EMP 8.0
Finite-element time-domain electromagnetics

PO Box 13595, Albuquerque, NM 87192 U.S.A.
Telephone: +1-505-220-3975
EMail: techinfo@fieldp.com  URL: http://www.fieldp.com
Contents

1 Introduction ........................................... 4
   1.1 Program functions .................................. 4
   1.2 Learning EMP ..................................... 6
   1.3 Finite-element solution procedure .................. 6
   1.4 Scripts and data files ............................... 8

2 Electromagnetic theory in EMP ......................... 10
   2.1 Electromagnetic waves in two dimensions .......... 10
   2.2 Courant stability condition and numerical dispersion . 12
   2.3 Current sources .................................... 14
   2.4 Standard boundary conditions ....................... 14
   2.5 Absorbing boundaries ................................ 15

3 Creating and analyzing a solution with EMP .......... 19
   3.1 Mesh generation ..................................... 19
   3.2 Creating the EMP script ............................ 20
   3.3 Finite-element solution ............................. 22
   3.4 Solution analysis .................................... 22

4 EMP solution reference .................................. 27
   4.1 Organizing EMP solutions ........................... 27
   4.2 EMP script format ................................... 28
   4.3 Control commands .................................... 30
   4.4 Region commands .................................... 31
   4.5 Diagnostic commands ................................ 35
   4.6 Running EMP ........................................ 36
   4.7 Format of the EMP output file ....................... 38

5 Mathematical functions for drives, sources and material properties ............ 41
   5.1 Function parser ...................................... 41
   5.2 Temporal variations of drives and sources .......... 41
   5.3 Spatial variations of material properties .......... 42

6 Solution analysis reference ............................. 44
   6.1 File menu commands ................................ 44
   6.2 Plot menu commands ................................ 45
   6.3 Analysis menu commands ............................. 50
   6.4 Scan plot menu ....................................... 52
   6.5 Analysis script commands ............................ 53

7 EMP tutorials ........................................... 56
   7.1 Coaxial transmission lines .......................... 56
   7.2 Transients in a high-voltage switch ................ 62
   7.3 Pulsed electric field probe ........................ 63
   7.4 Grating with a graded dielectric constant .......... 67
### 7.5 Modeling pulsed magnetic fields

---

### 8 Probe – history file plot utility

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.1 Introduction</td>
<td>72</td>
</tr>
<tr>
<td>8.2 Loading data files</td>
<td>73</td>
</tr>
<tr>
<td>8.3 Plot settings</td>
<td>73</td>
</tr>
<tr>
<td>8.4 Plot functions</td>
<td>75</td>
</tr>
<tr>
<td>8.5 Information</td>
<td>76</td>
</tr>
</tbody>
</table>
Introduction

1.1 Program functions

EMP is a versatile tool for calculations of time-dependent electromagnetic fields. The program runs on personal computers operating under any version of Windows. EMP 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 employs unique finite-element methods on conformal triangular meshes for accurate representations of curved and sloped material boundaries. Materials are characterized by values of relative dielectric constant $\varepsilon_r$, relative magnetic permeability $\mu_r$ and electrical conductivity $\sigma$. These quantities may be fixed over a region of the solution volume or may vary with position following specified mathematical functions. 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. Material types include vacuum, conductors, or lossy dielectrics and ferrites. In addition to the standard short-circuit and open-circuit boundary conditions, EMP
Figure 2: Benchmark test – radiation from a pulsed dipole antenna. Left: EMP results, $r$-$z$ plots of $|E|$ at 1 ns intervals. Right: cross-section plots of $|E|$ from J.G. Maloney, G.S. Smith and W.R. Scott, IEEE Trans. Antennas Propagation AP-38, 1059 (1990)

supports matched-termination absorbing boundaries to represent open-space conditions.

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 and material properties at specified times or temporal variations at up to 20 probe locations. All data files are in structured text format for easy transfer to spreadsheets or mathematical analysis programs. You can quickly analyze spatial data in an interactive, graphical environment (Fig. 1). The program generates quantitative data and a wide variety of plots. The utility program Probe is supplied for the analysis of temporal information. With care in preparation, EMP can quickly provide results of high accuracy. To illustrate, Fig. 2 shows a comparison of EMP results for a pulsed dipole antenna to published results.
1.2 Learning EMP

The size of this manual reflects the extensive capabilities of the EMP package. It is not necessary to read the entire document to address most applications. We have organized the chapters to help you get started quickly. You can investigate advanced topics as you gain more experience. The first step is to follow the walkthrough example in Chapter 3. This will give you an overview of the package organization and the solution process. Then, run some of the prepared examples supplied with the package, particularly the tutorials described in Chap. 7. The examples give insights into solution options and may serve as templates for your own work. Here are suggestions for additional activities:

- Browse remaining sections in this chapter to review basics of finite-element calculations. You may also want to read the following chapter on electromagnetic theory. You will probably return to the material to resolve issues (such as boundary conditions) as you get more involved with the programs.

- Scan the section on FPController (the component program launcher) in the Mesh manual.

- Be sure to read Chapter 2 of the Mesh manual, which reviews essential concepts for conformal meshes in finite-element solutions.

- Chapter 3 of the Mesh manual follows a walkthrough example to introduce construction of a solution geometry with the interactive Mesh drawing editor. The exercise introduces fundamental tools you will need for you own simulations.

- As you gain experience you will want to take advantage of the full range of Mesh/EMP capabilities. Chapter 4 of the Mesh manual is a comprehensive reference on the Mesh drawing editor, while Chap. 5 covers processing, plotting and repair of meshes. Chapters 4 and 6 in this manual summarize the solution and analysis functions of EMP. Methods for setting material properties that vary smoothly in space are described in Chap. 5.

- Later chapters in the Mesh manual cover advanced techniques. Chapters 6 and 7 show how to make direct entries in scripts to invoke advanced control features. Chapter 8 describes how to create meshes directly from photographic and data images. This feature is useful to model complex or irregular systems that are difficult to describe with simple geometric specifications.

1.3 Finite-element solution procedure

An in-depth understanding of finite-element numerical methods is not necessary to use EMP. Nonetheless, it is important to have a clear idea of fundamental concepts to create effective solutions. This section describes background material to understand the steps in an EMP solution.

The term field indicates a quantity (scalar or vector) defined over a region of space. Examples of fields include the vector electric field $E$ in an electrostatic solution, electric and magnetic fields in an electromagnetic solution and the scalar temperature $T$ in a thermal solution. Variations of field quantities are usually described by continuous partial differential equations, such
as the Helmholtz equation. These equations can be solved directly by analytic methods if the system geometry and material properties are simple. Analytic solutions are extremely difficult in systems with asymmetric structures or non-linear materials. Furthermore, analytic results are often be expressed in terms of series expansions that must be evaluated numerically. For all but the simplest problems, it is usually quicker and more accurate to employ a direct numerical approach.

The fundamental issue in numerical field solutions is that digital computers cannot directly solve continuous equations. On the other hand, computers are well suited to solving large sets of coupled linear equations. The goal of all numerical field methods (finite-difference, finite-element or boundary-element) is to convert the governing differential equations into a set of coupled linear equations. The solution of the linear equations approaches the results of the differential equation when the set becomes large.

The basis of the finite-element approach is to divide the full solution volume into a number of small volumes, or elements. Here, the term small indicates that element dimensions are much less than the scale length for variations of field quantities. The division of the volume is called the computational mesh. Figure 3 shows the type of mesh used for the two dimensional solutions of EMP. The figure defines three terms that will be used throughout this manual:

- **Element.** Volume divisions of the system.
- **Nodes.** Points where elements intersect.
- **Facets.** Surfaces between two elements.

The elements in Fig. 3 have triangular cross-sections. In a planar solution, the cross-section lies in the x-y plane and the elements extend an infinite distance in the z direction. In a cylindrical
solution, an element is a figure of revolution about the axis with a triangular cross-section in the $z$-$r$ plane.

The mesh in Fig. 3 has the important property of *conformality*. The term means that the triangles have been specially shaped to conform to the boundaries between materials (*regions*). As a result, each element has an unambiguous material identity. The finite-element method is based on two approximations: 1) material properties in an element are uniform and 2) elements are small enough so that the field quantities can be approximated by simple interpolation functions. With these assumptions, the governing differential equation can be integrated over elements surrounding a node to yield a linear equation. This equation relates the field quantity at the node to those at the surrounding nodes. The coupled set has one linear equation for each node in the mesh. In EMP, solution of the set representing the Helmholtz equation gives either $E_z$, $rE_\theta$, $H_z$ or $rH_\theta$ (depending on the system symmetry and field polarity). We can then perform two-dimensional interpolations to find the field quantities at intervening points, or take numerical spatial derivatives to find the other field components.

An EMP solution involves the following steps:

1. You define the boundaries of material objects in a solution volume for your application. The task is usually accomplished with the drawing editor of Mesh. You can also specify control information such as the target element sizes that may affect the accuracy and run time of the solution. The result is a text record (*script*) with sets of line and arc vectors that outline materials and metal boundaries.

2. Mesh analyzes the boundary specifications and automatically generates a set of conformal triangles such as those of Fig. 3. The program creates an output file that lists the locations of nodes and the identities of elements.

3. You define the material properties of regions in the solution volume. Usually, this task is performed through an interactive dialog in EMP.

4. EMP reads the mesh geometry and applies the material parameters to generate the linear equation set.

5. EMP advances the coupled equations in time using a centered differencing method with second-order accuracy. The program records field values at all points in space at specified times or records time variations of quantities at specified positions. The resulting files serve as a permanent record of the solution that are always available for analysis.

6. You can use the interactive graphical environment of EMP to explore the solution. The program creates a wide variety of plots and performs quantitative calculations of field quantities. You can also write scripts for automatic control of complex analyses.

### 1.4 Scripts and data files

Mesh and EMP read and generate several types of files. For a typical solution it is not necessary for you to deal directly with the files – the user interface takes care of data organization. On the other hand, if you generate a large amount of data it’s a good practice to archive application results in individual folders. Furthermore, there are advanced features of Mesh and EMP that require direct entries to the input scripts. For convenience, all input scripts and
Table 1: **EMP** files

<table>
<thead>
<tr>
<th>Name</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>MName.MIN</td>
<td><strong>Mesh</strong> input script (definition of foundation mesh and region outlines)</td>
</tr>
<tr>
<td>MName.MLS</td>
<td><strong>Mesh</strong> diagnostic listing</td>
</tr>
<tr>
<td>MName.MOU</td>
<td><strong>Mesh</strong> output (node locations and element identifies)</td>
</tr>
<tr>
<td>EName.EIN</td>
<td><strong>EMP</strong> input script (run control and material properties)</td>
</tr>
<tr>
<td>EName.ELS</td>
<td><strong>EMP</strong> diagnostic listing</td>
</tr>
<tr>
<td>EName.001,..</td>
<td><strong>EMP</strong> data dumps, field quantities at all positions at specified times</td>
</tr>
<tr>
<td>EName.P01,..</td>
<td><strong>EMP</strong> probe files, field quantities over time at specified positions</td>
</tr>
<tr>
<td>AName.SCR</td>
<td><strong>EMP</strong> script for automatic data analysis</td>
</tr>
</tbody>
</table>

Output data files are in text format. Both **Mesh** and **EMP** feature integrated text editors. Table 1 lists the file types and functions in the **EMP** package. Note that the suffixes indicate the file function.
2 Electromagnetic theory in EMP

2.1 Electromagnetic waves in two dimensions

The reference S. Humphries, *Field Solutions on Computers* (CRC Press, Boca Raton, 1997) gives detailed information on the application of finite-element methods to two-dimensional electromagnetic pulses. This section summarizes the basic theory underlying EMP. The program applies the following form of the Maxwell equations:

\begin{align}
\nabla \times \mathbf{H} &= \varepsilon \frac{\partial \mathbf{E}}{\partial t} + j_0 + \sigma \mathbf{E}, \\
\n\nabla \cdot \mu \mathbf{H} &= 0, \\
\n\nabla \times \mathbf{E} &= -\mu \frac{\partial \mathbf{H}}{\partial t}, \\
\n\n\nabla \cdot \varepsilon \mathbf{E} &= 0.
\end{align}

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 \( (j_0) \) may represent drive structures like coupling loops and capacitive probes. Dielectrics and ferromagnetic materials are isotropic and linear. Furthermore, the magnetic permeability \( \mu \), dielectric constant \( \varepsilon \) and conductivity \( \sigma \) are constant in time and have a uniform value within an element. The inclusion of conductivity enables modeling of pulse attenuation and implementation of absorbing boundaries.

