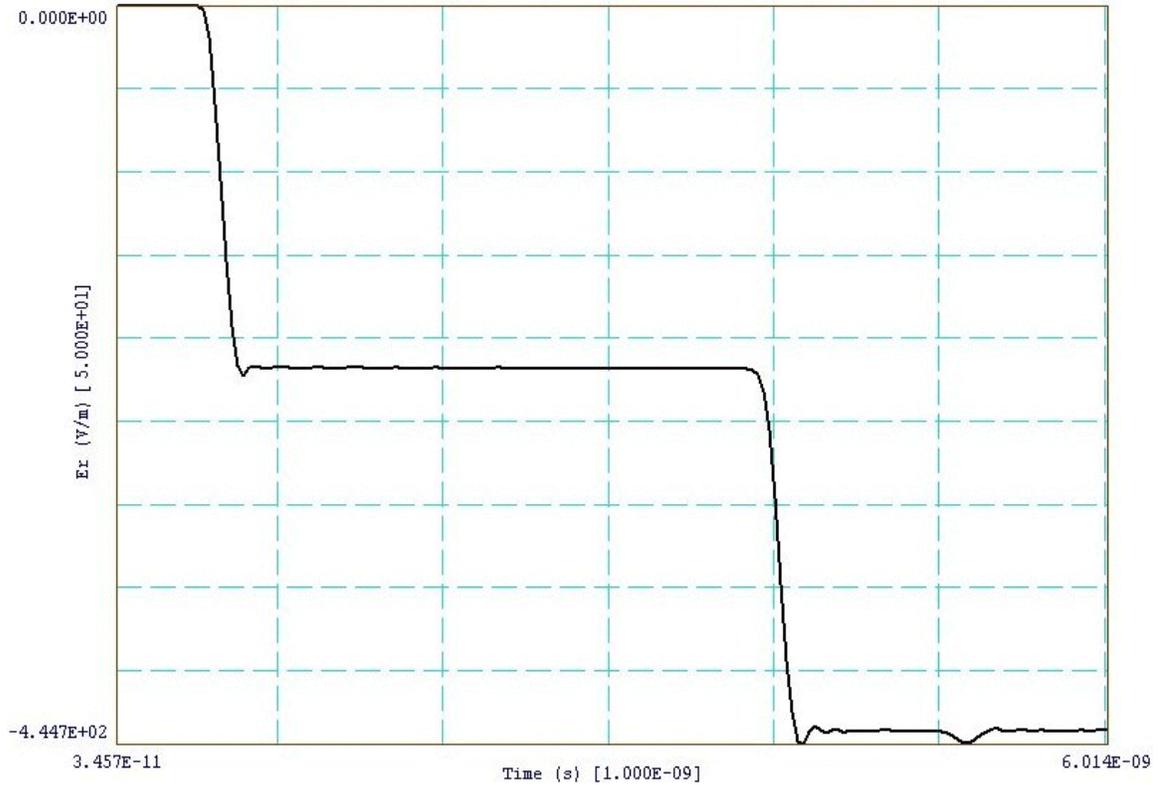


Figure 18. COAX01, radial electric field at $r = 6.5''$ and $z = 12.0''$.

```
REGION 4 Source Smooth.SRC 0.25E-9 125.3
REGION 4 Beta 90.0
```

The file `Smooth.SRC` contains a normalized step function table – the quantity rises from 0.0 to 1.0 over the time interval 0.0 to 1.0. The first factor following the file name converts the rise time to 0.25 ns. The second factor gives a peak total current of 2.0 A. For a radial current density j_r , the total layer current is

$$I = \Delta z 2\pi (r j_r) . \quad (23.4)$$

Taking $I = 2.0$ A and $\Delta z = 2.54 \times 10^{-2}$ m, the quantity rj_r has the value 125.3 A/m. The choice $Beta = 90^\circ$ sets current flow in the positive radial direction. The resulting electric and magnetic fields in the downstream pulse have negative polarity. Finally, the diagnostic statements specify data dumps at times $t = 1.0, 2.0, 3.0$ and 4.0 ns and probes at positions ($r = 6.5''$, $z = 4.0''$) and ($r = 6.5''$, $z = 12.0''$).

