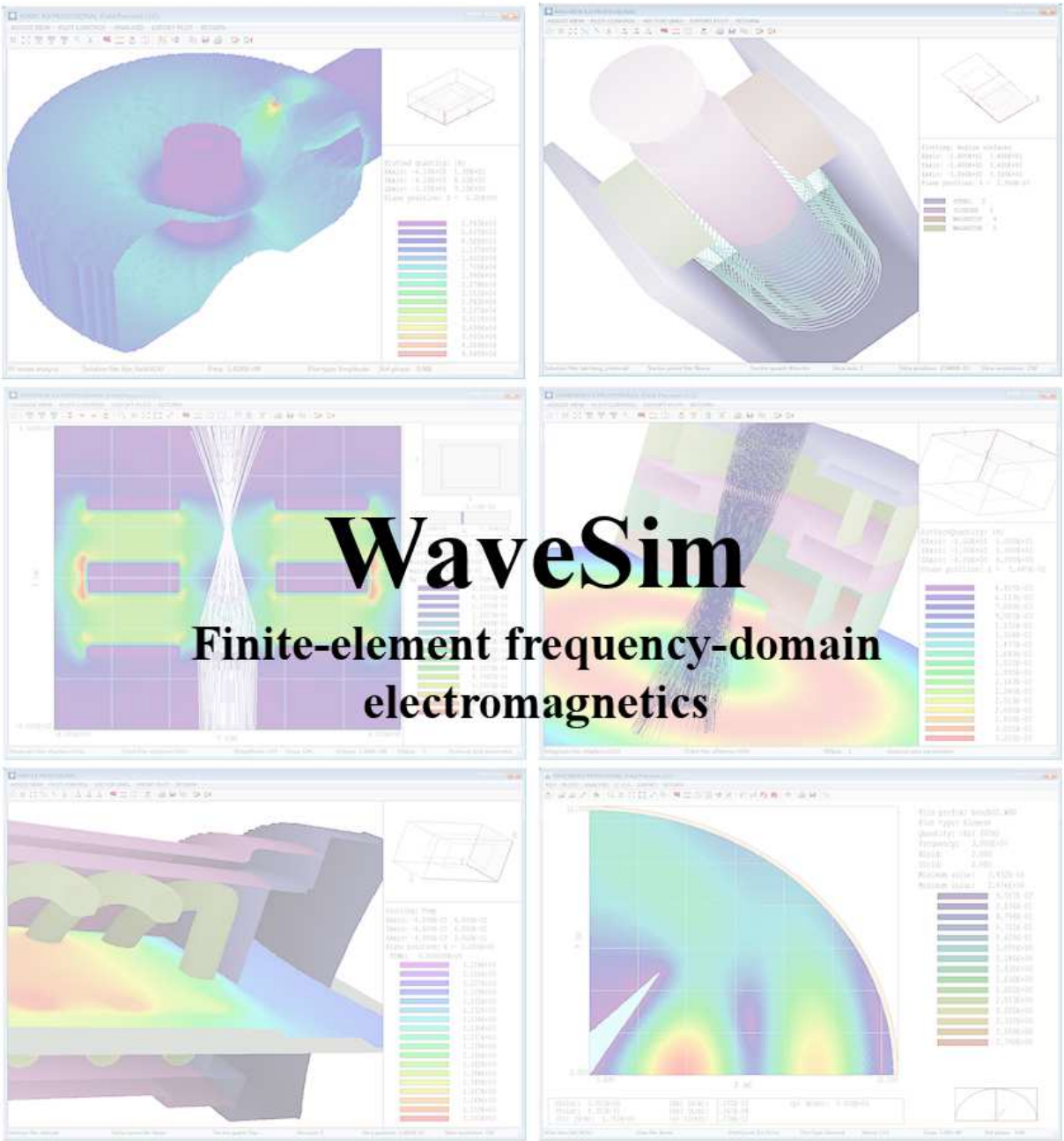
Field Precision LLC



WaveSim

Finite-element frequency-domain electromagnetics

EMail: techinfo@fieldp.com

URL: http://www.fieldp.com

Contents

1 Introduction **4**
- 1.1 Program functions . . . 4
- 1.2 Learning **WaveSim** . . . 4
- 1.3 Finite-element solution procedure . . . 5
- 1.4 Scripts and data files . . . 8

2 Creating and analyzing a WaveSim solution **9**
- 2.1 Mesh generation . . . 9
- 2.2 Solution setup . . . 11
- 2.3 Running and analyzing the scan solution . . . 15
- 2.4 Setting up the resonant mode calculation . . . 15
- 2.5 Plotting and analyzing the solution . . . 19

3 Electromagnetic theory in WaveSim **23**
- 3.1 Electromagnetic equations . . . 23
- 3.2 Properties of plane electromagnetic waves . . . 24
- 3.3 Reflection of plane electromagnetic waves . . . 25
- 3.4 Electromagnetics on a conformal triangular mesh . . . 27
- 3.5 Absorbing boundary . . . 30
- 3.6 Energy loss in materials . . . 32
- 3.7 Basic properties of resonators . . . 34
- 3.8 Waveguides and resonant cavities . . . 39
- 3.9 Power loss in resonant cavities . . . 41
- 3.10 Techniques for scattering solutions . . . 45

4 WaveSim solution reference **49**
- 4.1 **WaveSim** script format . . . 49
- 4.2 Program control commands . . . 51
- 4.3 Commands for material properties . . . 54
- 4.4 Running **WaveSim** . . . 56
- 4.5 Format of the **WaveSim** output file . . . 57

5 Solution analysis reference **60**
- 5.1 File menu commands . . . 60
- 5.2 Plot menu commands . . . 61
- 5.3 Analysis menu commands . . . 66
- 5.4 Scan plot menu . . . 70
- 5.5 Analysis script commands . . . 71

6 Resonant mode calculation tutorials **73**
- 6.1 Basic resonant cavity . . . 73
- 6.2 Waveguide cutoff modes . . . 77
- 6.3 Resonant cavity with material losses . . . 80

7 Electromagnetic scattering tutorials **83**
7.1 Coaxial transmission line 83
7.2 Two slit diffraction pattern 87
7.3 Electric dipole antennas 87
7.4 Magnetic dipole antenna 91
7.5 Wave scattering from objects 93

1 Introduction

1.1 Program functions

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

- Frequency scans to investigate the response of bounded systems.
- Automatic searches to locate resonances of closed structures, with or without material power loss.
- Scattering solutions at a given frequency for open and lossy systems.

WaveSim applies the method of matched termination boundary layers in scattering calculations. These layers accurately represent free-space boundaries. They are more versatile and efficient than the look-back technique used in finite-difference calculations. The method of distributed sources is another advanced feature of **WaveSim**. It allows you to generate ideal incident plane waves in a bounded anechoic chamber for studies of electromagnetic wave scattering. The material model of **WaveSim** is flexible. You can specify real and imaginary parts of the dielectric constant and magnetic permeability at the target frequency for up to 127 material regions.

An interactive graphical environment gives plots and scans of primary and secondary field components with automatic labeling of the mode options. The program calculates volume and surface integrals to determine material and wall losses in microwave devices.

1.2 Learning WaveSim

The size of this manual reflects the extensive capabilities of the **WaveSim** 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 2. 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. The examples give insights into solution options and may serve as templates for your own work. Here are suggestions for additional activities:

- Chapter 3 reviews the numerical methods applied in **WaveSim** for two-dimensional electromagnetics. The material is included for reference. You need not understand all details to get started with the program. You can check relevant sections as you move to advanced solutions.

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

- Be sure to read Chap. 2 of the **Mesh** manual which covers important concepts for conformal meshes and finite-element solutions.

- The finite-element approach involves three basic steps: 1) divide the solution space into small volumes (that is, create a *mesh*), 2) use the mesh to convert the electromagnetic equations into a discrete form and carry out a numerically solution and 3) analyze the results. As an introduction to the first task, read Chap. 3 pf the **Mesh** manual. The chapter 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.

- Chapters 4 and 5 cover basics of program operation. Chapter 4 discusses how to supply information to the code and how to perform the finite-element solution. Chapter 5 covers code capabilities plotting and quantitative analysis.

- The best shortcut to the effective application of **WaveSim is** to follow the tutorials in Chapters 6 and 7. The examples cover the full range of code applications. Chapter 6 concentrates on closed structures (resonant cavities and waveguides) and Chap. 7 addresses open structures (diffraction, electromagnetic scattering,...).

- As you gain experience, you can take advantage of the full range of **Mesh** capabilities. Chapter 4 of the **Mesh** manual is a comprehensive reference on the **Mesh** drawing editor. Chapter 5 covers processing, plotting and repairs of meshes. Other chapters cover advanced techniques in **Mesh**. 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 **WaveSim**. 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 **WaveSim** solution.

The term *field* indicates a quantity (scalar or vector) defined over a region of space. Examples of fields include the vector electric field $\mathbf{E}$ 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 may be solved analytically if the system geometry and material properties are simple. Analytic solutions are extremely difficult in systems with asymmetric structures or nonlinear materials. Furthermore, closed-form results are often 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.

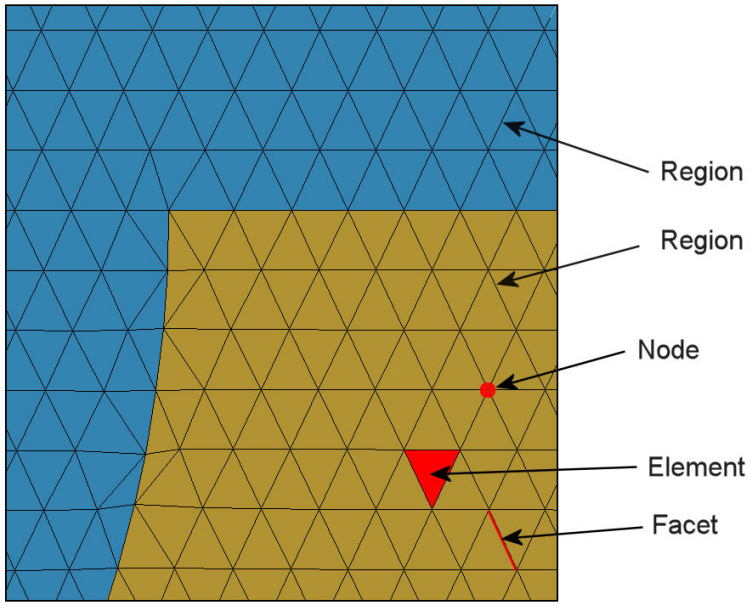


Figure 1: Conformal triangular mesh – definition of terms.

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 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 1 shows the type of mesh used for the two dimensional solutions of **WaveSim**. 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. 1 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. 1 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 **WaveSim**, solution of the set representing the Helmholtz equation gives either E_z, rE_θ, H_z or rH_θ (depending on the system symmetry and field polarization). 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 component.

With this background, we can understand the steps in an **WaveSim** solution:

- 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 electrodes and dielectrics.

- **Mesh** analyzes the boundary specifications and automatically generates a set of conformal triangles such as those of Fig. 1. The program creates an output file that lists the locations of nodes and the identities of elements.

- You define the material properties of regions in the solution volume. Usually, you can peform this task with the **WaveSim** interactive *Setup* dialog.

- **WaveSim** reads the mesh geometry and applies the material parameters to generate the linear equation set.

- **WaveSim** solves the boundary-value problem using a direct matrix inversion technique and records primary field quantities as a function of position. The resulting file serves as a permanent record of the solution that can be re-loaded for latter analysis.

- You can use the interactive graphical environment of **WaveSim** 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.

Table 1: **WaveSim** files

Name	Function
`MName.MIN`	**Mesh** input script (definition of foundation mesh and region outlines)
`MName.MLS`	**Mesh** diagnostic listing
`MName.MOU`	**Mesh** output (node locations and element identifies)
`WName.WIN`	**WaveSim** input script (run control and material properties)
`WName.WLS`	**WaveSim** diagnostic listing
`WName.WOU`	**WaveSim** data dump, field quantities at all positions
`AName.SCR`	**WaveSim** script for automatic data analysis

1.4 Scripts and data files

Mesh and **WaveSim** read and generate several types of files. For a typical solution it is not necessary for your 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 **WaveSim** that require direct entries to the input scripts. For convenience, all input scripts and output data files are in text format. Both **Mesh** and **WaveSim** feature integrated text editors. Table 1 lists the file types and functions in the **WaveSim** package. Note that the suffixes indicate the file function.

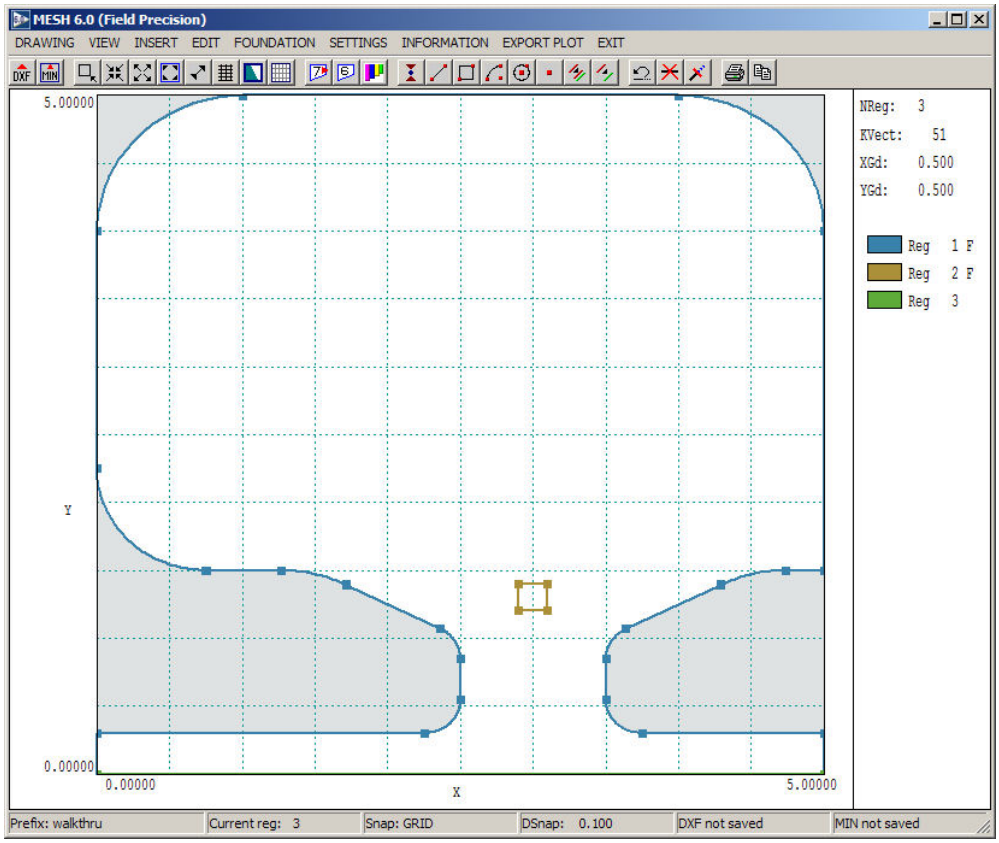


Figure 2: Screen display of the **Mesh** drawing editor – `WALKTHRU` example. Dimensions in cm.

2 Creating and analyzing a WaveSim solution

As an introduction to **WaveSim** procedures, we shall follow an example in detail. We want to calculate the resonant frequency and quality factor (Q) for the klystron cavity shown in Fig. 2. Note that the cavity is a figure of revolution about the axis at the bottom. In cylindrical **WaveSim** calculations, the horizontal axis is associated with z and the vertical axis with r. To begin, move the files `WALKTHRU.DXF` and `WALKTHRU.SCR` to a working directory. Make sure that the *Data directory* in **TC** points to the working directory.

2.1 Mesh generation

We will first use the **Mesh** to create a script that describes the geometry. Don't worry if you make a mistake in the following sequence. You can always use the prepared script `WALKTHRU.MIN` included with the examples. The file `WALKTHRU.DXF` is a standard Drawing Exchange File prepared with LibreCAD (a freeware utility available at https://librecad.org/). Advanced features of the program were employed to generate the fillets (rounded corners) on the cavity boundaries. We shall add region names and an additional region in the **Mesh** drawing editor. Launch **Mesh** and choose the command *Create script/DXF import*. After you pick the `DXF` file, the

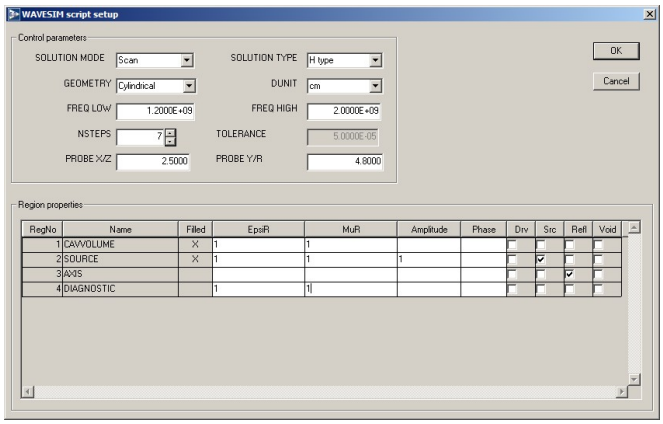


Figure 3: Region properties dialog.

program enters the drawing editor mode and displays the vectors that outline the boundaries of regions. (Fig. 2). We added shading to mark the void areas that are not included in the solution volume.

Initially, the drawing contains a layout region and three active regions. Click on *Setting/Region properties* to display the dialog of Fig. 3. The vectors of Region 1 comprise the closed outline of the cavity. The region has the *Filled* property so the region number is assigned to all enclosed nodes and elements. This region will have the properties of vacuum in the **WaveSim** calculation. Region 2 (a small square) is a drive-current region required for resonant-cavity calculations. This filled vacuum region will have an RF current density of arbitrary amplitude to initiate the mode. Region 3 consists of a single vector that extends along the axis. This region will be used to set nodes to the fixed condition $rH_\theta = 0.0$.

Before we generate a **Mesh** geometry script from the drawing, we shall add a region. The intent is to assign region number 4 to nodes on the physical cavity boundary (excluding the axis). This identification will be used for a calculation of resistive wall losses. Pick the command *Edit/Copy region*. To select Region 1, move the mouse inside the drawing close to a vector on the region. When you click the left button, **Mesh** highlights all vectors that constitute the boundary. The program then prompts for two points that define a displacement for the copied region. We want no displacement, so click the left button twice with the mouse in the same position. The display is updated and the list on the right-hand side of the screen shows that a new region has been added. To work on the region, return to the *Settings/Region properties* dialog. Assign the name *Diagnostic* to Region 4. Also, check the *Locked* attribute for Regions 0, 1, 2 and 3 and uncheck the *Visible* attribute. Click *OK* to exit the dialog. The display shows only the blue outline of Region 4 (Fig. 4).

We need to delete the on-axis vector. Choose *Edit/delete/vector*, move the mouse close to the axis and click the left button. You can check that the operation was successful by comparing the number of vectors in Regions 1 and 4 in the *Settings/Region properties* dialog. To save the modified drawing, use the *Drawings/Export DXF* file command. To leave the drawing editor and to create a **Mesh** geometry script, pick the *Exit* command and choose the option *Save*.

If you did not complete the above procedure, copy the file `WALKTHRU.MIN` from the example directory to continue the demonstration. In the main menu of **Mesh**, click on the command *File/Load script (MIN)* and choose `WALKTHRU.MIN`. Click the *Process* command. **Mesh** reads the boundary specifications, shapes elements to conform to the material divisions and assigns

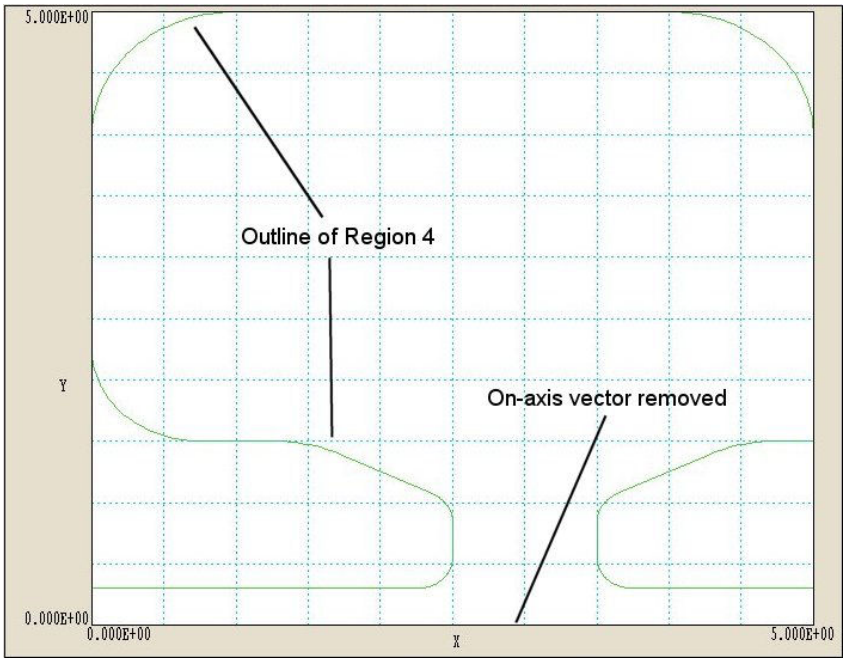


Figure 4: `WALKTHRU` example – Region 4 with on-axis vector removed.

region numbers to nodes and elements. Choose the command *File/Save mesh (MOU)* to create the output file `WALKTHRU.MOU`. This file will supply the geometric information for the **WaveSim** solution. At this point, you can experiment with some of the plot capabilities of **Mesh** by clicking on *Plot-repair*. Figure 4 shows a detail of the conformal elements near the axis. The box to the right shows region names along with the number of elements (for filled regions) or the number of nodes (for open regions).

2.2 Solution setup

The **WaveSim** script sets control parameters and assigns physical properties of materials to the regions. The simplest way to generate a script is with the interactive dialog in **WaveSim**. You can also use a text editor to write a script, make changes or add advanced directives. Chapter 4 covers the script format and gives a detailed discussion of commands. This section covers the interactive dialog and gives a brief discussion of options.

Run **WaveSim** 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:

- Set up program controls and material properties.
- Generate and solve the finite-element equations.
- 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 `WALKTHRU.MOU`.

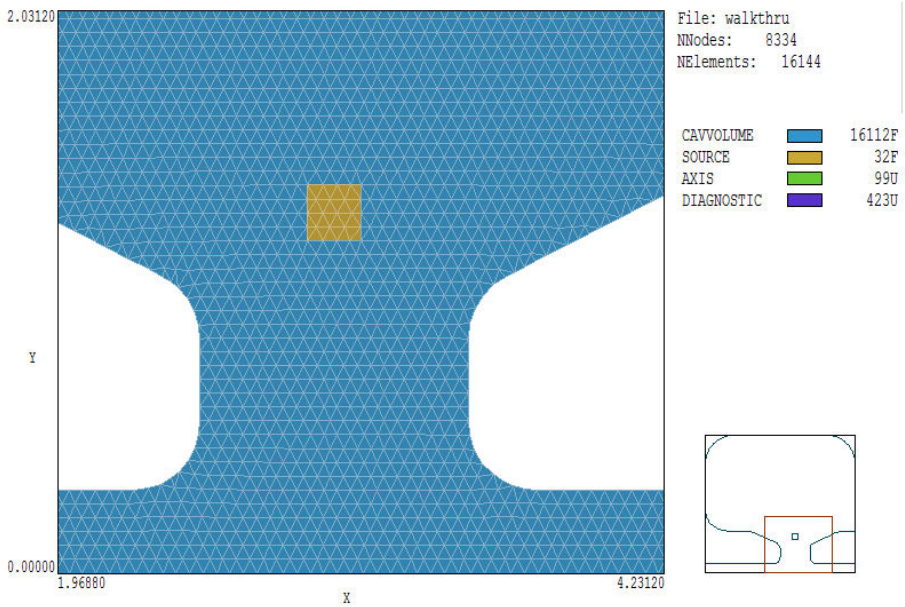


Figure 5: `WALKTHRU` example – detail of completed mesh.

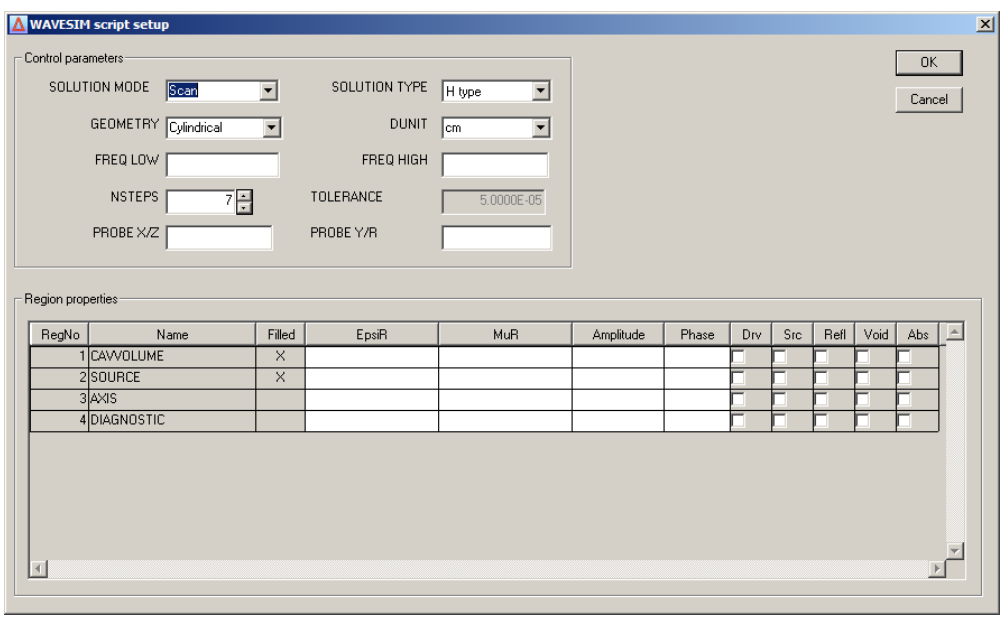


Figure 6: Dialog to set **WaveSim** control parameters and region properties.

WaveSim loads the information and displays the dialog of Fig. 6. 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:

- SOLUTION MODE. Set the type of calculation. There are three options in **WaveSim**: 1) **Search** makes a high-resolution search for the frequency and field distribution of a resonant electromagnetic mode in a closed system, 2) **Scan** excites a closed system over a broad frequency range to search for resonance conditions and 3) **Scatter** excites an open system at a specific frequency to calculate scattered electromagnetic waves. Following *Search* and *Scan* calculations, **WaveSim** creates a file of field values at all points in the solution volume.

- SOLUTION TYPE. Choose the field polarization, E or H. For an E type solution in planar geometry, the electric field points along z and there are H_x and H_y components of magnetic field intensity. Table 2 summarizes the full set of options.

- GEOMETRY. The symmetry of the solution: *planar* or *cylindrical*. A planar solution applies to a system with variations in x-y and infinite length in z, while a cylindrical system has symmetry in θ.

