Weak form finite element formulation for the Helmholtz equation

Lian Yuh Tio, Andrew A. P. Gibson, Bernice M. Dillon and Lionel E. Davis
Electromagnetics Centre for Microwave and Millimetre-wave Design & Applications, Department of Electrical Engineering and Electronics, University of Manchester Institute of Science and Technology (UMIST), Manchester, UK
Email: lianyuh@iee.org

Abstract Modern numerical software packages have potential as development tools for research and educational purposes. One such package is FEMLAB®, a finite element based environment with a user-defined, flexible equation solver. This paper describes the implementation and use of a vector Helmholtz solver in FEMLAB® for educational purposes.

Keywords FEMLAB®; finite element method; Helmholtz equation; weak form

Numerical modelling is unquestionably a major part of engineering design and analysis in the 21st century. Over the past ten years in particular, modelling based on finite element, finite volume, and finite difference analyses has progressively transformed the methods that engineers use in prototype design, research, and development.[1–7] Once the exclusive domain of researchers skilled in numerical science, these advanced modelling methods are now available for engineers and educators at all levels through ready-to-use software packages. State of the art commercial computer aided design (CAD) software for high frequency design problems includes field solvers, equivalent circuit solvers and some thermal analysis software.[7]

FEMLAB® by Comsol is mainly promoted as a research tool for solving partial differential equations (PDEs), in science and engineering.[1] It is a finite element based program with in-built mesh generator, solvers and post-processing facilities. Coupled problems can also be tackled using a multi-physics feature, which allows the flexible use of solutions from different physical equations. Unlike many other CAD software packages it is an open environment where the user can define problem-specific variational equations, which are then processed and solved using the built-in finite element routines.

FEMLAB's inherently modular and open environment also provides an opportunity for augmenting educational courses on numerical methods with practical applications. Problems can be set which allow the student to start by transforming a boundary value problem into a variational expression, before using FEMLAB® to solve the problem and to visualise the solutions. The student can examine the effects of boundaries and materials and explore a range of matrix solvers and applications. The purpose of this paper is to demonstrate how this can be done for the straightforward example of the vector Helmholtz equation in electromagnetic wave propagation.

Boundary value problem

Wave propagation in a dielectric loaded waveguide is governed by the Helmholtz equation and boundary conditions. The modal propagation characteristics of the waveguide can be calculated by describing the boundary value problem in terms of equations with transverse and axial field component variables. It is most convenient for practical purposes if the problem is formulated so that the solutions are evaluated at a specified frequency, with the propagation constants of the modes as the eigenvalues.

Adopting the usual convention, the wave is assumed to propagate in the z-direction ($e^{-j\beta z}$) with a time harmonic dependency ($e^{j\omega t}$) where ω is the real angular frequency. The analysis starts from Maxwell's curl equations,

$$\nabla \times \boldsymbol{E} = -j\omega\mu_o\mu_r\boldsymbol{H} \tag{1}$$

$$\nabla \times \boldsymbol{H} = j\omega\varepsilon_o\varepsilon_r\boldsymbol{E} \tag{2}$$

where ε_r and μ_r are the relative permittivity and permeability; ε_o and μ_o are the scalar vacuum permittivity and permeability constants respectively. By taking the curl of both eqns (1) and (2), the electric field vector $\boldsymbol{E}$ can be eliminated to form the magnetic field double-curl equation as[2,3,4,8,9]

$$\nabla \times \frac{1}{\varepsilon_r}\nabla \times \boldsymbol{H} - \omega^2\varepsilon_o\mu_o\mu_r\boldsymbol{H} = 0 \tag{3}$$

Equation (3) is decomposed into a transverse component,

$$\nabla_t \times \frac{1}{\varepsilon_r}\nabla_t \times \boldsymbol{H}_t - j\beta\frac{1}{\varepsilon_r}\left(\nabla_t H_z + j\beta\boldsymbol{H}_t\right) - \omega^2\varepsilon_o\mu_o\mu_r\boldsymbol{H}_t = 0 \tag{4}$$

and a z-directed longitudinal component as

$$\nabla_t \times \frac{1}{\varepsilon_r}\left(\nabla_t H_z + j\beta\boldsymbol{H}_t\right) \times \boldsymbol{z} - \omega^2\varepsilon_o\mu_o\mu_r H_z\boldsymbol{z} = 0 \tag{5}$$