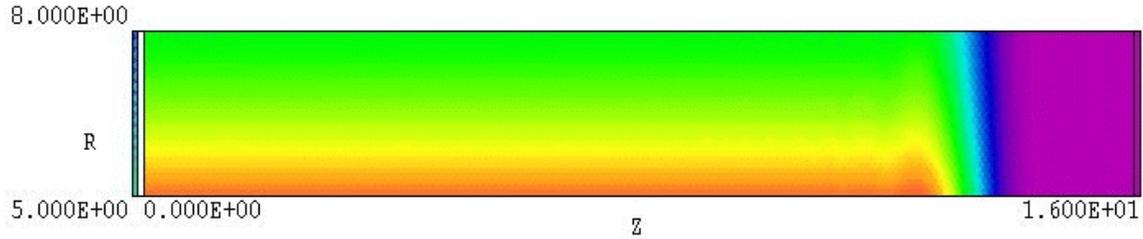


Figure 19. Element plot of $|\mathbf{E}|$ at 2.0 ns for example COAX01.

The predicted field values for a 1 A pulse at a radius of 6.5" are $H_\theta = -0.9640$ A/m and $E_r = 217.1$ V/m. Figure 18 shows the time-dependent electric field measured by the probe at $z = 12.0$ ". The field value in the smooth part of the pulse is 216 V/m. The ratio of the code-predicted electric and magnetic fields is $E_r/H_\theta = 225.2 \Omega$. The signal delay time between the probes at $z = 4.0$ " and 12.0" gives a propagation velocity of 1.783×10^8 m/s, close to the predicted value of 1.793×10^8 m/s. The small bump in the signal at 2.9 ns results from imperfect absorption of the pulse leading edge. The reflected energy is about 0.09% of the incident pulse energy. Figure 19 shows an element plot of electric field as the pulse front moves down the transmission line. The color changes radially because the electric field magnitude is proportional to $1/r$. There is a region of axially-uniform electric and magnetic fields behind the front. At $t = 4.0$ ns, the uniform region fills the entire solution volume. For this data dump, the volume integral function of **VEMP** gives the following values for electric and magnetic field energy in Region 1: $U_e = 1.879 \times 10^{-8}$ J and $U_m = 1.866 \times 10^{-8}$ J. The predicted value of electric field energy is $U_e = CV_o^2/2$, where C is the capacitance of Region 1 and V_o is the pulse voltage (16.87 V). The capacitance per length of the transmission line is 3.313×10^{-10} F/m and the length of Region 1 is 15.7" (0.3988 m). The predicted energy is 1.879×10^{-8} J. In an ideal pulse the electric and magnetic energies should be equal. It is important to note that **VEMP** electric field calculations in Region 4 are invalid because the region contains source current. Energy values should be ignored for this region. The interpolation routines may return anomalously high values for the electric field. It was necessary to set field limits manually to generate the plot of Fig. 19.

The calculation becomes more interesting if we replace the downstream absorber with an open-circuit termination (example COAX02). For the H type solution, we set the properties of Region 3 to $\epsilon_r = 2.8 \times 10^{-6}$ and $\mu_r \times 10^6$. In this case, the pulse reflects with positive polarity in E and negative polarity in H . The reflected pulse is absorbed in the upstream resistive layer a double transit. For long times, the solution is equivalent to a DC



Figure 20. Example COAX02, radial electric field at $r = 6.5''$ and $z = 4.0''$.

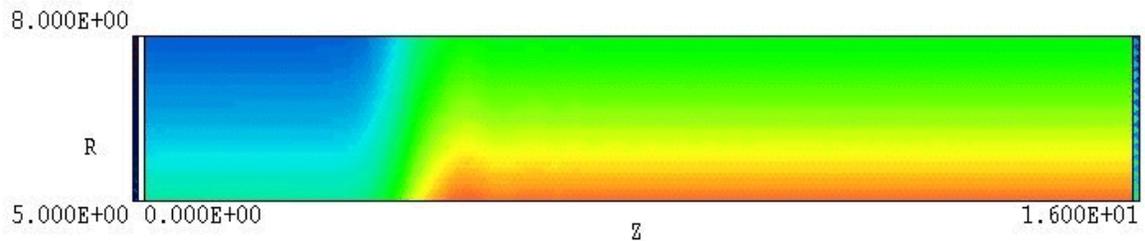


Figure 21. Example COAX02, radial electric field at $r = 6.5''$ and $z = 4.0''$.

voltage on the transmission line. Figure 20 shows the time-dependent electric field measured at position ($r = 6.5''$, $z = 4.0''$). As expected, the reflection results in doubling of the electric field and cancellation of the magnetic field. Figure 21, a plot of the spatial variation of $|\mathbf{E}|$ at $t = 4.0$ ns, shows the reflected pulse front moving backward. A **VEMP** energy integral for the 6.0 ns solution gives the values $U_e = 7.515 \times 10^{-8}$ J and $U_m = 1.242 \times 10^{-12}$ J. The theoretical values are $U_e = 7.516 \times 10^{-8}$ J and $U_m = 0.0$.