Equations 1 through 4 may be combined into individual equations for \( \mathbf{E} \) and \( \mathbf{H} \). The electric field equation is

\[ -\nabla \times \frac{1}{\mu} \nabla \times \mathbf{E} = \frac{\partial}{\partial t} (\nabla \times \mathbf{H}) = \varepsilon \frac{\partial^2 \mathbf{E}}{\partial t^2} + \frac{\partial j_0}{\partial t} + \sigma \frac{\partial \mathbf{E}}{\partial t}. \]

Using Stoke’s theorem, we can rewrite Eq. 5 as

\[ -\oint \frac{1}{\mu} \nabla \times \mathbf{E} = \int \int dS \left[ \varepsilon \frac{\partial^2 \mathbf{E}}{\partial t^2} + \frac{\partial j_0}{\partial t} + \sigma \frac{\partial \mathbf{E}}{\partial t} \right]. \]

The surface integral on the right hand side extends over the region bounded by the circuital 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 polarization classes:

- **E** type, with electric field polarization in the \( z \) direction, \( E_z \).
- **H** type with magnetic intensity polarization in the \( z \) direction, \( H_z \).
Equation 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. 6 to a large number of coupled linear equations. A linear equation may be generated for each node in the solution space by taking surface and line integrals of the governing equation in the $x$-$y$ plane over the region shown in Fig. 4 (blue line). The right-hand side of Eq. 6 is converted using time-centered difference operators. The solution procedure involves the simultaneous advance of all equations in small time steps $\Delta t$ to determine new values of $E_z$ at all positions. It is important to recognize that $E_z$ values are defined at nodes while the quantities $\mu$, $\sigma$, 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,

$$H_x = -\frac{1}{\mu} \int_0^t dt' \frac{\partial E_z}{\partial y},$$  
(7)

$$H_y = \frac{1}{\mu} \int_0^t dt' \frac{\partial E_z}{\partial x}.$$  
(8)

Equations 7 and 8 imply that lines of $H$ in the $x$-$y$ plane lie on contours of $E_z$. This fact is used to create field line plots in EMP.

The equation for $H$ type pulses in planar geometry is similar to Eq. 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:

$$E_x = \frac{1}{\epsilon} \left[ \int_0^t dt' \frac{\partial H_z}{\partial y} + \int_0^t dt' j_{ox} + \sigma \int_0^t dt' E_x \right],$$  
(9)

$$E_y = \frac{1}{\epsilon} \left[ -\int_0^t dt' \frac{\partial H_z}{\partial x} + \int_0^t dt' j_{oy} + \sigma \int_0^t dt' E_y \right].$$  
(10)
Table 2: Solution types in EMP, primary and second field quantities

<table>
<thead>
<tr>
<th>Type</th>
<th>Function</th>
<th>Primary</th>
<th>Secondary</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E$</td>
<td>Planar, variations in $x$ and $y$</td>
<td>$E_z$</td>
<td>$H_x$, $H_y$</td>
</tr>
<tr>
<td></td>
<td>Cylindrical, variations in $r$ and $z$</td>
<td>$rE_\theta$</td>
<td>$H_r$, $H_z$</td>
</tr>
<tr>
<td>$H$</td>
<td>Planar, variations in $x$ and $y$</td>
<td>$H_z$</td>
<td>$E_x$, $E_y$</td>
</tr>
<tr>
<td></td>
<td>Cylindrical, variations in $r$ and $z$</td>
<td>$rH_\theta$</td>
<td>$E_r$, $E_z$</td>
</tr>
</tbody>
</table>

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 (Sect. 7.1). Because of the nature of the curl operator, in cylindrical solutions EMP determines the quantities $rE_\theta$ and $rH_\theta$. It is easy to show that contours of these quantities are parallel to lines of $H$ and $E$ respectively in the $r$-$z$ plane. For $H$ type pulses, the correspondence holds only in regions with zero conductivity and source current.

2.2 Courant stability condition and numerical dispersion

The choice of $\Delta t$ in a time-domain electromagnetic solution is critical. For numerical stability, it must satisfy the Courant condition,

$$\Delta t < \min\left[\sqrt{\frac{\varepsilon_i \mu_i}{l_i}}\right] = \min \left[\frac{l_i}{v_i}\right]. \tag{11}$$

The quantity $v_i$ is the speed of light in element $i$ and $l_i$ is the minimum dimension of the element. Equation 11 states that $\Delta t$ 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 terminates 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. 5a illustrates the long-distance propagation of a Gaussian pulse. A one-dimensional pulse with spatial width 0.067 m travels 9 m. Although energy is conserved, 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 5b 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
Figure 5: Numerical dispersion in time-domain electromagnetic solution. 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).
Table 3: Interpretation of source quantities

<table>
<thead>
<tr>
<th>Pulse type</th>
<th>Geometry</th>
<th>Quantity</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E$</td>
<td>Planar</td>
<td>$dj_z/dt$</td>
<td>A/m$^2$-s</td>
</tr>
<tr>
<td></td>
<td>Cylindrical</td>
<td>$dj_\theta/dt$</td>
<td>A/m$^2$-s</td>
</tr>
<tr>
<td>$H$</td>
<td>Planar</td>
<td>$J_x, J_y$</td>
<td>A/m$^2$</td>
</tr>
<tr>
<td></td>
<td>Cylindrical</td>
<td>$rJ_z, rJ_r$</td>
<td>A/m</td>
</tr>
</tbody>
</table>

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 $2w_e^2/w_e$, where $w_e$ is the element width.

2.3 Current sources

Source current density may be assigned to any material region in the solution space. The current density $j$ is specified by $H$ type solutions and the time derivative $dj/dt$ for $E$ type solutions. EMP assumes uniform spatial distribution and a time dependence given by either a table of values or a mathematical function. Following the discussion of Sect. 2.1, the current-density component $dj_z/dt$ in a planar geometry generates $E$ 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 $\beta$ represents the angle of the current density relative to the $x$ axis.

In cylindrical coordinates, an azimuthal current-density variation $dj_\theta/dt$ drives $E$ type pulses and current in the $z$-$r$ plane drives $H$ type pulses. In the latter case, the reference parameter $\beta$ is the angle of the current density relative to the $z$ axis. Instead of taking a value of $j$ in $H$ type solutions that is constant over a region, the convention in EMP is to use a constant value of the quantity $\Gamma = rj$. 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 $\Delta z$ and a current flux $\Gamma = \Gamma_0 \hat{r}$. The total radial current in the region is $I_r = 2\Gamma_0 \Delta z$. Similarly, the axial current through an annular region with inner and outer radii $r_i$ and $r_o$ with an axial flux $\Gamma = \Gamma_0 \hat{z}$ equals $I_z = 2\pi \Gamma_0 (r_o - r_i)$. Table 3 shows input quantities for source current in the different program solution modes.

2.4 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 primary field component ($E_z$, $rE_\theta$, $H_z$ or $rH_\theta$) equals zero. For $E$ type pulses, the condition is equivalent to an open-circuit termination on the boundary. In this case, the magnetic intensity $H$ 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 $rE_\theta$ 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 Void command.

You can also define an internal region with the open-circuit property for $E$ type pulses by assigning a high value of the medium characteristic impedance:

$$\frac{Z}{Z_0} = \sqrt{\frac{\mu_r}{\epsilon_r}} \gg 1.0,$$

where $Z_0 = 377.3 \ \Omega$. When defining a high or low impedance material, be sure to pick values of the relative dielectric constant and magnetic permeability such that

$$\sqrt{\epsilon_r \mu_r} \simeq 1.0.$$  

Equation 13 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 not be possible to find a practical time step consistent with the Courant condition. 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 Fixed command. In this case the calculated quantity ($E_z$, $rE_\theta$, $H_z$ or $rH_\theta$) 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).

### 2.5 Absorbing boundaries

We can approximate 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 – the termination acts like an infinite extension of the line. The boundary region beyond the termination layer should have the following properties for the two types of pulses:

- **E type**: special Neumann condition.
- **H type**: fixed condition $H_z = 0.0$ or $rH_\theta = 0.0$.

In most cases the termination layer is located on the outer surface of the solution volume so that the Neumann condition for $E$ type pulses is automatically applied. For $H$ type pulses, it is necessary to add a line region with the Fixed condition.

As shown in Fig. 6a, a thin resistive layer with conductivity $\sigma$ and width $\Delta$ is located between the boundary and a material with electrical properties $\epsilon$ and $\mu$. In the limit that $\Delta$ is much smaller than the spatial width of the pulse, the electric field is approximately uniformly over the layer while $H$ falls to zero. If the resistor maintains the same conditions on $E$ and $H$ as an infinite extension of the material, there is no reflected wave from the boundary. We can derive the required value of $\sigma$ by noting that a field $E_x$ creates a linear current density equal to
\[ J_x = \sigma \Delta E_x. \]  
(14)

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

\[ Z_0 = \sqrt{\mu/\epsilon} = \frac{1}{\sigma \Delta}. \]  
(15)

The conductivity for a matched termination is

\[ \sigma = \frac{1}{\Delta \sqrt{\mu/\epsilon}}, \]  
(16)

where \( \epsilon \) and \( \mu \) are the electrical properties assigned to both the termination layer and the adjacent material.

Figure 7 shows the performance of a termination boundary for normally incident pulses. A Gaussian pulse with spatial width 0.067 m enters from a vacuum region. The figure plots the reflected pulse amplitude as a function of the termination element width \( \Delta \). 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. 6b). For an angle \( \theta \) 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
Figure 7: 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).

\[
\frac{E_r}{E_0} = \frac{1 - \sin \theta}{1 + \sin \theta}.
\]  

(17)

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 8 shows a test of Eq. 17 using a Gaussian pulse with 0.067 m width incident from vacuum. Figure 18a shows results for a 45$^\circ$ 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 $\sigma = 0.5301$ S/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. 17. Figure 18b shows an example with $\theta = 30^\circ$. The amplitude of the reflected pulse is -0.37 compared to a prediction of -0.33. One advantage of the termination layer method is that we can improve the performance if we know the general direction of the pulses. For example, if pulses strike the surface at an angle $\theta$, we can optimize absorption by setting the layer conductivity equal to

\[
\sigma = \frac{\sin \theta}{\Delta \sqrt{\mu / \varepsilon}}.
\]  

(18)

Figure 18c shows improvement of the solution of Fig. 18b with a layer conductivity of $\sigma = 0.2650$ S/m. In this case, the reflection coefficient is only -0.025.
Figure 8: 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 $30^\circ$ pulse absorption by adjusting the layer conductivity to $\sigma = 0.2650$ S/m.
Figure 9: Geometry for the CONICAL example. A pulse on a transmission line in a finite-width ground plane drives a conical antenna. Dimensions in cm.

3 Creating and analyzing a solution with EMP

3.1 Mesh generation

As a quick introduction to EMP techniques and capabilities, we shall work through an example. Move the files CONICAL.MIN and SQPULSE.SRC (supplied with the example library) to a working directory such as \tricomp\buffer. The first file (input for the Mesh program) describes the geometry while the second defines the pulse waveform. All files are in text format – they may be inspected and modified with an editor. Figure 9 shows a z-r plot of the cylindrical system. (Note that parts are figures-of-revolution about the z axis). A source layer carrying a radial current (near the bottom left-hand side) creates a TEM pulse in the 70 Ω, 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 free-space condition.

The strategy for generating the mesh is influenced by the nature of the electromagnetic pulse solution (Chapter 2 reviews the theory of electromagnetic waves in two-dimensional systems.) A TEM pulse in a coaxial transmission line is an $H$ type solution. EMP calculates the quantity $rH_\theta$ from a finite-element solution of the Helmholtz equation and then determines the electric field components ($E_z$, $E_r$) in terms of derivatives of the primary quantity. 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. Therefore, we use the following strategy for mesh generation:

- Fill the active solution volume (areas not shaded in gray in Fig. 9) with a resistive medium (Region 1) that will comprise the absorption layers in the transmission line and propagation volume. Note that the inner and outer conductors of the transmission line are
excluded from the solution volume.

- Carve out the inside of the solution volume with Region 2 representing air. The 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 line absorber terminates any reflected energy traveling backward.

- Define the thin radial slice that carries the current to initiate the pulse (Region 3).

- Include a line region to represent the fixed condition \( r H_\theta = 0.0 \). The region has three segments: 1) the axis \( r = 0.0 \), 2) the outer boundary of the transmission line absorber and 3) the outer boundary of the absorber on the periphery of the solution volume.

Run Mesh and load CONICAL.MIN. Pick the Edit script/graphics command to enter the drawing editor. Here, you can use the display capabilities to confirm the region assignments:

- Region 1: complex outline of the solution volume including the absorbing layers (Filled)
- Region 2: the vacuum portion of the solution volume excluding the absorbing layers (Filled)
- Region 3: the source layer to initiate the pulse (Filled)
- Region 4: the lines that will assume the condition \( r H_\theta = 0.0 \) on the axis and adjacent to the absorbing layers (Open).

Because sources should not contact absorbers, there is a small separation between the source and the absorbing layer in the transmission line. Abandon the drawing and return to the main menu. Click the Process command. Choose the Save mesh (MOU) command. You can now close or minimize Mesh.

3.2 Creating the EMP script

Run EMP from TC. The screen is initially blank and the status bar indicates that the program is waiting for input. Note the prominent tools marked 1, 2 and 3. The notation is a reminder of the three steps in an electromagnetic solution:

1. Set up program controls and material properties.
2. Generate and solve the finite-element equations.
3. Analyze the solution.

To start the first operation, we must identify the Mesh output file that defines the system geometry. Click the 1 tool or the Setup menu command. In the dialog, choose the file CONICAL.MOU. EMP loads the information and displays the dialog of Fig. 10. Note that the grid contains a row for each mesh region with a notation of the Fill status.

The entries in the Control parameter section of the dialog have the following functions:
Figure 10: Dialog to set EMP control parameters and region properties.

- **SOLUTION TYPE.** Choose the field polarization. An entry of $E$ designates that the main field component is $E_z$ or $rE_\theta$ and $H$ corresponds to $H_z$ or $rH_\theta$.

- **GEOMETRY.** Set the two-dimensional symmetry of the solution: **planar** or **cylindrical**. A planar solution varies in $x$-$y$ and has infinite length in $z$, while a cylindrical solution has symmetry in $\theta$.

- **TMAX.** The duration of the initial-value solution in seconds.