where $\partial/\partial z = -j\beta$. These component equations are clearly coupled in $\boldsymbol{H}_t$ and H_z and ω^2 is in the eigenvalue position. In order to calculate phase constant (β) as the eigenvalue a simple variable transformation is introduced,[2]

$$\boldsymbol{h}_t = \beta\boldsymbol{H}_t \tag{6}$$

$$h_z\boldsymbol{z} = -jk_oH_z\boldsymbol{z} \tag{7}$$

Substituting eqns (6) and (7) into eqn (4) yields

$$\nabla_t \times \frac{1}{\varepsilon_r}\nabla_t \times \boldsymbol{h}_t - k_o^2\mu_r\boldsymbol{h}_t = -\frac{\beta^2}{\varepsilon_r}\left(\nabla_t\frac{h_z}{k_o} + \boldsymbol{h}_t\right) \tag{8}$$

and similarly for eqn (5)

$$\beta^2\left[\frac{1}{k_o}\nabla_t \cdot \frac{1}{\varepsilon_r}\left(\nabla_t\frac{h_z}{k_o} + \boldsymbol{h}_t\right) + \mu_r h_z\right] = 0 \tag{9}$$

The final pair of coupled differential eqns (8) and (9) can now be solved for β^2, subject to the following boundary conditions:

$$\begin{aligned} \boldsymbol{n} \times \boldsymbol{h_t} &= 0 \\ h_z &= 0 \end{aligned} \tag{10}$$

on perfect magnetic conductors (PMC), and

$$\begin{aligned} (\nabla_t h_z + \boldsymbol{h_t}) \cdot \boldsymbol{n} &= 0 \\ \nabla_t \times \boldsymbol{h_t} &= 0 \end{aligned} \tag{11}$$

on perfect electric conductors (PEC).

Variational formulation

To obtain a weak variational formulation from the boundary value problem defined by eqns (8) and (9) and boundary conditions (10) and (11), each partial differential equation is multiplied by a *test function* and the result integrated over the waveguide cross-sectional area. Boundary conditions (10) and (11) are applied to the result to produce the final formulation or *functional equation*. In general, the functional is an energy-related integral, whose stationary point yields the correct field solution for the given boundary value problem.[10] In this case, eqn (8) is dot-multiplied with a test function, $\boldsymbol{h}_t^*$, and then integrated by parts over the whole sub-domain of the problem to give

$$\int_s \boldsymbol{h}_t^* \cdot \left[\nabla_t \times \frac{1}{\varepsilon_r} \nabla_t \times \boldsymbol{h}_r - k_o^2 \mu_r \boldsymbol{h}_t \right] ds = \int_s \left[-\boldsymbol{h}_t^* \cdot \left(\frac{\beta^2}{\varepsilon_r} \left(\nabla_t \frac{h_z}{k_o} + \boldsymbol{h}_t \right) \right) \right] ds \tag{12}$$

Similarly, the longitudinal component eqn (9) is scalar multiplied with a test function, h_z^*

$$\int_s h_z^* \beta^2 \left[\frac{1}{k_o} \nabla_t \cdot \frac{1}{\varepsilon_r} \left(\nabla_t \frac{h_z}{k_o} + \boldsymbol{h}_t \right) + \mu_r h_z \right] ds = 0 \tag{13}$$

By integrating by parts and applying Green's theorems,[4] eqns (12) and (13) can be simplified to

$$\int_s \left[\nabla_t \times \boldsymbol{h}_t^* \cdot \frac{1}{\varepsilon_r} \nabla_t \times \boldsymbol{h}_t \right] ds - \int_s k_o^2 \mu_r \boldsymbol{h}_t^* \cdot \boldsymbol{h}_t ds = \int_s \left[-\boldsymbol{h}_t^* \cdot \left(\frac{\beta^2}{\varepsilon_r} \left(\nabla_t \frac{h_z}{k_o} + \boldsymbol{h}_t \right) \right) \right] ds \tag{14}$$

$$-\int_s \frac{\beta^2}{\varepsilon_r} \left[\left(\nabla_t \frac{h_z^*}{k_o} \right) \cdot \left(\nabla_t \frac{h_z}{k_o} \right) \right] ds - \int_s \frac{\beta^2}{\varepsilon_r} \left[\left(\nabla_t \frac{h_z^*}{k_o} \right) \boldsymbol{h}_t \right] ds + \int_s \mu_r \beta^2 h_z h_z^* ds = 0 \tag{15}$$