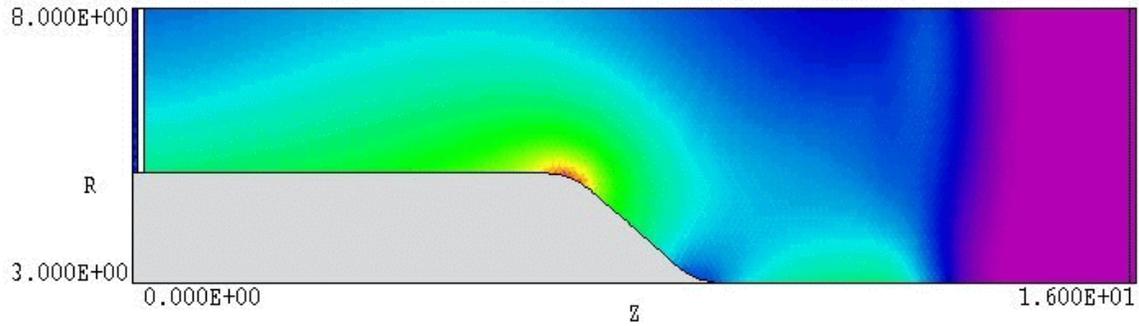


Figure 22. Geometry and distribution of $|\mathbf{E}|$ for the example COAX03: non-ideal impedance transition of a coaxial transmission line.

The example COAX3 (Fig. 22) addresses a calculation that requires a two-dimensional simulation. A coaxial transmission line with impedance $Z_1 = 16.87 \Omega$ undergoes a transition over a short axial length to a line of impedance $Z_2 = 35.20 \Omega$. A step pulse with risetime 0.25 ns and current amplitude 1.0 A is created by a current layer near the left-hand boundary. Ideal absorbing layers are placed adjacent to the left and right-hand boundaries. Probes are located near the left-hand boundary ($r = 6.5''$, $z = 2.0''$) and right-hand boundary ($r = 6.5''$, $z = 14.0''$).

For an ideal discontinuity we expect the following values of reflection and transmission coefficients:

$$\rho = \frac{Z_2 - Z_1}{Z_2 + Z_1} = 0.352, \quad (23.5)$$

$$\tau = \frac{2Z_2}{Z_2 + Z_1} = 1.352.$$

The incident pulse produces an electric field $E_r = 216$ V/m at the upstream probe. The sum of the incident and reflected waves should have an average amplitude $E_r = 292$ V/m. Figure 23 shows the code-generated voltage waveform. The average level of the long-term pulse is close the predicted figure. The delayed rise time of and oscillations on the reflected pulses result from the extended length of the transition and two-dimensional wave reflection effects. Figure 24 shows the electric field in the downstream probe (transmitted pulse). The voltage of the transmitted pulse is predicted to be $V_t = V_o \tau = (16.87)(1.352) = 22.81$ V. The radial electric field is related to the voltage by

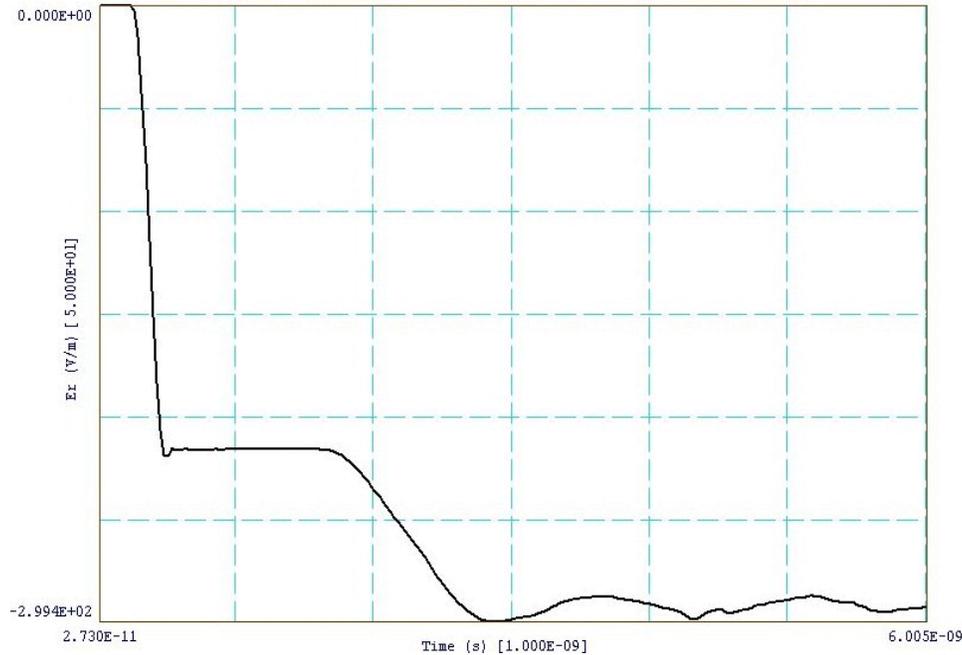


Figure 23. Variation of radial electric field in the upstream transmission line, example COAX03 ($r = 6.5''$, $z = 2.0''$).