- **DT.** The time-step for the difference solution. If you leave the field blank, EMP estimates a value that satisfies the Courant stability condition in all elements.

- **DUNIT.** A factor to convert the units used for coordinates in the Mesh file to meters. Choose the appropriate units in the list box. For other units, pick Custom and enter a value for DUnit directly in the script with an editor. The quantity DUnit equals the number of mesh units per meter: 39.37 for inches, 100.0 for cm.

The entries in the column options of the region grid box determine the nature of the corresponding region. The first five columns apply to standard materials: $EpsiR$ ($\epsilon_r$, the relative dielectric constant), $MuR$ ($\mu_r$, the relative magnetic permeability), $Cond$ ($\sigma$, the electrical conductivity in S/m), $Source$ (the drive current density or its derivative) and $Beta$ (the source direction of the current in degrees for $H$ type solutions). The definition of the Source quantity depends on the field polarity and solution symmetry. Section 2.3 gives a detailed discussion.

The values shown in Fig. 10 define an $H$ type solution with cylindrical symmetry. Dimensions are in cm and the maximum run time is 8.01 ns. Regions 1, 2 and 3 have the dielectric
properties of air ($\varepsilon_r = 1.0, \mu_r = 1.0$). The conductivity in Region 1 is determined from Eq. 16 as
\[ \sigma = \frac{1}{Z_0\Delta}, \tag{19} \]
where $Z_0$ is the characteristic impedance of Region 2 ($377.3 \, \Omega$) and the layer thickness is $\Delta = 2.5 \times 10^{-3} \, \text{m}$. The source current $rj_r$ of Region 3 points in the radial direction ($\beta = 90.0^\circ$) and is determined by values in the tabular file SQPULSE.SRC. The Fixed designation for Region 4 sets the condition $rH_\theta = 0.0$. The entries in the Diagnostics section call for four data dumps of spatial information at a time interval of 2.0 ns and two probes at positions $[r = 1.5 \, \text{cm}, \, z = -40.0 \, \text{cm}]$ (inside the transmission line) and $[r = 30.0 \, \text{cm}, \, z = 30.0 \, \text{cm}]$ (in the pulse propagation volume). When you have completed data entry, click OK and save the information in the file CONICAL.EIN.

We shall choose a peak source current $I_0 = 2.0 \, \text{A}$. In this case, 1.0 A is lost in the upstream absorber adjacent to the source and 1.0 A travels down the transmission line as a TEM pulse. For the cylindrical solution, we enter $rj_r$, where the peak value is given by:
\[ rj_r = \frac{I_0}{2\pi\Delta}. \tag{20} \]
For a layer thickness $\Delta = 2.5 \times 10^{-3} \, \text{m}$, the source magnitude is $rj_r = 127.3 \, \text{A/m}$. The table SQPULSE.SRC defines a normalized function that rises smoothly from 0.0 to 1.0 over the interval $0.0 \leq t \leq 1.0$ and falls to 0.0 over the interval $4.0 \leq t \leq 5.0$. We must introduce normalization factors for the desired amplitude and duration. We must edit the input file directly to implement this advanced feature. Click the EMP command File/Edit script (EIN) and choose CONICAL.EIN. Modify the Source command for Region 3 to read:
\[ \text{Source}(3) = \text{SQPULSE.SRC (0.20E-9, 127.3)} \]
This first number is a multiplication factor for time values input from the file. The resulting full-width of the pulse is 1.0 ns. The second number adjusts the dependent variable. Save the modified file. Table 4 shows the final appearance of the script.

### 3.3 Finite-element solution

The next step is to generate and to solve the finite-element equations. Click the tool marked 2 or the menu command Solve. Accept the default entry of CONICAL.EIN in the dialog by clicking OK. The screen turns blue to indicate that the program is in solution mode and the status bar reports the progress of the operation. During the run, EMP creates five data dumps (CONICAL.001,...) and two probe files (CONICAL.P01,...). Section 4.7 describes the file formats.

### 3.4 Solution analysis

To transfer to the analysis menu, click the tool marked 3 or the menu command Analyze. The first step is to pick a solution series. Click on the command File/Series – start file and choose CONICAL.001. After the file is loaded, you should see the plot shown on the top of Fig. 11. It shows electric field lines (contours of $rH_\theta$) at $t = 2.0 \, \text{ns}$. The leading edge of the
Table 4: EMP script CONICAL.EIN.

* ---- CONTROL ----
Mesh = conical
Solution = H
Geometry = Cylin
DUnit = 1.0000E+02
TMax = 8.0100E-09

* ---- REGIONS ----
* Region 1: ALAYER
  Epsi(1) = 1.0000E+00
  Mu(1) = 1.0000E+00
  Cond(1) = 1.0600E+00
* Region 2: VACUUM
  Epsi(2) = 1.0000E+00
  Mu(2) = 1.0000E+00
  Cond(2) = 0.0000E+00
* Region 3: SOURCE
  Epsi(3) = 1.0000E+00
  Mu(3) = 1.0000E+00
  Cond(3) = 0.0000E+00
  Source(3) = SQPULSE.SRC
  Beta(3) = 90.0
* Region 4: AXIS
  Fixed(4)

* ---- DIAGNOSTICS ----
DTime = 2.0000E-09
History = -4.0000E+01 1.5000E+00
History = 3.0000E+01 1.5000E+00

EndFile
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. 11. The launched electromagnetic pulse impinging on the absorbing layer is visible as well as energy moving upstream along 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. 12. For comparison, the theoretical electric field of a 1.0 A pulse at a radius of 1.51 cm in a 70 $\Omega$ 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 radial transmission line. The characteristic impedance of a conical line is:
\( Z_0 = \sqrt{\mu / \varepsilon} \frac{\tan \alpha}{2\pi}, \) (21)

where \( \alpha \) is the angle of the center conductor. 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 \( Z_0 < 60\Omega \) giving a reflection coefficient \( R_E > -0.08 \). The bottom trace of Fig. 12 shows the radial electric field of the radiated pulse at the second probe (\( r = 1.5 \) cm, \( z = 30.0 \) cm).
Figure 12: Radial electric field as a function of time for the CONICAL example in the transmission line (top) and in free space (bottom).
4 EMP solution reference

4.1 Organizing EMP solutions

The program emp.exe has two functions: 1) finite-element calculation of initial-value solutions for electromagnetic pulses and 2) analysis of output dumps (spatial information at specified times). The package also includes the program probe.exe to display information in history files (temporal information at specified locations). EMP can run in two modes: interactively in a window or autonomously in the background with the option for batch file control. The latter mode allows automatic processing of large or repetitive data sets.

EMP may use the following 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. (Required)
- A command script that sets control parameters for the solution program and describes the physical properties associated with region numbers. (Required)
- Tables that describe waveforms of drives and sources (Optional).

The mesh output file always has a name of the form MPREFIX.MOU, where MPREFIX is a valid file prefix (1 to 32 characters). The command script must have a name of the form EPREFIX.EIN. 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 EPREFIX.

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

- Prepare a Mesh script (MPREFIX.MIN) following the instructions in the Mesh manual. You can create the file directly with a text editor or graphically using the drawing editor.
- Run Mesh either interactively from the TC program launcher or from the Windows Command Prompt to create the file MPREFIX.MOU.
- Prepare a command script (EPREFIX.EIN) using either the Setup command in the EMP main menu (Sect. 3.2) or your own text editor. The allowed file commands are described in this chapter.
- Run EMP to create data dumps (EPREFIX.001, EPREFIX.002,...) or history files (EPREFIX.P01, EPREFIX.P02,...). All output files are in text format. The data dumps contain information on the mesh geometry, the physical properties of regions, and values of computed quantities. In electromagnetic solutions the region properties are the relative magnetic permeability, the relative dielectric constant and the conductivity. The computed field quantities depend on the field polarity and the solution symmetry.
• Analyze data dumps in the Analysis menu of EMP. You can also transfer the information in the data dumps to your own analysis programs.

• Plot history files using Probe.

4.2 EMP script format

The EMP script is a text file composed of data lines that contain commands and parameters. You can construct a script using the Setup dialog in EMP (Sect. 3.2). You can also write and modify scripts with an editor. Direct script editing is required for some advanced EMP capabilities. Section 4.3 covers commands that control general program operation. Section 4.4 describes commands to set simple properties of material regions. Advanced methods to define regions with material properties that vary in space are discussed in Chap. 5. Finally, Sect. 4.5 covers diagnostic commands that control the generation of data dumps.

The script must end with the EndFile command. The entries on a line may be separated by the standard delimiters introduced in the Mesh manual:

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

Any number of delimiters may be used in a line. Blank lines and comment lines are ignored. Comment lines begin with an asterisk (*). EMP accepts commands in any order. Table 5 shows an example of complete script.

There are three classes of commands: program control, region properties and diagnostics. Control and diagnostic commands contain a keyword and a value. Region commands set the physical properties associated with elements and nodes. They have the format:

Keyword RegNo Value

Here, the integer RegNo is the region number defined in the Mesh input file. The string Keyword specifies the physical property. The value may be one or more numbers. As an example, the command

EPSI(2) = 7.2

sets the relative dielectric constant in elements with region number 2 to the value 7.2. (Note: For back-compatibility, EMP recognizes command formats from Version 1.0 through 5.0 where control commands start with the keyword Set, region commands with Region and diagnostic commands with Diag.)
Table 5: Example of an EMP control script

* EMP Test
* Plane wave absorption, normal incidence
* Gaussian pulse with FWHM of 0.222 ns
* Propagation distance: 0.125 + 0.100 m
* Matched conductance: Sigma = 1/Zo*Deltaz = 0.5301 mhos/m

TMax = 1.95E-9
Geometry = Rect
DUnit = 1.0
Solution = E
dT = 5.0e-12

Mu(1) = 1.0
Epsi(1) = 1.0
Cond(1) = 0.5301

Mu(2) = 1.0
Epsi(2) = 1.0

Drive(3) = GAUSS.SRC

* Propagate 0.125 m
SetTime = 1.084E-9
History = 0.30 0.125

ENDFILE
4.3 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 begin with a keyword followed by one or more parameters. In the following, each command is written symbolically and as it might appear in a script:

**MESH MPrefix**

MESH = SparkGap

Specify the **Mesh** output file that defines the geometry of the simulation. The file must have a name of the form **MPREFIX.MOU** and be available in the working directory. If this command does not appear in a script with the name **EPREFIX.EIN**, then **EMP** will seek the default mesh file **EPREFIX.MOU**.

**GEOMETRY [Rect,Cylin]**

GEOMETRY = Cylin

Specify whether the solution has planar symmetry (variations in \(x\) and \(y\) with infinite length in \(z\)) or cylindrical symmetry (variations in \(r\) and \(z\), uniform in \(\theta\)).

**DUNIT DUnit**

DUnit = 1.0E6

Set the interpretation of spatial coordinates from the mesh input file and the **History** command. The spatial conversion factor equals the number of spatial units per meter. For example, if coordinates in **Mesh** are given in mm, set **DUnit** = 1000.0. (Default, **DUnit** = 1.0).

**SOLUTION [E,H]**

SOLUTION = H

Set the field polarization of the solution. An **E** type solution has primary field components \(E_z\) (planar) or \(rE_\theta\) (cylindrical) with secondary components \(H_x\) and \(H_y\) (or \(H_z\) and \(H_r\)). An **H** type solution has primary field components \(H_z\) (planar) or \(rH_\theta\) (cylindrical) with secondary components \(E_x\) and \(E_y\) (or \(E_z\) and \(E_r\)).

**TMAX Tmax**

TMAX = 34.5E-9

The real number **TMax** equals the duration of the solution in seconds.

**NSTEP NStep**

NStep = 4500

An alternate way to limit the run time by setting the maximum number of computational steps. Enter an integer value. The default is 100,000.
INTERPOLATION [Linear, Spline]
INTERPOLATION = Linear
Sets the interpolation method for input tables that define time variations of drives and sources. 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. (The default is Spline.)

4.4 Region commands
Region commands set the physical properties of the regions of the solution volume defined in Mesh. The second entry in all commands is the region number, an integer. This section discusses how to assign uniform properties over the region volume. Chapter 5 covers spatial variations of properties over regions defined by mathematical functions. The first three commands discussed in this section 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.

MU RegNo MuR
MU(1) = 250.0
Assigns a value of the relative magnetic permeability ($\mu_r = \mu/\mu_0$) to all elements with the corresponding region number. The default is $\mu_r = 1.0$.

EPSI RegNo EpsiR
EPSI(7) = 7.3
Assigns a value of the relative dielectric constant ($\varepsilon_r = \varepsilon/\varepsilon_0$) to all elements with the corresponding region number. The default is $\varepsilon_r = 1.0$.

COND RegNo Sigma
COND(3) = 1.060
Assign a value of electrical conductivity to all elements with the corresponding region number. Enter the conductivity in S/m (siemens per meter). In SI units, the conductivity is the inverse of the volume resistivity, $\sigma = 1/\rho$, with $\rho$ specified in $\Omega$-m. The default is $\sigma = 0.0$.

FIXED RegNo
FIXED(4)
Assign the fixed value of 0.0 to the primary quantity ($E_z, H_z, rE_\theta$ or $rH_\theta$) for all nodes of the region. The fixed field value holds throughout the run. In cylindrical solutions, set the axis ($r = 0.0$) to the Fixed condition.

VOID RegNo
VOID(8)
Set an internal region to the Void condition. The special Neumann condition applies on the
boundaries with standard material regions. In this case, 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 \( \epsilon_r = 10^6 \) and \( \mu_r = 10^{-6} \) manually. The difference is that EMP will not try to plot field lines inside a void.