Re-combining these equations produces a full vector functional as,

$$\begin{aligned} &\int_s \left[\nabla_t \times \boldsymbol{h}_t^* \cdot \frac{1}{\varepsilon_r} \nabla_t \times \boldsymbol{h}_t \right] ds - \int_s k_o^2 \mu_r \boldsymbol{h}_t^* \cdot \boldsymbol{h}_t ds \\ &= -\beta^2 \int_s \left[\left(\nabla_t \frac{h_z}{k_o} + \boldsymbol{h}_t \right) \cdot \frac{1}{\varepsilon_r} \left(\nabla_t \frac{h_z^*}{k_o} + \boldsymbol{h}_t^* \right) - \mu_r h_z h_z^* \right] ds \end{aligned} \tag{16}$$

This eigenvalue problem has β^2 as the eigenvalue and the vector magnetic fields as the eigenvectors. Thus for a specified geometry, material characteristics and frequency the propagation characteristics of the modal fields can be evaluated. A similar variational expression can also be derived for the vector electric field.

One of the major advantages of FEMLAB® is that the FE solver can be set up simply by entering the variational expression eqn (16), and specifying the boundary conditions and required elements and matrix solvers. Then for a defined geometry, FEMLAB® will automatically generate a mesh, construct the matrix equation from the variational expression, element type and boundary conditions. It then solves the matrix problem and the post processing facilities allow the calculated fields to be investigated. Use of this facility by students gives them a real insight into the mechanics of the finite-element solution process without them having to get to the level of programming solvers for themselves. A detailed description of the specification of the eqn (16) in FEMLAB® is provided in the Appendix.

Numerical examples

For illustrative purposes, an inhomogeneous dielectric loaded rectangular waveguide, as shown in Fig. 1, is solved using the FE solver described in the Appendix. The results are then compared with the analytical results.[5,11]

The model is set-up in FEMLAB® by drawing the geometry using the user-friendly graphical user interface (GUI). Next, global constants such as ε_o, μ_o, frequency (f), angular frequency ($\omega = 2\pi f$) and propagation constant of free space ($k_o = \omega\sqrt{\varepsilon_o \mu_o}$) are defined. The boundary conditions and the subdomain material properties are defined before meshing and solving.

Figure 2 depicts the dispersion characteristic of the partially filled rectangular waveguide shown in Fig. 1. The TM to y case (TM^y) is illustrated in Fig. 2(a) and

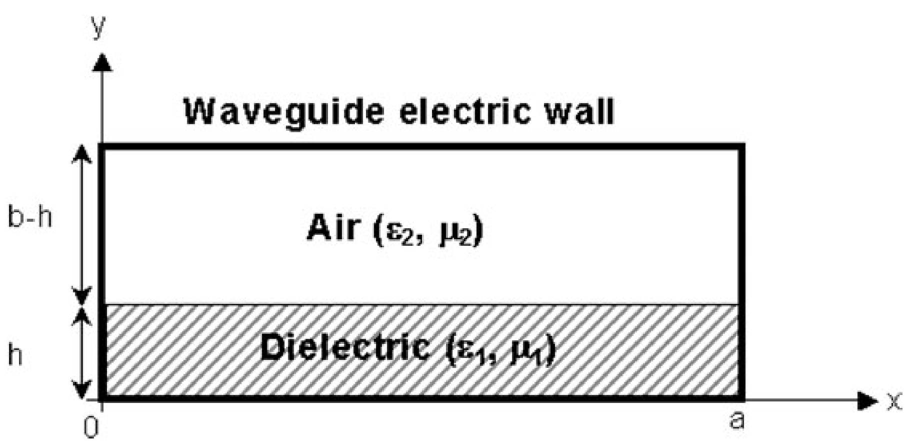

Fig. 1 *Inhomogeneously dielectric filled rectangular waveguide. [a = 22.86 mm, b = 10.16 mm, h = 1/3b, $\varepsilon_1 = 2.56\varepsilon_o$, $\varepsilon_2 = \varepsilon_o$, $\mu_1 = \mu_2 = \mu_o$]*