$$E_r(r) = \frac{V_o}{r \ln (R_o/R_i)} . \quad (23.6)$$

The predicted field value is 140.9 V. Figure 24 shows the downstream waveform with the **VEMP** digital oscilloscope cursor located near the predicted value. The average value of E_r is close to the expectation with superposed oscillations generated at the transition.

To conclude, we shall use **EMP** to determine the current rise time and transients in a high-voltage Blumlein line control by a spark gap. Figure 25 shows the geometry. The transmission line that we have discussed in the previous examples is connected to ground through a shorting switch. The line ($R_i = 5.0$, $R_o = 8.0''$) is filled with oil ($\epsilon_r = 2.8$) and has a characteristic impedance of 16.87 Ω . The gas-filled switch has a ceramic housing ($\epsilon_r = 6.0$). The switch electrodes and arc column are represented as perfect conductors (Neumann regions in the H type solution). The critical question is how to initiate the solution, because **EMP** handles only dynamic electromagnetic pulses. A DC charge on the line corresponds to $rH_\theta = 0$ at all locations, a null solution that would not evolve in time. We can resolve the problem by noting that a DC charge of voltage V_o on a

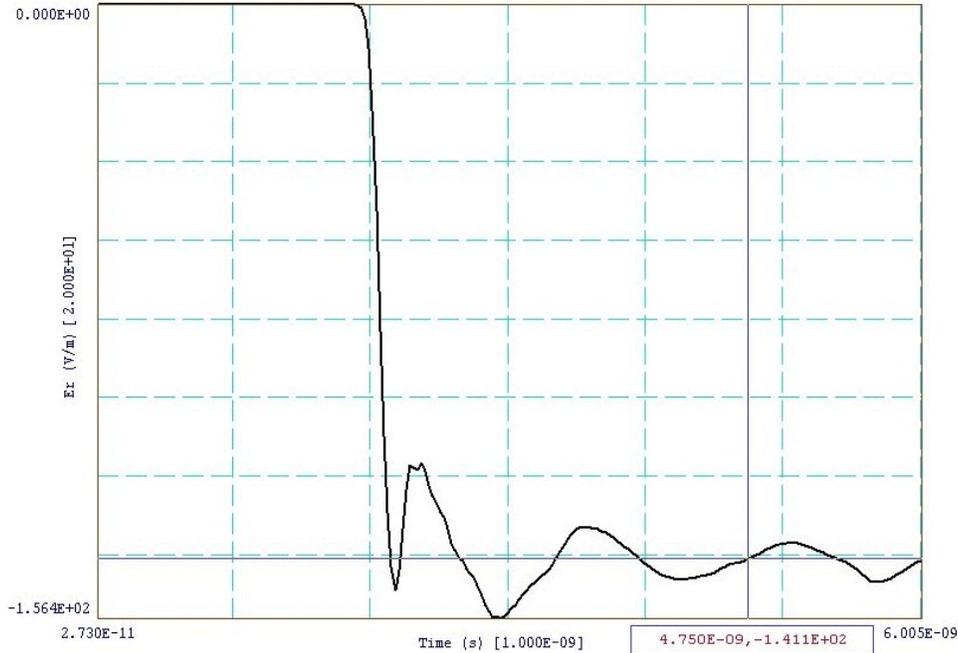


Figure 24. Variation of radial electric field in the downstream transmission line, example COAX03 ($r = 6.5''$, $z = 14.0''$).

transmission line can be represented as the summation of two square pulses with amplitude $V_o/2$ traveling in opposite directions and reflecting from open-circuit terminations at each end of the line. If we suddenly short one end of the line, only the pulse component traveling toward the switch will interact. Therefore, we can estimate the current through and voltage across the switch by generating a dynamic incident step pulse with voltage $V_o/2$.