There are two important considerations for using internal voids:

- If you create a void region in a mesh by over-writing a portion of a material region, then Mesh will assign the void region number to all nodes including those on the boundary. For a correct EMP solution, 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 extra region. The example RODH 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 CONICAL walkthrough example.

**DRIVE RegNo FileName [TMult DMult]**

**DRIVE(5) = Gauss.DRV**

This command defines a region where the primary field quantity follows a specified variation in time. Drive regions are normally used to generate pulses. The string parameter is the full name of a tabular function file in the current directory that specifies the time variation of \( E_z, H_z, rE_\theta \) or \( rH_\theta \). Note that the tabular function routines return values of zero when the time exceeds the largest entry on the table. Therefore, a drive boundary assumes the Fixed condition after its active period. To define an open circuit condition after the drive pulse, 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.

A tabular function is a text file consisting of up to 256 entry lines of values for the time and the dependent variable. You can prepare table files with a text editor or spreadsheet. You can also use published data or digitized experimental traces. As an example, Table 6 shows the tabular function GAUSS.DRV supplied with the code to define a Gaussian pulse. Note that the file syntax conforms to the same rules as the EMP script. The free-form parser accepts real numbers in any format with a choice of delimiters. You can add documenting comment lines starting with an asterisk (*). The end of the data is marked with the EndFile command. Although the data lines of the example are ordered by increasing value of the independent variable, this ordering is not required. EMP sorts the list before use and records the final order in the listing file, FPREFIX.ELS. Furthermore, the independent variable intervals need not be uniform. A table requires a minimum of 5 entries. The interpolation routines return 0.0 for values of the \( t \) outside the range of the table.
Table 6: Table to define a Gaussian pulse.

* Gaussian pulse shape, 0.222 ns FWHM, amplitude 1.0
* Peak at 0.6667 ns

<table>
<thead>
<tr>
<th>Time (ns)</th>
<th>Amplitude</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0000E+00</td>
<td>1.3886E-11</td>
</tr>
<tr>
<td>1.3333E-11</td>
<td>3.7370E-11</td>
</tr>
<tr>
<td>2.6667E-11</td>
<td>9.8581E-11</td>
</tr>
<tr>
<td>4.0000E-11</td>
<td>2.5490E-10</td>
</tr>
<tr>
<td>5.3333E-11</td>
<td>6.4605E-10</td>
</tr>
<tr>
<td>6.6667E-11</td>
<td>1.6050E-09</td>
</tr>
<tr>
<td>...</td>
<td></td>
</tr>
<tr>
<td>5.3333E-10</td>
<td>3.6787E-01</td>
</tr>
<tr>
<td>5.4667E-10</td>
<td>4.4485E-01</td>
</tr>
<tr>
<td>5.6000E-10</td>
<td>5.2728E-01</td>
</tr>
<tr>
<td>5.7333E-10</td>
<td>6.1261E-01</td>
</tr>
<tr>
<td>5.8667E-10</td>
<td>6.9766E-01</td>
</tr>
<tr>
<td>6.0000E-10</td>
<td>7.7879E-01</td>
</tr>
<tr>
<td>6.1333E-10</td>
<td>8.5213E-01</td>
</tr>
<tr>
<td>6.2667E-10</td>
<td>9.1392E-01</td>
</tr>
<tr>
<td>6.4000E-10</td>
<td>9.6078E-01</td>
</tr>
<tr>
<td>6.5333E-10</td>
<td>9.9005E-01</td>
</tr>
<tr>
<td>6.6667E-10</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>6.8000E-10</td>
<td>9.9005E-01</td>
</tr>
<tr>
<td>6.9333E-10</td>
<td>9.6080E-01</td>
</tr>
<tr>
<td>7.0666E-10</td>
<td>9.1394E-01</td>
</tr>
<tr>
<td>7.2000E-10</td>
<td>8.5215E-01</td>
</tr>
<tr>
<td>7.3333E-10</td>
<td>7.7881E-01</td>
</tr>
<tr>
<td>7.4666E-10</td>
<td>6.9769E-01</td>
</tr>
<tr>
<td>...</td>
<td></td>
</tr>
<tr>
<td>1.7000E-09</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>1.8000E-09</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>1.9000E-09</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>2.0000E-09</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>2.5000E-09</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>3.0000E-09</td>
<td>0.0000E+00</td>
</tr>
</tbody>
</table>

ENDFILE
DRIVE RegNo > FunctionString
DRIVE(5) > 10.5*SIN(6.0E8*$t)
Define the time variation of a drive region from a mathematical function. Section 5.2 gives a
detailed description of the command format.

TEXPIRE RegNo TExp
TEXPIRE(7) = 3.5E-9
Define an expiration time for a drive region. The boundary reverts to an unspecified (Neumann)
condition after a time limit. The quantity \( TExp \) is the expiration time in seconds. The drive
condition determined from a table or mathematical function holds in the interval \( 0.0 \leq t \leq T_{exp} \).
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
type solution) or conducting (H type solution) boundaries.

SOURCE RegNo FileName [TMult SMult]
SOURCE(3) = Coupler.SRC (2.0E-9, 150.0)
Include a spatially-uniform current source with a specified time variation in a material region.
Sources represent current loops or capacitive probes that drive RF pulses. The string is 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 create a library of
normalized waveforms that can be incorporated into a variety of solutions. The default values
are \( TMult = 1.0 \) and \( SMult = 1.0 \). The interpretation of the source function value in EMP
depends on the geometry and solution type. Table 3 lists the options. Important note: source
regions should not be in direct contact with absorbing regions (\( \sigma > 0.0 \)).

SOURCE RegNo > FunctionString
SOURCE(6) > 0.5*(1.0 - COS(6.7E8*$t))
Define the time variation of a source region from a mathematical function. Section 5.2 gives a
detailed description of the command format.

BETA RegNo Beta
BETA(2) = 90.0
Set the direction of the region current density in the \( x-y \) or \( z-r \) plane for \( H \) type solutions. The
angle \( \beta \) (in degrees) is the direction of the current density relative to the \( x \) or \( z \) axis. The angle
equals \( \tan^{-1}(j_y/j_x) \) or \( \tan^{-1}(j_r/j_z) \). The default is \( \beta = 0.0^\circ \).

TABSHIFT RegNo TMult TOff VMult VOff
TABSHIFT(7) = (1.0, 5.0, 1.0E4, 0.0)
Adjust the interpretation of time-dependent functions from tabular files or mathematical func-
tions for drive and source regions. There are four real number parameters. The first, \( (TMult) \)
is a multiplication factor for values of time and the second \( (TOff) \) is an offset. The default
values are 1.0 and 0.0. The third \( (VMult) \) is a multiplication factor and the fourth \( (VOff) \)
is an offset for the dependent variable. This program function allows you to apply normalized tables to a variety of applications. If $t$ is the program time, the value passed to the tabular function or mathematical function is

$$t_{pass} = T_{mult} \times (t - T_{off}).$$

(22)

If the interpolation or function returns a value $v$, the quantity used in the program is

$$v_{pass} = V_{mult} \times v + V_{off}.$$  

(23)

The command affects only Drive or Source regions that are associated with a table or mathematical function. Note that the TabShift command supplements the $T_{Mult}$, $D_{Mult}$ and $S_{Mult}$ parameters in the Drive and Source commands.

4.5 Diagnostic commands

Diagnostic commands control data output from the program. EMP produces three types of data, all in text format:

- From 1 to 999 data dump files with the names of the form FPREFIX.001, FPREFIX.002, .... Each file contains a complete record of spatial information (the mesh geometry and distributions of field and material quantities) at a specified time. These files are used as input in the Analyze menu of EMP.

- A listing file with the name FPREFIX.ELS that can be inspected with a text editor. This file contains a variety of information about the EMP run, useful mainly to check the validity of input parameters and to diagnose problems.

- From 1 to 20 history files with the names of the form FPREFIX.P01, ..., FPREFIX.P20. Here, the magnetic and electric field components at one or more probe locations are recorded at each time step of the solution. The information may be inspected with a text editor or displayed with the utility program Probe (described in Chap. 8).

**SETTIME STime**

SETTIME = 2.5E-9

Record a data dump at or immediately after a specified time. The quantity $S_{Time}$ is the simulation time (in seconds) for the data record. When using automatic time step selection, it is uncertain that EMP will complete a cycle exactly at the specified time. The convention is that the code records a file when the time is equal to or greater than $S_{Time}$. You can define up to 100 record times with multiple SetTime commands. The only restriction is that the commands must appear in chronological order.

**DTIME DTime**

DTIME = 1.5E-10

Record data dumps at a uniform time interval. Enter the interval $D_{Time}$ in seconds. As with the SetTime command, 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 calculated target time.

\[ \text{NSTEP NStep} \]
\[ \text{NSTEP = 250} \]
Create data dumps separated by a uniform number of computational steps. The integer \textit{NStep} is the number of cycles per data dump. You can record data at exact intervals by using this option and setting the time step with the \textit{Dt} command.

\[ \text{HISTORY XPos YPos} \]
\[ \text{HISTORY ZPos RPos} \]
\[ \text{HISTORY} = (2.50, 3.75) \]
Record field values as a function of time at a specified location. Enter the target coordinates for the probe location in distance units set by \textit{DUnit}. The real number inputs are \([x, y]\) for planar problems and \([z, r]\) for cylindrical problems. The probe coordinates that appear in the history file usually are 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 to find secondary quantities are determined by a linear fit using primary field values at the three element nodes. The input file can contain up to 20 \textit{History} commands in any order.

\[ \text{NHSTEP NHStep} \]
\[ \text{NHSTEP = 10} \]
Set the number of computational time steps per record in the history files, where \textit{NHStep} is an integer. 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 \textit{NHStep} = 1.

\[ \text{MOVIE NFrame [Nx Ny] [QMin QMax]} \]
\[ \text{MOVIE 100 800 600 0.0 110.0} \]
Create a set of \(N_{frame}\) graphics files in \textit{BMP} format separated by time interval \(t_{\text{max}}/N_{frame}\). The plots show electric or magnetic field lines (depending on the solution mode) in the full solution volume. The number of lines is determined by any previous setting in the \textit{Analysis} menu. The integer parameters \(N_x\) and \(N_y\) give the horizontal and vertical image size in pixels. If they do not appear, the default is \(N_x = 1024\) and \(N_y = 768\). The two real-number parameters are the upper and lower temperature limits for the generating function. Autoscaling is applied to each frame if they are absent. You can use our utility program \textit{Cecil}\_\textit{B} to stitch the files into a movie in \textit{AVI} format. The program is available at

\[ \text{http://www.fieldp.com/cecil_b.html} \]

4.6 Running EMP

EMP can run as an interactive program in a window or as a background task. To run the program from the command prompt, use a command of the form:
where the file EPrefix.EIN and the appropriate Mesh output file are available in the data directory. With this capability, you set up extended autonomous EMP runs using a DOS batch file or a Perl script.

The remainder of this section discusses commands in the main menu when EMP runs in the interactive mode. The program enters this mode when launched from TC or run with no input file prefix. The following commands appear in the File popup menu:

**EDIT SCRIPT (EIN)**
**EDIT LISTING (ELS)**
**EDIT FILE**

These commands call the internal editor to inspect or to modify EMP input and output files. Choosing a file from an alternate directory does not change the working directory. The Edit script (EIN) command shows a list of all files with names of the form FPREFIX.EIN while Edit listing (ELS) displays files with names FPREFIX.ELS.

**RUN ANALYSIS SCRIPT**

An analysis script allows you to perform complex or repetitive analyses on a set of similar solutions. This command displays a dialog listing files with the suffix SCR. Pick a file and click OK. The script can load data files, open and close data records, and perform any of the quantitative analysis functions described in this chapter. The script command language is described in Sect. 6.5. Note that the analysis script must be in the same directory as the data files.

**SETUP**

The function of this command is to create an EMP script to control an electromagnetic solution. The program first prompts for a Mesh output file to define the system geometry. The prefix of the file will be used as the argument of the Mesh script command. The program then displays the dialog shown in Fig. 11. The number of regions in the dialog is determined by the Mesh file. The functions of the control parameters in the upper box are described in Sect. 4.3. You can enter basic physical properties of regions in the grid box (see Sect. 4.4). You must edit the script directly to invoke advanced functions.

The Solve popup menu includes the following two commands.

**RUN**

Pick an input file (such as EPrefix.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 EPrefix.MOU or the file specified in the Mesh command is present. During the solution, the screen color is blue and the progress is shown in the status bar.

**STOP**

This command terminates the EMP run. Any data dumps created before this time will be available.
SOLUTION DISPLAY
When solution display is active, EMP plots a screen plot of electric or magnetic field (depending on the solution mode) whenever a data dump is created. The number of lines is determined by the last setting in the Analysis menu.

ANALYZE
Call up the analysis menu for plotting and numerical analysis.

EMP MANUAL
Display this manual in your default PDF display software. The file emp.pdf must be available in the same directory as emp.exe.