- DUNIT. Set a factor to convert the units used for coordinates in the **Mesh** file to meters. The value is the number of mesh units per meter: 39.37 for inches, 100.0 for cm.

- FREQUENCY or FREQ LOW. The function of the dialog field depends on the setting of the solution type. In the *Search* mode, enter the lower limit of the search range in Hz. In the *Scan* mode, the value equals the lower limit for the frequency scan. Finally, in the *Scatter* mode the value equals the frequency of electromagnetic waves generated by sources or drives.

- FREQ HIGH. This field is inactive in the *Scatter* mode. Enter the upper frequency limit in Hz for the *Search* or *Scan* mode.

- NSTEPS. This field is inactive in the *Scatter* mode. In the *Search* mode the value equals the number of steps in the iterative search for a resonant mode. Higher values give better accuracy at the expense of a longer run time. In the *Scan* mode the value equals the number of frequency intervals in the scan.

- PROBEX/Z and PROBEY/R. The fields are active for the *Search* and *Scan* modes. The values give the point in the solution volume to sample fields. Enter values in units set by *DUnit*. In the *Search* mode, changes in the probe response with frequency are used to identify a resonance. In the *Scan* mode, probe values at different frequencies are reported in the listing file `WPREFIX.WLS`.

The entries in the column options of the region grid box determine the physical properties of the corresponding region. The first three columns show the region number, the name assigned in **Mesh** and the *Fill* status. Entries in the next two columns define the relative dielectric constant ϵ_r and magnetic permeability μ_r of a material region. Define a lossy material by entering complex values. The fields accept either one real number or two real numbers separated by any of the standard delimiters. Here are some examples

Table 2: Options for two-dimensional solutions, showing the primary and secondary field quantities.

Type	**Symmetry**	**Primary**	**Secondary**
E	Planar	E_z	H_x, H_y
E	Cylindrical	rE_θ	H_z, H_r
H	Planar	H_z	E_x, E_y
H	Cylindrical	rH_θ	E_z, E_r

- Vacuum or air: $\epsilon_r = 1.0$, $\mu_r = 1.0$.
- Ideal dielectric: $\epsilon_r = 81.0$, $\mu_r = 1.0$.
- Lossy dielectric or material with non-zero conductivity: $\epsilon_r = (5.0, 26.7)$, $\mu_r = 1.0$.
- Lossy magnetic material: : $\epsilon_r = 1.0$, $\mu_r = (25.0, 0.92)$.

The four check boxes on the right-hand side signal special properties for **WaveSim** solutions:

- **Drv**. In a drive region the primary field quantity at nodes has a fixed complex value. If the *Drv* box is checked, you must enter values for the amplitude and phase of the primary field value in the sixth and seventh columns. Enter the phase in degrees and the amplitude in the following units: E_z (V/m), rE_θ (V), H_z (A/m) or rH_θ (A).
- **Src**. A source region has a current density to generate electromagnetic radiation. If the *Src* box is checked, you must enter values for the amplitude and phase of the current in the sixth and seventh columns. Depending on the field type and symmetry, enter amplitude values for the following quantities: (E type, planar) j_z in A/m^2, (E type, cylindrical) rj_θ in A/m, (H type, planar) j_x and j_y in A/m^2, (H type, cylindrical) rj_z and rj_r in A. An addition parameter is required for H type solutions, the direction of the current in the x-y or z-r plane. The **WaveSim** dialog sets a default value for the angle $\beta = 0^o$ (aligned along x or z). You can change the value in the scrip0t with a text editor. Note that you can also specify values of ϵ_r and μ_r for a source region. The default is $\epsilon_r = 1.0$ and $\mu_r = 1.0$.
- **Refl**. The primary field at nodes of the region has the fixed value $[0.0, 0.0]$. The reflection condition represents a metal surface (short-circuit boundary) for E type solutions and an ideal open non-radiating boundary (open-circuit) for H type solutions.
- **Void**. This condition is the inverse of the *Reflect* condition. A void represents a metal wall for H type solutions and an open-circuit for E type solutions. All unspecified external boundaries of the solution volume automatically assume the *Void* condition.
- **Abs**. An absorbing layer is a region of uniform thickness Δ that absorbs almost all the energy of a traveling wave. Absorbing layers often cover the outer boundary of the solution volume to represent the free-space condition. The setup dialog makes an entry in the script with a symbolic value of Δ. You must set the actual value by editing the script. Enter the layer thickness in the current units (defined by *DUnit*).

To continue, fill the dialog entries with the values shown in Fig. 6. We seek the TM_{010} mode of the cylindrical cavity, an H type solution with field components rH_θ, E_z and E_r. The first calculation is a scan over a broad frequency range to find the approximate location of the resonant mode by observing zero crossings of the probe signal. The cavity has an outer radius $R_0 = 0.05$ m. An upper limit on the frequency of the TM_{010} mode is:

$$f = \frac{2.405c}{2\pi R_0}. \tag{1}$$

or $f = 2.3$ GHz. The re-entrant beam tube adds capacitance near the axis and lowers the frequency. As an initial guess we shall use the range 1.2 to 2.0 GHz. The cavity response is calculated at 31 evenly-spaced values over the frequency interval. Zero-crossings of the probe signal indicate resonances. For good sensitivity, the probe is located in the vacuum region at a position where the primary field (rH_θ) has a high value.

The values in the grid field assign the following physical properties to the regions:

- **Region 1**. Vacuum with $\epsilon_r = 1.0$ and $\mu_r = 1.0$.
- **Region 2**. A vacuum region with a source current density aligned along z to generate the field component E_z. The amplitude is $rj_z = 1.0$ A/m and the phase is 0.0^o. (Note that the source amplitude is arbitrary in resonant-cavity calculations with no losses.)
- **Region 3**. The boundary condition $rH_\theta = 0.0$ on the axis.
- **Region 4**. A diagnostic line of nodes along the cavity wall with the properties of vacuum.

When entries are complete, click *OK* and save the file as `WALKTHRU.WIN`.

2.3 Running and analyzing the scan solution

In the main **WaveSim** menu, click the command *Solve/Run* and choose the input file `WALKTHRU.WIN`. The program loads the input data files, sets up node equations and carries out a series of sparse matrix inversions to find complex values of rH_θ at all nodes. The scan may take several minutes – the status bar shows the progress of the calculations. When the run is complete, click on the command *File/Edit listing (WLS)* and choose `WALKTHRU.WLS`. The file contains a listing of the probe response (shown in Table 3) at 31 frequencies (30 uniform intervals). Figure 7 shows a plot of the real part of the probe response. The zero crossing near 1.5 GHz indicates the presence of a resonance. Close the editor to continue.

2.4 Setting up the resonant mode calculation

Based on the information from the scan, we shall search for the fundamental cavity resonance in the frequency interval 1.45 to 1.55 GHz. A quick way to set up the run is to use the internal program editor to make small changes in the **WaveSim** script. Choose the command *File/Edit script (WIN)* and load `WALKTHRU.WIN`. Note how the information entered in the dialog has been encoded in a permanent text record. Make the following changes:

- Replicate (copy and paste) the lines beginning with *Mode*, *Range* and *FStep*.

Table 3: Results of the frequency scan – `WALKTHRU` example

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

NStep	Frequency	Probe(Real)	Probe(IMag)	Probe(Mag)
0	1.200000E+09	7.733434E-04	0.000000E+00	7.733434E-04
1	1.226667E+09	8.492374E-04	0.000000E+00	8.492374E-04
2	1.253333E+09	9.425691E-04	0.000000E+00	9.425691E-04
3	1.280000E+09	1.060607E-03	0.000000E+00	1.060607E-03
4	1.306667E+09	1.214461E-03	0.000000E+00	1.214461E-03
5	1.333333E+09	1.423041E-03	0.000000E+00	1.423041E-03
6	1.360000E+09	1.720786E-03	0.000000E+00	1.720786E-03
7	1.386667E+09	2.183340E-03	0.000000E+00	2.183340E-03
8	1.413333E+09	2.991458E-03	0.000000E+00	2.991458E-03
9	1.440000E+09	4.769071E-03	0.000000E+00	4.769071E-03
10	1.466667E+09	1.192175E-02	0.000000E+00	1.192175E-02
11	1.493333E+09	-2.353390E-02	0.000000E+00	2.353390E-02
12	1.520000E+09	-5.901643E-03	0.000000E+00	5.901643E-03
13	1.546667E+09	-3.362803E-03	0.000000E+00	3.362803E-03
14	1.573333E+09	-2.349347E-03	0.000000E+00	2.349347E-03
15	1.600000E+09	-1.803289E-03	0.000000E+00	1.803289E-03
16	1.626667E+09	-1.462793E-03	0.000000E+00	1.462793E-03
17	1.653333E+09	-1.229809E-03	0.000000E+00	1.229809E-03
18	1.680000E+09	-1.060601E-03	0.000000E+00	1.060601E-03
19	1.706667E+09	-9.322275E-04	0.000000E+00	9.322275E-04
20	1.733333E+09	-8.315093E-04	0.000000E+00	8.315093E-04
21	1.760000E+09	-7.502444E-04	0.000000E+00	7.502444E-04
22	1.786667E+09	-6.836829E-04	0.000000E+00	6.836829E-04
23	1.813333E+09	-6.279686E-04	0.000000E+00	6.279686E-04
24	1.840000E+09	-5.807320E-04	0.000000E+00	5.807320E-04
25	1.866667E+09	-5.401655E-04	0.000000E+00	5.401655E-04
26	1.893333E+09	-5.050357E-04	0.000000E+00	5.050357E-04
27	1.920000E+09	-4.742479E-04	0.000000E+00	4.742479E-04
28	1.946667E+09	-4.471136E-04	0.000000E+00	4.471136E-04
29	1.973333E+09	-4.230709E-04	0.000000E+00	4.230709E-04
30	2.000000E+09	-4.015633E-04	0.000000E+00	4.015633E-04

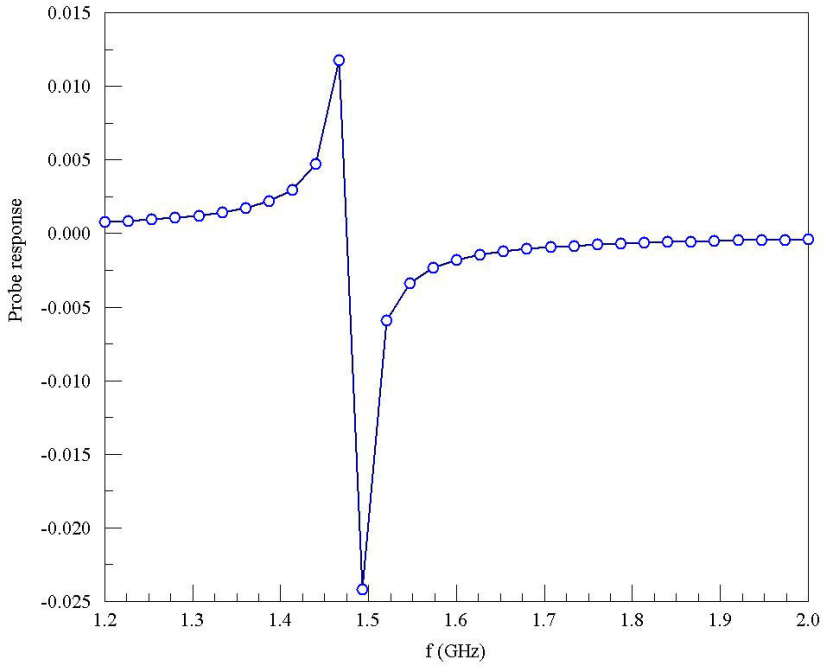


Figure 7: Probe response versus frequency for the `WALKTHRU` example.

- Deactivate the original lines by turning them into comments (add an asterisk at the beginning).
- In the new lines, set the mode to *Search*, the range to $1.45 \times 10^9 \leq f \leq 1.55 \times 10^9$ and the number of frequency steps to 7.
- Save the file and exit the editor.

The final script should look like the example in Table 4. Note that the order of the commands is not important – **WaveSim** reads and stores the full content of the script before beginning operations.

Start the second solution using the *Process* command. The resonance search takes less than a minute. The modified listing file has the following entry:

```
--- Response at frequency interval boundaries ---
     Frequency (low):   1.450000E+09
      Response (low):   1.624939E+02
    Frequency (high):   1.550000E+09
     Response (high):  -3.134437E+02

Iteration:   1   Frequency:   1.484142E+09   Response:   1.280748E+00
Iteration:   2   Frequency:   1.484412E+09   Response:  -9.944601E-02
Iteration:   3   Frequency:   1.484374E+09   Response:   1.367313E-01
Resonance search successful.
     Final frequency:   1.484412E+09
     Writing data to file walkthru.WOU
```

In this run, **WaveSim** identified the fundamental mode frequency as 1.4847 GHz and created an output file of field values at that frequency. As shown in the list extract, the program

Table 4: Modifed script WALKTHRU.WIN

```
Mesh = walkthru
Geometry = Cylin
DUnit =   1.0000E+02
Solution = H
Mode = Search
Range =   1.4500E+09    1.5500E+09
FStep =     7
Probe =   2.5000E+00    4.8000E+00
* Mode = Scan
* Range =   1.2000E+09    2.0000E+09
* FStep =     30
* Region 1: CAVVOLUME
Epsi(1) =   1.0000E+00    0.0000E+00
Mu(1) =   1.0000E+00    0.0000E+00
* Region 2: SOURCE
Source(2) =   1.0000E+00      0.000   0.0
* Region 3: AXIS
Reflect(3)
* Region 4: DIAGNOSTIC
Epsi(4) =   1.0000E+00    0.0000E+00
Mu(4) =   1.0000E+00    0.0000E+00
EndFile
```

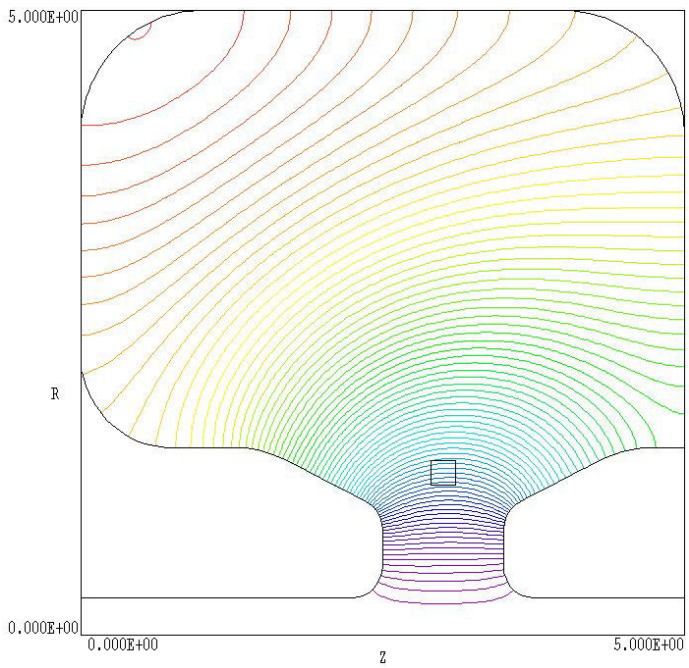


Figure 8: Electric field lines separated by equal intervals of magnetic field intensity flux – `WALKTHRU` example.

first calculates the probe response at the ends of the frequency range to ensure that there is a positive/negative transition. If the condition is met, **WaveSim** seeks a zero in the probe signal. The probe method is more versatile than a direct solution for the eigenvalues of the finite-element node matrix. Using the method, **WaveSim** can locate resonances in structures with material losses.

2.5 Plotting and analyzing the solution

Click on the "3" tool or the *Analyze* menu command. In the dialog, pick the file `WALKTHRU.WOU`. The program loads the data and displays the default plot of electric field lines shown in Fig. 8. Contours of rH_θ are separated by equal intervals of magnetic-field flux; therefore, the spacing is larger near the axis. We shall perform two tasks:

- Plot the magnitude and direction of the electric field near the beam gap for a given amplitude of deceleration field.
- Determine the Q value for an ideal copper cavity.

In the *Search* mode, the magnitude of field values is arbitrary. To begin, we shall normalize the solution so that it has a desired peak deceleration field. Click on *Analyze/Analysis settings/Scan plot quantity* and choose $|E_z|peak$. Next, choose *Analysis/Line scan* and move the mouse pointer into the plot area. Note that the cursor changes to a cross-hair pattern. Observing the coordinates listed in the status bar at the bottom of the screen, move the pointer to a position near the point $(z = -1.5, r = 0.0)$ and click the left mouse button. Then move the mouse to the position $(z = +1.5, r = 0.0)$ and click again. **WaveSim** calculates a sequence of

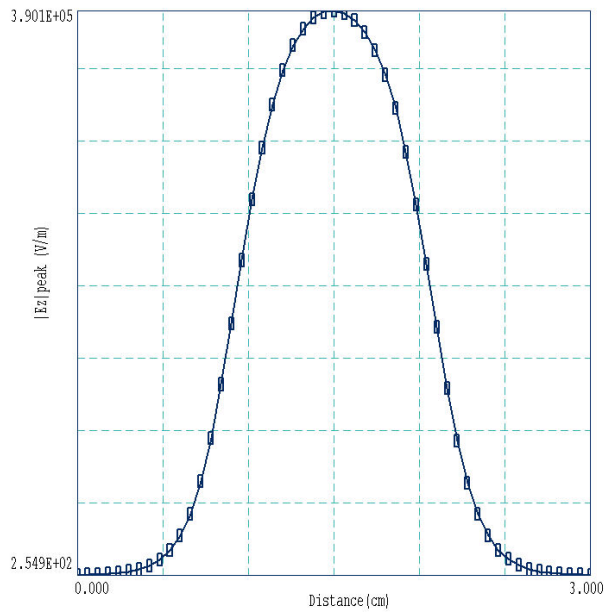


Figure 9: Line scan of $|E_z|peak$ along the axis from $z = -1.5$ cm to $z = +1.5$ cm – `WALKTHRU` example.

values of $|E_z|$ amplitude and displays the results (Fig. 9). The peak field shown in the figure is 3.901×10^5 V/m. (Note that the relative value may not be the same on your computer because of differences in mathematics coprocessors). We shall normalize the solution so that the peak electric field is 7.5×10^6 V/m. Exit the scan plot menu by clicking *Return* and choose the command *Analysis/Analysis settings/Renormalize fields*. Enter the value $(7.5 \times 10^6)/(3.901 \times 10^5) = 19.23$ in the box and click *OK*. **WaveSim** multiplies all values of rH_θ by the normalization factor. Use the *File/Save solution file* command to create a file `WALKTHRUNORM.WOU` with the modified values.

To create the field plot, click on the command *Plot/Plot settings/Type* and choose the option *Vector*. **WaveSim** automatically changes the plot quantity to $|\mathbf{E}|peak$ (peak value of $\sqrt{E_z^2 + E_r^2}$). Initially, the plot has color-coded field information but no vectors. The reason is that the instantaneous electric field at all positions equals 0.0 at the default reference phase of $\phi = 0.0^o$. Click on the command *Analysis/Analysis settings/Reference phase* and set $\phi = 90.0^o$. You can use the *Zoom* tool to create the plot of Fig. 10.

To conclude, we shall calculate the quality factor for the cavity using automatic analysis features of **WaveSim**. Click on *File/Run script* and choose `WALKTHRU.SCR`. The script has the contents:

```
INPUT WALKTHRUNORM.WOU
OUTPUT WALKTHRUNORM
VOLUMEINT
ENDFILE
```

The script loads the normalized solution file, opens a data file with the name `WALKTHRUNORM.DAT` and records the results of a volume analysis. To see the results, click on *File/Close data record* and then view `WALKTHRUNORM.DAT` in the internal editor. Table 5 shows the results of the analysis.

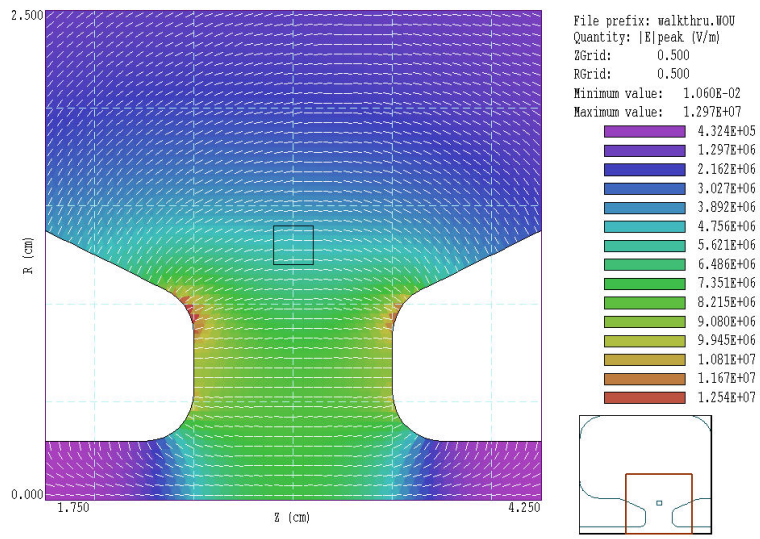


Figure 10: Electric field magnitude and direction near the deceleration gap – normalized example `WALKTHRUNORM`.

At the chosen field normalization, the electromagnetic energy inside the cavity is $U = 3.292 \times 10^{-3}$ joules. The other quantity to find the Q-factor is the following integral taken over the surface of Region 4:

$$\int_S \int dS \; H_{\parallel}^2/2. \tag{2}$$

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

$$P = R_s \int_S \int dS \; H_{\parallel}^2/2, \tag{3}$$

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

$$R_s = 2.61 \times 10^{-7} \sqrt{f} \quad (\Omega/\mathrm{m}). \tag{4}$$

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

$$Q = \frac{2\pi f \; U}{P}, \tag{5}$$

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

Table 5: Volume integrals for the WALKTHRU example

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

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

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

3 Electromagnetic theory in WaveSim

This chapter reviews some basic electromagnetic theory and summarizes the numerical methods employed in **WaveSim**. Section 3.1 lists the form of the Maxwell equations used in the program. Sections 3.2 and 3.3 review the nature of plane waves and reflection at boundaries. The finite-element form of the Maxwell equations on a conformal triangular mesh is discussed in Sect.3.4. It is not necessary for you to understand all details of the complex expressions. The main purpose is to introduce the organization of modes and the primary and secondary quantities in numerical solutions. Section 3.5 covers an important application of reflection conditions, the definition of ideal absorbing boundaries to approximate free-space conditions. Section 3.6 discusses how complex values of dielectric constant and magnetic permeability represent material energy losses and the application to absorbing boundaries in **WaveSim**. Sections 3.7 and 3.8 discuss some properties of resonant structures and methods applied in **WaveSim** to determine resonant frequencies. Section 3.9 covers procedures to determine power losses in and Q factors of resonators. Finally, Sect. 3.10 discusses scattering solutions and the distributed source method to separate small scattered signals from an incident-wave background.

3.1 Electromagnetic equations

WaveSim applies the following form of the Maxwell equations:

$$\nabla \times \mathbf{H} = \epsilon \frac{\partial \mathbf{E}}{\partial t} + \mathbf{j}_o, \tag{6}$$

$$\nabla \cdot \mu \mathbf{H} = 0, \tag{7}$$

$$\nabla \times \mathbf{E} = -\mu \frac{\partial \mathbf{H}}{\partial t}, \tag{8}$$

$$\nabla \cdot \epsilon \mathbf{E} = 0. \tag{9}$$

The magnetic field intensity is related to the magnetic flux density by $\mathbf{H} = \mathbf{B}/\mu$. We assume that the solution volume has no free space-charge. Time-dependent source current densities ($\mathbf{j}_0$) may represent drive structures like coupling loops and capacitive probes. Dielectrics and ferromagnetic materials are isotropic and linear. The magnetic permeability μ and dielectric constant ϵ may be complex quantities. You can represent losses from conductivity and non-ideal material response by assigning imaginary parts. Values of ϵ and μ are spatially uniform through a region of the solution volume.

Under the assumption that all current sources vary harmonically at frequency $f = \omega/2\pi$ and that material properties do not depend on the field amplitude, all field quantities vary as:

$$F(t) = \exp(j\omega t). \tag{10}$$

Using Eq. 10, Eqs. 6 and 8 can be expressed as:

$$\nabla \times \mathbf{H} = j\omega\epsilon\mathbf{E} + \mathbf{j}_o, \tag{11}$$

$$\nabla \times \mathbf{E} = -j\omega\mu\mathbf{H}. \tag{12}$$

We can convert Eqs. 11 and 12 to a single equation that involves only $\mathbf{H}$ by taking the curl of $1/\epsilon$ times Eq. 11 and substituting for $\nabla \times \mathbf{E}$ from Eq. 12:

$$\nabla \times \left(\frac{1}{\epsilon} \nabla \times \mathbf{H} \right) = \omega^2 \mu \mathbf{H} + j\omega \nabla \times \left(\frac{\mathbf{j}_0}{\epsilon} \right). \tag{13}$$

A similar equation holds for $\mathbf{E}$:

$$\nabla \times \left(\frac{1}{\mu} \nabla \times \mathbf{E} \right) = \omega^2 \epsilon \mathbf{E} - j\omega \mathbf{j}_0. \tag{14}$$

We can simplify the mathematics of two-dimensional solutions by dividing waves into two polarization classes:

- In planar geometry, E type waves have a single component of electric field pointing in the z direction (normal to the derivatives involved in the curl operations). The $\mathbf{H}$ field, which must be normal to $\mathbf{E}$, has components H_x and H_y. The strategy is to use Eq. 14 to find E_z, and then to find H_x and H_y from Eq. 12. For an E type solution in cylindrical coordinates, we solve Eq. 14 for the quantity rE_θ and then determine magnetic field components H_z and H_r from Eq. 12. We shall call E_z and rE_θ the *primary* field components and refer to H_x, H_y, H_z and H_r as the *secondary* field components.

- An H type wave in planar geometry has magnetic field component H_z and electric field components E_x and E_y. In this case, we use Eq. 13 to find H_z and Eq. 11 to find E_x and E_y. In a cylindrical solution, the primary field component is rH_θ and the secondary components are E_z and E_r.

A general electromagnetic disturbance may be separated into E and H components. In practical numerical solutions, we normally seek a single solution type determined by the nature of the physical system.

3.2 Properties of plane electromagnetic waves

Consider wave propagation in a uniform medium without sources and with no dissipation (*i.e.*, ϵ and μ are uniform and real). In this limit, Eq. 14 has the form:

$$\nabla^2 \mathbf{E} + \omega^2 \mu \epsilon \mathbf{E} = 0. \tag{15}$$

We seek plane wave solutions to Eq. 15 where quantities vary only in z. The condition that $\nabla \cdot \mathbf{E} = 0$ implies that there is no component E_z. We choose a coordinate system with the electric field along x, so that

$$\frac{\partial^2 E_x}{\partial z^2} + \omega^2 \mu \epsilon E_x = 0. \tag{16}$$

The function

$$E_x(z,t) = E_o \exp[j(\omega t \pm kz)], \tag{17}$$

is a general solution of Eq. 16 if

$$k = \pm\omega\sqrt{\mu\epsilon}. \tag{18}$$

Equation 17 represents a traveling wave, a harmonic function of space that moves in the positive or negative z direction. The spatial wavelength is $\lambda = 2\pi/\mathrm{k}$ and the velocity of a point of constant phase is

$$v_{phase} = \mp\frac{\omega}{k} = \mp\frac{1}{\sqrt{\mu\epsilon}}. \tag{19}$$

The phase velocity depends on the properties of the medium. In vacuum where $\epsilon = \epsilon_0$ and $\mu = \mu_0$, the phase velocity equals the speed of light, $c = 2.997925 \times 10^8$ m/s.