The properties of the upstream absorber and current source layer are the same as those in previous examples. The step pulse has a rise time of 0.25 ns and a current of 1.0 A. The simulation run time is extended to 15 ns to investigate long-term behavior. Figure 26 shows a measurement of the radial electric field in the upstream transmission line ($r = 6.5''$, $z = 6.0''$). Initially, the inductive switch acts as an open circuit giving a reflected pulse with positive polarity and voltage doubling. Equivalently, the voltage across the switch equals V_o at $t = 0^+$. The line voltage decreases with time approximately as

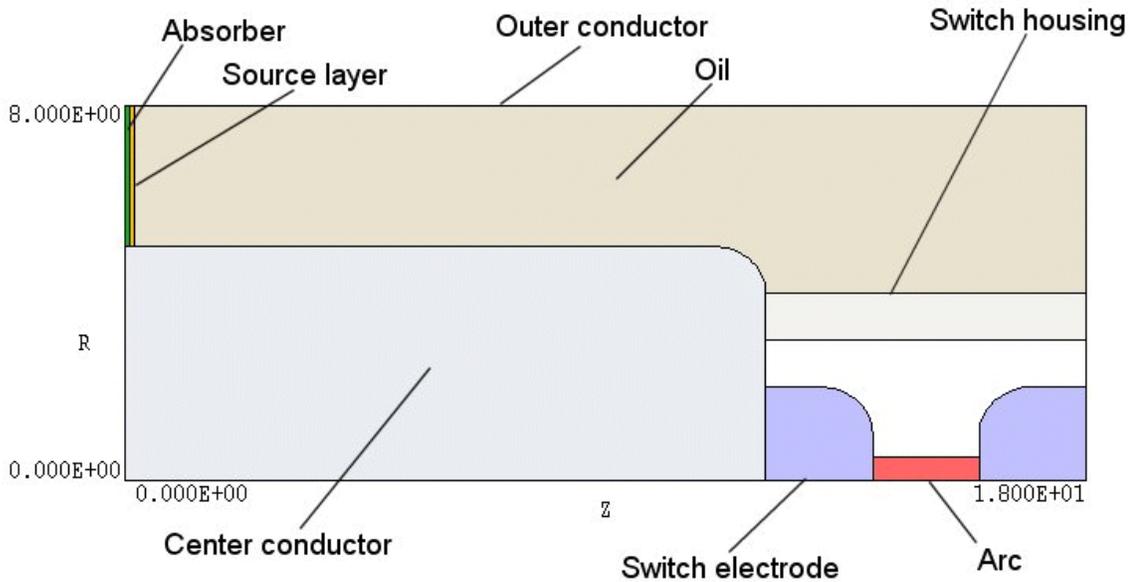


Figure 25. Geometry for example SWITCH.

$$\frac{V(t)}{V_o} = \exp \left[-\frac{t}{L_s / Z_o} \right].$$

In the equation L_s is the switch inductance and Z_o is the characteristic impedance of the transmission line. Inspection of the curve shows that $L_s / Z_o \approx 2.9 \text{ ns}$, or $L_s \approx 49 \text{ nH}$. The oscillations superimposed on the waveform result from the capacitive mismatch of the switch assembly. Figure 27 shows the time-variation of H_0 near the plasma column, proportional to the switch current. To conclude, Figure 28 shows element plots of $|\mathbf{E}|$ at times $t = 1.0, 2.0, 3.0$ and 4.0 ns as the pulse moves down the transmission line and strikes the switch assembly.

Note: The theory of transmission lines and Blumlein lines as pulse modulators is discussed in S. Humphries, **Principles of Charged-particle Acceleration** (John Wiley and Sons, New York, 1986), Chap. 9. The text is available at no charge from Field Precision at www.fieldp.com/cpa.html.