4.7 Format of the EMP output file
The EMP data dump files (EPREFIX.001, EPREFIX.002, ...) are in text format. The files have three sections:

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

The header section consists of a title line and 11 data lines:

--- Run parameters ---
XMin: -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 1 and 2 list \( x_{\text{min}} \) and \( x_{\text{max}} \), the limits along the horizontal axis (\( x \) or \( z \)) of the solution volume. Dimensions are recorded in meters. The quantity \( k_{\text{max}} \) in Line 3 is the number of nodes along the horizontal direction. Lines 4-6 describe the vertical axis (\( y \) or \( r \)). Line 7 contains the quantity \( DUnit \), the conversion factor from dimensions used in Mesh to meters. Line 8 gives the number of regions in the solution volume, while Line 9 specifies the symmetry (0: planar, 1: cylindrical). Line 10 is the simulation time when the file was created and Line 11 is the polarity of the solution. The values in Lines 9-11 are used to set labels and calculated quantities in the Analyze menu.

The node section (Table 7) consists of 4 title lines and \( k_{\text{max}} \times l_{\text{max}} \) data lines, one for each node of the solution space. 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 \((RgUp\text{ and } RgDn)\). The upper element lies above the line between nodes \((K,L)\) and \((K+1,L)\) and the lower element lies below the line.

• The coordinates of the node in meters, \((x,y)\) or \((z,r)\).

• The primary field quantity \(A\). The recorded quantity depends on the solution field polarity and symmetry: \(E_z\text{ (V/m)}, rE_\theta\text{ (V)}, H_z\text{ (A/m)}\) or \(rH_\theta\text{ (A)}\).

• The \(x\) or \(z\) component of the secondary field quantity in the upper and lower elements \((AxzUp, AxzDn)\). The recorded quantity depends on the solution field polarity and symmetry: \(H_x\text{ (A/m)}, H_z\text{ (A/m)}\), \(E_x\text{ (V/m)}\) or \(E_z\text{ (V/m)}\).

• The \(y\) or \(r\) component of the secondary field quantity in the upper and lower elements \((AyrUp, AyrDn)\). The recorded quantity depends on the solution field polarity and symmetry: \(H_y\text{ (A/m)}, H_r\text{ (A/m)}, E_y\text{ (V/m)}\) or \(E_r\text{ (V/m)}\).

• The material properties in the upper and lower elements, \(\epsilon_r\) (relative dielectric constant), \(\mu_r\) (relative magnetic permeability) and \(\sigma\) (electrical conductivity).

The region section consists of four title lines followed 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 (Dirichlet) regions, 0 otherwise.

• \(Void\): 1 if the region is a void, 0 otherwise.

• \(Epsi\): relative dielectric constant for uniform regions.

• \(Mu\): relative magnetic permeability for uniform regions.

• \(Cond\): electrical conductivity in S/m for uniform regions.

• \(Area\): cross-section area of the region in m\(^2\).

• \(Beta\): angle of the current density in degrees relative to the \(x\) or \(z\) axis (quantity meaningful only for source regions in \(H\) type solutions).
Table 7: Example – node section of the EMP data dump

--- Vertices ---

<table>
<thead>
<tr>
<th>k</th>
<th>l</th>
<th>RgNo</th>
<th>RgUp</th>
<th>RgDn</th>
<th>x</th>
<th>y</th>
<th>A</th>
</tr>
</thead>
<tbody>
<tr>
<td>242</td>
<td>1</td>
<td>4</td>
<td>2</td>
<td>0</td>
<td>1.000000E-01</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>243</td>
<td>1</td>
<td>4</td>
<td>2</td>
<td>0</td>
<td>1.025759E-01</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>244</td>
<td>1</td>
<td>4</td>
<td>2</td>
<td>0</td>
<td>1.051469E-01</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>245</td>
<td>1</td>
<td>4</td>
<td>2</td>
<td>0</td>
<td>1.077111E-01</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>246</td>
<td>1</td>
<td>4</td>
<td>2</td>
<td>0</td>
<td>1.102677E-01</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
</tbody>
</table>

--- AxzUp, AxzDn, AyrUp, AyrDn, EpsiUp ---

<table>
<thead>
<tr>
<th>AxzUp</th>
<th>AxzDn</th>
<th>AyrUp</th>
<th>AyrDn</th>
<th>EpsiUp</th>
</tr>
</thead>
<tbody>
<tr>
<td>-2.411373E-23</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>8.854188E-12</td>
</tr>
<tr>
<td>-2.283328E-23</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>8.854188E-12</td>
</tr>
<tr>
<td>-2.068102E-23</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>8.854188E-12</td>
</tr>
<tr>
<td>-1.793478E-23</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>8.854188E-12</td>
</tr>
</tbody>
</table>

--- EpsiDn, MuUp, MuDn, CondUp, CondDn ---

<table>
<thead>
<tr>
<th>EpsiDn</th>
<th>MuUp</th>
<th>MuDn</th>
<th>CondUp</th>
<th>CondDn</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000000E+00</td>
<td>1.256637E-06</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>0.000000E+00</td>
<td>1.256637E-06</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>0.000000E+00</td>
<td>1.256637E-06</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>0.000000E+00</td>
<td>1.256637E-06</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
<td>0.000000E+00</td>
</tr>
</tbody>
</table>
5 Mathematical functions for drives, sources and material properties

5.1 Function parser

You can define temporal variations of drive and source regions and set spatial variations of material properties over a region using mathematical functions. For this purpose, EMP features a flexible and robust algebraic function interpreter. A function is a string (up to 230 characters) that may include the following entities:

- Independent variables. For temporal functions, the variable is $t$ (the adjusted simulation time). Spatial functions are defined in terms of $x$ and $y$ (planar solutions) and $z$ and $r$ (cylindrical solutions).
- Real and/or integer numbers in any valid format (e.g., $3.1415$, $476$, $1.367E23$, $6.25E-02$, $8.92E+04$, ... ). Integers are converted to real numbers for evaluation.
- Binary operations: + (addition), - (subtraction), * (multiplication), / (division) and $\wedge$ (exponentiation).
- Functions: abs (absolute value), sin (sine), cos (cosine), tan (tangent), ln (normal logarithm), log (base 10 logarithm), exp (normal exponent) and sqrt (square root).
- Up to 20 sets of parentheses to any depth.
- Any number of space delimiters.

The parser conforms to the standard algebraic rules and includes error checking features. Errors may include unbalanced parentheses, unrecognized characters and sequential binary operations. As an example, the expression

$$1 - \exp(-1.0*(($z^2 + r^2)/24))$$

corresponds to

$$1 - \exp\left[-\left(\frac{z^2 + r^2}{24}\right)\right].$$  \hspace{1cm} (24)

5.2 Temporal variations of drives and sources

Section 4.4 described how to use the Drive and Source commands with tabular functions. The commands have alternate forms if you want to set temporal variations using mathematical functions.
Define the time variation of a drive region from a mathematical function. The function is a string up to 230 characters in length that follows the ‘>’ symbol. The only allowed variable is the time $t$.

Define the time variation of a source region from a mathematical function. The function is a string up to 230 characters in length that follows the ‘>’ symbol. The only allowed variable is the time $t$.

The time $t$ normally corresponds to the simulation time $t$. If the script includes a TabShift command for the region, EMP substitutes the value:

$$t = T_{mult} \times (t - T_{off}).$$

The value returned by the function is adjusted according to:

$$f' = V_{mult} \times f + V_{off}.$$ (26)

### 5.3 Spatial variations of material properties

EMP can represent continuous spatial variations of the material quantities $\varepsilon_r$, $\mu_r$ and $\sigma$ over regions that follow mathematical functions. Use the following command to set a spatial variation of relative dielectric constant.

Set values of $\varepsilon_r$ in the elements of a volumetric material region according to a specified function of space. The keyword Epsi followed by the region number and the ‘>’ symbol designates that a function string occupies the remainder of the line. The function may be up to 230 characters in length and follows the format described in Sect. 5.1. For spatial functions, the parser recognizes the variables $x$ and $y$ for planar solutions and $z$ and $r$ for cylindrical solutions. The dielectric constant in an element equals the value of the function evaluated at the center-of-mass of the element. You must ensure that the function returns a physically valid value over all elements of the region.

Note that there is no limit on the number of temporal and spatial region functions. Any drive or source region may be associated with a temporal function and the physical properties of all material regions may be defined with spatial functions. Positions are passed to the function in units set by DUnit (cm, inches, µm,...). You can model discontinuous functions by dividing a volume into two or more regions.
Set values of $\mu_r$ in the elements of a volumetric material region according to a specified function of space.

SIGMA RegNo > Function
SIGMA(2) = 15.2*exp($x^2/3.4$)*exp($y^2/2.3$)
SIGMA(6) = 1.0 + 0.25*$r^3$
Set values of $\sigma$ (in S/m) over the elements of a volumetric material region according to a specified function of space.
Table 8: EMP standard units

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spatial dimensions</td>
<td>meters or units set by DUnit</td>
</tr>
<tr>
<td>Electrical field, $E$</td>
<td>volts/meter (V/m)</td>
</tr>
<tr>
<td>Magnetic field, $H$</td>
<td>amperes/meter (A/m)</td>
</tr>
<tr>
<td>Current density, $j$</td>
<td>amperes/meter$^2$ (A/m$^2$)</td>
</tr>
<tr>
<td>Electrical conductivity, $\sigma$</td>
<td>siemens/meter (S/m)</td>
</tr>
<tr>
<td>Energy density</td>
<td>joules/meter$^3$ (J/m$^3$)</td>
</tr>
<tr>
<td>Power density</td>
<td>watts/meter$^3$ (W/m$^3$)</td>
</tr>
</tbody>
</table>

6 Solution analysis reference

Click the Analyze command in the EMP main menu to enter the analysis menu. Here, you can load data dumps to create plots or to perform numerical calculations. The menu contains the following main entries: File, Plots, Analysis, Scans, Export and Return. The commands of the Export menu (which generate hardcopy output and plot files) are identical to those in Mesh. The Return command restores the main menu where you can perform additional solutions.

6.1 File menu commands

SET SOLUTION SERIES
A dynamic run may create several data files with the same prefix, FPREFIX. Use this command to specify a file prefix for subsequent load operations. Moving to a new directory in the dialog changes the program working directory. The command must be the first activity in an analysis session. EMP counts the number of files in the series, records the times and displays the dialog of Fig. 13. Pick a solution file to load and click OK.

LOAD SOLUTION
Load a different solution file in a series. EMP displays the dialog of Fig. 13. Pick a different solution and click OK. This command is active only when a series has been specified.

OPEN DATA RECORD
Commands such as Point calculation and Line scan generate quantitative information. You can automatically record the data generated during an analysis session by opening a data file. Supply a file prefix in the dialog or accept the default. The data file has a name of the form FPREFIX.DAT and will be stored in the working directory. The file is in text format. You can use an editor to view the file or to extract information to send to mathematical analysis programs or spreadsheets.
CLOSE DATA RECORD
Close the current data file. Use this command if you want to start a new file. Note that you must close the data file before opening it with the internal editor.

RUN SCRIPT
A script allows you to perform complex or repetitive analyses on a set of similar solutions. This command displays a dialog listing files with the suffix SCR. Pick a file and click OK. The script can load data files, open and close data records, and perform any of the quantitative analysis functions described in this chapter. The script command language is described in Sect. 6.5. Note that the analysis script must be in the same directory as the data files.

CREATE SCRIPT
Use this command to create an analysis script with the internal text editor. Supply a file prefix SPrefix in the dialog – the resulting script will be saved with the name SPrefix.SCR. The program opens the file in the editor and writes a reference list of allowed commands. The list follows the EndFile command and will be ignored by the script parser. Enter commands above the EndFile command.

EDIT SCRIPT
EDIT DATA FILE
EDIT FILE
Use these commands to view or to modify an existing file. The dialog shows files with suffix SCR for the Edit script command and DAT for the Edit data file command. Changing directories in the dialog does not change the working directory of the program.

6.2 Plot menu commands
Spatial plots show variations of quantities over the two-dimensional space of the simulation. The following plot types are available:

- Mesh. Element facets of the computational mesh.
• **Region.** Computational mesh with elements color-coded by region number.

• **Contour.** Lines that follow constant values of a computed quantity. In electromagnetic solutions, contours of the primary field quantity lie along lines of the secondary field quantity. The correspondence depends on the solution type and symmetry. In $E$ type solutions, contours of $E_z$ or $rE_\theta$ lie along lines of $H$ in the $x-y$ or $z-r$ planes. In $H$ type solutions, contours of $H_z$ or $rH_\theta$ lie along lines of $E$ in the $x-y$ or $z-r$ planes.

• **Element.** Elements of the solution space color-coded according to a computed quantity (such as the field energy density).

• **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 represented as height 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. For large meshes, you may notice a delay regenerating a Surface plot. The program must map the current quantity to a rectangular grid, performing a large number of interpolations.

The Settings popup menu contains the following commands.

**TYPE**
Choose the plot type from the above list. A plot type may not support some plotted quantities. If you receive a message when you switch plot types that the current quantity is not allowed, use the Quantity command to pick a valid option.

**QUANTITY**
A dialog shows a list of available quantities (Table 9) consistent with the current plot type. Note that the specific quantity may depend on the solution type and symmetry. Spatial plots of $\varepsilon_r$, $\mu_r$, and $\sigma$ are included to check the validity of assignment of spatial variations from mathematical functions (Chap. 5).

**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 specific limits. In the dialog uncheck the Autoscale box and supply the minimum and maximum values. Note that the program does not check that the values are physically reasonable. This operation will not affect scaling of other plot quantities. Check the box to return to autoscale mode.