Equation 11 implies that there is also a magnetic intensity associated with the wave that is normal to both the electric field and the direction of wave propagation. For negative k (wave propagation in the positive z direction), the relationship is

$$H_y = \frac{j}{\omega\mu}\frac{\partial E_x}{\partial z} = \frac{k}{\omega\mu}E_x = E_x\frac{1}{\sqrt{\mu/\epsilon}}. \tag{20}$$

The cross-product $\mathbf{E}\times\mathbf{H}$ points in the direction of propagation. Noting that the dimension of $\mathrm{E_x}$ is V/m and $\mathrm{H_y}$ is A/m, the quantity in the denominator on the right hand side of Eq. 20 has units of ohms. This quantity is called the *impedance* of the medium:

$$Z = \sqrt{\frac{\mu}{\epsilon}}. \tag{21}$$

The impedance is complex if ϵ or μ have complex values (*i.e.*, material losses). In vacuum, the value is

$$Z_o = \sqrt{\frac{\mu_0}{\epsilon_0}} = 377.3\ \Omega. \tag{22}$$

Note that could have derived an equation for $\mathrm{H_y}$ similar to Eq. 15 and then calculated $\mathrm{E_x}$. In one-dimensional solutions, both choices lead to the same results.

3.3 Reflection of plane electromagnetic waves

Consider a wave traveling in the positive z direction incident on a plane boundary at $z = 0.0$ between two media. Figure 11 shows the labeling conventions. The incident wave generates a reflected wave traveling backward in Material 1 and a transmitted wave moving forward in Material 2. We denote the electric-field amplitudes in the incident, reflected and transmitting waves as E_i, E_r and E_t. The electric field is continuous across the boundary:

$$E_{xi} + E_{xr} = E_{xt}. \tag{23}$$

The common factor of $\exp[j\omega t]$ has been suppressed in Eq. 23. The magnetic field parallel to the interface, must also be continuous. Therefore the amplitudes are related by

$$H_{yi} - H_{yr} = H_{yt}. \tag{24}$$

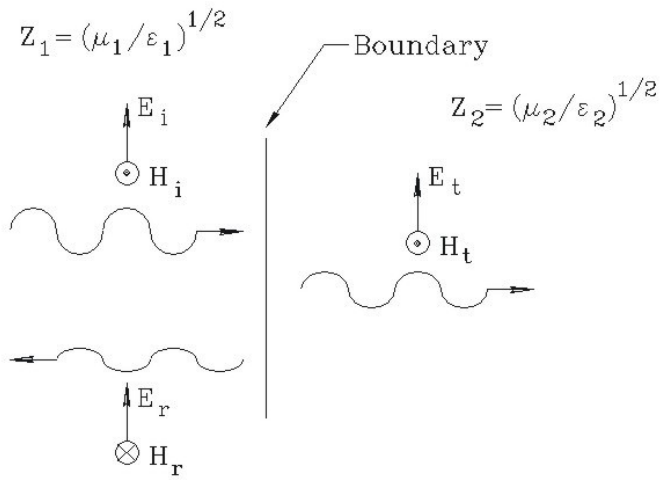


Figure 11: Traveling wave incident on a boundary between media with different impedances.

Note the minus sign in Eq. 24. A reflected wave traveling in the $-z$ direction with positive E_{xr} has negative H_{yr}. Substituting from Eq. 21, we can rewrite Eq. 24 as

$$\frac{1}{Z_1}\ (E_{xi} - E_{xr}) = \frac{1}{Z_2}\ E_{xt}. \tag{25}$$

Combining Eqs. 23 and 25, we find reflection and transmission coefficients for the electric field and magnetic intensity in terms of the impedances of the two media:

$$R_E = \frac{E_{xr}}{E_{xi}} = \frac{Z_2 - Z_1}{Z_2 + Z_1}, \tag{26}$$

$$T_E = \frac{E_{xt}}{E_{xi}} = \frac{2Z_2}{Z_2 + Z_1}. \tag{27}$$

and

$$R_H = \frac{H_{yr}}{H_{yi}} = \frac{Z_1 - Z_2}{Z_2 + Z_1}, \tag{28}$$

$$T_H = \frac{H_{yt}}{H_{yi}} = \frac{2Z_1}{Z_2 + Z_1}. \tag{29}$$

Note that conditions of Eqs. 27 and 29 are independent of frequency.

Consider a wave incident on a material with low impedance ($Z_2 \ll Z_1$) such as a dielectric with high ϵ_r. The *short circuit* boundary condition gives total reflection of the wave with inversion of the electric field. At the other extreme, the *open circuit* condition is $Z_2 \gg Z_1$. The pulse is again totally reflected but with positive electric field and inverted magnetic intensity. The cancellation of **H** near the interface is analogous to the condition of zero current at an open circuit. The special case where $Z_1 = Z_2$ is called an *impedance match*. Here the boundary has no effect and the wave is totally transmitted. Section 3.5 describes how to define absorbing boundaries in numerical solutions.

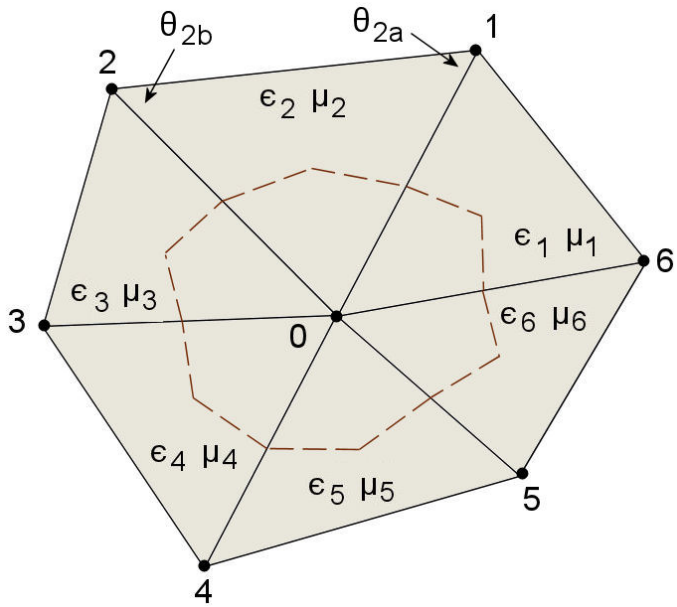


Figure 12: Relative numbering of nodes and elements in the structured, conformal triangular mesh. The dashed line passes through the midpoints of facets and the centers-of-mass of elements.

3.4 Electromagnetics on a conformal triangular mesh

To carry out a field solution on a digital computer we must convert the governing partial differential equation to a set of coupled linear equations (Sect. 1.3). This section reviews the method to convert Eqs. 13 and 14 using finite-element techniques applied on the structured conformal mesh shown in Fig.12. Each node is surrounded by six elements. The finite-element method is based on the following conditions:

- The values of ϵ and μ are uniform over the cross-section area of an element.
- The variation of the primary field quantity over an element can be represented by a linear interpolation function that depends on the field values at the three nodes.
- Instead of requiring that the governing equation holds at each point in the solution space, we apply the less-restrictive condition that the integral of the equation holds over a finite region surrounding a node.

The dashed line in Fig. 12 shows the region of the surface integral at a node. The integral extends over one-third of the area of the six surrounding triangles. Details of the calculation are described in S. Humphries, **Field Solutions on Computers** (CRC Press, Boca Raton, 1997), Chap. 14. In this section, we quote the results without proof. There is no need to study the equations – they are listed for reference in the following sections on computational techniques.

E type pulse, planar geometry

The equation for the primary field quantity E_z at a node is:

$$\sum_{i=1}^{6} E_{zi} W_i - E_{z0} \left[\sum_{i=1}^{6} W_i - A_i \right] = S_{o.} \tag{30}$$

where

$$W_i = \frac{1}{2} \left[\frac{\cot \theta_{bi+1}}{\mu_{i+1}} + \frac{\cot \theta_{bi}}{\mu_i} \right], \tag{31}$$

$$A_i = \omega^2 \sum_{i=1}^{6} \frac{\epsilon_i a_i}{3}, \tag{32}$$

and

$$S_o = j\omega \sum_{i=1}^{6} \frac{J_i a_i}{3}. \tag{33}$$

The quantity E_{z0} is the primary field at the node and the quantities E_{zi} represent field values at surrounding nodes. The other quantities refer to the six surrounding elements: θ_{bi} (angle shown in Fig. 12), ϵ_i (complex dielectric constant), μ_i (complex magnetic permeability), J_i (source current density in the z direction) and a_i (area of the triangle). Equation 30 represents a set of complex-number equations, one for each node in the solution volume. **WaveSim** solves the equations by direct sparse-matrix inversion to find E_{zi}. The secondary field quantities are determined from numerical spatial derivatives of the primary quantity:

$$H_x = \frac{j}{\mu\omega} \frac{\partial E_z}{\partial y}, \tag{34}$$

$$H_y = -\frac{j}{\mu\omega} \frac{\partial E_z}{\partial x}. \tag{35}$$

One implication of Eqs. 34 and 35 is that contour lines of fixed E_z lie along vector lines of $\mathbf{H}$ in the x-y plane.

E type pulse, cylindrical geometry

The equation for the primary field quantity (rE_θ) at a node is:

$$\sum_{i=1}^{6} (rE_{\theta i}) W_i - (rE_{\theta 0}) \left[\sum_{i=1}^{6} W_i - A_i \right] = S_{o.} \tag{36}$$

where

$$W_i = \frac{1}{2} \left[\frac{\cot \theta_{bi+1}}{\mu_{i+1} R_{i+1}} + \frac{\cot \theta_{bi}}{\mu_i R_i} \right], \tag{37}$$

$$A_i = \omega^2 \sum_{i=1}^{6} \frac{\epsilon_i a_i}{3 R_i}, \tag{38}$$

and

$$S_o = j\omega \sum_{i=1}^{6} \frac{\Gamma_i a_i}{3R_i}. \tag{39}$$

In the equations, R_i is the average radial position of the element and $\Gamma_i = J_i r$. In **WaveSim** the quantity Γ_i is assumed to have a fixed value over a region. The secondary field quantities are given by:

$$H_z = \frac{j}{\mu\omega r}\frac{\partial(rE_\theta)}{\partial r}, \tag{40}$$

$$H_r = -\frac{j}{\mu\omega r}\frac{\partial(eE_\theta)}{\partial z}. \tag{41}$$

Equations 40 and 41 imply that contour lines of rE_θ lie along vector lines of $\mathbf{H}$ in the z-r plane.

H type pulse, planar geometry

The primary field quantity H_z at a node is given by

$$\sum_{i=1}^{6} H_{zi}W_i - H_{zo}\left[\sum_{i=1}^{6} W_i - A_i\right] = -S_{o.} \tag{42}$$

where

$$W_i = \frac{1}{2}\left[\frac{\cot\theta_{b,i+1}}{\epsilon_{i+1}} + \frac{\cot\theta_{a,i}}{\epsilon_i}\right], \tag{43}$$

$$A_i = \omega^2 \sum_{i=1}^{6} \frac{\mu_i a_i}{3}, \tag{44}$$

and

$$S_o = \sum_{i=1}^{6} \frac{J_{0i}}{2\epsilon_i}\left[\cos\beta_i(x_i - x_{i-1}) + \sin\beta_i(y_i - y_{i-1})\right]. \tag{45}$$

In Eq. 45, the quantity β_i gives the direction of source current in the x-y plane relative to the x axis, and the quantities (x_i, y_i) are the coordinates of surrounding nodes. The secondary field quantities are given by

$$E_x = \frac{j}{\omega\epsilon}\left[-\frac{\partial H_z}{\partial y} + j_{ox}\right],$$

$$E_y = \frac{j}{\omega\epsilon}\left[\frac{\partial H_z}{\partial x} + j_{oy}\right]. \tag{46}$$

H type pulse, cylindrical geometry

The primary field quantity rH_θ at a node is given by

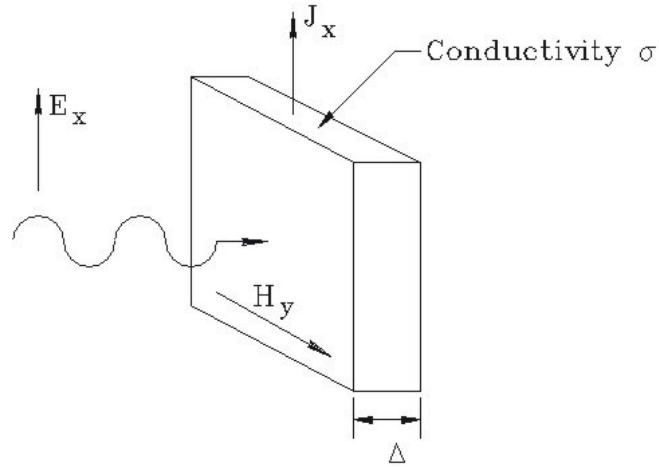


Figure 13: Traveling wave incident on a matched resistive layer.

$$\sum_{i=1}^{6}(rH_{\theta i})W_i - (H_{\theta 0})\left[\sum_{i=1}^{6}W_i - A_i\right] = -S_{o.} \tag{47}$$

where

$$W_i = \frac{1}{2}\left[\frac{\cot\theta_{b,i+1}}{\epsilon_{i+1}R_{i+1}} + \frac{\cot\theta_{a,i}}{\epsilon_i R_i}\right], \tag{48}$$

$$A_i = \omega^2 \sum_{i=1}^{6}\frac{\mu_i a_i}{3R_i}, \tag{49}$$

and

$$S_o = \sum_{i=1}^{6}\frac{\Gamma_{0i}}{2\epsilon_i R_i}\left[\cos\beta_i(z_i - z_{i-1}) + \sin\beta_i(r_i - r_{i-1})\right]. \tag{50}$$

In Eq. 45, $\Gamma_{0z} = rj_{0z}$ and $\Gamma_{0r} = rj_{0r}$. The convention of specifying Γ in **WaveSim** ensures conservation of current for radial flow. The quantity β_i gives the direction of source current in the z-r plane relative to the z axis and the quantities (z_i, r_i) are the coordinates of surrounding nodes. The secondary field components are given by

$$E_z = \frac{j}{\omega\epsilon r}\left[-\frac{\partial(rH_\theta)}{\partial r} + \Gamma_{0z}\right], \tag{51}$$

$$E_y = \frac{j}{\omega\epsilon r}\left[\frac{\partial(rH_\theta)}{\partial z} + \Gamma_{0r}\right]. \tag{52}$$

3.5 Absorbing boundary

In the discussion of wave reflection in Sect. 3.3, the special case where $Z_1 = Z_2$ is an *impedance match*. Here the boundary has no effect and the wave is totally transmitted. A related solution

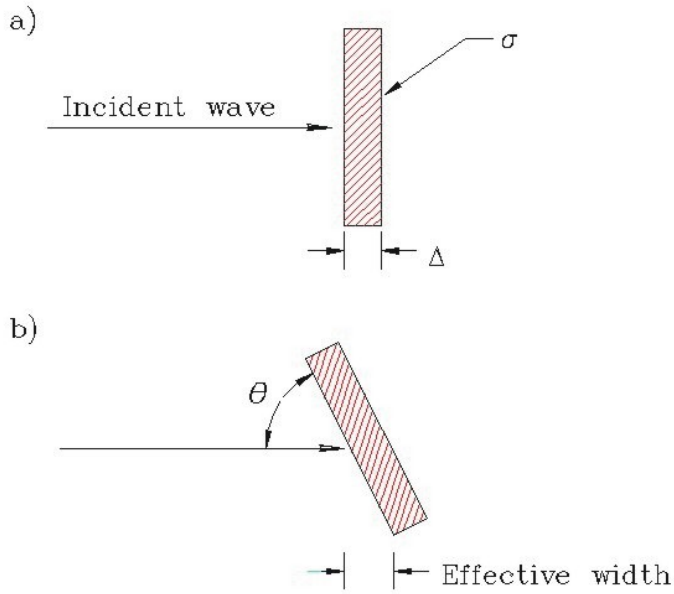


Figure 14: Absorbing boundary for two-dimensional solutions. *a*) Normal incidence, $\theta = 90^o$. *b*) Increase in effective layer thickness for $\theta < 90^o$.

that is important for numerical calculations is the termination of a wave by a lumped element resistor. Figure shows the geometry. We introduce a resistive layer with conductivity σ and thickness Δ adjacent to an open circuit boundary of a material with ϵ and μ. Assume that the layer has the same values of dielectric constant and magnetic permeability as the adjacent material. In this case, the layer has complex dielectric constant $[\epsilon - \sigma/\omega]$, where ω is the angular frequency of the wave. In the limit that $\Delta \ll \lambda$, the electric field is approximately uniform over the layer depth. There is no reflected wave if the resistor maintains the same conditions on $\mathrm{E_x}$ and $\mathrm{H_y}$ as an infinite extension of the medium. We can derive the correct value of σ by noting that the field $\mathrm{E_x}$ creates a linear current density of

$$J_x = (\sigma\Delta)\ E_x. \tag{53}$$

The quantity $1/\sigma\Delta$ (with units of ohms) is called the *surface resistance* of the termination layer. Assuming zero magnetic field on the right-hand side of the resistor, the value of magnetic intensity on the left-hand side is $H_y = J_r$. Substituting in Eq. 20 gives the condition for a matched termination,

$$Z_0 = \sqrt{\frac{\mu}{\epsilon}} = \frac{1}{\sigma\Delta}. \tag{54}$$

The above derivation is valid for a wave normally incident on a thin layer (Fig. 14*a*). Here, the propagation vector of the wave makes an angle $\theta = 90^o$ with respect to the layer surface. For optimum performance, we must modify the conductivity if the wave arrives at an angle $\theta < 90^o$. In this case, the effective layer thickness is $\Delta/\sin\theta$, reducing the surface resistance by a factor of $\sin\theta$. In this case the termination is under-matched. Applying Eq. 27, the reflection coefficient for the electric field is

$$\frac{E_r}{E_0} = -\frac{1-\sin\theta}{1+\sin\theta}. \tag{55}$$

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

The best way to ensure good absorption is to construct the solution volume so that waves arrive close to normal incidence. For example, in a free-space planar scattering problem, the termination layer should be a cylindrical shell with center at the scattering object. If oblique waves are unavoidable, the performance of the absorption layer can be optimized by adjusting the conductivity. Figure 14*b* shows that if a wave is incident at angle θ, then the effective thickness of the layer increases. We can compensate the effect by setting the layer conductivity equal to

$$\sigma = \frac{\sin\theta}{\Delta\sqrt{\mu/\epsilon}}. \tag{56}$$

3.6 Energy loss in materials

Material responses at high frequency may lag behind the driving fields. In this case, the fields created by shifts of dielectric charge or reorientation of magnetic domains may not be in phase with applied fields. This process leads to energy losses in the material. We can represent phase differences with complex values of dielectric constant and magnetic permeability. The standard notation is

$$\epsilon = \epsilon' + j\epsilon'', \tag{57}$$

$$\mu = \mu' + j\mu''. \tag{58}$$

As an illustration, Fig. 15 shows the variation of ϵ'/ϵ_0 and ϵ''/ϵ_0 in purified water. At low frequency the medium is an ideal dielectric with $\epsilon'\epsilon_0 = 81$. At high frequency inertial effects in the reorientation of polar molecules causes a drop in the real part of the dielectric constant and increasing losses.

Poynting's theorem describes conservation of electromagnetic energy flow,

$$-\nabla\cdot(\mathbf{E}^*\times\mathbf{H}) = \mathbf{E}^*\cdot\epsilon\frac{\partial\mathbf{E}}{\partial t} + \mathbf{H}^*\cdot\mu\frac{\partial\mathbf{H}}{\partial t} + \mathbf{E}^*\cdot\mathbf{J}_{c.} \tag{59}$$

Equation 59 is derived in most introductory texts, such D.K. Cheng, **Field and Wave Electromagnetics, Second Edition** (Addison-Wesley, Reading, 1992), 38. The quantity in parenthesis on the left-hand side is the *Poynting vector*, equal to the flux of electromagnetic power. Quantities on the right-hand side are volumetric power losses. The quantity $\mathbf{J}_c$ in the last term is the conductive current driven in resistive materials. Asterisks denote complex conjugates. Assuming a harmonic field variation and substituting $\mathbf{E} = \mathbf{E}_{\mathrm{r}} + \mathrm{j}\mathbf{E}_{\mathrm{i}}$, $\mathbf{H} = \mathbf{H}_{\mathrm{r}} + \mathrm{j}\mathbf{H}_{\mathrm{i}}$, the right hand side of Eq. 59 becomes

$$2j\omega\frac{[\epsilon'(\mathbf{E}_r^2+\mathbf{E}_i^2)+\mu'(\mathbf{H}_r^2+\mathbf{H}_i^2)]}{2}$$
$$-\omega\epsilon"(\mathbf{E}_r^2+\mathbf{E}_i^2) - \omega\mu"(\mathbf{H}_r^2+\mathbf{H}_i^2) + \sigma(\mathbf{E}_r^2+\mathbf{E}_i^2). \tag{60}$$

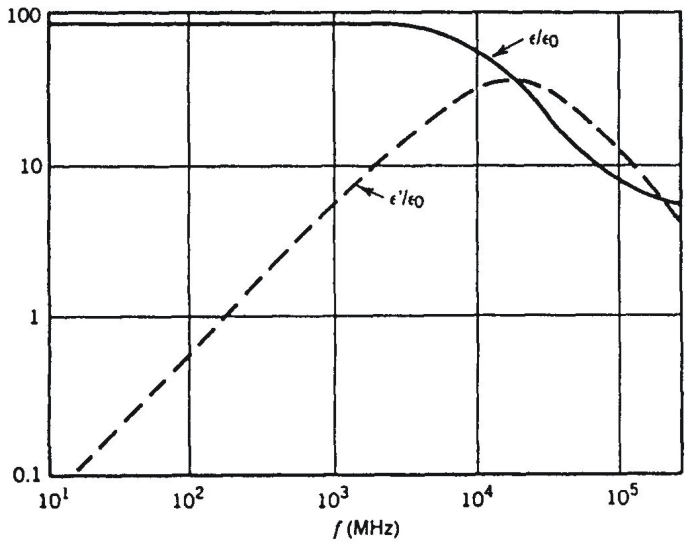


Figure 15: Real and imaginary parts of the relative dielectric constant of purified water as a function of frequency. (Adapted from J.B. Halstead, *Liquid Water – Dielectric Properties* **in Water – a Comprehensive Treatise**, F. Franks, ed., Plenum, New York, 1972).

The field quantities in Eq. 60 represent time averages. The first term is the time derivative of the total field energy. The remaining three terms represent power loss in materials. In the electric field terms we can combine contributions of material response and resistivity into a single expression for the complex part of dielectric constant,

$$\epsilon" \rightarrow \epsilon" - \frac{\sigma}{\omega}. \tag{61}$$

We can apply Eq. 60 to determine the properties on an ideal absorbing layer for a two-dimensional **WaveSim** solution. Consider first an E type solution where a plane wave moves from a non-absorbing medium (Material 1) into an absorbing layer material of thickness Δz (Material 2). Using Eq. 20 and to evaluating the Poynting vector, the power per unit area entering the later is $Z_1(E_r^2 + E_i^2)$. The characteristic impedance of Medium 1 is given by $Z_1 = \sqrt{\mu_1'/\epsilon_1'}$. If we express dissipation in the layer with a complex dielectric constant, the power absorbed per unit area is given by -$\omega\epsilon_2''(E_r^2 + E_i^2)\Delta z$. The condition for total absorption of E waves is

$$\epsilon_2" = -\frac{1}{\omega Z_1 \Delta z}. \tag{62}$$

The real parts of the material properties for Medium 2 are matched to those of Medium 1, $\epsilon_2' = \epsilon_1'$ and $\mu_2' = \mu_1'$. The procedure for H type waves is to use a complex magnetic permeability. The condition for perfect absorption is

$$\mu_2" = -\frac{Z_1}{\omega \Delta z}. \tag{63}$$

To summarize, use the following procedure to include an absorbing boundary in a **WaveSim** solution. During mesh generation, be sure to include an extra material region of uniform thickness Δ along an absorbing boundary. To ensure that the condition $\Delta \ll \lambda$ is satisfied, the

Table 6: Termination layer material properties

Type	ϵ_2'	ϵ_2"	μ_2'	μ_2"
E	ϵ_1'	-1/$\omega\Delta Z_1\epsilon_0$	μ_1'	0
H	ϵ_1'	0	μ_1'	$-Z_1/\omega\Delta\mu_0$

layer should be only a single-element thick. The outer surface of the termination layer should have an unspecified (open-circuit) boundary. Include a script command of the form:

```
ABSLAYER(RegNo) Delta [EpsiR, MuR, Theta]
```

WaveSim automatically sets ϵ and μ in the region, depending on the current frequency. Enter the real-number parameters *EpsiR* and *MuR* if there is a non-vacuum medium adjacent to the layer. The quantities μ_r and ϵ_r are the real parts of the relative magnetic permeability and electrical permittivity of the medium. The final optional parameter introduces a tuning factor for non-normal incidence. The quantity *Theta* is the angle between the average wave propagation vector and the layer surface. The default is $\theta = 90^o$.

3.7 Basic properties of resonators

Resonators are closed systems with trapped traveling waves. Certain values of frequency give constructive interference, resulting in high field values for relatively weak excitation. One strategy to find resonant modes is to set up frequency-domain solutions at several frequencies, looking for characteristic signs of resonance. This section treats simple one-dimensional resonators. Application of the techniques to two and three-dimensional systems are discussed in the following section.

Figure 16 shows the simplest one-dimensional resonator, a vacuum region between two metal boundaries separated by a distance L. The boundary condition for E type waves is $E_x(0) = E_x(L) = 0$. The condition for constructive interference of traveling waves with wavelength λ is

$$\lambda = \frac{2L}{n}, \quad n = 1, 2, 3, ... \tag{64}$$

The wavelength is related to frequency by $\lambda = c/f$. For $L = 1$ m the frequencies of the first three modes are $f = 149.8962$ MHz, 299.7925 MHz and 449.6887 MHz. Figure 16 shows the corresponding variations of E_x. The figure also illustrates a common method to excite an E type mode, a drive current near the expected location of maximum magnetic field. To check for a resonance, we sweep the drive through a range of frequencies and monitor the signal on a probe near the point of maximum electric field. The probe response rises sharply near the resonant frequency.