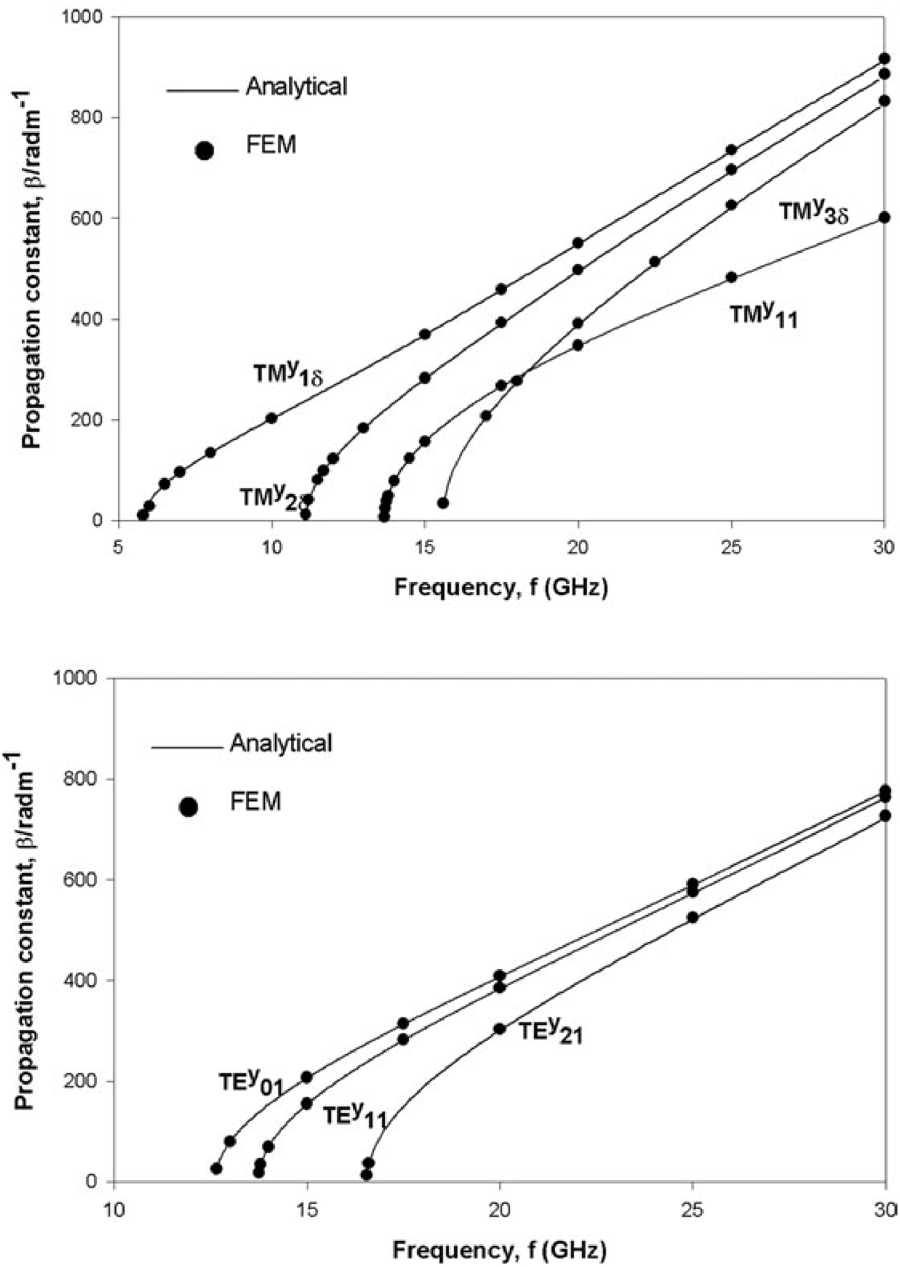

Fig. 2 *Dispersion characteristics of TM^y and TE^y modes for the partially filled rectangular waveguide shown in Fig. 1.*