**TOGGLE GRID DISPLAY**
Use this command to activate or to suppress the display of grid lines in Mesh, Region, Contour, Element and Vector plots. Grid lines corresponding to the axes ($x = 0.0$ or $y = 0.0$) are plotted as solid lines.
Table 9: Available plot quantities

<table>
<thead>
<tr>
<th>Plot type</th>
<th>Quantity (Depends on solution type, symmetry)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contour</td>
<td>Lines of $\mathbf{H}$ or $\mathbf{E}$</td>
</tr>
<tr>
<td>Element</td>
<td>$E_z, rE_\theta, H_z$ or $rH_\theta$ (primary quantity)</td>
</tr>
<tr>
<td></td>
<td>$E_z, E_\theta, H_z$ or $H_\theta$ (primary field)</td>
</tr>
<tr>
<td></td>
<td>$</td>
</tr>
<tr>
<td></td>
<td>$H_x, H_z, E_z$ or $E_z$ (secondary field, $x$ or $z$ component)</td>
</tr>
<tr>
<td></td>
<td>$H_y, H_r, E_y$ or $E_r$ (secondary field, $y$ or $r$ component)</td>
</tr>
<tr>
<td></td>
<td>$U$ (field energy density)</td>
</tr>
<tr>
<td></td>
<td>$P$ (resistive power density)</td>
</tr>
<tr>
<td></td>
<td>$</td>
</tr>
<tr>
<td></td>
<td>$\epsilon_r$ (relative dielectric constant)</td>
</tr>
<tr>
<td></td>
<td>$\mu_r$ (relative magnetic permeability)</td>
</tr>
<tr>
<td></td>
<td>$\sigma$ (electrical conductivity)</td>
</tr>
<tr>
<td>Vector</td>
<td>$</td>
</tr>
<tr>
<td></td>
<td>$</td>
</tr>
<tr>
<td>Surface and scans</td>
<td>$E_z, rE_\theta, H_z$ or $rH_\theta$ (primary quantity)</td>
</tr>
<tr>
<td></td>
<td>$E_z, E_\theta, H_z$ or $H_\theta$ (primary field)</td>
</tr>
<tr>
<td></td>
<td>$</td>
</tr>
<tr>
<td></td>
<td>$H_x, H_z, E_z$ or $E_z$ (secondary field, $x$ or $z$ component)</td>
</tr>
<tr>
<td></td>
<td>$H_y, H_r, E_y$ or $E_r$ (secondary field, $y$ or $r$ component)</td>
</tr>
<tr>
<td></td>
<td>$U$ (field energy density)</td>
</tr>
<tr>
<td></td>
<td>$P$ (resistive power density)</td>
</tr>
<tr>
<td></td>
<td>$</td>
</tr>
<tr>
<td></td>
<td>$\epsilon_r$ (relative dielectric constant)</td>
</tr>
<tr>
<td></td>
<td>$\mu_r$ (relative magnetic permeability)</td>
</tr>
<tr>
<td></td>
<td>$\sigma$ (electrical conductivity)</td>
</tr>
</tbody>
</table>
GRID CONTROL
This command displays the dialog of Fig. 14 to set properties of the grid. In the default autoscale mode, EMP automatically chooses intervals and positions so that lines occur at convenient intervals in x or y (for example, 0.01 rather than 0.01153). The grid intervals change as the view is zoomed. To set the grids manually, uncheck the Autoscale box and enter values for the intervals in x and y.

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

TOGGLE SNAP MODE
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 (5.500, -2.500). By default, snap mode is active. 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 tip of the cursor arrow, giving misleading results.

SNAP DISTANCE
Set the distance scale DSnap for the mouse snap mode.

TOGGLE ELEMENT OUTLINE
This command determines whether the element facets are included in Element and Vector plots. It may be necessary to deactivate outlines for a clear view of large meshes.
CONTOUR 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 (Fig. 15). Overlapping labels on closely-spaced lines may look better in a zoomed view.

NUMBER OF CONTOURS
Change the number of plotted contour lines. This command is active only when the current plot type is Contour.

The following commands, described in the Mesh manual, 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
ZOOM IN
EXPAND VIEW
GLOBAL VIEW
PAN

The following commands control the appearance of Surface plots. The commands are active only when a Surface plot is displayed.

ROTATE 3D IMAGE
Rotate the Surface plot by 90° in the spatial plane.
VIEW ANGLE 3D
Set the elevation angle for the view point.

SET GRID 3D
Change the resolution of the surface plot. To create a Surface plot, a quantity is mapped to a rectangular grid with dimensions $N_x \times N_y$. The numbers also determine the total number of grid lines in the Surface plot. The default values are $N_x = N_y = 40$.

6.3 Analysis menu commands
The commands in the Analysis popup 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 plot is displayed.

POINT CALCULATION
After clicking on the command, point to any position in the solution volume with the mouse. Note that snap mode is deactivated for coordinate input. The program writes a subset of interpolated quantities to the information area below the plot and also records complete information if a data file is open. To enter point coordinates by keyboard, press the F1 key or change the program mode with the Toggle mouse/keyboard command.

LINE SCAN
The line scan is one of the most useful functions of EMP. 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). To enter point coordinates by keyboard, press the F1 key or change the program mode with the Toggle mouse/keyboard command. The program computes a series of values of field quantities at equal intervals along the line. The information is recorded if a data file is open. The program also makes a screen plot of the currently selected quantity versus distance along the scan and activates the commands in the Scan plot menu (Sect. 6.4). The default is to include all computed quantities. Use the Set recorded quantities command to limit the information.

ELEMENT PROPERTIES
Pick an element with the mouse (or keyboard) and the post-processor writes material and field properties to the screen. The information is recorded if a data file is open.

REGION PROPERTIES
To find 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. EMP calculates volume integrals of the electric and magnetic field energy density and the total resistive power dissipation in the region.
VOLUME INTEGRAL
No input is needed for this command. EMP automatically computes integrals of quantities over the full solution volume and over individual regions. Information is recorded on the screen or in a history file.

MATRX FILE
EMP can make matrix files of field values and material properties to help you create your own analysis routines. Although information is available in the output file of the solution program, it may be difficult to deal with the conformal triangular mesh. The Matrix file command uses the interpolation capabilities of the program to create a text data file of field quantities on a rectangular grid in $x$-$y$ or $z$-$r$. The command displays a dialog 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. The default is to include all computed quantities. Use the Set recorded quantities command to limit the information.

The Settings popup menu contains the following entries.

INTERPOLATION METHOD
The default interpolation method for the Point calculation and Line scan commands is a second-order least-squares fit. This method 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. The status bar reports the current interpolation type.

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 data file records which include all field quantities. The available line scan quantities depend on the solution type and symmetry. The choices are listed in Table 9.

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.

SET RECORDED QUANTITIES
The default for data records of line scans and matrix files is to include all plotted quantities. The may result in large files with unnecessary information. Use this command to set the quantities that will be include. In the dialog, activate the check box to include the quantity. There are a few rules:

- Quantity selection is available when the program runs interactively in a window. It does not apply when the program is run in the background.
• In the interactive mode, quantity selection applies to direct calls to the Line scan or Matrix file commands and indirect calls from an analysis script.

• All quantities are active when the program starts.

• The state of active quantities is preserved when a new data file is loaded.

6.4 Scan plot menu

The commands of the Scan menu become active when a plot is created following the Line scan command.

OSCILLOSCOPE MODE

In the oscilloscope mode, a scan plot assumes characteristics of a digital oscilloscope (Fig. 16). 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 EMP. This command closes the scan plot and returns the program to the previous spatial plot.

6.5 Analysis script commands
Scripts to control analysis sessions have a name of the form FPrefix.SCR. They should be in the same directory as the data files. Scripts are text files that follow the TriComp syntax conventions. The program ignores blank lines and indentations. Data lines use the standard delimiters and comment lines begin with an asterisk [*]. Processing ends when the EndFile command is encountered. To run a script, choose the Run script command in the File menu. The program shows a list of available scripts. 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.

EMP can perform analyses autonomously under script file control from the Command Prompt. If the file GTest.SCR is in the data directory, then use a command of the form:

[ProgPath]\EMP GTEST <Enter>

The main application of the command prompt mode is to generate data files and to perform extended analyses under batch file control.

The following commands may appear in a script:

INPUT FileName
INPUT Convolute.002
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 Convolute.001. You can load several files for sequential analysis.

OUTPUT FPrefix
OUTPUT ACavity
Close the current data file and open an output file ACavity.DAT.

POINT X Y
POINT Z R
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 in Mesh units.
SCAN Xs Ys Xe Ye
SCAN Zs Rs Ze Re
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.

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

**ELEMENT X Y**
**ELEMENT Z R**
**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 in **Mesh** units.

**NSCAN NScan**
**NSCAN = 150**
Set the number of points in a line scan. The default is 50 and the maximum number is 500.

**REGION RegNo**
**REGION = 5**
Write volume and surface integrals for a region to the data file. The integer parameter is the region number.

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

**MATRIX FPrefix Nx Ny Xs Ys Xe Ye**
**MATRIX FPrefix Nz Nr Zs Rs Ze Re**
**MATRIX = Switch1 (10, 20) (0.00, 0.00, 5.00, 10.00)**
Open a matrix file and record values. The command requires seven parameters: 1) The prefix of the matrix file **FPrefix.MTX** (string), 2) the number of intervals along the x or z direction (integer). 3) the number of intervals along the y or r direction (integer), 4-7) coordinates of the corners of a box in the solution volume (real).

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

The following is an example of a script to compare field values along the axes of four different solutions and to write the results to the file **COMP.DAT**.
NSCAN 200
OUTPUT COMP
INPUT SWITCH01.001
SCAN 0.00 -50.00 0.00 50.00
INPUT SWITCH01.002
SCAN 0.00 -50.00 0.00 50.00
INPUT SWITCH01.003
SCAN 0.00 -50.00 0.00 50.00
INPUT SWITCH01.004
SCAN 0.00 -50.00 0.00 50.00
ENDFILE
<table>
<thead>
<tr>
<th>Example</th>
<th>Technique</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSORB</td>
<td>Defining absorbing layers ion $E$ and $H$ type pulses, normal and non-normal incidence.</td>
</tr>
<tr>
<td>INLINE</td>
<td>Inline resistor in a coaxial transmission line, comparison to predicted reflection and transmission coefficients ($H$ type, cylindrical)</td>
</tr>
<tr>
<td>RODREFLECT</td>
<td>Plane wave reflection from a metal rod, definition of an internal metal body in $E$ and $H$ type solutions. Use of an internal void.</td>
</tr>
<tr>
<td>FUNCTEST</td>
<td>Entering time-dependent drives and sources with table and mathematical functions, effect of <code>TabShift</code> command.</td>
</tr>
<tr>
<td>TM01</td>
<td>Excitation of the TM01 mode in a circular waveguide. Source specified by a mathematical function.</td>
</tr>
<tr>
<td>WGUIDEDIEL</td>
<td>Excitation of the TM01 mode in a circular dielectric waveguide.</td>
</tr>
<tr>
<td>PARABOLA</td>
<td>Linear parabolic transmitter. Illustration of the <code>TExpire</code> command for a drive region.</td>
</tr>
</tbody>
</table>

7 EMP tutorials

This section contains a number of walkthrough examples that illustrate solution techniques. All input files are contained in the example library included with the program. The library includes a variety of additional examples that illustrate basic setup techniques in EMP (Table 10).

7.1 Coaxial transmission lines

This section introduces some useful EMP techniques to generate TEM waves in coaxial transmission lines. 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

$$Z_0 = \frac{1}{2\pi} \sqrt{\mu \epsilon} \ln \left( \frac{R_o}{R_i} \right), \quad (\Omega)$$

(27)

and a capacitance per length of

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

(28)
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 $rH_\theta$.

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. Boundary layers on the left and right ends are set to the condition $rH_\theta$ to complete the definition of the absorbing layer for $H$ type pulses (Sect. 2.5). A radial current source layer just downstream from the absorber drives the pulse. The absorbing and source 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. The inner and outer radial boundaries assume the default Neumann condition (Sect. 2.4), a perfect conductor for $H$ type pulses.

The EMP input file COAX01.EIN is listed in Table 11. We shall review the motivations for choosing the parameters in the file. The speed of light in the oil is $1.793 \times 10^8$ 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 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 $Z_0 = 16.87\ \Omega$. The absorbing layers have the same dielectric properties and a nonzero conductivity. The matched conductivity for total absorption is given by

$$\sigma = \frac{1}{\Delta z \sqrt{\mu/\epsilon}}.$$  \hspace{1cm} (29)

Inserting values in Eq. 29, the matched conductivity is $\sigma = 1.746 \text{ S/m}$. The resistance of a coaxial resistor is given by
Table 11: Contents of the file COAX01.EIN

TMax = 4.01E-9
Geometry = Cylin
DUnit = 39.37
Solution = H
  * Solution volume
  Epsi(1) = 2.8
  Mu(1) = 1.0
  * Upstream absorber
  Epsi(2) = 2.8
  Mu(2) = 1.0
  Cond(2) = 1.746
  * Downstream absorber
  Epsi(3) = 2.8
  Mu(3) = 1.0
  Cond(3) = 1.746
  * Source layer
  Epsi(4) = 2.8
  Mu(4) = 1.0
  Source(4) = SMOOTH.SRC (0.50E-9, 125.3)
  Beta(4) = 90.0
  * Boundary
  Fixed(5)
  DTime = 1.0E-9
  History = 12.0 6.5
  History = 4.0 6.5
EndFile

\[ R = \frac{1}{2\pi\sigma\Delta z} \ln\left(\frac{R_o}{R_i}\right). \]  
\( (30) \)

Equation 30 gives the value \( R = 16.87 \, \Omega \), the characteristic impedance predicted by Eq. 27.