We can understand some features of resonance response by studying the behavior of the circuit shown in Fig. 17. The circuit is a lumped-element model for the lowest frequency E wave mode. Here, the capacitive energy (electric field) is highest near the center of the cavity and the inductive energy (magnetic field) at the edges. The drive current couples into a portion of the cavity inductance denoted L_1. The total system inductance is $L = L_1 + L_2$. If a harmonic

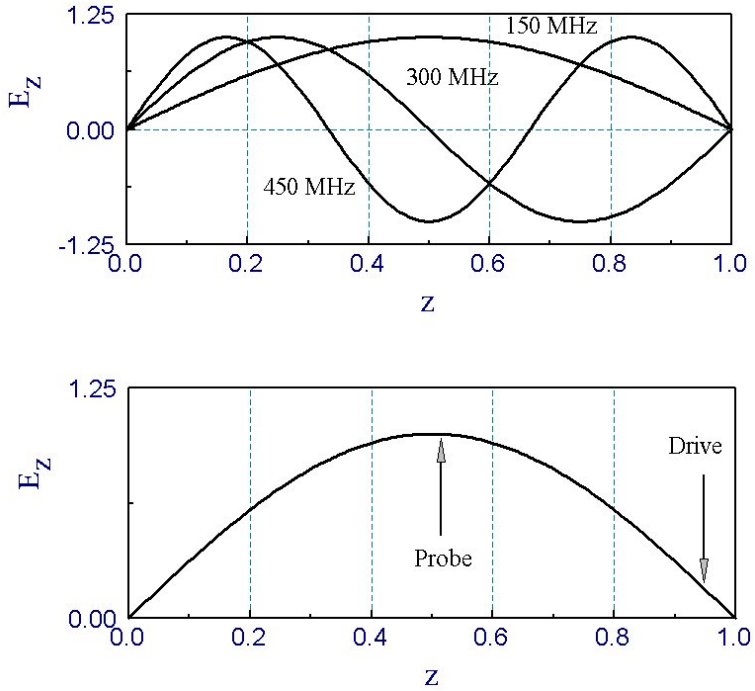


Figure 16: Lower order E wave modes of a one-dimensional vacuum resonator. For $L = 1$ m, the predicted resonant frequencies are $f_1 = 149.8962$ MHz, $f_2 = 299.7925$ MHz and $f_3 = 449.6887$ MHz.

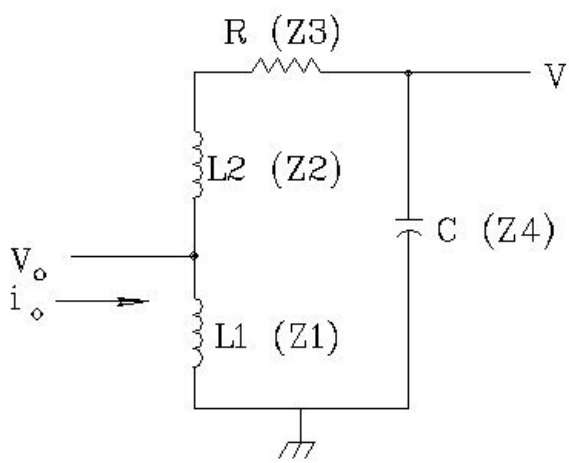


Figure 17: Driven LCR resonant circuit.

current I_0 is applied at the drive point (with the sign convention shown) we can find the drive voltage by calculating the complex circuit impedance. The impedances of the individual circuit components are $Z_1 = j\omega L_1$, $Z_2 = j\omega L_2$, $Z_3 = R$ and $Z_4 = 1/j\omega C$. The total impedance at the drive is Z_1 in parallel with $(Z_2 + Z_3 + Z_4)$, so that

$$V_0 = -\frac{Z_1(Z_2 + Z_3 + Z_4)}{Z_1 + Z_2 + Z_3 + Z_4}\ i_0. \tag{65}$$

We measure the voltage across the capacitive region of the cavity, as shown. The probe voltage V is given in terms of V_0 by the law of voltage division, $V = V_0 Z_4/(Z_2+Z_3+Z_4))$. Alternatively, we can write

$$V = -\frac{Z_1 Z_4}{Z_1 + Z_2 + Z_3 + Z_4}\ i_0. \tag{66}$$

Inserting expressions for the component impedances in Eq. 66 gives

$$V = i_0\ \frac{-j\omega L_1}{1 - \omega^2 LC + j\omega RC}. \tag{67}$$

We can generalize Eq. 67 by expressing it in terms of the circuit resonant frequency ω_0, the resonator characteristic impedance Z_0, the quality factor Q and the dimensionless frequency $\Omega = \omega/\omega_0$. The quantities are given by:

$$\omega_0 = \frac{1}{\sqrt{LC}}, \quad Z_0 = \sqrt{L/C}, \quad Q = Z_0/R. \tag{68}$$

The parameter Q equals the ratio of stored electromagnetic energy U in the resonator multiplied by ω and divided by the average resistive power dissipation P:

$$Q = \frac{\omega U}{P}. \tag{69}$$

The quantity $2\pi/Q$ is the approximate fraction of stored energy lost per cycle. A practical resonator has $Q \gg 1$.

Equation 67 takes the form

$$V = i_0\ \left[Z_0 \frac{L_1}{L}\right]\ \frac{[-\Omega^2/Q + j\Omega(\Omega^2 - 1)]}{[(1 - \Omega^2) + \Omega^2/Q^2]}. \tag{70}$$

The resonance condition is that $\Omega = 1$. Figure 18 plots the variation of the real and imaginary parts of V as a function of Ω near resonance. In the plot the drive current has phase 0° ($I_0 = [1, 0]$) and the circuit has low damping ($Q = 250$). The imaginary part of the probe voltage has a sign change at resonance. At this point, the real part of the voltage has maximum amplitude $V_m = -QZ_0(L_1/L)I_0$. Substituting $\Omega = 1 + \Delta\Omega$ and making binomial expansions of terms in Eq. 70, we find that the amplitude of the real part of V drops to half its maximum value at $\pm\Delta\Omega = 1/2Q$. This implies that the Q factor equals the reciprocal of the frequency difference between the half amplitude points.

The above discussion suggests a numerical technique to find the properties of damped and undamped resonators. As an example, consider E wave solutions in the system of Fig. 16. We set up a frequency-domain solution with a current source [1,0] in an element near the wall and monitor the real and imaginary parts of E_x near the expected field maximum. For the

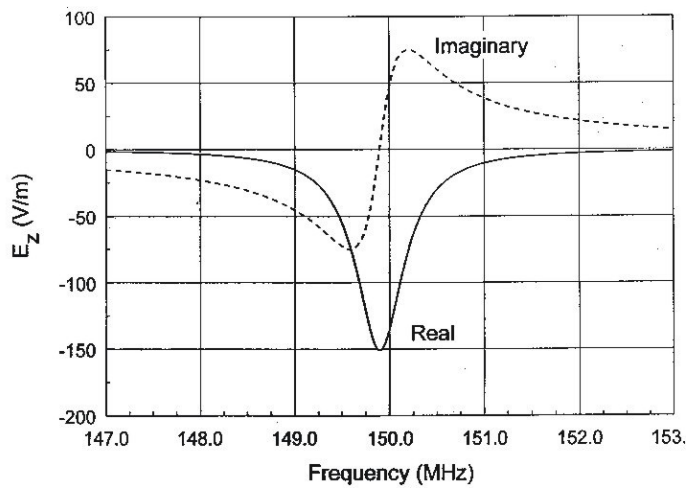


Figure 18: Resonant circuit – variation of V as a function of Ω near resonance. Drive current $I_0 = [1, 0]$, $Q = 250$.

fundamental mode ($n = 1$), the best position is $z = L/2$. The next step is to pick upper and lower frequencies that bracket the anticipated value of resonant frequency, f_u and f_d. Initial solutions are performed to check that the imaginary part of the probe response is positive at f_u and negative at f_d. This condition guarantees that there is a resonance in the range. Subsequently, a frequency search for the sign change of the imaginary part gives the resonant frequency $f_0 = 2\pi\omega_0$. Damping of the fields may result from volume resistivity in the medium, wall resistivity or imperfect materials. To find the effect of dissipation, we can make several solutions near f_0 to determine the frequency spread between the half amplitude points of the real part of the probe signal. The total quality factor is given by $Q = f_0/(2|f - f_0|)$.

The plot of Fig. 18 was derived by a numerical solution of the system of Fig. 16 with 250 elements. The predicted frequency of the $n = 1$ mode is $f_0 = 149.89623$ MHz. A uniform conductivity of σ_0 was assigned to give damping. The stored energy density in an element is $u = \epsilon_0 E_x^2/2$ and the time-averaged power loss density is $\mathrm{P} = \sigma_0 E_x^2$. Therefore, the predicted quality factor is $Q = 2\pi f_0 \epsilon_0/\sigma_0$. A value $\sigma_0 = 3.33 \times 10^{-5}$ mhos/m gives $Q = 250$. Note that the imaginary part of the probe voltage reverses sign at f_0 in Fig. 18 and that the full-width at half-maximum of the real part is 0.6 MHz as expected. Far from the resonance, the probe electric field is purely imaginary, 90° out of phase from the drive current. The sign of the imaginary part is negative at low frequency implying that system acts like a capacitor. The impedance is imaginary and positive at high frequency, implying inductive behavior.

For most applications we want to determine the resonant frequency to high accuracy. Each point in a search involves a solution of finite element equations. Solutions can be time-consuming in two and three-dimensional problems. The implication is that we should seek a method to find the zero-crossing that minimizes the number of steps. Figure 19*a* illustrates the *bisection method*. The plot shows the imaginary part of the probe signal as a function of frequency, $P_i(f)$. Initially, the frequency values f_d and f_u define a search range, $[f_d, f_u]$. Bisection of the range gives $f_1 = (f_u + f_d)/2$. If $P_i(f_1) < 0$, we use $[f_1, f_u]$ as the search range; otherwise, the range is $[f_d, f_1]$. The bisection continues either for a maximum number of steps or until the frequency width of the bisection region drops below a target frequency error. If Δf is the target error and Δf_0 is the initial range, the maximum number of steps is

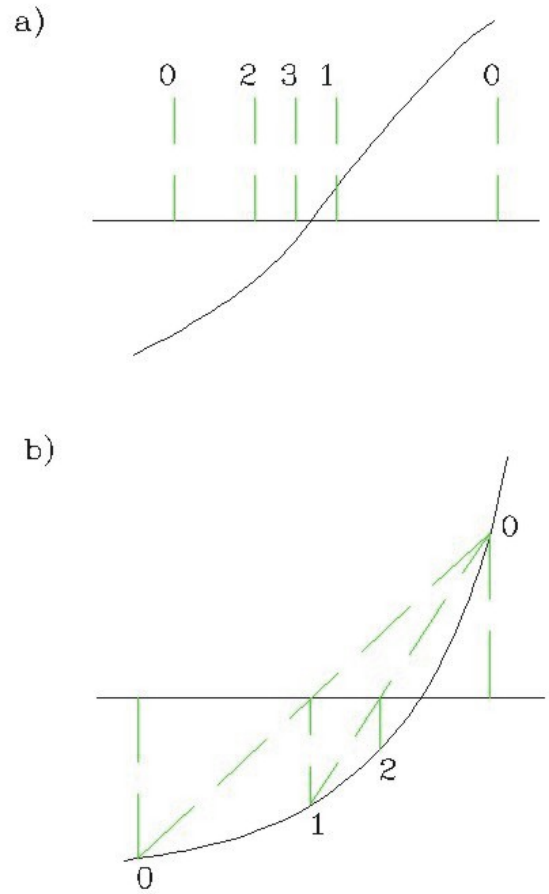


Figure 19: Numerical determination of zero crossing point of a function. *a*) Bisection method. *b*) False position method.

Table 7: Resonant frequency calculation - 250 elements

Mode	f (theory) (MHz)	f (calc) (MHz)	Error
1	149.89525	149.89623	0.0007%
2	299.78457	299.79246	0.0026%
3	449.66204	449.68869	0.0059%
4	599.52185	599.58492	0.0105%

$$n = \log_2(\Delta f_0/\Delta f). \tag{71}$$

The advantage of the bisection method is that it never fails to converge - the decreasing intervals always bracket the root. For well-behaved functions, alternative methods can achieve a target accuracy in fewer steps. Figure 19*b* shows the *false position method* for root finding. Starting again from points f_d and f_u that bracket the zero crossing, we interpolate the frequency as shown and calculate the value f_1 and the corresponding probe response $P_i(f+1)$. We choose the range $[f_d, f_1]$ or $[f_1, f_u]$ that encloses the root and repeat the interpolation.

The bisection method was used for the baseline calculation of Fig. 18 with an initially broad range of $f_d = 120$ MHz to $f_u = 180$ MHz. The search converged in 16 steps to the value 149.89496 MHz. The accuracy of 1 part in 1.2×10^5 was limited by the mesh size. Table 13.1 shows predicted resonant frequencies and numerical results for the first five modes of the one-dimensional resonator. The error is approximately proportional to the ratio $\Delta z/\lambda_n$ where Δz is the element size and λ_n is the mode wavelength.

3.8 Waveguides and resonant cavities

This section reviews resonance calculations in two-dimensional structures. The theory applies to the cutoff modes of uniform waveguides or axisymmetric modes of cylindrical resonators. The procedures follow from the discussion of the previous section. We excite a cavity at a set of frequencies with a drive current and sense the response with a probe at the expected position of maximum primary field. Depending on the mode and the phase of the drive, we search for a zero crossing of the real or imaginary part of the probe signal with the correct polarity change.

To test the method we shall solve for the pillbox cavity of Fig. 20, the simplest cylindrical resonator. The resonant modes are field solutions consistent with the wall conditions that $\mathbf{E}_\parallel = 0$ and $\mathbf{H}_\perp = 0$. We divide the modes into two classes, depending on the disposition of field components relative to the z axis. E type solutions for rE_θ with subsidiary field components B_r and B_z are called *TE modes*. H type solutions for rH_θ yielding E_r and E_z are called *TM modes*. The modes have names of the form TM_{mnp}, where the indices denote the field characteristics. The index m designates an azimuthal variation of the form $\exp(jm\theta)$. We assume azimuthal symmetry so that $m = 0$. The indices n and p indicate the complexity of field variations in the r and z directions respectively. Low numbers correspond to simple variations and generally have low frequencies. The predicted primary field variations for TM_{0np} modes in a pillbox cavity of radius R and length D are given by

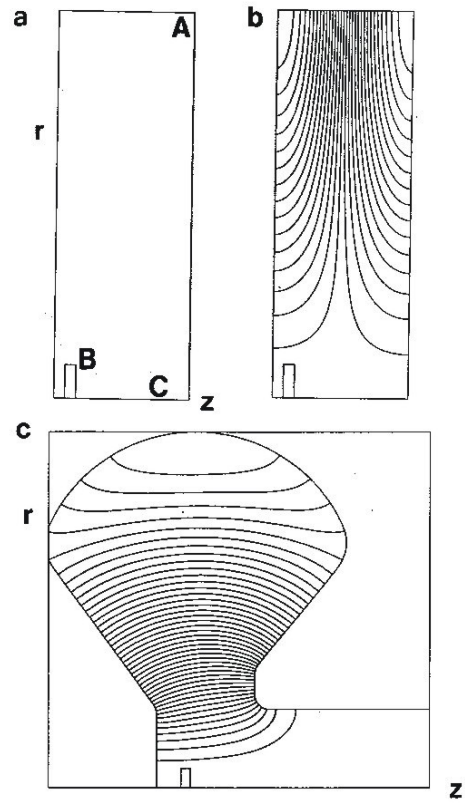


Figure 20: Resonant modes in cylindrical cavities. *a*) Pillbox geometry. For H type solutions metal walls are open circuit boundaries and $rH_\theta = 0$ on axis. Location of drive current and sensing probe shown. Cavity radius: 0.70 m. Cavity length: 0.25 m. *b*) Contours of rH_θ (electric field lines) for the TM_{011} mode at 621.329 MHz. *c*) Contours of rH_θ for the TM_{010} mode in a klystron cavity.

$$H_\theta = H_0 \ \mathrm{J}_1(\chi_n r/R) \ \cos(p\pi z/D), \tag{72}$$

with frequencies

$$f_{mp} = \frac{1}{2\pi\sqrt{\mu\epsilon}} \left[\frac{\chi_n^2}{R^2} + \frac{p^2\pi^2}{D^2}\right]. \tag{73}$$

In Eq. 72, the quantity J_1 is a Bessel function and χ_i represents zeros of the J_0 Bessel function: $\chi_1 = 2.4048$, $\chi_2 = 5.5201$, $\chi_3 = 8.6537$, ...

The benchmark geometry of Fig. 20 has $R = 0.70$ m and $D = 0.25$ m with an element size of about 0.01 m. As in the one-dimensional solutions, the accuracy increases with smaller elements and decreases for higher mode numbers. For H wave solutions the metal walls are represented by natural boundary conditions on the left, right and top. The Dirichlet condition $rH_\theta = 0$ holds on the axis (bottom boundary). For modes of type TM_{0n0}, we expect an electric field E_z concentrated near the axis. These modes are excited by a small drive region with an axial current ($\beta = 0.0$). The probe is located near the outer radius where we expect the maximum value of rH_θ according to Eq. 72. The numerical result for the TM_{010} mode is 163.919 MHz (accurate to 0.002 per cent). The values are 589.366 MHz for the TM_{030} mode (0.08 per cent

accuracy) and 621.329 MHz for the TM_{011} mode (0.04 per cent accuracy). Figure 20*b* shows contours of rH_θ for the TM_{011} mode which lie along electric field lines.

Figure 21 illustrates a practical calculation for a superconducting proton accelerator. The accelerating structure consists of five coupled cavities – the simulation of Fig. 21*a* represents half of the structure with a symmetry boundary on the right-hand side. The cavities are excited by an axial current at 0.0° phase in the small disk-shaped region at the bottom-right. In the lossless structure, the drive creates purely real values of the quantity rH_θ. A probe at the position marked with a circle senses the reciprocal of the real part of the primary field, $V_p = 1/\mathrm{Re}(rH_\theta)$. The resonance search involves location of the zero crossings of V_p from positive to negative polarity. The theory of coupled cavity arrays (see, for instance, S. Humphries, **Principles of Charged Particle Acceleration** (Wiley, New York, 1986), Sect. 14.3) shows that a five-cavity system has five resonant modes with TM_{010} type fields at different frequencies. The one with the highest frequency is called the π-mode because the phase of the axial electrical field reverses by 180° between adjacent cavities. The other modes have different values of phase shift: 0, $\pi/4$, $\pi/2$ and $3\pi/4$. In the simulation of Fig. 21 we expect to detect only three resonances because the $\pi/4$ and $3\pi/4$ modes are excluded by the symmetry boundary. Figure 21*b* shows a scan of probe output as a function of frequency. Arrows show the locations of the 0, $\pi/2$ and π modes. The calculation gives values of 682.60 MHz for the 0 mode, 694.09 MHz for the $\pi/2$ mode and 701.62 MHz for the π mode.

3.9 Power loss in resonant cavities

In this section we shall discuss how to apply Eq. 60 to calculate RF power deposition and the Q factors of resonant structures. The procedures to calculate stored electromagnetic energy and volume power dissipation on a triangular mesh involves scans through elements of the solution volume. As an illustration, consider an E wave solution in a planar geometry. For each element, the first step is to find the field components. The amplitude of the primary field component is the average of values at the corners of the triangle. If i_1, i_2 and i_3 are the indices of the element vertices, the average values of the real and imaginary parts of the primary field are

$$E_{zr} = \frac{E_{zr}(i_1) + E_{zr}(i_2) + E_{zr}(i_3)}{3},$$

$$E_{zi} = \frac{E_{zi}(i_1) + E_{zi}(i_2) + E_{zi}(i_3)}{3}. \tag{74}$$

The subsidiary field components are given by derivatives of the primary components. For example, the following approximation gives the real part of H_x:

$$H_{xr} = \frac{[E_{zr}(i_2) - E_{zr}(i_1)](x_3 - x_1) - [E_{zr}(i_3) - E_{zr}(i_1)](x_2 - x_1)}{(y_2 - y_1)(x_3 - x_1) - (y_3 - y_1)(x_2 - x_1)}. \tag{75}$$

Following Eq. 60, the total field energy in the element per unit length in z is

$$dU = \left[\frac{\epsilon'(E_{zr}^2 + E_{zi}^2) + \mu'(H_{xr}^2 + H_{xi}^2 + H_{yr}^2 + H_{yi}^2)}{4}\right] A. \tag{76}$$

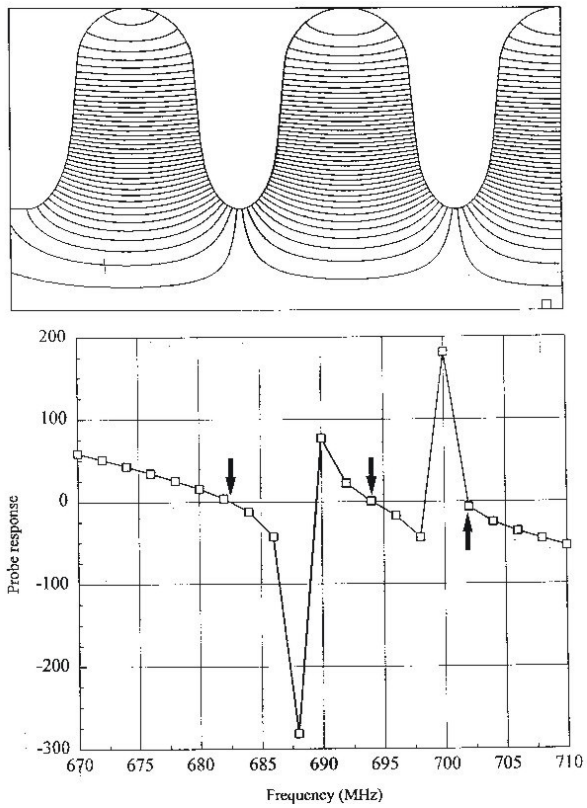


Figure 21: Resonant modes in a proton accelerator structure, coupled superconducting cavities. *a*) Simulation geometry for half of a 5-cavity assembly. Symmetry boundary on the right-hand side, drive current region at the bottom-right. Electric field lines for the π-mode. *b*) Scan of probe output as a function of frequency – arrows show the locations of the 0, $\pi/2$and π modes.

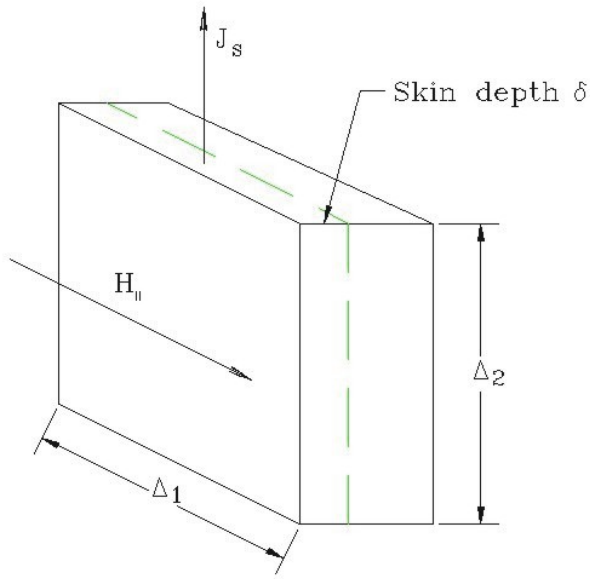


Figure 22: RF power loss by currents driven in a metal wall.

In Eq. 76, the quantity A is the element area in the x-y plane and ϵ' and μ' are the real parts of the dielectric constant and magnetic permeability. The quantity in brackets is the field energy density. The additional factor of 2 in the denominator follows from time averaging of the harmonic functions. Similarly, the power dissipated in the element per unit length in z is

$$dP = \omega \left[\frac{\epsilon''(E_{zr}^2 + E_{zi}^2) + \mu''(H_{xr}^2 + H_{xi}^2 + H_{yr}^2 + H_{yi}^2)}{2} \right] A. \tag{77}$$

The quantities ϵ'' and μ'' are the imaginary parts of the element material properties and ω is the angular frequency of the electromagnetic fields. The quantities in Eqs. 76 and 77 can be summed over elements to give total energy deposition in regions or used to generate plots of energy and power distributions.

In vacuum cavities for charged particle acceleration power losses result mainly from resistive dissipation from currents driven in the metal walls. At the field levels required in accelerators power losses may be high even for good conductors. Currents driven by high-frequency electromagnetic fields are confined to a layer on the surface of metals with thickness equal to the *skin depth*. For copper lined cavities in the range 250 to 1000 MHz, the skin depth is only a few μm. Therefore it is impractical to apply the volume method of Eq. 77. Surface integrals give more accurate results.

Consider a small segment of a metal surface shown in Fig. 22. The boundary condition on the surface of a good conductor is that the magnetic intensity is parallel to the surface. We denote the field amplitude as $H_\parallel$. Exclusion of the field from the volume of the material implies that the surface carries a linear current density of amplitude $J_s = H_\parallel$. The surface segment has dimensions Δ_1 along the direction of magnetic intensity and Δ_2 along the direction of current. If the metal has volume resistivity ρ, the total resistance of the segment is $R = \rho\Delta_2/\Delta_1\delta$. The time-averaged power deposited in the segment is $R(J_s\Delta_1)^2/2$. Dividing by $\Delta_1\Delta_2$ gives the time-average power per unit area of the surface,

$$p = \frac{\rho H_\parallel^2}{2\delta} = \frac{R_s H_\parallel^2}{2}. \tag{78}$$

The quantity R_s in Eq. 78 with units of Ω is called the *surface resistance*. It is given by the expression

$$R_s \;=\; \sqrt{\pi f \mu_o \rho}\;, \tag{79}$$

where f is the RF frequency.

The following procedure is used in **WaveSim** to find wall losses. Consider first H type solutions where metal walls are open boundaries. During the mesh generation process we define one or more line regions with unspecified boundary conditions where we want to evaluate power deposition. After completing the solution, we identify all line segments of the mesh where both nodes have the target region number. In a planar geometry, suppose the node coordinates of one such segment are (x_1, y_1) and (x_2, y_2). The power loss on the segment (per unit length in z) is

$$dp = \frac{1}{2}[(H_1 + H_2)/2]^2 \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2}. \tag{80}$$

The expression for a segment with coordinates (z_1, r_1) and (z_1, r_1) in a cylindrical system is

$$dp = \frac{1}{2}[(H_1 + H_2)/2]^2 \frac{2\pi(r_1 + r_2)}{2} \sqrt{(z_2 - z_1)^2 + (r_2 - r_1)^2}. \tag{81}$$

The procedure is more involved for E wave solutions. After finding vectors that lie on lines of $E_z = 0$ or $rE_\theta = 0$, we identify the two elements adjacent to each vector. Because the vector lies on a metal boundary, the routine to calculate magnetic intensity returns valid values only for the material element that lies inside the solution volume. It is sufficient to take the magnitude of **H** in this element because we know that the magnetic intensity must lie parallel to the conducting surface. This value is used in place of the averages in Eqs. 80 or 81.