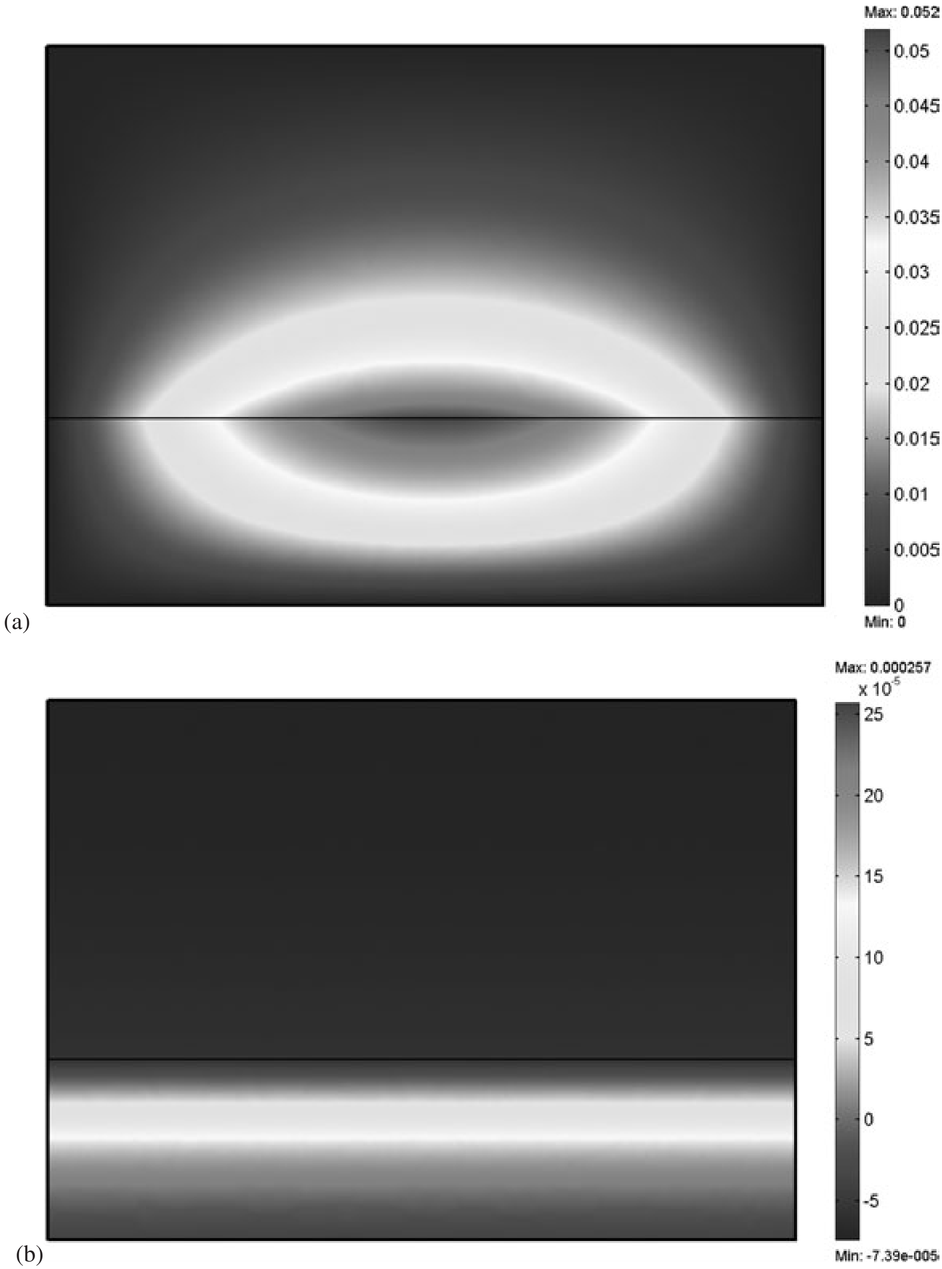

Fig. 3 *(a) $\boldsymbol{E}_z$ field distribution for the dominant $TM^y_{1\delta}$ mode at 20 GHz calculated using an $\boldsymbol{E}$-vector formulation. (b) $\boldsymbol{H}_z$ field distribution for the TE^y_{01} mode at 20 GHz calculated using a $\boldsymbol{H}$-vector formulation eqn (16).*

TE to y case (TE^y) case is depicted in Fig. 2(b). Note that the expression 'TM (TE) to y' means that this type of mode has no H_y (E_y) field component. The FEM solutions show good agreement with the analytical solutions.[5,11] Figure 3(a, b) illustrates the variation of the fields over the cross-section of the inhomogeneously dielectric filled waveguide at 20 GHz for the dominant $TM^y{}_{1\delta}$ mode and the first $TE^y{}_{01}$ mode.

Conclusions

FEMLAB® is a finite element based multi-physics research tool, which can also play a role in the teaching of numerical methods for science and engineering. This program provides an open environment for students to specify variational equations for solution by the FE method. The automatic meshing, matrix solving and post-processing frees the student to concentrate on the underpinning variational method and the numerical solutions. A tutorial example has been presented on how to transform the vector Helmholtz equation into a variational functional. This functional was entered into FEMLAB® before solving an inhomogeneous dielectric waveguide structure.

Acknowledgement

The authors wish to acknowledge the assistance and support of FEMLAB®.