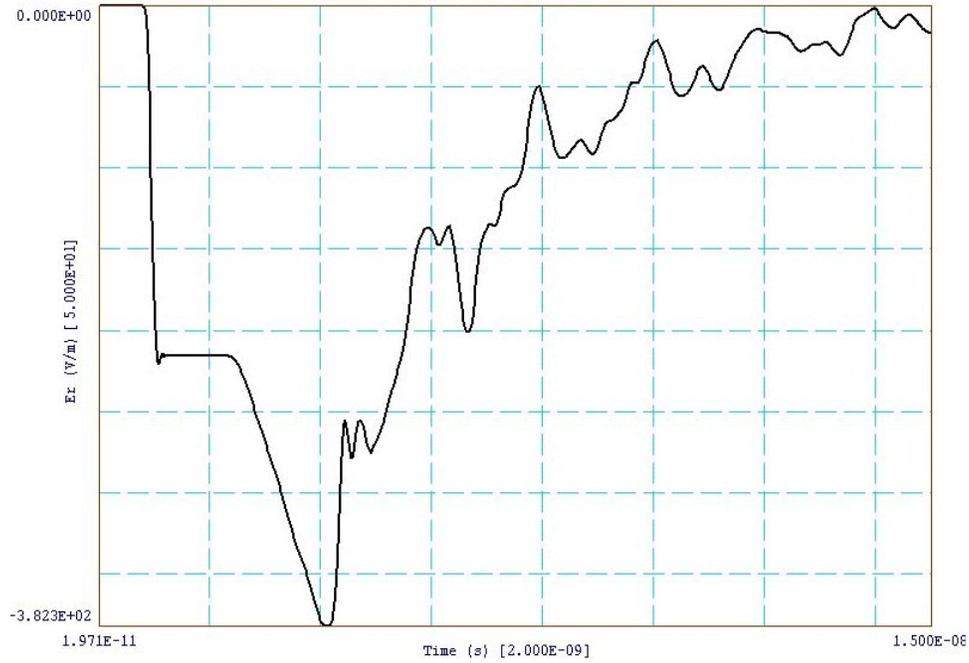


Figure 26. Radial electric field at position ($r = 6.5''$, $z = 6.0''$), example SWITCH

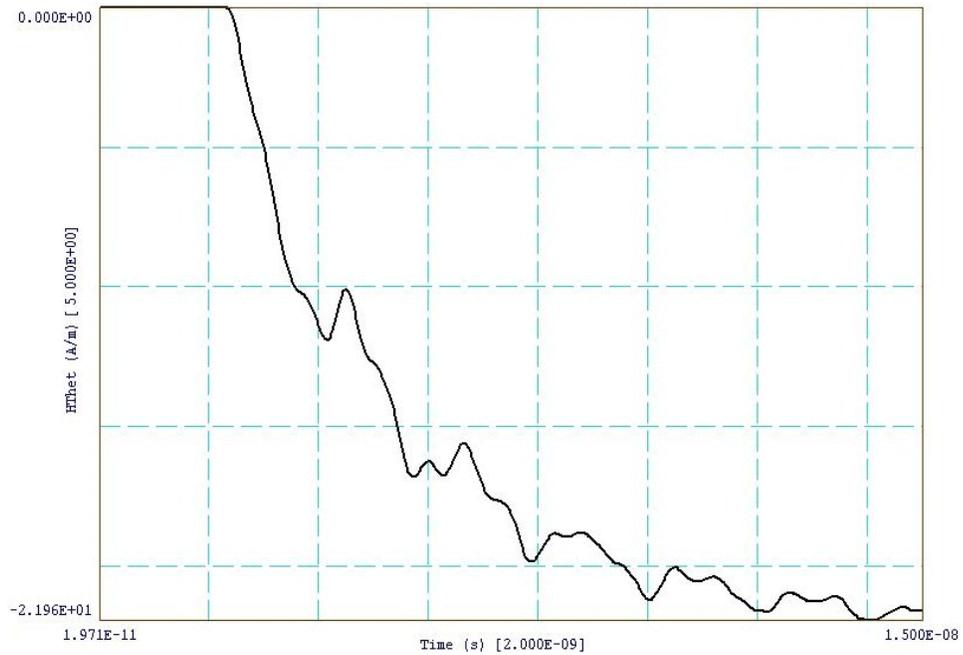


Figure 26. Azimuthal magnetic field near the plasma column at position ($r = 0.6''$, $z = 15.0''$), example SWITCH.