To illustrate the method, consider calculating Q factors for TM_{010} modes in accelerator cavities. Figure 23 shows two 303 MHz cavities. The first is a pillbox of length $d = 0.2$ m and radius $R = 0.3789$ m. The second has the same axial length and a rounded outer boundary. We shall find that this cavity has a higher Q factor; therefore, it consumes less power to achieve the same accelerating gradient. In the solution, both cavities are driven by a small current source near the axis. Resonant calculations in lossless structures give relative field levels – we can adjust the solution to represent any excitation by scaling all values of the primary field. For the pillbox cavity the value of electric field on axis is $E_z(0, z) = E_0 = 3.6103 \times 10^4$ V/m. The predicted stored energy [see, for instance, J.D. Jackson, **Classical Electrodynamics** (Wiley, New York, 1975), Sect. 8.8] is

$$U = \pi R^2 d \left[\frac{\epsilon E_o^2}{2}\right] J_1^2(2.405). \tag{82}$$

Equation 82 predicts a value $U = 1.410 \times 10^{-4}$ J, close to the numerical value of $U = 1.369 \times 10^{-4}$ J from Eq. 76. The integral of $H_\parallel^2/2$ over the metal surface of the cavity using Eq. 77 equals 1684 A^2. Suppose the inside of the cavity is lined with high-purity, polished copper with $\rho = 1.712 \times 10^{-8}$ Ω-m. At 303 MHz, the surface resistance is $R_s = 4.531 \times 10^{-3}$ Ω. The product

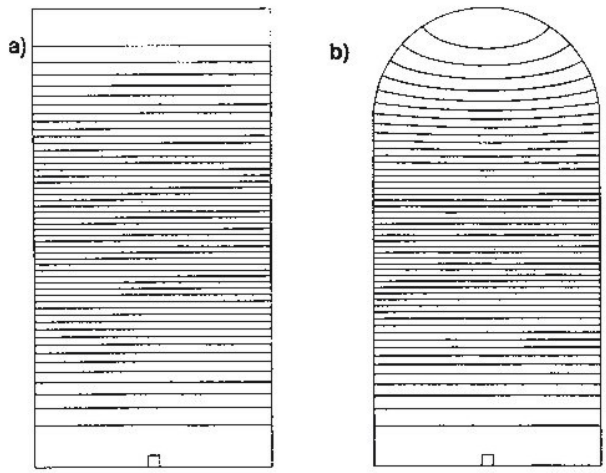


Figure 23: Comparison of the Q factors for two 303 MHz accelerators cavities with the same axial length cavities. Contours of rH_θ, lower boundary is the z axis. a) Pillbox cavity, outer radius 0.3789 m. b) Cavity with contoured outer wall, outer radius 0.4000 m.

of the surface resistivity and surface current integral gives a total power loss of $P = 7.630$ W. The quality factor is therefore $Q = 2\pi f U/P = 34,165$. The analytic formula for the TM_{010} mode in a cylindrical cavity is

$$Q = \left(\frac{d}{\delta}\right)\left(\frac{1}{1+d/R}\right), \tag{83}$$

where δ is the skin depth:

$$\delta = \sqrt{\frac{\rho}{\pi\mu_0 f}}. \tag{84}$$

At 303 MHz, the skin depth in copper is $\delta = 3.778$ μm. Inserting values in Eq. 83 gives $Q = 35,290$, close to the code value. Similar calculations for the rounded cavity of Fig. 23 give $U = 1.080 \times 10^{-4}$ J, $\int\int dA\ H_{\|}^2/2 = 1211$ A^2, and $P = 3.750$ W. These figures imply an improved quality factor of $Q = 37497$.

3.10 Techniques for scattering solutions

WaveSim can find solutions for scattering of electromagnetic radiation from objects in free space. There are two requirements to accomplish such solutions with a numerical method that uses a bounded computational volume:

- An absorbing boundary around the volume.
- A source inside the volume that creates ideal plane waves but does not interfere with the propagation of scattered waves.

The first requirement is relatively easy. We construct an anechoic chamber by surrounding the solution volume with a matched termination layer (Sect. 3.6). The second is more challenging.

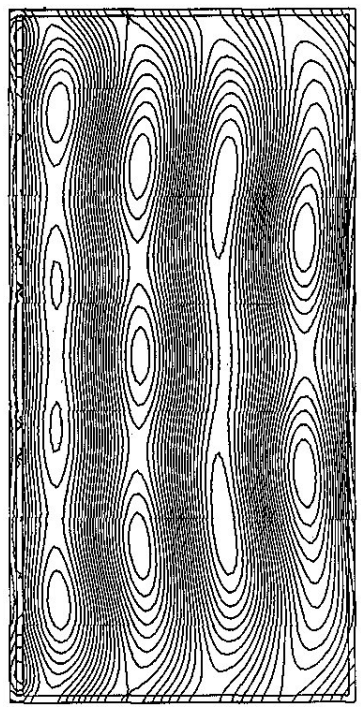


Figure 24: Unsuccessful approach to generate plane waves in an anechoic chamber. Vacuum region surrounded by matched termination layer with a planar drive current layer at the left. The plot shows lines of **H** – waves travel from left to right.

Fixed field drive boundaries inside the solution region may not be used because they reflect scattered waves. Therefore, a complex distribution of internal current sources is necessary. We can determine the optimal spatial distribution of current with the *distributed source method*. To appreciate the technique it is informative to examine first an unsuccessful approach. Figure 24 shows the geometry. The intention is to use a planar current layer to generate E waves traveling to the right. Waves moving to the left are immediately absorbed in the adjacent termination layer. The figure shows the resulting solution for a radiation wavelength equal to half the box width. The waves approximate traveling plane waves but are clearly far from ideal. The upper and lower boundaries cause the problem. The discontinuities of the current sheet at the top and bottom give small transverse field components that reflect at low incidence angle from the termination layers. The pattern of Fig. 24 results from the interference of these components with the propagating plane waves.

To achieve perfect plane waves in the presence of an absorbing boundary we must rethink the problem. Rather than try to guess the correct current density distribution, we will work backward starting from the desired waveform. To begin, consider an anechoic chamber with an absorbing layer but no scattering objects. The desired field variation of an E wave is

$$E_z(x, y) = \xi \exp[-j(k_x x + k_y y)]. \tag{85}$$

If $k_x > 0$ and $k_y = 0$, the expression of Eq. 85 represents a traveling wave with amplitude ξ and wavelength $\lambda = 2\pi/k_x$ moving in the $+x$ direction. Suppose we seek an E wave solution with no sources and a field of the form of Eq. 85. Substitution into Eq. 30 gives

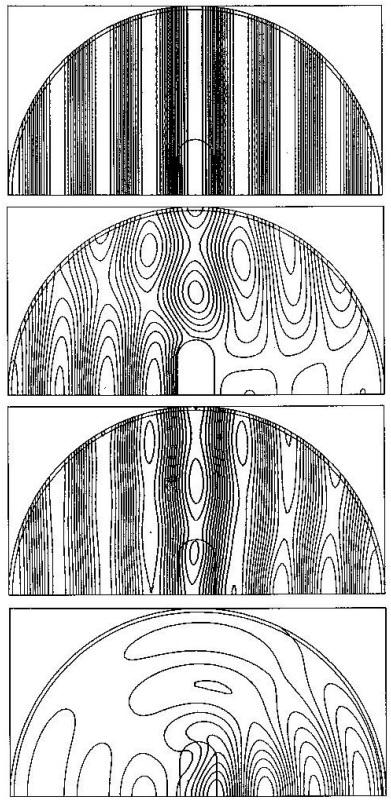


Figure 25: Scattering solutions using the distributed source method. *a*) Ideal plane E waves created by a distributed source. *b*) Total field in the presence of a conducting body ($\epsilon = 10^5\epsilon_0$). *c*) Total field in the presence of a dielectric body ($\epsilon = 1.5\epsilon_0$). *d*) Isolated scattering fields from the dielectric body of part *c*.

$$\sum_i E_{zi}W_i - \xi \sum_i \exp[-j(k_x x_i + k_y y_i)W_i -$$

$$E_{zo}(\sum_i W_i - A_i) + \xi \exp[-j(k_x x_o + k_y y_o)(\sum_i W_i - A_i) = 0. \tag{86}$$

Moving the known terms in Eq. 86 to the right-hand side gives

$$\sum_i E_{zi}W_i - E_{zo}(\sum_i W_i - A_i) =$$

$$\xi \left[\sum_i \exp[-j(k_x x_i + k_y y_i)W_i - \exp[-j(k_x x_o + k_y y_o)(\sum_i W_i - A_i) \right]. \tag{87}$$

Equation 87 has the same form as Eq. 30 if we view the right-hand side as a source function. Noting that the total field must be zero, a numerical solution of Eq. 87 gives the desired plane wave. The important feature is that solutions inside the chamber follow Eq. 85 with *absorbing wall effects included*.

The above discussion suggests the following steps for an ideal scattering solution.

- Set up an anechoic chamber with no scattering objects and calculate the source terms on the right-hand side of Eq. 87 using the standard subroutines to evaluate W_i and A_i.
- Introduce the scattering objects and include their contributions to ϵ and μ when calculating values of W_i and A_i for the left-hand side of Eq. 87.
- Apply the standard matrix inversion to find the field solution with the sources and scattering objects. To isolate the contribution of the scattering objects, subtract the right-hand side of Eq. 85 from the total field.

The procedure is implemented automatically in **WaveSim**. Commands in the input script list regions corresponding to scattering objects. In the initial calculation of W_i and A_i to find the source terms, the region numbers of the scattering objects are replaced by that of the uniform background medium.

Figure 25 illustrates the procedure for E wave scattering from dielectric and metal bodies in vacuum. To optimize the performance of the termination layer, we construct a cylindrical anechoic chamber with an axis centered on the object. Figure 25a shows the ideal plane wave solution with no object ($\epsilon = \epsilon_0$). Figures 25b and c show E_z lines for the total solution for a metal body ($\epsilon = 10^5\epsilon_0$) and a dielectric ($\epsilon = 1.5\epsilon_0$). Finally, Figure 25$d$ plots scattering fields isolated from the solution of Fig. 25c.

4 WaveSim solution reference

A **WaveSim** calculation requires a minimum of two input files:

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

The mesh file always has a name of the form `MNAME.MOU`, where `MNAME` is a valid file prefix (1 to 32 characters). The script must have a name of the form `WNAME.WIN`. **WaveSim** issues an error message if both input files are not available in the working directory. To organize data, the resulting output files have the names `WNAME.WLS` (listing) and `WNAME.WOU` (field and mesh data).

A **WaveSim** run consists of several steps:

- Prepare a **Mesh** input script with a name of the form `MNAME.MIN` following the instructions in the Mesh manual. You can create the file directly with a text editor or graphically using the drawing editor of **Mesh**.
- Run **Mesh** either interactively from the TC program launcher or from the Windows Command Prompt to create the file `MNAME.MOU`.
- Prepare a command script (`WNAME.WIN`) using the *Setup* command in **WaveSim** or a text editor. The allowed file commands are described in this section.
- Run **WaveSim** to create listing output and the output file `WNAME.WOU` in the *Search* and *Scatter* modes. The output file is in text format and contains information on the mesh geometry, the physical properties of regions and values of the real and imaginary parts of the primary field component.
- Investigate the solution using the *Analyze* menu in **WaveSim**. You can also create data files and and transfer information to your own analysis programs.

4.1 WaveSim script format

You can create a script for **WaveSim** using the interactive dialog described in Sect. 2.2 or by writing the commands directly with a text editor. Direct script editing is required for some advanced **WaveSim** capabilities. This chapter gives a detailed description of the syntax and functions of script commands. Section 4.2 covers commands that control program operation. These commands are created by the entries in the *Control parameters* section of the dialog (Fig. 6). Section 4.3 reviews commands to set simple material properties. These commands are created by entries in the *Region properties* grid of the dialog. The remaining sections describe advanced capabilities of **WaveSim** as well as program operation and output file format.

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 (*). **WaveSim** accepts commands in any order. The following example illustrates a complete script:

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

There are two classes of commands: program control and region properties. A control command contains 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) = (50.0, 0.76)
```

sets the relative dielectric constant of elements with region number 2 to $\epsilon_r' = 50.0$ and $\epsilon_r'' = 0.76$. (**Note**: For back-compatibility, **WaveSim** recognizes command formats from Version 1.0 through 5.0 where control commands start with the keyword *Set* and region commands start with *Region*.)

Table 8: **WaveSim** solution modes

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

4.2 Program control commands

In the following sections, commands are written symbolically and as they might appear in the **WaveSim** script:

MESH MPrefix
MESH = Cavity05
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 `FPrefix.WIN`, then **WaveSim** will seek the default mesh file `FPrefix.MOU`.

SOLUTION [H, E]
SOLUTION = E
This command sets the primary field quantity following the choices for two-dimensional solutions listed in Table 2. The parameters are the characters E or H in either upper or lower case. In E type solutions the primary field quantity is either E_z or rE_θ. In H type solutions the primary field quantity is either H_z or rH_θ.

MODE [Search, Scan, Scatter]
MODE = Scatter
Sets the nature of the calculation according to the options of Table 8. The parameter options are the strings *Search*, *Scan* or *Scatter*. *Scan* and *Search* calculations involve several solutions and may involve long run times for large meshes.

DUNIT DUnit
DUNIT = 1.0E4
You may use any convenient distance units in **Mesh**. This command defines a factor to convert coordinates supplied by **Mesh** to the standard distance units of meters used in **WaveSim**. The quantity *DUnit* equals the number of **Mesh** units per meter. For example, if the **Mesh** dimensions are entered in microns, set $DUnit = 1.0 \times 10^6$. The default value is 1.00. (**Note**. Spatial quantities recorded in the output file `FPrefex.WOU` are always in meters. In an analysis session with **WaveSim**, spatial quantities in graphs and listing files are scaled to the **Mesh** units. For example, if the **Mesh** dimensions are in cm and $DUnit = 100.0$, the spatial quantities in plots will be in cm.)

GEOMETRY [Rect, Cylin]
GEOMETRY = Cylin
WaveSim handles problems in rectangular (planar) or cylindrical geometries. Rectangular systems have variations in x and y with infinite length in z. Cylindrical systems have variations in r and z with azimuthal symmetry. The parameter options are *Rect* and *Cylin*. In cylindrical solutions the program takes the z axis along the **Mesh** x direction and the r axis along y. In this case the program issues an error message if any node has a y coordinate less than 0.0.

FREQ Freq
FREQ = 3.45E6
Set a value for the wave frequency (f) in Hz for *Scatter* type calculations. This command has no effect in the *Search* and *Scan* modes.

RANGE FreqLow FreqHigh
RANGE = 120.0E6, 150.0E6
Defines a range of frequencies for *Scan* or *Search* calculations. The parameters are f_{min} and f_{max} in Hz. In the *Scan* mode, **WaveSim** performs a series of calculations from f_{min} to f_{max} in uniform frequency steps of $(f_{max} - f_{min})/f_{step}$, where f_{step} is described under the *FStep* command. In the *Search* mode, f_{min} and f_{max} define a frequency interval for the search that should bracket a single resonance. In this case, the program will not proceed unless the probe response changes sign over the interval. The command has no effect in the *Scatter* mode.

FSTEP FStep
FSTEP = 25
Set the number of frequency intervals in the *Scan* mode or the maximum of cycles in the *Search* mode. The integer parameter is the number of steps. The defaults values are 20 in the *Scan* mode and 7 in the *Search* mode. The command has no effect in the *Scatter* mode.

TOLERANCE FToler
TOLERANCE 5.0E-5
Set the target accuracy for a resonance calculation in the *Search* mode. The real number parameter *FToler* is the target accuracy. **WaveSim** exits the search when the relative change

in frequency between two cycles satisfies $\Delta f/f \leq FToler$. The default value is $FToler = 1.0 \times 10^{-4}$. The command has no effect in the *Scatter* and *Scan* modes.

PROBE XPos YPos
PROBE ZPos RPos
PROBE = (0.01, 0.02)
Sets a location for a virtual probe that senses the real and imaginary parts of the primary field component for the *Scan* and *Search* modes. The real number parameters are the approximate coordinates of the position, either (x, y) or (z, r). **WaveSim** places the probe at the nearest node. The positions should be entered in meters or alternate units if the *DUnit* command appears in the input file. This command has no effect in the *scatter* mode.

$$\omega \, \frac{\epsilon''}{\epsilon_0} = \frac{\sigma}{\epsilon_0} \, , \tag{88}$$

where σ is the conductivity is S/m.

LOWQ
For a resonance search in an ideal or high-Q structure, **WaveSim** seeks a zero in the reciprocal of either the real or imaginary part of the probe response. In structures with very low values of Q, it may be better to search for a direct zero of the probe signal. This command activates the low Q search option.

PLANEWAVE kx ky A [T]
PLANEWAVE 13.6 0.0 5000.0 T
This command sets up an incident plane electromagnetic wave for scattering solutions in rectangular geometry using the method of distributed sources (Sect. 3.10). There are three real number parameters:

- k_x, the wave number in m^{-1} along the x direction.
- k_y, the wave number in m^{-1} along the y direction.
- A, the amplitude of the primary field component in the wave.

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

$$E_z(x, y, t) = A \;\; \exp[j(k_x x + k_y y - \omega t)]. \tag{89}$$

Values of $k_x > 0.0$ and $k_y = 0.0$ give a plane wave propagating in the positive x direction. The optional single-character parameter T controls the field solution output. If it appears, **WaveSim** makes a data file of the total field – incident plus scattered. Otherwise, the programs records the scattered field only (total field minus the specified plane wave).

4.3 Commands for material properties

Because frequency-domain solutions are meaningful only for linear materials, the set of **WaveSim** commands to define region material properties is relatively simple. The commands may be generated with the *Setup* dialog in **WaveSim** or directly with a text editor.

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

VOID RegNo
VOID = 7
The *Void* condition is the inverse of the *Reflect* condition. It represents a metal wall for H type waves and an open-circuit for E type waves. **WaveSim** implements internal void regions by setting $\epsilon = \epsilon_0$ and $\mu = 10^{12}\mu_0$ for E type waves and $\epsilon = 10^{12}\epsilon_0$ and $\mu = \mu_0$ for H type waves. Note that all unspecified external boundaries automatically assume the *Void* condition.

EPSI RegNo EpsiR' EpsiR"
EPSI(2) = (5.8, -56.03)
This command sets the dielectric constant for elements of a region. The first real number parameter is the real part of the relative dielectric constant, ϵ'/ϵ_0. The optional second parameter equals the imaginary part of the dielectric constant, ϵ''/ϵ_0 when *CondMode* is not set. If the *CondMode* command appears, enter the quantity $\omega\epsilon''/\epsilon_0$. **WaveSim** takes the imaginary part of the dielectric constant as zero if there is no second parameter. The default value for all regions is $\epsilon_r = [1.0, 0.0]$.

MU RegNo MuR' MuR"
MU(2) = (100.0, -25.6)
This command sets the magnetic permeability for elements of a region. The first real number parameter is the real part of the relative magnetic permeability, μ'/μ_0. The optional second parameter equals the imaginary part of the magnetic permeability, μ''/μ_0, when *CondMode* is not set. If the *CondMode* command appears, enter the quantity $\omega\mu''/\mu_0$. **WaveSim** takes the imaginary part of the magnetic permeability as zero if there is no second parameter. The default value for all regions is $\mu_r = [1.0, 0.0]$.

DRIVE RegNo Amp Phase
FIXED(7) = 2500.0 (45.0)
Set a fixed-field condition. In this case, the region may act as a wave source. The two real parameters are the amplitude and phase of the primary field at all points in the region. Specify the amplitude in the standard units of the primary field and the phase in degrees.

Table 9: Source currents in **WaveSim**

Type	Geometry	Quantity	Units
E	Rect	j_z	A/m^2
	Cylin	rj_θ	A/m
H	Rect	j_x, j_y	A/m^2
	Cylin	rj_z, rj_r	A/m

SOURCE RegNo Amp Phase [Beta]
SOURCE(3) = 4500.0 (-90.0, 0.0)
The keyword *Source* indicates that the region has a source current density. The first real number is the amplitude of the source. The interpretation of the value and units depends on the geometry and solution type. Table 9 shows the options. The second real-number parameter is the phase in degrees. The third optional parameter gives the direction of the current for H wave initiation. The quantity is the angle of the current density vector (in degrees) in the x-y plane relative to the x axis for planar problems. For cylindrical problems, the quantity gives the angle in the r-z plane relative to the z axis. The parameter has no meaning for E wave solutions where the current density points along the direction of the primary field (z or θ). Note that you can also specify values of relative dielectric constant and magnetic permeability for the source region using the *Epsi* and *Mu* commands.

ABSLAYER RegNo Delta [EpsiR MuR Theta]
ABSLAYER(5) = 0.2 (4.0)
ABSLAYER(5) = 0.2 (4.0, 20.0, 75.0)
Define the region as an absorbing layer. The integer parameter *RegNo* must correspond to a filled mesh region of uniform thickness with a surface approximately normal to the direction of wave propagation. The real number *Delta* is the layer thickness in units set by *DUnit*. Enter values for the optional parameters *EpsiR* and *MuR* if the layer is adjacent to a non-vacuum region with relative electrical permittivity ϵ_r and magnetic permeability μ_r (real parts). The final optional parameter is the average angle between the wave propagation vector and the layer surface. Enter the value in degrees. The default is $\theta = 90^o$. **WaveSim** automatically sets the real and imaginary parts of ϵ and μ in the region according to Table 6. Values are updated to reflect changing frequency in the *Search* and *Scan* modes.

COND RegNo Sigma
COND(8) = 4.0
As an alternative to setting the imaginary part of ϵ for a region in a *Scatter* mode solution, you can specify the conductivity σ in S/m. **WaveSim** supplements the imaginary part of the electrical permittivity according to Eq. 61. Note that this command must follow the *Freq* command so that the value of ω is defined.

NREPLACE Reg1 Reg2
NREPLACE(5) = 2
This command is used to implement the distributed source method for free-space scattering solutions (Sect. 3.10). The source terms are computed with the scattering object(s) removed from a homogeneous region of space. In this command, the first integer specifies the number of a region that constitutes a part of the scattering object. The second integer is the region number of the surrounding solution volume. This volume carries a plane wave in the absence of the scattering object. Each region of the scattering object(s) must have an *NReplace* command.

4.4 Running WaveSim

WaveSim 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:

```
[ProgPath\]WaveSim [DataPath\]WPrefix <ENTER>
```

where the file `WPREFIX.WIN` and the **Mesh** output file are available in the data directory. With this capability, you set up extended autonomous **WaveSim** runs using a DOS batch file or a Perl script.

The remainder of this section discusses commands in the main menu when **WaveSim** 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 (WIN)
EDIT LISTING (WLS)
EDIT FILE
These commands call the internal editor to inspect or to modify **WaveSim** input and output files. Choosing a file from an alternate directory does not change the working directory. The *Edit script (WIN)* command shows a list of all files with names of the form `FPREFIX.WIN` while *Edit listing (WLS)* displays files with names `FPREFIX.WLS`.

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. 5.5. Note that the analysis script must be in the same directory as the data files.

SETUP
The function of this command is to create a **WaveSim** script to control a 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. 6. 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. 4.2. You can enter basic physical properties of regions in the grid box (see Sect. 4.3).

The *Solve* popup menu includes the following two commands.

RUN

Pick an input file (such as `FPrefix.WIN`) to start a solution. The working directory is changed if you pick a file from an alternate directory. The run begins if the file `FPrefix.MOU` 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 **WaveSim** and saves as much data as possible, depending on the solution mode.

ANALYZE

Pick a file of the type `FPrefix.WOU` and call up the analysis menu for plotting and numerical analysis.

WAVESIM MANUAL

Display this manual in your default PDF viewer. The file `WaveSim.pdf` must be available in the same directory as `wavesim.exe`.

4.5 Format of the WaveSim output file

The WaveSim output file `FPrefix.WOU` is in text format. The file has three sections:

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

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

```
--- Run parameters ---
XMin:  0.000000E+00
XMax:  5.000000E-02
KMax:  101
YMin:  0.000000E+00
YMax:  5.000000E-02
LMax:  101
DUnit:  1.000000E+02
NReg:    4
ICylin:  1
Frequency:  1.484412E+09
Type:  H
```

Lines 2 and 3 list x_{min} and x_{max}, the limits along horizontal axis (x or z) of the solution volume. Dimensions are given in meters. The quantity k_{max} in Line 4 is the number of nodes along the horizontal direction. Lines 5-7 describe the vertical axis (y or r). Line 8 contains the quantity $DUnit$, the conversion factor from dimensions used in **Mesh** to meters. Line 9 gives the number of regions in the solution, while Line 10 specifies the symmetry (0: planar, 1: cylindrical). Line 11 and 12 list the RF frequency (in Hz) and the solution type (E or H).

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

```
--- Nodes ---
  k    l  RgNo RgUp RgDn       x             y
=====================================================
  ...
  18   15    1    1    1 -5.673316E-02  3.500000E-02
  19   15    1    1    1 -5.415951E-02  3.500000E-02
  20   15    1    1    1 -5.164122E-02  3.500000E-02
  21   15    2    2    2 -5.000000E-02  3.500000E-02
  22   15    2    2    2 -4.718639E-02  3.500000E-02
  23   15    2    2    2 -4.461200E-02  3.500000E-02
  ...

        FReal         FImag        PDensUp       PDensDn
    ========================================================
    ...
     -7.331566E-03 -2.326431E-03  0.000000E+00  0.000000E+00
     -7.385575E-03 -2.342826E-03  0.000000E+00  0.000000E+00
     -7.439394E-03 -2.359163E-03  0.000000E+00  0.000000E+00
     -7.474633E-03 -2.369858E-03  2.050144E+01  2.051407E+01
     -7.570758E-03 -2.398376E-03  2.107301E+01  2.105082E+01
     -7.656379E-03 -2.423766E-03  2.162613E+01  2.156914E+01
    ...
```

Each data line contains the following quantities:

- The indices of the node (K,L)
- The region number of the node (*RgNo*) and region numbers for two associated elements (*RgUp* and *RgDn*). The upper element lies above the line between nodes (K,L) and (K+1,L) and the lower element lies below the line.
- The coordinates of the node in meters, (x,y) or (z,r).
- The real and imaginary parts of the principal field component: E_z in V/m, rE_θ in V, H_z in A/m or rH_θ in A.
- The power density (in W/m^3) of the upper and lower elements for transfer to TDiff.

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

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

   1    0    0    0    0    0  8.854188E-12  0.000000E+00  1.256637E-06
   2    0    0    0    0    0  8.854188E-12  0.000000E+00  1.256637E-06
   3    1    0    0    0    0  8.854188E-12  0.000000E+00  1.256637E-06
   4    0    0    0    0    0  8.854188E-12  0.000000E+00  1.256637E-06

         MuImag        AReal         AImag         JReal

       0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       0.000000E+00  0.000000E+00  0.000000E+00  1.000000E+00
       0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00

         JImag        JAngle          Area

       0.000000E+00  0.000000E+00  2.015532E-03
       0.000000E+00  0.000000E+00  3.999995E-06
       0.000000E+00  0.000000E+00  0.000000E+00
       0.000000E+00  0.000000E+00  0.000000E+00
```

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

Table 10: **WaveSim** standard units

Quantity	**Unit**
Spatial dimensions	meters or units set by *DUnit*
Frequency	Hz (cycles/second)
Electric field, **E**	V/m (volts/meter)
Magnetic intensity, **H**	A/m (amperes/m)
Current density, **j**	A/m^2 (amperes per square meter)
Energy density	J/m^3 (joules per cubic meter)
Power density	W/m^3 (watts per cubic meter)

5 Solution analysis reference

To create plots and to perform numerical analyses, click the *Analyze* command in the **WaveSim** main menu and choose a data file. The analysis routines automatically adjust labels and calculated quantities depending on whether the solution E type or H type, planar or cylindrical. Table 10 lists standard units for electromagnetic solutions.