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 total current should be 2.0 A. The following lines in the EMP input file define the current source:

Source(4) = Smooth.SRC (0.25E-9, 125.3)
Beta(4) = 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 0.50 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

\[ I = (2\pi r \Delta z)j_r. \]  
\( (31) \)
Figure 18: Example COAX01: temporal variation of $E_r$ at position $z = 12.0''$, $r = 6.5''$.

With $I = 2.0$ A and $\Delta z = 2.54 \times 10^{-2}$ m, the quantity $r_j$, 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 ($r = 6.5'', z = 12.0''$).

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 during the initial pulse propagation is 216.5 V/m. The ratio of the code-predicted electric and magnetic fields is $E_r/H_\theta = 227.3 \ \Omega$. The signal delay time between the probes at $z = 4.0''$ and $12.0''$ gives a propagation velocity of $1.783 \times 10^8$ m/s, close to the predicted value. 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 17b 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 $t = 3.0$ ns, the uniform region fills the entire solution volume. For this data dump, the volume integral function of the analysis menu gives the following values for electric and magnetic field energy in the solution volume: $U_e = 1.888 \times 10^{-8}$ J and $U_m = 1.863 \times 10^{-8}$ J. The predicted value of electric field energy is $U_e = CV_0^2/2$, where $C$ is the capacitance of the line and $V_0$ is the pulse voltage (16.87 V). The capacitance per length of the transmission line (Eq. 28) is $3.313 \times 10^{-10}$ F/m and the length of the solution volume is 16.0” (0.4064 m). The predicted energy is $U_e = 1.916 \times 10^{-8}$ J. In an ideal pulse the electric and magnetic energies should be equal. The predicted power dissipation in each absorbing layers is $I^2R$, where $I = 1.0$ A and $R = 16.87 \ \Omega$. The total is 33.37 W, close to the code calculation of 34.29 W.

The calculation becomes more interesting if we change the downstream absorber to an open-
circuit termination (example COAX02) by setting $\sigma = 0.0$ in Region 3. 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 after a double transit. For long times, the solution is equivalent to a DC voltage on the transmission line. Figure 19 shows the time-dependent electric field measured at position $(z = 4.0'', r = 6.5'')$. As expected, the reflection results in doubling of the electric field and cancellation of the magnetic field. An energy integral for the 6.0 ns solution gives the values $U_e = 7.467 \times 10^{-8}$ J and $U_m = 7.540 \times 10^{-10}$ J. The theoretical values are $U_e = 7.516 \times 10^{-8}$ J and $U_m = 0.0$.

The example COAX3 (Fig. 20) 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
Figure 21: Example COAX03: temporal variation of $E_r$.  

a) Position $r = 6.5''$, $z = 2.0''$.  
b) Position $r = 6.5''$, $z = 14.0''$

coefficients:

$$\rho = \frac{Z_2 - Z_1}{Z_2 + Z_1} = 0.352, \quad (32)$$

$$\tau = \frac{2Z_2}{Z_2 + Z_1} = 1.352. \quad (33)$$

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 20a shows the code-generated waveform of $E_r$ at the upstream probe. The average level of the long-term pulse is close to 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. The color coding of Fig. 20 shows the electric field of the pulse before reacing the downstream absorber.

The voltage of the transmitted pulse is predicted to be $V_t = V_0 = (16.87)(1.352) = 22.81$ V. The radial electric field is related to the voltage by

$$E_r(r) = \frac{V_0}{r \ln(R_o/R_i)}. \quad (34)$$
The predicted field value is 140.9 V. Figure 21b shows the downstream waveform. The average value of $E_r$ is close to the expected value with superposed oscillations generated at the transition.

As an exercise, modify the coaxial line solutions to investigate pulse reflections from open-circuit and close-circuit terminations. The TERMINATION example shows how to model both types of boundaries in one simulation.

### 7.2 Transients in a high-voltage switch

In this example, we shall use EMP to determine the current rise time and transients in a high-voltage Blumlein line controlled by a spark gap\(^1\). Figure 22 shows the geometry. The transmission line discussed in Sect. 7.1 is connected to ground through a shorting switch. The line ($R_i = 5.0''$, $R_o = 8.0''$) is filled with transformer oil ($\epsilon_r = 2.8$) and has a characteristic impedance of 16.87 $\Omega$. The gas-filled switch has a ceramic housing ($\epsilon_r = 6.0$). The outer boundary, line center conductor, switch electrodes and arc column are represented as perfect conductors (i.e., default Neumann condition for the $H$ type pulse over the boundary of the solution volume).

Because EMP handles only dynamic pulses, the key question is how to initiate the solution. A DC charge on the line corresponds to $rH_0 = 0.0$ at all locations; therefore, the null solution would not evolve in time. We can resolve the problem by noting that a DC charge of voltage $V_0$ on a transmission line may be represented as the summation of two square pulses with amplitude $V_0/2$ traveling in opposite directions and reflecting from open-circuit terminations.

---

\(^1\)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 [www.fieldp.com/cpa.html](http://www.fieldp.com/cpa.html).
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 electric field distribution within the switch by generating a dynamic incident step pulse with voltage \( V_0/2 \).

The properties of the upstream absorber and current source layer are the same as those in Sect. 7.1. 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. The color coding in Fig. 22 shows \(|\mathbf{E}|\) for the TEM wave arriving at the switch (2.0 ns). Figure 23a shows a measurement of the radial electric field in the transmission line near the switch. 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 approximately \( V_0 \) at \( t = 0^+ \). The line voltage decreases with time approximately as

\[
\frac{V(t)}{V_0} = \exp\left(-\frac{t}{L_s/Z_0}\right). \tag{35}
\]

In Eq. 35, \( L_s \) is the switch inductance and \( Z_0 \) is the characteristic impedance of the transmission line. Inspection of the curve shows that \( L_s/Z_0 \approx 2.9 \) ns, or \( L_s \approx 49 \) nH. The oscillations superimposed on the waveform result from the capacitive mismatch of the switch assembly. Figure 23a shows the time-variation of \( H_\theta \), proportional to the switch current.

### 7.3 Pulsed electric field probe

This example illustrates how to use EMP to model transients for low-frequency pulsed electric fields in media with conductive and dielectric properties (i.e., real and displacement current). Figure 24a shows the simulation geometry, a phantom experiment for tumor ablation studies. A cylindrical metal probe with a Teflon sheath is driven by a train of current pulses. The probe is immersed in a medium that simulates tissue \((\epsilon_r = 70.0, \sigma = 1.0 \text{ S/m})\). The medium fills a chamber with a grounded metal wall. Electric field, pulsed

In this simulation we use a step-function current pulse with risetime 0.5 ns to investigate both transient and long-term solutions. It is useful to do some preliminary calculations to understand the nature of the solution and to determine parameters to check accuracy. The transit time of an electromagnetic pulse over a distance of 30.0 mm in a medium with \( \epsilon_r = 70.0 \) is about 1.0 ns. We expect to observe transients on this time scale. Furthermore, we can get a good approximation to the steady-state solution by extending the calculation to 10.0 ns. To benchmark the steady-state result, we perform initial electrostatic calculations with EStat. In this case, we assign the fixed potentials \( \phi = 1.0 \) V to the probe and 0.0 V to boundary (top and right-hand sides in Fig. 24). Equipotential lines for a calculation where the Teflon and tissue are treated as ideal dielectrics \((\sigma = 0.0)\) are plotted in Fig. 24a. In contrast, Fig. 24b shows a solution where the materials are treated as conductors \((\sigma = 10^{-6} \text{ S/m for the Teflon, 1.0 S/m for the tissue})\) with no displacement current. This solution would hold at time \( t \gg RC \), where \( R \) and \( C \) are the resistance and capacitance of the system. Taking an integral of electrostatic field energy density over the solution of Fig. 24a, we find that \( C = 3.052 \times 10^{-11} \) F. Similarly, an integral of power density over the solution of Fig. 24b gives \( P = 0.04454 \) W for a drive voltage of 1.0 V. The corresponding resistance is \( R = 22.45 \) Ω, and the \( RC \) time is 0.685 ns. The implication is that the EMP solution at \( t = 10.0 \) ns should be almost identical to the
Figure 23: SWITCH example: **Probe** display of fields measured at the top of the spark gap \([z = 11.0", r = 5.2"]\).  
  
  a) \(E_r(t)\).  
  
  b) \(H_\theta(t)\).
solution of Fig. 24b. For a direct comparison, we renormalize the conductive solution for a probe voltage of 22.45 V, equivalent to a drive current of 1.0 A.

Now consider the calculation described by the input files PULSDE.MIN and PULSDE.EIN. In the $H$ type solution, the flow of real and displacement current is in axial and radial directions, creating the primary field component $rH_\theta$. The solution volume includes only the tissue and Teflon regions. The default Neumann condition applies on the metal surfaces of the probe and the top and right-hand boundaries. We add a line region with the condition $rH_\theta = 0.0$ on the portion of the solution volume along the axis ($r = 0.0$). With regard to the pulse drive, we recognize that if a current $I(t)$ flows in the infinite region to the left of the boundary, then the condition

$$rH_\theta(t) = \frac{I(t)}{2\pi},$$

holds. We can approximate the effect of a long probe by setting a Drive boundary condition on the left-hand boundary at some distance from probe. This region is controlled by the command:

Drive(3) = SMOOTH.DRV 0.5E-9 0.1592

Here, SMOOTH.DRV describes a normalized step function with a risetime of 1.0 s. The first real-number parameter changes the risetime to 0.5 ns and the second $(1/2\pi)$ sets the peak current to 1.0 A (Eq. 36).

Figure 25 shows the spatial distribution of electric field at times 0.25, 0.50, 1.00 and 2.00 ns. The effect of the electromagnetic transit time is apparent as the pulse propagates to the probe tip. The plot also shows the effect of the $RC$ time. The electric field is initially concentrated in the Teflon, which has relatively low dielectric constant. Later in time, the field migrates into the conductive tissue. A plot of $|\mathbf{E}|$ at 10.0 ns in indistinguishable from Fig. 24b. Figure 26 shows
Figure 25: Plots of $|E|$ for the PULSEDE example.
Figure 26: PULSEDE example: calculation of $E_z(t)$ at $z = 7.5 \text{ mm}, r = 0.5 \text{ mm}$.

$E_z(t)$ near the probe tip. Initial transients are damped by the conductivity of the tissue and approach a steady-state level of $E_z = 2266 \text{ V/m}$ over the expected time scale. For comparison, the EStat conductive calculation gives a value 2143 V/m.

### 7.4 Grating with a graded dielectric constant

This example illustrates how to set up a calculation where material properties vary over a region. We investigate the effect of a structure with a periodic variation of dielectric constant in $y$ on a plane pulse. The intent is to generate an approximately planar pulse with amplitude modulation in $y$. Figure 27a shows the geometry. A plane pulse with Gaussian waveform (0.22 ns full width at half maximum) impinges on a dielectric plate of thickness 30 cm with a periodic variation of dielectric constant $\epsilon_r(y)$. The simulation represents one period of the plate in the region $-D \leq y \leq D$, where $D = 10.0 \text{ cm}$. The relative dielectric constant varies as

$$\epsilon_r = 1.5 - 0.25 \left[1.0 - \cos(\pi y/D)\right].$$

(37)

The plate acts as a focusing lens about the position of maximum $\epsilon_r$ ($y = 0.0$).

The solution space for the $E$ type calculation has symmetry boundaries at the top and bottom and a drive boundary on the left-hand side. The drive produces a normalized pulse with peak amplitude $E_z = 1.0 \text{ V/m}$. The novel feature of the calculation is the definition of a spatially-varying dielectric constant through the command:

* Region 2: LENS  
Epsi(2) > 1.5 - 0.25*(1.0 - COS(0.3116*$y))

Figure 27a shows $\epsilon_r(x, y)$ for the resulting distribution. Figure 27b plots lines of $|\mathbf{H}|$ at 1.4 ns after the pulse has passed through the front face of the dielectric. Note the changed shape of the wavefront with resulting focusing. Finally, Fig. 27c shows the state of the pulse at 2.0 ns after emerging from the downstream boundary. The amplitude of pulse varies along $y$ from a minimum of 0.683 V/m at the boundaries to 1.223 V/m at the center.
Figure 27: Example GRATING, graded dielectric structure to produce an intensity-modulated plane pulse.
7.5 Modeling pulsed magnetic fields

The MAGNEAR example illustrates how to use EMP to determine pulsed magnetic fields in the near-field limit. In particular, we shall calculate the diffusive penetration of fields into a conductive material. The benchmark calculation has a simple geometry for comparison to theory and calculations with PerMag. Nonetheless, the example demonstrates the range of techniques in dealing with low-frequency magnetic fields.

Figure 28 shows the geometry. The field is created by a pulsed solenoid in free space with inner radius 5.0 cm, outer radius 8.8 cm and length 16.0 cm. The coil encloses a rod of radius 2.5 cm and length 25.0 cm. In the solution, Region 1 is a matched absorbing layer with \( \sigma = 1.060 \) S/m to approximate the free-space condition. Region 2 (air) and Region 3 (drive coil) have \( \sigma = 0.0 \), while the conductivity of the object may be varied to investigate field diffusion.