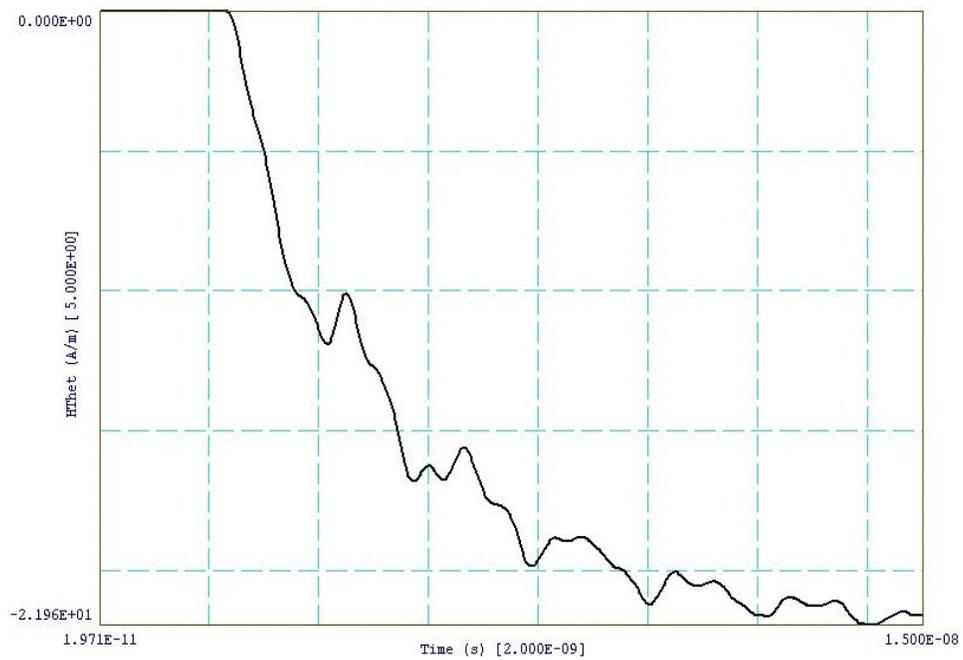


Figure 27. Azimuthal magnetic field near the plasma column at position ($r = 0.6''$, $z = 15.0''$), example SWITCH.

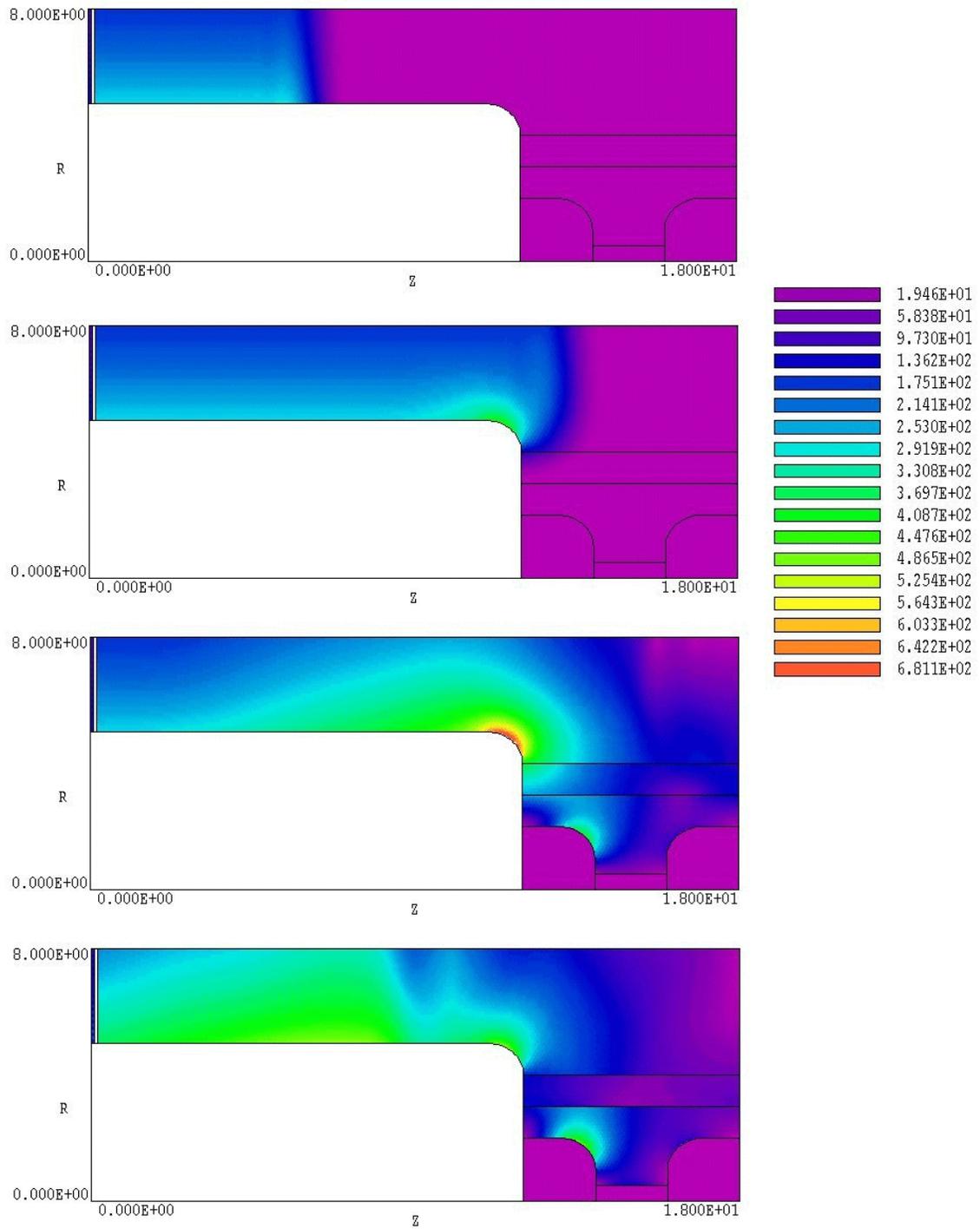


Figure 28. Element plots of $|E|$ at times $t = 0.0, 1.0, 2.0$ and 4.0 ns, example SWITCH.