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 generate additional solutions.

5.1 File menu commands

LOAD SOLUTION FILE

Load a different solution file for analysis without returning to the main menu. Pick a new file `WPREFIX.EOU` in the dialog. Changing the directory in the dialog changes the program working directory.

SAVE SOLUTION FILE

Save the currently-loaded values of the real and imaginary parts of the primary field in the standard solution file formal. Supply the same or a different file name. This command is used mainly after the *Renormalize fields* command.

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 analysis functions described in this chapter. The script command language is described in Sect. 5.5. 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.

5.2 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. One function of this type of plots is to show field lines. In E type solutions, contours of E_z or rE_θ lies along lines of $\mathbf{H}$ (Sect. 3.4). Conversely, the contours of H_z and rH_θ in H type solutions like along lines of $\mathbf{E}$. Note that contours of other quantities may exhibit regions of compressed lines because of field discontinuities at the boundaries of regions.
- **Element**. Elements of the solution space color-coded according to a computed quantity.

- **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 11) consistent with the current plot type. The list will be empty for *Mesh* and *Region* plots.

WaveSim supports a extensive set of calculated quantities compared to a static-field program like **EStat**. There are two reason for the diversity: 1) available quantities depend on whether the solution was E or H type, planar or cylindrical and 2) the time-dependent quantities have phase as well as amplitude. With regard to the first issue, **WaveSim** automatically adjusts the list of plot quantities to reflect the solution type and symmetry. With regard to time variation, it is useful to review some features of harmonic functions.

The `WOU` file contains real and imaginary parts of the primary field quantity, $[F_r, F_i]$. The real and imaginary parts of the secondary field quantities $[f_{xr}, f_{xi}]$ and $[f_{yr}, f_{yi}]$ (or $[f_{zr}, f_{zi}]$ and $[f_{rr}, f_{ri}]$) can be determined from spatial derivatives of the primary quantity (Section 3.4). The amplitude (or peak value) of a harmonic quantity is denoted as $|F|$ and is given by

$$|F| = \sqrt{F_r^2 + F_i^2}. \tag{90}$$

The amplitude is a positive number at all points. We shall use the following notation when the amplitude applies to both space and time:

$$||\mathbf{f}|| = \sqrt{f_{xr}^2 + f_{xi}^2 + f_{yr}^2 + f_{yi}^2}. \tag{91}$$

Another option is to plot a snapshot of field variations in a quantity at a reference value of phase ϕ_r. The value of the quantity calculated at the reference phase is:

$$F_{ref} = [F_r + jF_i][\cos\phi_r - j\sin\phi_r]. \tag{92}$$

Reference phase quantities may have both positive and negative values. Note that a reference phase quantity may have zero value at all points in space for a lossless solution. For example, in a H type resonant solution, the electric field equals zero at $\phi_r = 0^o$. To show the electric field in this case, choose $\phi_r = 90^o$.

Table 11: **WaveSim** plot quantities

Plot type	**E, planar**	**E, cylin**	**H, planar**	**H, cylin**
Contour	E_z (**H** line) $\|E_z\|$ E_z(ref)	rE_θ (**H** line) $\|E_\theta\|$ E_θ(ref)	H_z (**E** line) $\|H_z\|$ H_z(ref)	rH_θ (**E** line) $\|H_\theta\|$ H_θ(ref)
Element	$\|E_z\|$ E_z(ref) $\|\|\mathbf{H}\|\|$ u p $\|H_x\|$ H_x(ref) $\|H_y\|$ H_y(ref)	$\|E_\theta\|$ E_θ(ref) $\|\|\mathbf{H}\|\|$ u p $\|H_z\|$ H_z(ref) $\|H_r\|$ H_r(ref)	$\|H_z\|$ H_z(ref) $\|\|\mathbf{E}\|\|$ u p $\|E_x\|$ E_x(ref) $\|E_y\|$ E_y(ref)	$\|H_\theta\|$ H_θ(ref) $\|\|\mathbf{E}\|\|$ u p $\|E_z\|$ E_z(ref) $\|E_r\|$ E_r(ref)
Vector	$\|\mathbf{H}\|(ref)$	$\|\mathbf{H}\|(ref)$	$\|\mathbf{E}\|(ref)$	$\|\mathbf{E}\|(ref)$
Surface, Scan	$\|E_z\|$ E_z(ref) $\|\|\mathbf{H}\|\|$ u p $\|H_x\|$ H_x(ref) $\|H_y\|$ H_y(ref) E_z(real) E_z(imag) H_x(real) H_x(imag) H_y(real) H_y(imag)	$\|E_\theta\|$ E_θ(ref) $\|\|\mathbf{H}\|\|$ u p $\|H_z\|$ H_z(ref) $\|H_r\|$ H_r(ref) E_θ(real) E_θ(imag) H_z(real) H_z(imag) H_r(real) H_r(imag)	$\|H_z\|$ H_z(ref) $\|\|\mathbf{E}\|\|$ u p $\|E_x\|$ E_x(ref) $\|E_y\|$ E_y(ref) H_z(real) H_z(imag) E_x(real) E_x(imag) E_y(real) E_y(imag)	$\|H_\theta\|$ H_θ(ref) $\|\|\mathbf{E}\|\|$ u p $\|E_z\|$ E_z(ref) $\|E_r\|$ E_r(ref) H_θ(real) H_θ(imag) E_z(real) E_z(imag) E_r(real) E_r(imag)

The following equations are included for reference. The phase of a complex harmonic quantity is given by:

$$\phi = \tan^{-1}(F_i/F_r). \tag{93}$$

The expression for the time-averaged field energy density (J/m^3) in an electromagnetic solution is

$$u = \frac{1}{4}\left(\epsilon'||\mathbf{E}||^2 + \mu'||\mathbf{H}||^2\right). \tag{94}$$

where the magnitude operation extends over the real and imaginary parts of all spatial components of the field quantities (Eq. 91) . When a solution contains materials with losses, the time-averaged power density (W/m^3) is given by:

$$p = \frac{\omega}{2}\left(\epsilon''||\mathbf{E}||^2 + \mu''||\mathbf{H}||^2\right). \tag{95}$$

where ϵ'' and μ'' are the imaginary part of the material properties.

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.

GRID CONTROL

This command displays the dialog of Fig. 26 to set properties of the grid. In the default autoscale mode, **WaveSim** automatically chooses intervals and positions so that lines occur at convenient values of 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

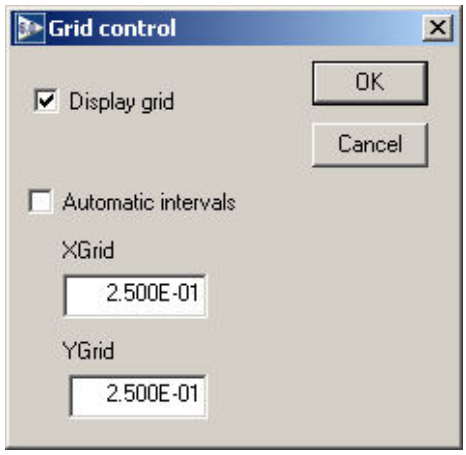


Figure 26: Grid control dialog

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. 27). 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.

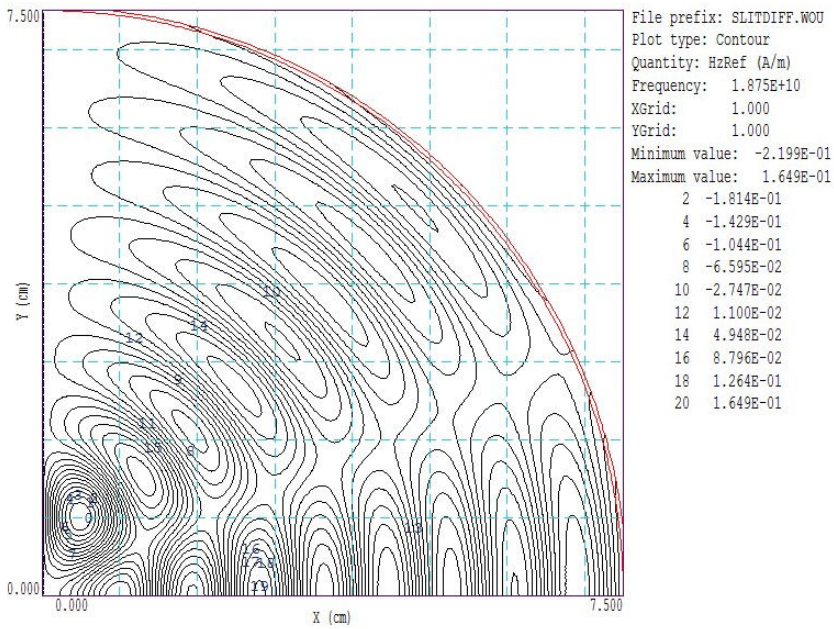


Figure 27: Monochrome contour plot with labels

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 $N_x = N_y = 40$.

5.3 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

Click on the command and move the mouse cursor to any point in the solution space. (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, use the *Toggle mouse/keyboard* command.

LINE SCAN

The line scan is one of the most useful functions of **WaveSim**. After clicking on the command, supply two points with the mouse in a view of a *Mesh*, *Region*, *Contour*, *Element* or *Vector* plot to define a scan line. The snap mode is useful in this application (for example, you may want a scan to extend from 0.000 to 5.000 rather than 0.067 to 4.985.) The program computes a series of values of field quantities at equal intervals along the line. The information is recorded if a 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.5.4).

ELEMENT PROPERTIES

Pick an element with the mouse (or keyboard). **WaveSim** highlights the elements and writes mesh and material 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. **WaveSim** calculates volume integrals of field energy density and power density over the specified region and shows maximum values of $|\mathbf{E}|$ and $|\mathbf{H}|$.

VOLUME INTEGRAL

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

- The volume (cylindrical) or cross-section area (planar).
- Volume integrals of electromagnetic field energy density (Eq. 94). The output units are J/m in rectangular solutions and J in cylindrical solutions.
- Volume integrals of power dissipation (Eq. 95). The output units are W/m in rectangular solutions and W in cylindrical solutions.
- The value and location of maxima of $||\mathbf{E}||$ and $||\mathbf{H}||$.
- If line regions are present, surface integrals of $H_{\|}^2/2$ to compute wall losses in resonators.

Table 12: Example of data created by the *Volume integral* command

```
 --- Volume Integrals ---
Volume:   9.020E-02 m3
Field energy:   2.947E+01 J
Volume power dissipation:   0.000E+00 W
EMax:   2.396E+07 V/m
    Z:   5.625E-04,  R:   3.324E-03
HMax:   2.557E+04 V/m
    Z:   9.431E-02,  R:   2.900E-01

    Integrals by region
  NReg    Volume     Energy     Power
           (m3)       (J)        (W)
  ====================================
     1  8.993E-02  2.932E+01  0.000E+00
     2  2.765E-04  1.481E-01  0.000E+00

 --- Line Integrals over Line Regions ---
  NReg     Int(Hp^2/2)
              (A^2)
 ====================
     3      0.000E+00
     4      3.609E+08
```

Table 12 shows an example of a data listing.

MATRIX FILE

WaveSim can make matrix files of field values to help you create your own analysis routines. Although information is available in the output file of the solution program, it may be difficult to deal with the conformal triangular mesh. The *Matrix file* command uses the interpolation capabilities of the program to create 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. Each data line contains the coordinates [(x,y) or (z,r)], the region number, and the real and imaginary parts of the primary and secondary field quantities. For example, in a E type solution in planar coordinates, the program lists $E_{zr}, E_{zi}, H_{xr}, H_{xi}, H_{yr}$ and H_{yi}.

The *Analysis settings* popup menu contains the following entries.

REFERENCE PHASE

Set the reference phase ϕ_r. Enter the value in degrees. The settings affects plots and calculations of reference phase quantities (Table 11).

INTERPOLATION METHOD

The default interpolation method for the *Point calculation* and *Line scan* commands is a second-order least-squares fit with intelligent collection of data points. For example, only points on the side of a dielectric boundary that contains the target point are included to give the correct field discontinuity at the boundary. The least-squares fit may fail in very small regions or enclosed areas if the program cannot identify enough data points. In this case, toggle to the linear mode. Here, field values are determined by a first order fit in the element that contains the target point. 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 of Table 11 and click *OK*. This setting has no effect on the data file records which include all field quantities.

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.

RENORMALIZE FIELDS

Multiply all stored real and imaginary values of the primary field by a specified factor. The command may be used to set a specific field normalization for a resonant mode calculation. You can save the modified values with the *Save solution file* command.

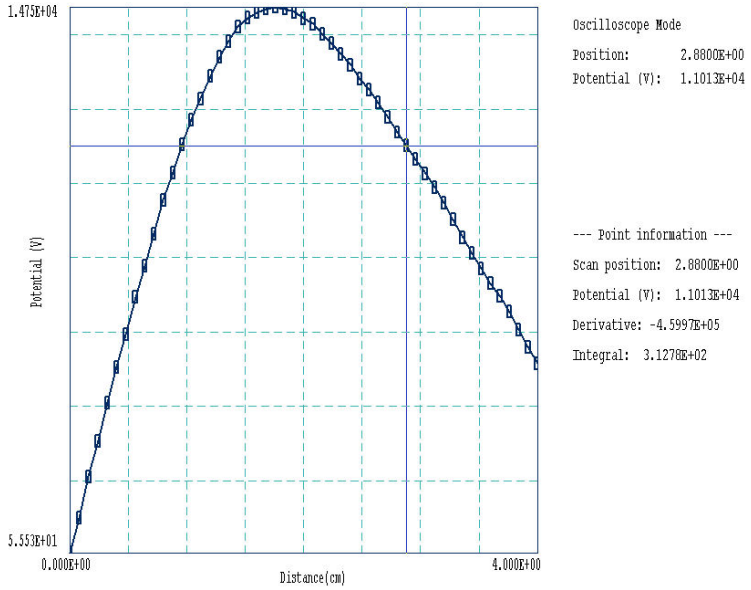


Figure 28: Scan plot in the oscilloscope mode

5.4 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. 28). 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 **WaveSim**. This command closes the scan plot and returns the program to the previous spatial plot.

5.5 Analysis script commands

Scripts to control analysis sessions have a name of the form `WPREFIX.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.

WaveSim 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]\WAVESIM GTEST <Enter>
```

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

The following commands may appear in a script:

INPUT FileName
INPUT Switch1.WOU
Close the current data file and load a file for analysis. The parameter is the full name of the data file. For the command illustrated, the post-processor would load the file `SWitch1.WOU`. You can load several files for sequential analysis.

OUTPUT FPrefix
OUTPUT SW02
Close the current data file and open an output file `SW02.DAT`.

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*.

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.

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

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.

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).

REFPHASE Phi
REFPHASE = 45.0
Change the reference phase for subsequent calculations. Enter the value in degrees.

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

6 Resonant mode calculation tutorials

This chapter reviews four calculations that illustrate the underlying physics and setup techniques for resonant mode calculations in **WaveSim**. The simulations employ the *Scan* and *Search* modes. The three sections emphasize different classes of solutions:

- **Basic resonant cavity**. The geometry (a right circular cylinder or pillbox) is simple to enable comparisons with theory. The example illustrates the basic setup for a resonant mode search and the method to calculate the Q factor determined by wall losses.
- **Waveguide cutoff modes**. Calculation of TE and TM mode cutoff frequencies for a circular waveguide, combining *Scan* and *Search* operations. An advanced example uses asymmetric structures to shift mode frequencies, depending on polarization.
- **Ferrite-loaded cavity**. The geometry is a synchrotron cavity with a tuning ferrite. The example shows how to find the resonant frequency and Q factor in the presence of volumetric power losses.

All input files are supplied in the **WaveSim** example library.

6.1 Basic resonant cavity

We want to find the resonance frequency and Q factor for the TM_{010} mode of the cavity shown in Fig. 29, a right circular cylinder of length $L = 20.0$ cm and radius $R = 40.0$ cm. In the cylindrical system, the primary field quantity is rH_θ, so we seek an H type solution. The first region in the **Mesh** file is a filled region that covers the solution volume. The natural boundary condition ($H_\perp = 0$) corresponds to a metal wall. The second filled region is a small volume near the point of maximum electric field that carries an axial drive current to excite the mode. It is a good practice to locate the drive a short distance from the axis. In this case, the elements near the axis are homogeneous, ensuring good field interpolations in the *Analysis* menu. The third region is a line along the axis to set the condition $rH_\theta = 0.0$. Finally, the fourth region is a line along the physical walls of the cavity for an estimate of resistive power losses.

Table 13 shows the **WaveSim** control script. The theoretical value of the resonant frequency, given by

$$f = \frac{2.404826\ c}{2\pi R}, \tag{96}$$

is 286.856 MHz. We choose a search range 250 MHz to 300 MHz. Because the mode has the lowest frequency, we need not worry about competing modes in the range and it is not necessary to do a preliminary scan. The probe to sense the cavity response is placed at $r = 35.0$ cm, near the point of maximum rH_θ. Regions 1, 2 and 4 have the dielectric properties of vacuum. The drive current density in Region 2 points in the z direction and has an arbitrary amplitude $rj_z = 1.0$ A/m at 0° phase.

With an element size of 0.5 cm, the search takes about 3 seconds and returns a resonant frequency of 286.821 MHz. The difference from the theoretical value is $\Delta f/f \cong 1.2 \times 10^{-4}$.

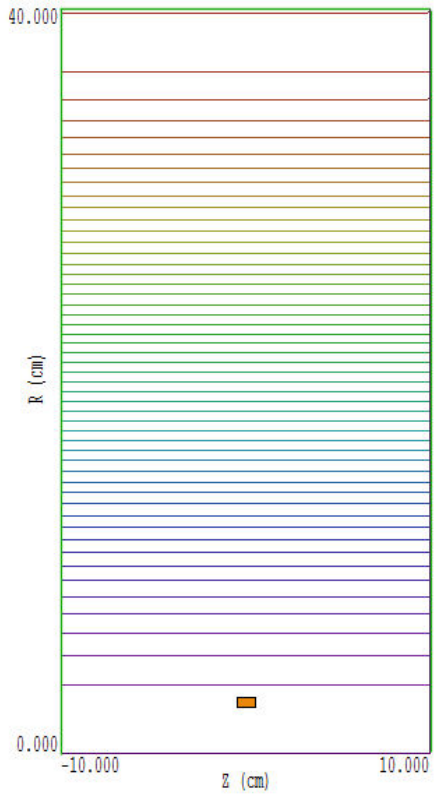


Figure 29: Geometry for the `SimpleCavity` example with electric field lines. The drive current region is shown in orange and the diagnostic boundary in green.

Table 13: Contents of the file SIMPLECAVITY.WIN

```
Geometry = Cylin
DUnit = 100.0
Solution = H
Mode = Search
Range = (250.0E6 300.0E6)
FStep = 9
Tolerance = 1.0E-6
Probe = (0.0, 35.0)
* Vacuum
Mu(1) = 1.0
Epsi(1) = 1.0
* Drive source
Mu(2) = 1.0
Epsi(2) = 1.0
Source(2) = (1.0, 0.0, 0.0)
* Axis
Reflect(3)
* Diagnostic
Epsi(4) = 1.0
Mu(4) = 1.0
ENDFILE
```

With an element size of 0.25 cm, the runtime is 32 seconds and the predicted frequency is 286.848 MHz ($\Delta f/f \cong 2.6 \times 10^{-5}$). During the second run, the program makes the following entries in the listing file SIMPLECAVITY.WLS while performing the search procedure described in Sect. 3.7:

```
--- Response at frequency interval boundaries ---
     Frequency (low):   2.500000E+08
      Response (low):   7.021391E+02
    Frequency (high):   3.000000E+08
     Response (high):  -2.361759E+02

Iteration:   1   Frequency:   2.874149E+08   Response:  -1.037878E+01
Iteration:   2   Frequency:   2.868448E+08   Response:   5.303467E-02
Iteration:   3   Frequency:   2.868477E+08   Response:  -4.179829E-05
Iteration:   4   Frequency:   2.868476E+08   Response:   2.791608E-03
Resonance search successful.
     Final frequency:   2.868477E+08
     Writing data to file SimpleCavity.WOU
```

When you transfer to the analysis menu and load SIMPLECAVITY.WOU, **WaveSim** creates the default plot of electric field lines shown in Fig. 29. The distance between lines is proportional to the magnitude of rH_θ. The field amplitudes have arbitrary value. Use the *Point calculation* and *Renormalize fields* command to set the on-axis electric field to $E_0 = E_z(z, 0) = 10.0$ MV/m (see Sect. 5.3). A volume integral produces the listing of Table 14 if a data file has been opened.

Table 14: Volume integral listing. Solution for the `SIMPLECAVITY` example with the peak electric field normalized to $E_0 = 10.0$ MV/m.

```
Volume:   1.005E-01 m3
Field energy:   1.200E+01 J
Volume power dissipation:   0.000E+00 W
EMax:   1.446E+07 V/m
    Z:  -9.944E+00,  R:   3.333E-01
HMax:   1.545E+04 V/m
    Z:   9.712E+00,  R:   3.058E+01

    Integrals by region
  NReg    Volume     Energy     Power
           (m3)       (J)        (W)
  ====================================
     1  1.005E-01  1.200E+01  0.000E+00
     2  8.639E-06  1.907E-03  0.000E+00

 --- Line Integrals over Line Regions ---
  NReg     Int(Hp^2/2)
              (A^2)
 ====================
     3      0.000E+00
     4      1.441E+08
```

Following the discussion of Sect. 3.9, the theoretical value of total field energy is $U = 11.991$ J. In comparison, **WaveSim** calculates 12.000 J. The theoretical value of the cavity Q for the TM_{010} mode is given by Eq. 83. For an ideal copper-lined cavity with volume resistivity $\rho = 1.712 \times 10^{-8}$ Ω/m. The skin depth (Eq. 84) at 286.8 MHz is 3.883×10^{-6} m and the surface resistivity (Eq. 79) is $R_s = 4.409 \times 10^{-3}$ Ω. The predicted Q value is 34,330. As an exercise, use the values in Table 14 to show that the code predicts an instantaneous power loss of 6.353×10^5 W and a Q factor of 34,040.

6.2 Waveguide cutoff modes

This section illustrates how to apply **WaveSim** to find the cutoff modes of waveguides of arbitrary shape. The field quantities vary with axial distance as $\exp(jkz)$. The wave number k approaches zero at the cutoff frequency, $\omega_c = 2\pi f_c$. In this limit, the field quantities do not vary in z so we can treat them as a resonant mode of an infinite structure. Waveguide calculations of cutoff modes always use the planar geometry option where field quantities vary in x and y but have no variation z.

If the fill medium of the waveguide is homogeneous, we can find the characteristics of a propagating mode from the cutoff mode. For example, when $\omega > \omega_c$, the phase velocity of a propagating waves is given by

$$\frac{\omega}{k} = \frac{1}{\sqrt{\epsilon\mu}} \frac{1}{\sqrt{1 - \omega_c^2/\omega^2}}. \tag{97}$$

To start, we shall make a comparison with theory by treating TE_{0n} and TM_{0n} modes in a circular waveguide of radius $R = 2.0$ cm. Modes of type TE have no axial component of electric field; therefore, the primary field component is H_z. The **Mesh** file `CIRCWAVEGUIDE.MIN` defines three regions: Region 1 fills a circle of radius 2.0 cm, Region 2 is a small drive current volume and Region 3 is a line around the outside of Region 1. Region 3 is not necessary for the H type solutions of the TE modes where the natural boundary represents a metal wall. The region will play a role when we investigate TM modes.

Table 15 lists the **WaveSim** input file `CIRCWAVEGUIDETE.WIN`. The frequency scan brackets the anticipated values of f_c,

$$f_{TE01} = \frac{1.841c}{2\pi R} = 4.395 \text{ GHz}, \quad f_{TE02} = \frac{5.330c}{2\pi R} = 12.72 \text{ GHz}. \tag{98}$$

For the H type solution, Regions 1, 2 and 3 have the physical properties of vacuum. Region 2 carries a drive current in the x direction to create electric fields which point predominantly in x. Note that we have displaced the region from $y = 0.0$ so that we can couple energy into the TE_{02} mode. The probe is located at $x = 0.0$ near the outer wall of the cavity.

Figure 30 shows a plot of the real part of the probe response as a function of frequency. There is a strong positive-to-negative transition at the frequency of the TE_{01} mode and a negative-to-positive transition at the TE_{02} mode. The intervening modes are of type TE_{n1}. The TE_{02} mode has a different transition polarity because H_z has a 180° difference in phase between the drive and probe positions. To calculate fields for the TE_{02} mode, replace the scan commands with the following set:

```
Mode = Search
```

Table 15: File `CIRCWAVEGUIDETE.WIN`, set up for a frequency scan.

```
Mesh = CircWaveguide
Geometry = Rect
DUnit = 100.0
Solution = H
Probe = (0.00 1.75)
Mode = Scan
Range = (2.0E9 16.0E9)
FStep = 70
* Vacuum
Mu(1) = 1.0
Epsi(1) = 1.0
* DriveSource
Mu(2) = 1.0
Epsi(2) = 1.0
Source(2) = (1.0, 0.0, 0.0)
* Boundary
Mu(3) = 1.0
Epsi(3) = 1.0
ENDFILE
```

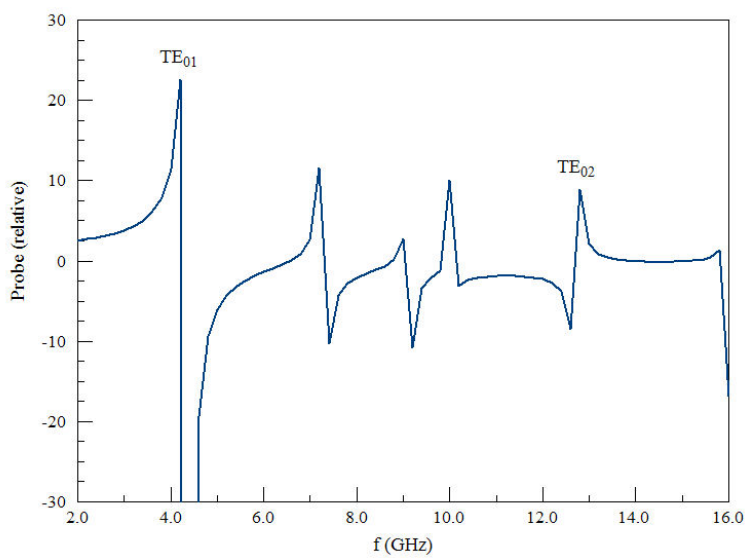


Figure 30: Frequency scan of TE cutoff modes in a circular waveguide of radius $R = 2.0$ cm.

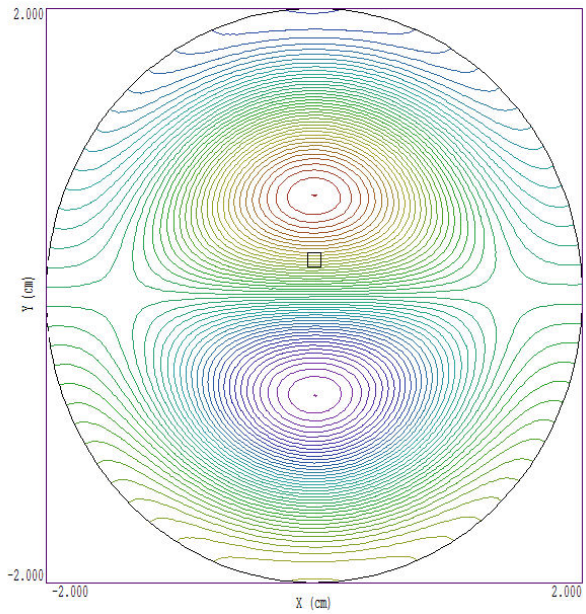


Figure 31: Electric field lines, TE_{02} mode at cutoff in a circular waveguide of radius $R = 2.0$ cm.