We shall run the calculation to \( t_{max} = 3.0 \) ns. This time is much longer than the electromagnetic transit time over the coil region and about twice the transit time over the solution volume. Because the drive current points in the \( \theta \) direction, the cylindrical solution has \( E \) type polarization. Following Sect. 2.3, the source quantity for the coil is \( \frac{dj_\theta}{dt} \). We shall use the normalized step function SMOOTH.SRC, which smoothly rises to a value of 1.0 at \( t = 1.0 \). The source command has the form:

\[
\text{Source(3) = SMOOTH.SRC 0.1E-9 5.575E12}
\]

The first parameter modifies the tabular function so that the risetime is \( t_{rise} = 0.1 \) ns. The second parameter normalizes the derivative of the current so that the total current at 3.0 ns is \( I_\theta = 100.0 \) A:

\[
I_\theta = \left[ (t_{max} - t_{rise}) + \frac{t_{rise}}{2} \right] \frac{dj_\theta}{dt} A_c,
\]

where the coil cross-section area is \( A_c = 6.08 \times 10^{-3} \text{ m}^2 \).

In the initial run we set \( \sigma = 0.0 \) and make a comparison to a static field calculation using PerMag. A common issue for a static solution is the choice of boundary condition – the free-space condition is not an option. The standard approach is to create a solution volume much larger than the coil and to apply a Dirichlet condition (equivalent to a perfectly-conducting
Figure 29: Field line calculations for MAGNEAR example. a) PerMag calculation with 100 A drive. b) EMP calculation at 3.0 ns

Figure 29a shows resulting field lines. The EMP solution at $t = 3.0$ ns is plotted at the bottom. The large transient from the sharp initial current rise has propagated through the absorbing layer, and the solution is approaching a steady-state. Although there are clear differences in the field lines, they occur at a distance from the coil where the field is quite weak. The magnetic fields near the coil are almost identical in the two solutions. PerMag gives a value $|\mathbf{H}| = 471.9$ A/m at $[z = 0.0 \text{ cm}, r = 0.0 \text{ cm}]$, while EMP gives 473.4 A/m (an difference of only 0.3%). The EMP solution would approach the ideal free-space solution if we let $dj/dt$ drop to zero and waited a few ns. In contrast to PerMag, EMP has the interesting feature that the near-field solution is relatively independent of the size of the solution volume. We can verify that long-term field level inside the coil is almost unchanged if we halve the size of the solution volume.

We can observe magnetic field diffusion effects by assigning a non-zero conductivity to the object (Region 4). For a pulsed current of duration $\Delta t$, the conductivity for skin depth $\delta$ is given approximately by

$$\sigma \cong \frac{2\Delta t}{\mu_0\delta^2}. \quad (39)$$

For $\Delta t = 3.0$ ns and $\delta = 0.02$ m, Eq. 39 yields the value $\sigma = 12.0$ S/m. Assigning this
Figure 30: MAGNEAR example: radial variation of $|\mathbf{H}|$ at $z = 0.0$ cm and $t = 3.0$ ns.

conductivity to the object gives the field variation shown in Figure 29. The figure plots a radial scan of magnetic field at the solution midplane from the axis to a position inside the coil. The field within the conducting object ($r < 2.5$ cm) follows the predicted diffusive pattern.
8 Probe – history file plot utility

8.1 Introduction

 Probe is the universal plotting program for all Field Precision initial-value solution codes. You can set from 1 to 20 probes by specifying positions in the solution program command script. The probes record quantities in an element or at a node as a function of time. The resulting text files have names of the form FPREFIX.P01,..., FPREFIX.P12, where FPREFIX is the run prefix.

Table 12 shows the standard probe file format. The first section is a header that contains the following information:

- Generating program name.
- Dimensionality of the generating program (1, 2 or 3).
- The spatial position of the probe (from 1 to 3 quantities).
- The index of the element sampled by the probe.
- The region number of the element.
- Conversion factors for the probe position and the recorded quantities.
- Labels for the recorded quantities.

Although the solution programs and their output files employ SI units (meters, kilograms,...), the graphical analysis displays often use practical units to make it easier to visualize results and to facilitate automatic grids. **Probe** multiplies file quantities by the conversion factors during the loading process. Note that the quantity $D_{Conv}$ and conversion factors for positions are equal to $D_{Unit}$, a variable used in many solution programs. After four lines of label information, the remainder of the file consists of data lines. Each line contains the time (in seconds) and one or more element or node quantities. Real numbers are recorded in E15.6 format.

### 8.2 Loading data files

When you start **Probe** the only active menu option is *Load probe*. Plotting and analysis functions become active when a probe file has been opened. The program displays a dialog showing all files with suffixes of the form P01, ..., P12. Pick a file to analyze and click *OK*. Changing directories in the dialog will change the working directory of the program. If the load is successful, **Probe** creates a default plot of the data (Fig. 31).

The status bar at the bottom of the window contains the name of the probe file, the current plot quantity, and the temporal range of data. The default plot shows the first quantity recorded in the probe file over the full range of time. The horizontal and vertical scales are chosen so that the plot fits on the screen and the grid lines are automatically adjusted so that they lie on even values of the plotted quantity with easily recognized intervals (*e.g.*, 0.02, 0.05, 0.10, ...). The grid intervals are shown in parentheses next to the labels of the horizontal and vertical axes. The title line at the top of the plot shows the following information: generating program, probe file name, element number, region number and position. This information is recorded in hardcopy plots to help you archive your data.

### 8.3 Plot settings

The commands of the *Plotsettings* menu control the quantities, ranges and appearance of the plot. The screen plot automatically updates whenever you make a change.

**PLOTTED QUANTITY**

A dialog shows a list of element quantities included in the probe file. Highlight your choice and click *OK*.

**TIME LIMITS**

By default **Probe** shows the full time-span recorded. You can narrow the range by supplying values for the minimum and maximum time. Uncheck *Autoscale* in the dialog and supply maximum and minimum values. To return to the full range, check the *Autoscale* box.
Table 12: Example of the **Probe** file format

**Field Precision probe file**

Program: KB1
NDimen: 1
XPosition: 4.357E-02
ElementNo: 70
RegionNo: 2
NQuant: 6
DConv: 100.0
QConv1: 100.0
QConv2: 0.001
QConv3: 1.0E-9
QConv4: 1.0
QConv5: 1.0E-6
QConv6: 0.001
QLabel1: x or r (cm)
QLabel2: Rho (gm/cm3)
QLabel3: P (GPa)
QLabel4: T (deg-K)
QLabel5: U (MJ/kg)
QLabel6: v (km/s)

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>Re (m)</th>
<th>Rho (kg/m3)</th>
<th>Press (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.250000E-09</td>
<td>4.357041E-02</td>
<td>1.000000E-01</td>
<td>-1.238464E+02</td>
</tr>
<tr>
<td>1.250000E-08</td>
<td>4.357041E-02</td>
<td>1.000000E-01</td>
<td>-1.238464E+02</td>
</tr>
<tr>
<td>1.874999E-08</td>
<td>4.357041E-02</td>
<td>1.000000E-01</td>
<td>-1.238464E+02</td>
</tr>
<tr>
<td>2.499996E-08</td>
<td>4.357041E-02</td>
<td>1.000000E-01</td>
<td>-1.238464E+02</td>
</tr>
<tr>
<td>3.124995E-08</td>
<td>4.357041E-02</td>
<td>1.000000E-01</td>
<td>-1.238464E+02</td>
</tr>
</tbody>
</table>

...<br><br>

<table>
<thead>
<tr>
<th>Temp (deg-K)</th>
<th>Epsi (J/kg)</th>
<th>V(Av) (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.000000E-03</td>
<td>8.620001E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>1.000000E-03</td>
<td>8.620001E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>1.000000E-03</td>
<td>8.620001E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>1.000000E-03</td>
<td>8.620001E+00</td>
<td>0.000000E+00</td>
</tr>
<tr>
<td>1.000000E-03</td>
<td>8.620001E+00</td>
<td>0.000000E+00</td>
</tr>
</tbody>
</table>

...
VERTICAL LIMITS
In the default mode Probe picks a scale to display the full range of the plotted quantity. You can narrow or expand the range by supplying minimum and maximum values. The program returns to full range if you check the Autoscale box or if you change quantities using Pick plotted quantity.

TOGGLE GRID
Switch between grid and fiducial lines in the plot.

TOGGLE PLOT SYMBOLS
Include or remove symbols to mark the recorded points.

8.4 Plot functions
The commands of the Plot functions menu activate the Oscilloscope mode of the program and also send plots to hardcopy devices or plot files.

OSCILLOSCOPE MODE
When you issue this command, Probe simulates a digital oscilloscope. As shown in Fig. 32, the mouse cursor changes to a cross-hair pattern when it is inside the plot window. The program adds movable fiducial lines to mark the current point. You can drag the fiducials along the time axis by moving the mouse. A box at the bottom of the plot shows values of the time and plotted quantity at the current position. If you press the left mouse button, the program displays a box with the following information about the current point:

- Time, \( t \).
- Value of the plotted quantity, \( V(t) \).
- Derivative of the plotted quantity, \( dV(t)/dt \).
- Definite integral of the plotted quantity, \( \int_0^t V(t')dt' \).

The definite integral is taken from the time of the first recorded value in the probe file to that of the current point. You can find integrals between points by subtracting values. Other functions of the program are deactivated in the Oscilloscope mode. Press the right mouse button or the Esc key to return to normal program operation.

SMOOTH DISPLAY
Use this command one or more times to smooth the currently-displayed trace. Smoothing applies to the screen display and exported plots, but does not affect the data values in the probe file.

DEFAULT PRINTER
Probe can port copies of the plot to any installed Windows printer. The program sends output
to the default printer, so be sure to select the correct device using the *Settings/Printer* function of Windows before making the plot.

**SAVE PLOT FILE**
Use this command to create a graphics file of the current plot in either Windows Bitmap (BMP) or Portable Network Graphics (PNG) formats. In the dialog, specify the format, the size in pixels and the file prefix. The graphics file is created in the current directory.

**COPY TO CLIPBOARD**
Copy the plot to the clipboard in Windows MetaFile format.

### 8.5 Information

**PROBE FILE INFORMATION**
Display information on the probe file in a message box (Fig 33). The quantity \( N_{Skip} \) in line 7 is used for long files. There is no reason to store more than 1000 points for plots on typical screens and hardcopy devices. When there are less than 1000 data lines, *Probe* loads all points \( (N_{Skip} = 1) \). When the file contains 1000 to 2000 data lines, the program loads every second
point ($N_{Skip} = 2$), and so forth. In this way the **Probe** can handle probe files of any length without exceeding memory limits.

**VIEW PROBE FILES**
Load a probe file into the internal editor so you can inspect the data directly. The editor runs in read-only mode so that you cannot change the file. Exit the editor to return to program operation.

**PROBE MANUAL**
Show this document in your default PDF viewer.
Index

absorbing boundary, 15
analytic solutions, 7

capacitance, 56, 65
characteristic impedance, 15, 56
c coaxial transmission line, 56
comment lines, 28
computational mesh, 7
conformal mesh, 8
conical transmission line, 24
Courant stability condition, 5, 12, 15
current density, 14, 22
current source, 58
delimiters, 28
dielectric constant, spatial variation, 67
Dirichlet boundary, 15
element, 7

EMP
  absorbing boundary, 15
  analysis script commands, 53
  boundary conditions, 14
  choosing time step, 12
  command prompt, 36
  current sources, 14
  examples, 56
  features, 4
  file types, 9
  finite-element solution, 22
  input files, 27
  instructions, 38
  learning, 6
  materials, 5, 11, 31
  Movie, 36
  output file format, 38
  output files, 35
  physical approximations, 10
  physical units, 44
  picking element size, 12
  plot quantities, 47
  plot types, 45
  running in a window, 37
  script, 22, 28
script dialog, 20, 37
script example, 23, 29, 58
Solution display, 38
solution procedure, 8, 27
solution types, 12, 21
spatial variations from functions, 42
steps in a solution, 20
tabular functions, 22, 32
time variations from functions, 41

EMP commands
  Analyze, 38
  Beta, 34
  Cond, 31, 43
  Contour style, 49
  Create analysis script, 45
  Drive, 32, 42
  Dt, 21
  DTime, 36
  DUnit, 21, 30
  Edit script, 22, 37
  Element properties, 50
  EMP manual, 38
  Epsi, 31, 42
  Fixed, 31
  Geometry, 21, 30
  Grid control, 48
  History, 36
  Interpolation, 31
  Interpolation method, 51
  Line scan, 50
  Load solution, 44
  Matrix file, 51
  Mesh, 30
  Mouse/keyboard, 48
  Mu, 31, 43
  NHStep, 36
  NStep, 30, 36
  Number of contours, 49
  Open data record, 44
  Oscilloscope mode, 52
  Plot limits, 46
  Plot quantity, 46
  Plot type, 46
point calculation, 50
Region properties, 50
Run, 37
Run analysis script, 37, 45
Set solution series, 44
Set'Time, 35
SetUp, 37
Snap distance, 48
Solution, 21, 30
Source, 34, 42
Stop, 37
TabShift, 35, 42
TExpire, 34
TMax, 21, 30
Toggle element outline, 48
Toggle grid, 46
Toggle snap mode, 48
Void, 15, 32
Volume integral, 51

facet, 7
field
definition, 6
equations, 7
field line plots, 11, 24, 46
finite-element method, 6, 7
basis, 8
free-space boundary, 15

inductance, 63
internal void region, 32

magnetic diffusion, 70
magnetic field intensity, 10
mathematical function syntax, 41
Maxwell equations, 10

Neumann boundary, 14
node, 7
node equations, 7, 8, 11, 22
numerical dispersion, 12
numerical solutions, 7

open circuit boundary, 15

polarization classes, 10
Probe, 24, 72
file format, 72