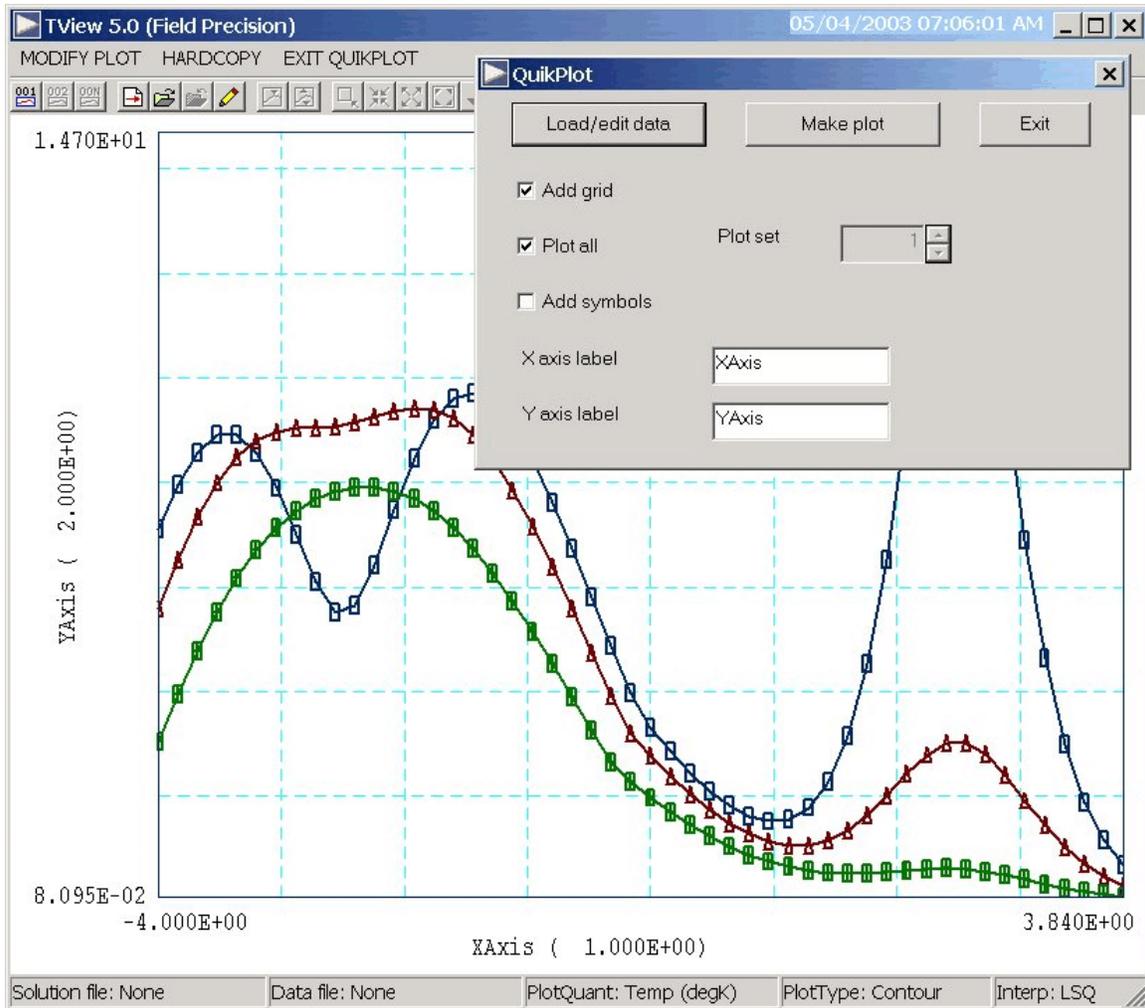


Figure 29. Control dialog superimposed on QuikPlot screen.

21. TView 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 a temporal variation of current density. Click on *QUIK PLOT* in the *TOOLS* menu to bring up a dialog (Fig. 29). Initially, all commands are deactivated except *LOAD/EDIT DATA*. Clicking this command brings up an editor window. You can paste numerical data that you have copied from text files or spreadsheets into the window. You can also use the editor function to make changes or to save the data to a file.

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

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

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

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