```
Range = (12.0E9 13.0E9)
FStep = 9
Tolerance 1.0E-7
```

It is also necessary to change the source current to

```
Source(2) = (-1.0, 0.0, 0.0)
```

so that the real part of the probe signal has a positive-to-negative transition. The **WaveSim** prediction for the mode cutoff frequency is $f_c = 12.711$ GHz. Electric field lines are plotted in Figure 31. The file `CIRCWAVEGUIDETM.WIN` illustrates how to generate TM type modes in the circular waveguide. The solutions are of type E with the primary field quantity is E_z. In this case, the condition on the metal wall is $E_z = 0.0$ and we must set it explicitly. The boundary (Region 3) has the property

```
Region(3) = Reflect
```

There are two other differences from the TE solution: 1) it is not necessary to supply an angle for the source current which points in z, 2) the imaginary part of the probe signal is non-zero because the electric field is 90°out of phase relative to the drive source.

We shall conclude this section with an example where a numerical solution is necessary. Suppose we want to propagate a TE_{01} wave in the circular waveguide and ensure that the electric field points in the x direction. In this case, we introduce a structure that has little effect on the mode when the electric field points along x, but significantly modifies the frequency when $\mathbf{E}$ points along y. Figure 32 shows a modified circular waveguide with thin fins. When the electric field points along x, the fins have little effect. They make a small change in the cross-section area of the guide, raising the cutoff frequency from 4.395 GHz to 4.516 GHz. We can generate a wave with y polarization of electric field with the source command:

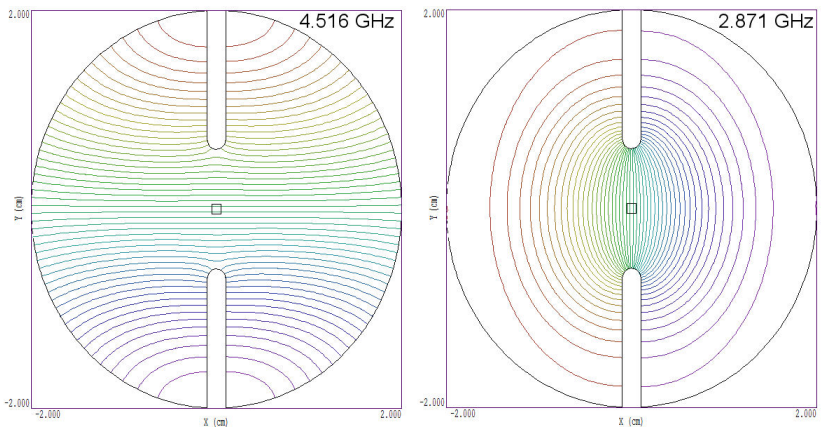


Figure 32: Electric field lines in a modified circular waveguide, with fins to shift the frequency of the TE_{01} mode, depending on the polarization of electric fields.

```
Source(2) = (1.0, 0.0, 90.0)
```

In this case, the fins significantly increase the capacitance for the mode, reducing the cutoff frequency to 2.871 GHz.

6.3 Resonant cavity with material losses

The example of this section demonstrates how to use **WaveSim** to find a resonant mode when a system includes material power loss. Figure 33 shows the geometry, a low-frequency synchrotron acceleration cavity with a tunable ferrite. The cavity is a figure of revolution about the z axis. The ferrite has relative dielectric constant $\epsilon_r = 5.0$ and complex relative magnetic permeability $\mu_r =$[200.0, 1.0].

We can estimate the frequency of the TM_{010} mode by viewing the area near the ferrite as a quarter wave resonator. The length is about 0.25 m and the velocity of propagation in the ferrite is $v = c/\sqrt{200} = 2.12 \times 10^7$ m/s. The single transit time through the ferrite is $\tau = 11.8$ ns. The resonant frequency is $f \cong 1/4\tau = 21$ MHz. Table 16 shows the **WaveSim** script for an initial resonant search with no material loss. The fundamental resonant mode occurs at 15.3105 MHz and has the electric field line distribution shown in Fig. 33.

To introduce material losses, the magnetic permeability for the ferrite is changed to

```
Mu(2) =  (2.0000E+02, 1.0000E+00)
```

If the ferrite completely filled the cavity volume, we expect a Q factor equal to

$$Q = \frac{\mu_r'}{\mu_r''} = 200. \tag{99}$$

Using the *Volume integral* command in the *Analysis* menu, we find that the 63% of the field energy is concentrated in the ferrite. Therefore, the Q value for the system of Fig. 33 should be close to the value of Eq. 99. The resonant search at low Q is more demanding because the reduced slope of the probe response. Therefore, we narrow the search range to 14.0 MHz $\leq f \leq$ 16.0 MHz and set $Tolerance = 1.0 \times 10^{-4}$. The resulting resonant frequency is 15.3108

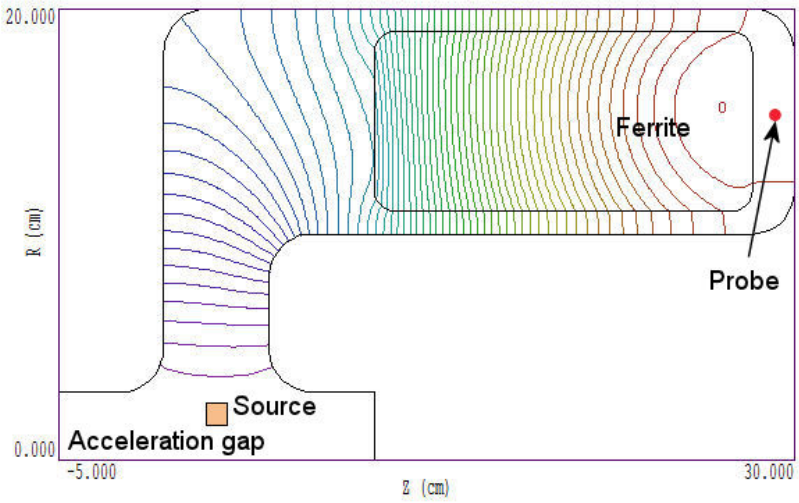


Figure 33: Geometry of the SYNCHCAV example, electric field lines at 15.31 MHz.

Table 16: File SYNCHCAV.WIN, frequency search with no material loss.

```
Mesh = synchcav
Geometry = Cylin
DUnit =   1.0000E+02
Solution = H
Mode = Search
Range =  10.0E6  25.0E6
FStep =   9
Tolerance =   1.0000E-05
* Region 1: VACUUM
Epsi(1) =   1.0000E+00    0.0000E+00
Mu(1) =   1.0000E+00    0.0000E+00
* Region 2: FERRITE
Epsi(2) =   5.0000E+00    0.0000E+00
Mu(2) =   2.0000E+02
* Region 3: SOURCE
Source(3) =   1.0000E+00      0.000   0.0
* Region 4: AXIS
Reflect(4)
EndFile
```

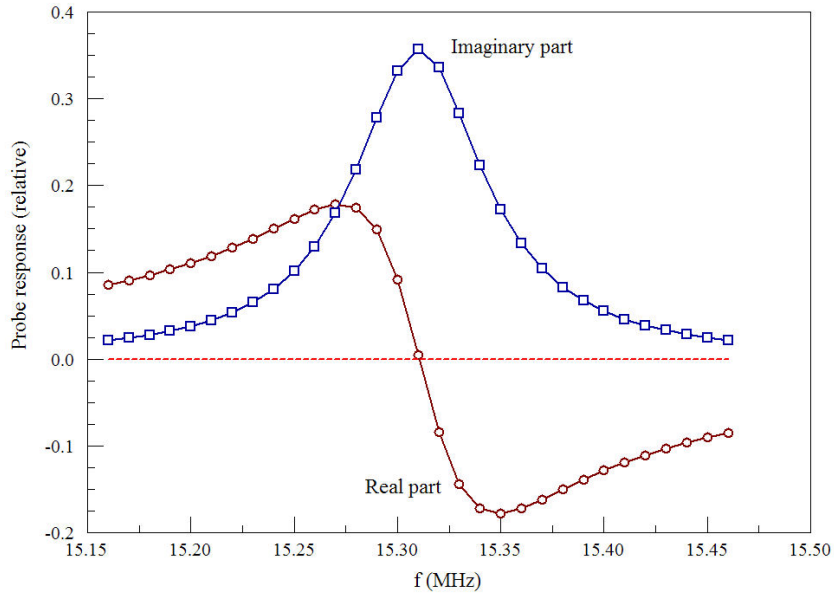


Figure 34: Probe response versus frequency for the `SYNCHCAV` example.

MHz, close to the value without losses. Again using the *Volume integral* command, we find that $U = 6.494 \times 10^{-6}$ J and $P = 3.091$ W (note that the absolute values may be different on your computer). Inserting the values in Eq. 69, we find that $Q = 202.1$.

We can check the results by running a frequency scan and comparing the response to the resonance curve of Fig. 18. The expected value of the frequency shift to reduce probe response to half its peak value is

$$\Delta f \cong \frac{f_0}{2Q} = \pm 37.9 kHz. \tag{100}$$

To create a plot we pick a scan range that covers 300 kHz centered at the resonance: 15.16 MHz to 15.46 MHz. The result, shown in Fig. 34, is a classic resonance response with the expected width. Note that the roles of the real and imaginary parts of the probe response are reversed compared to Fig. 18 because we are treating an H type solution.

7 Electromagnetic scattering tutorials

This section discusses **WaveSim** calculations in the *Scatter* mode. Here, electromagnetic waves generated by specified sources are either absorbed or propagate into free space. Typical calculations include transmission lines, antennas and wave scattering from objects. Absorbing boundaries play a significant role in *Scatter* solutions. This chapter contains four sections that illustrate different classes of solutions:

- **Coaxial transmission line**. The example shows how to generate TEM waves of specified amplitude in coaxial transmissions lines and how to define ideal terminations.
- **Two slit diffraction pattern**. The classic two-slit geometry illustrate the method for approximating a free-space boundary.
- **Electric dipole antennas**. The first example treats an ideal radiating dipole in free space, with comparisons to theory. The second example illustrates a practical example, the radiation resistance of a half-dipole antenna over a ground plane driven by a transmission line.
- **Magnetic dipole antenna**. How to set up a magnetic dipole antenna (circular current loop). Code performance when the wavelength is much large than the solution volume.
- **Wave scattering from objects**. The examples demonstrate how to apply the distributed source method for precision two-dimensional scattering calculations.

Input files for all examples are supplied in the **WaveSim** example library.

7.1 Coaxial transmission line

The example files have the prefix `TCONNECT`. Although the geometry is simple, the example introduces many useful techniques. For example, a coaxial transmission line is used to drive an antenna in Sect. 7.3. Figure35 illustrates the geometry. The coaxial transmission line has inner radius $R_i = 0.1$ m and outer radius $R_o = 0.23$ m. The characteristic impedance is

$$Z_0 = \frac{1}{2\pi}\sqrt{\frac{\mu}{\epsilon}}\ \ln\left(\frac{R_o}{R_i}\right). \tag{101}$$

In vacuum ($\epsilon = \epsilon_0$, $\mu = \mu_0$), the line dimensions give $Z_0 = 50\ \Omega$. The body of the transmission line in Fig. 35 is divided into two sections (Regions 3 and 4) so we can experiment with transitions between regions with different values of characteristic impedance. We will start with a calculation where the relative dielectric constant is $\epsilon_r = 1.0$ in both regions, and then change the value in the downstream region to $\epsilon_r = 4.0$.

We want to generate a traveling TEM wave at 600 MHz with a current amplitude of 1.0 A. This wave has field components E_r and H_θ, so we seek an H type solution. In this case, all external boundaries automatically assume the properties of a metal wall. A matched absorbing layer at the downstream boundary (Region 5) ensures that waves in Region 4 propagate only

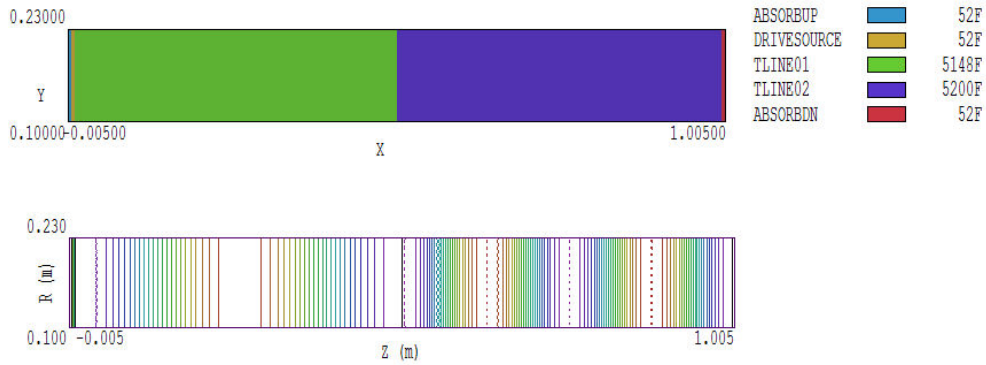


Figure 35: Example TCONNECT. *a*) Geometry showing region divisions. *b*) Electric field lines in a line with $\epsilon_r = 1.0$ in the left volume and $\epsilon_r = 4.0$ on the right.

in the $+z$ direction. The layer thickness ($\Delta = 0.005$ m) is much smaller than the wavelength of the radiation, $\lambda = 2\pi/f = 0.5$ m, so we expect good absorption. There are two layers at the left-hand boundary: Region 2 is a current source to create the wave and Region 1 is a termination layer to absorb reflected waves that may be created at the interface of Regions 3 and 4.

The main setup tasks are to calculate the amplitude of the drive current and values for ϵ_r'' in the upstream and downstream absorbers. The source layer (Region 2) has thickness $\Delta = 0.005$ m. A radial current creates waves with component E_r. The total current amplitude must equal 2.0 A because half is absorbed in the adjacent termination layer. According to Table 9, we must specify the quantity rj_r for an H type solution in cylindrical coordinates. The total radial current is given in terms of the weighted current density by

$$2\pi(rj_r)\Delta = I, \tag{102}$$

where $I = 2.0$ A. Inserting values for the example gives $rj_r = 63.66$ A/m. Regarding absorber properties, we must set an imaginary part of the magnetic permeability for an H type solution. Referring to Table 6, the value for a matched absorber of thickness Δ adjacent to a vacuum volume is

$$\mu_r'' = -\frac{377.3}{2\pi f \mu_0 \Delta}. \tag{103}$$

With parameter values for the example, we find that $\mu_r'' = -15.89$ for a layer adjacent to a vacuum region. Table 17 shows the complete input file.

When Regions 3 and 4 have $\epsilon_r' = 1.0$, we expect a pure traveling wave with equal power loss in both the upstream and downstream absorbers. The predicted value is $P = 377.3I^2/2 = 25.0$ W. The *Volume integral* command in the *Analysis* menu gives the following results:

Table 17: **WaveSim** input file `TCONNECT.WIN`, set up for a vacuum transmission line

```
Geometry = Cylin
DUnit = 1.00
Solution = H
Mode = Scatter
Freq = 600.0E6
* Upstream absorber
Epsi(1) = 1.0
Mu(1) = (1.0, -15.89)
* Drive region
Mu(2) = 1.0
Epsi(2) = 1.0
Source(2) = (63.66, 0.0, 90.0)
* Region 1 of transmission line
Epsi(3) = 1.0
Mu(3) = 1.0
* Region 2 of transmission line
Epsi(4) = 1.0
Mu(4) = 1.0
* Downstream absorber
Epsi(5) = 1.0
Mu(5) = (1.0, -15.89)
ENDFILE
```

```
  Integrals by region
NReg    Volume     Energy     Power
         (m3)       (J)        (W)
=====================================
   1  6.739E-04  1.178E-10 -1.119E+01
   2  6.739E-04  1.975E-10  0.000E+00
   3  6.671E-02  4.593E-08  0.000E+00
   4  6.739E-02  7.480E-08  0.000E+00
   5  6.739E-04  4.713E-10 -2.230E+01
```

A practical problem is how to be sure that we have picked the correct value of μ_r'' for the absorbing layers. If the setup is correct, there should be no component of standing wave in the line. If we use the *Line scan* command in the *Analysis* menu to determine $|E_r|$ at fixed radius, the plot should show a constant value. In the event of partial reflection, the standing wave components will introduce variations in $|E_r|$. In this case, the *standing wave ratio* for line voltage is defined as:

$$S = \frac{|E_{rmax}|}{|E_{rmin}|} = \frac{1 + |R_E|}{1 - |R_E|} \tag{104}$$

where R_E is the electric-field reflection coefficient defined in Eq. 27. The reflection coefficient is given in terms of the standing wave ratio by

$$|R_E| = \frac{S - 1}{S + 1}. \tag{105}$$

A scan at $r = 0.17$ m with the plot quantity $|E|$ gives $E_{rmax} = 361.6$ V/m and $E_{rmin} = 345.4$ V/m. The values imply that $S = 1.047$ and $R_E = 0.023$. Therefore, the termination layer absorbs 99.95% of the incident wave power. For detailed information of wave propagation in a transmission, see D.K. Cheng, **Field and Wave Electromagnetics, 2nd Edition** (Addison-Wesley, Reading, 1992), 461.

To complete the exercise, we shall set $\epsilon_r' = 4.0$ in the downstream region of the transmission line, reducing the characteristic impedance to $Z_0 = 188.7\,\Omega$. We must also change the properties of the downstream absorber to match the adjacent medium:

```
Epsi(5) = 4.0
Mu(5) = (1.0, -7.943)
```

Figure 35*b* shows electric field lines. Note that the wavelength in the downstream region is half the value in the upstream region. There will be partial wave reflection at the junction of the media. The electric field reflection coefficient predicted by Eq. 27 is $R_E = -0.333$; therefore, we expect to see a standing wave in Region 3 with $S = 2.0$. A scan over Region 3 for the modified calculation gives the following values a $r = 0.17$ m: $E_{rmax} = 470.1$ V/m and $E_{rmin} = 236.5$ V/m, corresponding to $S = 1.988$. The transmitted wave should have $V = 33.33$ V and $I = 0.333$ A; therefore, the predicted power dissipation in the downstream absorber is 11.10 W. The calculated WaveSim value is 11.18 W.

7.2 Two slit diffraction pattern

The `SLITDIFF` example illustrates the definition of a free-space boundary and exercises the ability of code to handle complex two dimensional geometries. The example treats the familiar two-slit diffraction pattern, where a plane wave passing through narrow slits in a metal plate. The numerical method allows us to see features that are not represented in the simple theory:

- the effect of the metal plate on the near fields, and
- the transition to the far field pattern for slits of finite width.

Will study diffraction of microwaves at frequency 18.75 GHz ($\lambda = 1.6$ cm). **WaveSim** can handle optical frequencies equally as well – it is simply a matter of scaling.

Figure 36 shows planar geometry described by `SLITDIFF.MIN`. The bulk of the solution volume is a half-cylinder centered on average position of the slits. The choice ensures that propagating waves intersect the boundary at approximately normal incidence. We first fill the entire solution volume with a medium that will constitute absorber (Region 1) and then overwrite the central portion with a vacuum medium (Region 2). The process leaves a thin shell of absorber ($\Delta = 0.05$ cm) on the circular boundary. Regions 3 and 4 overwrite nodes on the left-hand boundary to define strips of height 0.10 cm with centers separated by a distance $D = 2.0$ cm.

If a plane wave moving in $+x$ is incident on the left-hand boundary, then slits with orientation along z will preferentially transit the polarization component with E_y and H_z. Therefore, we shall seek an H type solution. In this case, unspecified external boundaries assume the properties of metal. Therefore, the left-hand boundary acts like a conducting plate. Table 18 shows the input file `SLITDIFF.WIN`. The value of μ_r'' in the absorbing layer (Region 1) was determined from Table 6 with $\Delta = 5.0 \times 10^{-4}$ m. The nodes of the slits (Regions 3 and 4) are set to the simple drive condition $H_z = 1.0 \cos(2\pi f t)$. This specification is adequate because wave travel away from the drive and do not return. In simulations with reflected waves, it is better to excite waves with a current source adjacent to an absorbing layer (Sect. 7.1) to avoid spurious reflections. Figure 36 shows an element plot of $|H_z|$. For comparison, the predicted angle (with respect to the x axis) of the first minimum in the far field region is given by

$$\sin(\theta) \cong \frac{\lambda}{2D}. \tag{106}$$

For the parameters of the example, $\theta = 23.6^{\circ}$.

7.3 Electric dipole antennas

An antenna is a current-carrying structure that radiates electromagnetic energy, generally into free space. This section discusses a benchmark calculation and a practical application of **WaveSim** to calculate the properties of electric dipole antennas. The first example, `DIPOLE`, addresses radiation by an ideal dipole. Figure 37 shows the geometry. A region of axial oscillating current is located inside an air-filled, spherical anechoic chamber of radius 100 cm. An absorbing layer of thickness 1.0 cm covers the boundary. The current-carrying region with the material properties of air has radius $R = 1.0$ cm and length $L = 6.0$ cm. The current

Table 18: WaveSim input file SLITDIFF.WIN

```
Geometry = Rect
DUnit = 100.0
Solution = H
Mode = Scatter
Freq = 1.875E10
Mu(1) = (1.0, -5.084)
Epsi(1) = 1.0
Mu(2) = 1.0
Epsi(2) = 1.0
Drive(3) = (1.0, 0.0)
Drive(4) = (1.0, 0.0)
EndFile
```

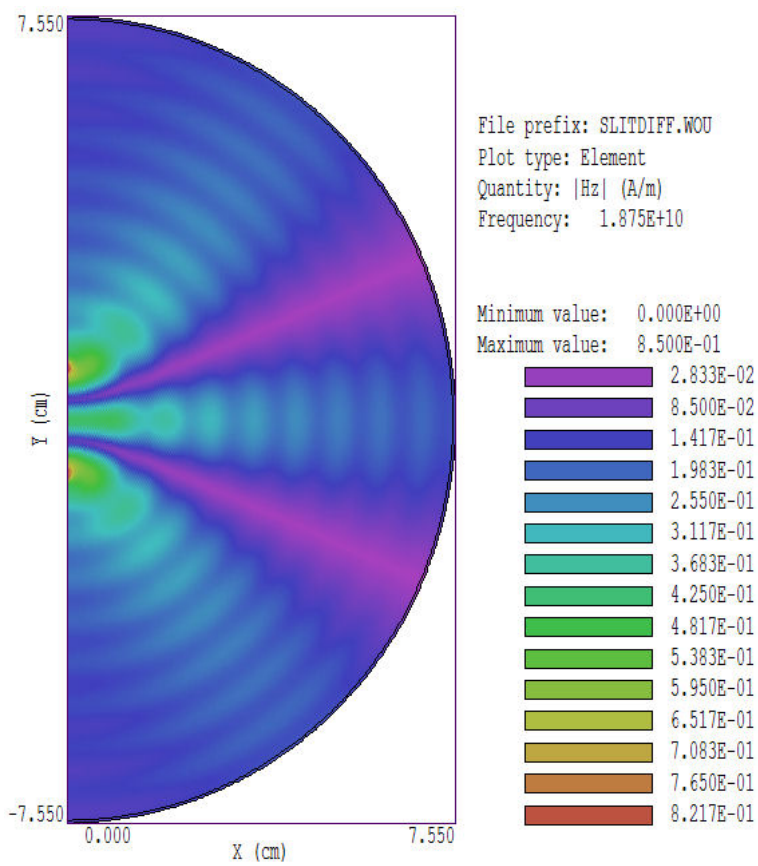


Figure 36: Example SLITDIFF, plot of $|H_z|$.

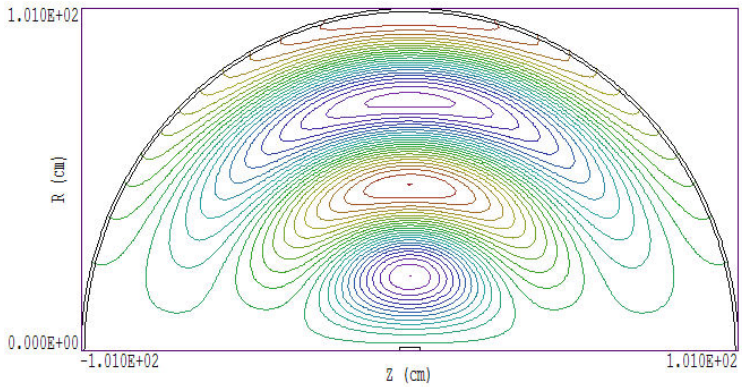


Figure 37: Electric field lines from a dipole current, $f = 600$ MHz, phase $\phi = 90^o$.

creates a component H_θ of magnetic field. Therefore, we seek an H type solution in cylindrical coordinates. The magnitude of the total current is related to the weighted current density by

$$I_0 = \int_0^R dr 2\pi (r j_r) = 2\pi R (r j_r). \tag{107}$$

For $I_0 = 1.0$ A, Eq. 107 implies that $(rj_r) = 15.92$ A/m. We shall first perform a high-frequency calculation at 600 MHz ($\lambda = 50$ cm). Referring to Table 6, the imaginary part of the relative magnetic permeability in the absorbing layer should have the value $\mu_r'' = -7.944$. The reference S. Ramo, J. Winnery, and T. Van Duzer, **Fields and Waves in Communication Electronics** (Wiley, New York,1965), 645 gives the following expression for the total radiated power:

$$U = \frac{377.3\pi I_0^2}{3} \left(\frac{L}{\lambda}\right)^2. \tag{108}$$

The predicted value at $f = 600$ MHz is 5.69 W.

Figure 37 shows calculated electric field lines at phase $\phi = 90^o$. The *Volume integral* command in the *Analysis* menu gives an integral of power over the absorbing layer of 5.485 W, close to the prediction. We can use this example to demonstrate that **WaveSim** generates accurate solutions, even when the free-space boundary is in the near-field region. We lower the frequency to 100 MHz ($\lambda = 3.0$ m) and change the imaginary magnetic permeability of the absorber to $\mu_r'' = -47.664$. Figure 38 shows resulting contours of H_θ at $\phi = 0^o$. The values of H_θ on the outside of the source region are close to the static-field prediction. Although the near-field region occupies the entire solution volume, the radiated power absorbed in termination layer agrees with theory. Prediction of Eq. 108 at 100 MHz is 0.1581 W compared to the **WaveSim** value of 0.1559 W.

Next, consider a practical example of a metal antenna driven by a transmission line. Figure 39 shows a 1.5 GHz quarter wave cartop antenna on a mount. (In the cylindrical geometry, the top of the antenna is on the right-hand side.) Following S. Ramo, J. Winnery, and T. Van Duzer, **Fields and Waves in Communication Electronics** (Wiley, New York,1965), Sect. 12.07, the ideal quarter wave antenna above a ground plane has length 5.5 cm and radiation resistance $R = 36.54\Omega$. The power generated with a normalized current input $I_0 = 1.0$ A is 18.27 W.

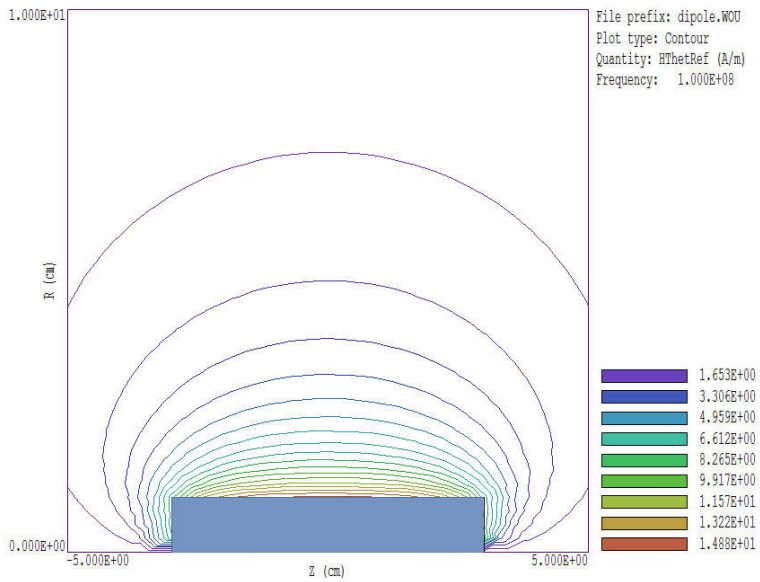


Figure 38: Contours of $|H_\theta|$ at $\phi = 0^o$ for the `DIPOLE` example at 100 MHz.

The file `QUATWAVE.MIN` defines the geometry. Region 1, shown as a colored band in Fig. 39 is a thin ($\Delta = 0.1$ cm) absorbing layer on the surface of a hemispherical anechoic chamber of radius 15.0 cm. Region 2, the shaded area in Fig. 39, has the properties of air. It outlines all the major objects in the simulation (propagation volume, shaped ground plane, inner and outer conductors of the transmission line and the antenna). In the H type solution, the boundaries in contact with void elements or the edge of the solution volume act as metal walls. Region 3 is a small absorbing layer at the end of the transmission line and Region 4 contains a radial current to generate a TEM wave (Sect. 7.1). Both regions have axial thickness $\Delta = 0.1$ cm. Finally, Region 5 is a line on axis between the tip of the antenna and the right-hand edge of the solution volume that will assume the property $rH_\theta = 0.0$.

In this demonstration calculation, we assume that transmission line is air-filled with dimensions that give a characteristic impedance that matches the expected radiation resistance. A detailed design study would probably use standard dielectric-filled transmission lines with a transition that minimizes power reflection at the antenna. With the choice $R_o = 0.5$ cm and $R_i = 0.272$ cm, Eq. 101 gives a characteristic impedance $Z_0 = 36.56\ \Omega$. Following the methods described in previous sections, we determine that the absorbers should have the relative magnetic permeability $\mu_r = [1.0, -31.77]$. We pick a total source current of 2.0 A, half of which is lost to the transmission line absorber. The associated source current density is $rj_r = 318.3$ A/m.

Figure 39 shows the time-averaged radiation power density absorbed in the outer layer. The variation follows the familiar half-wave dipole pattern with a peak in the direction normal to the antenna. Electric field lines in transmission line and radiated into space are plotted in Fig. 40. In an ideal solution, we expect to observe that the spatial integral of power loss in both absorbing regions of 18.27 W. The *Volume integral* command gives values of 16.51 W for the radiated power and 13.01 W in the line absorber. The antenna performance is good, but clearly there is some wave reflection at the transition. The interference of initial and reflected waves at the line absorber reduces the current amplitude. To search for optimum conditions,

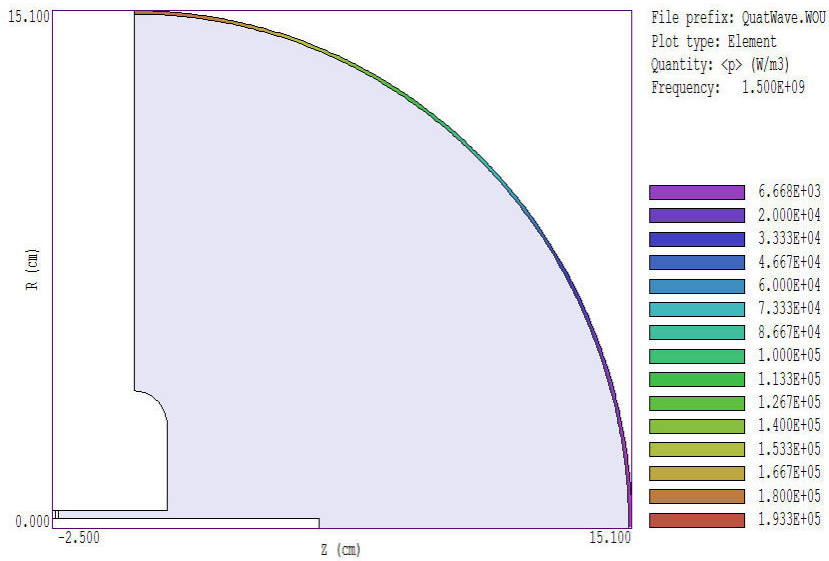


Figure 39: Geometry of the `QUATWAVE` example. The air volume is shaded in blue. The absorption layer is color-coded by time-averaged power density to illustrate the radiation patter.

the length of the antenna was varied keeping the frequency and transmission line properties fixed. The radiated power versus antenna length is plotted in Fig. 41. The optimum length of 5.6 cm is close to $\lambda/4$.

7.4 Magnetic dipole antenna

The simplest form of a magnetic dipole antenna is a circular current loop with an RF drive current. Figure 42 shows the setup for a **WaveSim** model defined by the file `MAGDIPOLE.MIN`. A toroidal loop with minor radius $r_c = 1.5$ cm and major radius $R_c = 10.0$ cm carries a uniformly distributed oscillating current with peak amplitude $I_0 = 1.0$ A. The current moves in the θ direction and generates field components E_θ, H_z and H_r. Therefore, the solution is of type E. The antenna is located at the center of a spherical vacuum volume of radius 100.0 cm. The absorbing layer on the outside has thickness 0.5 cm.

The file `MAGDIPOLE.WIN` covers a range of frequencies from 62.5 MHz ($\lambda = 4.80$ m) to 500.0 MHz ($\lambda = 0.60$ m). For an E type solution in cylidrical coordinates, the quantity in the *Source* command for the loop region is rj_θ, given approximately by

$$rj_\theta \cong I_0 \, \frac{R_c}{\pi r_c^2}. \tag{109}$$

Using values for the simulation geometry, we find that $rj_\theta = 141.5$ A/m.

Figure 42 shows magnetic field lines at $\lambda = 0.60$ m (top) and $\lambda = 2.4$ m (bottom). The high-frequency result exhibits the standard dipole field pattern, while the bottom approaches the field distribution of a static current loop. In the limit $2\pi R/\lambda \ll 1.0$, the radiated power is given by

$$P = \frac{4\pi^5}{3} \sqrt{\frac{\mu_0}{\epsilon_0}} \left(\frac{R}{\lambda}\right)^4 I_0^2. \tag{110}$$

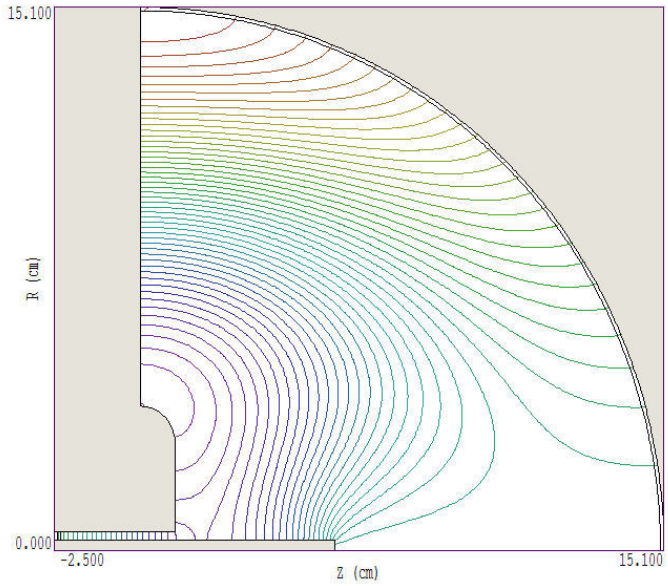


Figure 40: Example `QUATWAVE`, electric field lines

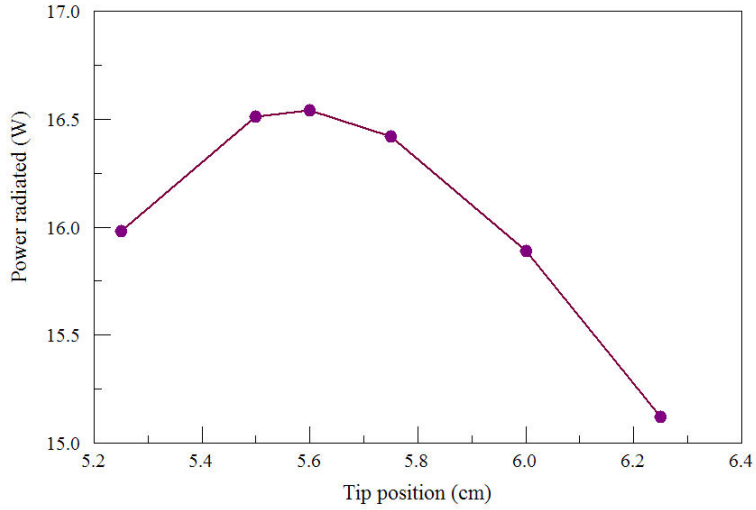


Figure 41: Example `QUATWAVE`, radiated power as a function of the antenna length

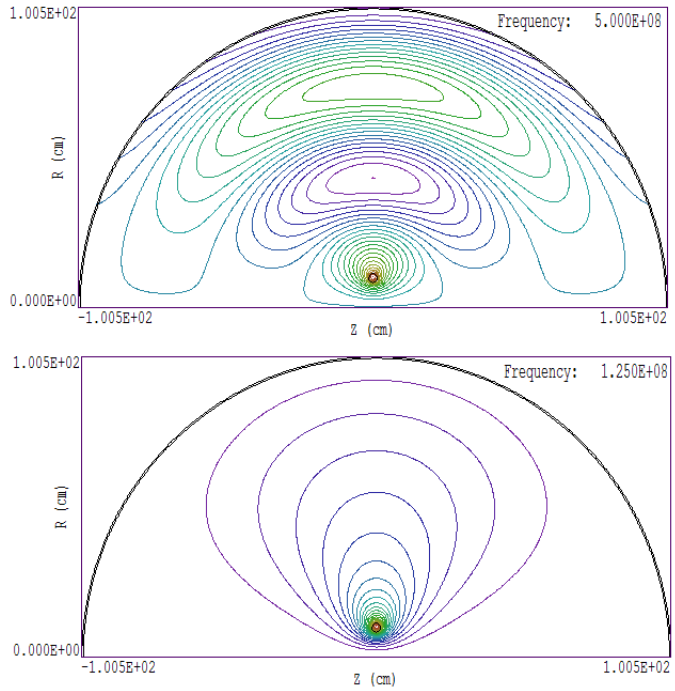


Figure 42: Magnetic dipole antenna model, lines of $|\mathbf{H}|$. Circular wire loop inside a spherical anechoic chamber.

We can find the radiated power from the average power deposition in the absorbing layer. Figure 43 shows a plot of radiated power versus wavelength. The dashed red line is a prediction fronm Eq. 110. Over a wide range, the results from **WaveSim** are in good absolute agreement with theory. The deviation at short wavelength probably results from limitations of the analytic model. The discrepancy at long wavelength reflects numerical errors in evaluating small electric fields and non-ideal performance of the absorbing layer.

7.5 Wave scattering from objects

Electromagnetic scattering calculations generally involve an incident plane wave interacting with objects. In this application it is important to recognize a limitation of two-dimensional codes. Plane waves do not have azimuthal symmetry about an axis of propagation. Therefore, plane waves may be included only in calculations of the type:

```
Geometry = Rect
```

In other words, **WaveSim** can handle scattering from a cylindrical rod, but not from a sphere.

High-accuracy calculations of electromagnetic scattering present one the great challenges for finite-element solutions. Plane waves must be represented in a bounded volume in a way that allows identification of the relatively weak scattered signal. **WaveSim** achieves unprecedented performance through the use of the distributed source method discussed in Sect. 3.10 and in the reference S. Humphries, *Scattering simulations in inhomogeneous volumes for Scanning Near-field Optical Microscopy*, J. Modern Optics **47**, 805 (2000). The examples in this section illustrate features of the method and techniques for scattering solutions in **WaveSim**.

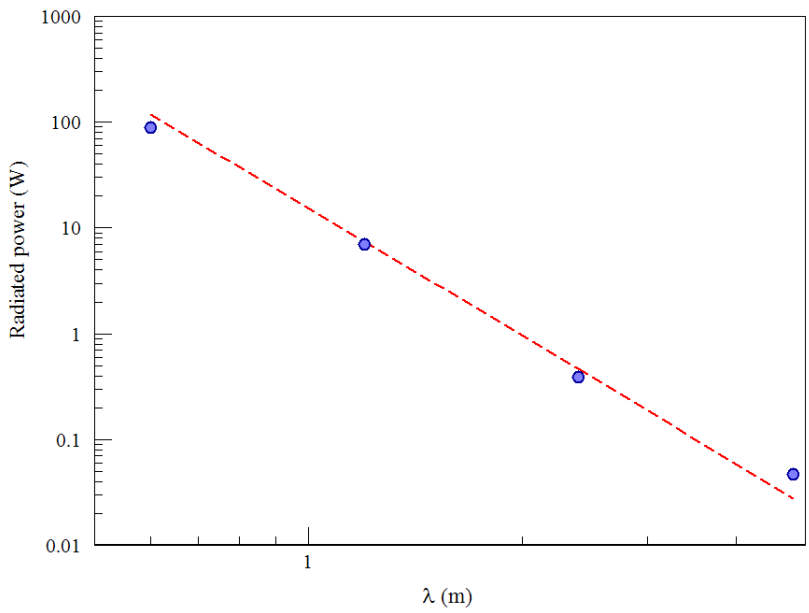


Figure 43: Magnetic dipole antenna, radiated power as a function of vacuum wavelength. Dashed line: theory for a small loop. Points: **WaveSim** calculations.

In electromagnetics even the simplest solutions may be interesting and informative. The `WIRE` example illustrates the basic steps in a scattering calculation. The file `WIRE.MIN` defines a cylindrical vacuum region of radius 5.0 cm with a termination layer of thickness $\Delta = 0.05$ cm. The absorbing layer is divided into two parts (Region 1 on the right and Region 2 on the left) to allow identification of the power carried by backscattered radiation. The scattering object is a metal rod on axis with radius $R = 0.5$ cm. Table 19 shows the **WaveSim** input file `WIREE.WIN`, set up for an incident plane wave polarized with the electric field parallel to the wire. At frequency $f = 20.0$ GHz, the wavelength is $\lambda = 0.015$ m and the wavenumber is $k = 418.9$ m^{-1}. The *PlaneWave* command signals that the code should perform a distributed source calculation and defines a plane wave with amplitude $E_0 = 1.0$ V/m traveling in the $+x$ direction. The other new command,

```
NReplace(4) = 3
```

specifies that the scattering object (Region 4) should be replaced with the properties of vacuum (Region 3) for the calculation of the distributed source terms. For the E type solution, the values $\epsilon' = 1.0 \times 10^12$ and $\mu' = 1.0$ in Region 4 give the boundary conditions for a perfect conductor.

Figure 44 shows a plot of $|E_z|$ for the total field pattern. (The total field output file was created by including the letter T in the *PlaneWave* command.) In the absence of the scattering object, the electric field amplitude would be uniform in space (*i.e.*, a pure traveling wave). The metal object creates the standing wave variations shown in Fig. 44. The is constructive interference on the front side at distance $\lambda/2$ from the object and a diffraction shadow. We can calculate the backscatter cross section by saving only the scattered field and again using *Volume integral* command. The average power dissipation in Region 2 (upstream absorber) is $P = -1.132 \times 10^{-05}$ W/m. The time-averaged power flux in the plane wave is

Table 19: WaveSim input file `WIREE.WIN`

```
Geometry = Rect
Mesh = Wire
DUnit = 100.0
Solution = E
Mode = Scatter
Freq = 2.0E10
PlaneWave = (418.9, 0.00, 1.00)
* Absorber front
Mu(1) = 1.0
Epsi(1) = (1.0, -4.766)
* Absorber rear
Mu(2) = 1.0
Epsi(2) = (1.0, -4.766)
* Vacuum region
Mu(3) = 1.0
Epsi(3) = 1.0
* Wire
Mu(4) = 1.0E-6
Epsi(4) = 1.0E6
NReplace(4) = 3
EndFile
```

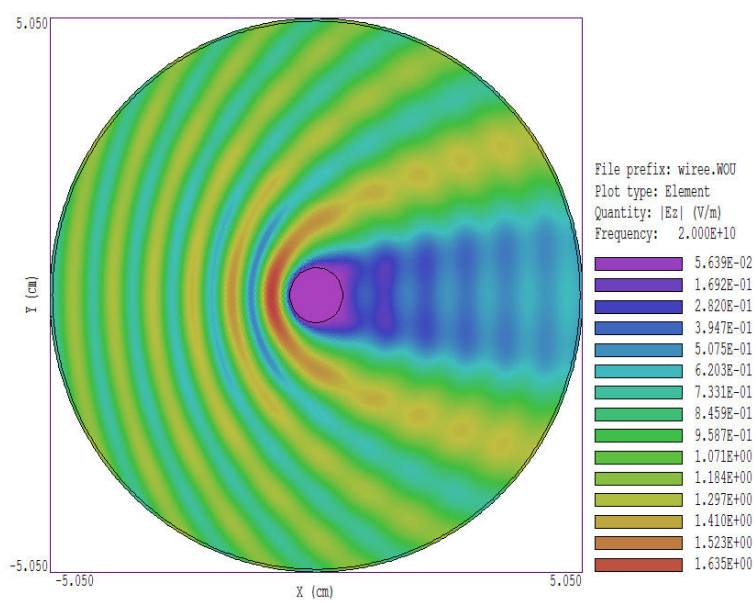


Figure 44: Example WIREE, plot of $|E_z|$, total fields.

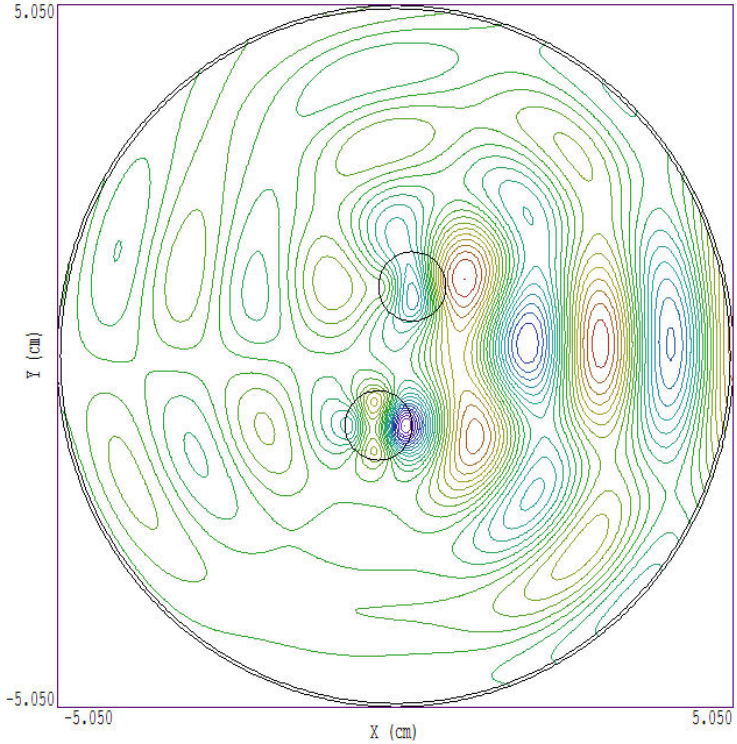


Figure 45: Example DOUBLEBAR, **H** lines of the scattered field.

$$S = \frac{E_0 H_0}{2} = \frac{E_0^2}{2 \times 377.3} = 1.32 \times 10^{-3} \text{ W/m}^2. \tag{111}$$

The backscatter cross-section (the effective height of the object) is therefore $1.132 \times 10^{-5}/1.32 \times 10^{-3} = 0.00858$ m $= 0.858$ cm, slightly less than the physical height. The file WIREH.WIN illustrates how to set up the same problem with the electric field of the incident wave normal to the axis of the wire. Again, the values $\epsilon' = 1.0 \times 10^12$ and $\mu' = 1.0$ are used to set properties of a perfect conductor for Region 4.

The example DOUBLEBAR illustrates a setup with multiple dielectric objects. Again, we use a cylindrical anechoic chamber with 5 cm radius and an absorbing layer with thickness $\Delta = 0.05$ cm. The incident plane wave has frequency at $f = 15.0$ GHz ($\lambda = 2.0$ cm) and electric field along z. Scattering is from two dielectric rods of radius 0.5 cm with $\epsilon'_r = 5.0$. The rods have vertical separation 2.0 cm and horizontal separation 0.5 cm ($\lambda/4$). The spacing was chosen so that the interference of the reflected signals minimized direct backscattered radiation. The difference from the previous run is that an *NReplace* command is required for both object regions. Figure 45 shows the resulting scattered field distribution. Note the reduced wavelength and concentration of field energy in dielectrics, the strong forward signal representing the diffraction shadow and the small field amplitude in the backward direction.

Index

absorbing boundary, 30, 33, 45, 55, 84
analytic solutions, 5
antenna, 87, 91

bisection search method, 37

circuit impedances, 36
command prompt, 56
comment lines, 50
computational mesh, 6
conductivity, 31, 55
conformal mesh, 6
coupled resonant cavities, 41

delimiters, 49
dielectric constant, 23
    complex, 32, 54
    water, 32
diffraction pattern, 87
distributed source method, 4, 46, 53, 56, 93

E type wave, 24, 91
electric dipole antenna, 87
electromagnetic energy, 21, 33, 64
electromagnetic power loss, 21, 33, 64, 91
electromagnetic scattering, 45, 83, 93
element, 6

facet, 6
false position search method, 39
field
    definition, 5
    equations, 5
field lines, 19, 75
field normalization, 20
finite-element method, 5, 6
    basis, 6
free-space boundary, 4

H type wave, 24

impedance, 25
impedance match, 26, 30

klystron, 9

magnetic dipole antenna, 91
magnetic permeability, 23
    complex, 32, 54, 80, 84
material power loss, 32
Maxwell equations, 23
    conformal mesh, 27
Mesh
    drawing editor, 10
    filled regions, 10
    open regions, 10
    plots, 11
Mesh commands
    Load script (MIN), 10
    Save mesh (MOU), 11

node, 6
node equations, 6, 7
normal incidence, 31
numerical solutions, 5

open circuit condition, 26

phase velocity, 25
pillbox cavity modes, 39
plane wave, 24, 46
polarization, 24
Poynting
    theorem, 32
primary field component, 24

Q factor, 9, 19, 36, 41, 77
    expression for, 21

radiation cross section, 96
radiation resistance, 89, 90
reference phase, 20, 69
reflection coefficient, 26, 86
resonance width, 37, 82
resonant cavity, 34, 39
    example, 73
    power loss, 41
resonant frequency, 9, 34, 75
resonant mode location, 36
resonant response, 34, 73, 77

secondary field component, 24
short circuit condition, 26
skin depth, 45
standing wave ratio, 86
surface resistance, 31, 44
surface resistivity, 21
symmetry
    cylindrical, 6
    planar, 6

TEM wave, 84
termination layer, 4
text editor
    internal, 56, 61
TM010 mode, 15, 73
transmission coefficient, 26
transmission line, coaxial, 83
traveling wave, 25, 86

wall losses, 21, 43, 67, 76
wave number, 25
wave reflection, 25
waveguid, 39
waveguide
    cutoff modes, 77
WaveSim
    analysis script, 20, 71
    applications, 4
    command prompt, 56
    file format, 57
    file types, 8
    frequency scan, 15
    input files, 49
    instructions, 57
    learning, 4
    listing file, 13, 15
    material models, 4
    output file format, 57
    plot quantities, 62
    plot types, 61
    resonance search, 15
    running in a window, 56
    script, 11
    script dialog, 56
    script format, 49
    setup dialog, 11
    solution modes, 51
    solution options, 14
    solution procedure, 7
    solution steps, 11, 49
    units of physical quantities, 60
WaveSim commands
    AbsLayer, 34, 55
    Analyze, 19, 57
    Close data record, 61
    Cond, 55
    Contour style, 65
    Create script, 61
    Drive, 14, 54
    DUnit, 13, 52
    Edit file, 61
    Edit script (WIN), 56
    Element properties, 67
    Epsi, 54
    Freq, 13, 52
    FStep, 13, 52
    Geometry, 13, 52
    Grid control, 64
    Grid display, 64
    Interpolation method, 69
    Line scan, 67
    Load solution file, 60
    LowQ, 53
    Manual, 57
    Matrix file, 69
    Mesh, 51
    Mode, 13, 51
    NReplace, 56
    Open data record, 61
    Oscilloscope mode, 70
    Planewave, 53
    Plot limits, 64
    Plot quantity, 62
    Plot type, 62
    Point calculation, 67
    Probe, 13, 53
    Range, 13
    Reference phase, 69
    Reflect, 14, 54
    Region properties, 67
    Renormalize fields, 20, 69
    Run, 57

Run analysis script, 56
Run script, 20, 61
Save solution file, 60
Scan plot quantity, 69
Setup, 57
Snap, 65
Solution, 13, 51
Source, 14, 55
Stop, 57
Tolerance, 52
Void, 14, 54
Volume integral, 67