References

1 FEMLAB® Electromagnetics Module, Version 2.3. Comsol Ltd. 2002.
2 J. L. Volakis, A. Chatterjee and L. C. Kempel, *Finite Element Methods for Electromagnetics: With Applications to Antennas, Microwave Circuits, and Scattering* (IEEE Press, New York, 1998).
3 F. A. Fernandez and Yilong Lu, *Microwave and Optical Waveguide Analysis by The Finite Element Method* (John Wiley, Chichester, New York, 1996).
4 P. P. Silvester and R. L. Ferrari, *Finite Elements for Electrical Engineers*, 3rd edn (Cambridge University Press, Cambridge, 1990).
5 R. F. Harrington, *Time Harmonic Electromagnetic Fields* (McGraw-Hill, New York, 1961).
6 B. M. A. Rahman and J. B. Davies, 'Finite-element analysis of optical and microwave waveguide problems,' *IEEE Trans. Microw. Theory Tech*., **32** (1984), 20–28.
7 M. B. Steer, J. W. Bandler and C. M. Snowden, 'Computer-aided design of rf and microwave circuits and systems', *IEEE Trans. Microw. Theory Tech*., **50** (2002), 996–1005.
8 D. M. Pozar, *Microwave Engineering,* 2nd edn (John Wiley, Chichester, New York, 1998).
9 S. Ramo, J. R. Whinnery and T. Van Duzer, *Fields and Waves in Communication Electronics* (Wiley, New York, 1994).
10 A. A. P. Gibson and B. M. Dillon, 'Variational solution of electric and magnetic circuits', *IEE Eng. Sci. Educ. J*., **4**(1) (1995), 5–10.
11 C. A. Balanis, *Advance Engineering Electromagnetics* (John Wiley, Chichester, 1989).

Appendix

Implementation of weak formulation in FEMLAB

To set up a vector Helmholtz solver within FEMLAB® the field variables, variational expression and boundary conditions must be specified to define the problem. The element type and matrix solver must also be specified or left as default values to define the solution process.

1. At FEMLAB® startup, the *Weak Subdomain Eigenvalue Mode* application mode is chosen and the dependent variables: hx, hy and hz are defined.
2. The variational expression is setup in the *Subdomain Settings* dialog box. In this box there are text fields for each variable with both '*weak*' and '*dweak*' coefficients. The left-hand side of eqn (16) is called the '*weak*' coefficient and the right-hand side is the '*dweak*' coefficient. β^2 is of course the eigenvalue. It does not matter in which field you enter the different terms. They are all added together eventually. It is noteworthy that no space is allowed when writing an expression in the *Subdomain Settings*. The notations for test functions are: hx_test, hy_test and hz_test. Derivatives are given by appending the independent variable, e.g. $\partial hx/\partial y$ is denoted hxy and $\partial hx\_test/\partial y$ is denoted hxy_test. As an example, the scalar product ***h_test.h*** is written hx_test*hx+hy_test*hy. The '*dweak*' coefficient has a little peculiarity. The dependent variables should have *_time* dependent to their names, e.g. write hz_time instead of hz. The reason is that '*dweak*' is also used in time dependent problems, and then *_time* stands for a time derivative. Table 1 illustrates the '*weak*' and '*dweak*' coefficients for eqn 16.
3. Element type is chosen from the *Element* tab in the *Subdomain Settings* dialog box. The default element type is Lagrangian, but vector elements can be specified to avoid spurious modes and simplify boundary conditions at material interfaces. The FEMLAB® settings for vector elements are summarised in Table 2. The '*gporder*' is the number of Gauss points, when doing the numerical integration, see page 3–50 in the FEMLAB® Reference Manual.[1] While the

TABLE 1 *Subdomain setting for 'weak' and 'dweak' terms in FEMLAB®*

Page in subdomain setting	Term	Expression
Weak	hx	(hyx − hxy)/epsilonr*(hyx_test-hxy_test) − k0^2*mur*(hx*hx_test + hy*hy_test)
	hy	0
	hz	0
Dweak	hx	((hzx_test/k0 + hx_test)*(hzx_time/k0 + hx_time) + (hzy_test/k0 + hy_test)*(hzy_time/k0 + hy_time))/ epsilonr − mur*hz_test*hz_time
	hy	0
	hz	0

TABLE 2 *Elements setting in Subdomain Settings*

Page in subdomain setting	Term	Expression
Element	shape	shvec({'hx' 'hy'}) shlag(1,'hz')
	gporder	2 2 2
	cporder	1 1 1

TABLE 3 *Boundary conditions setting for **H**-vector formulation*

Description	Equation	Constraint term	Expression
PMC	$n \times H = 0$	Hx	Nx*thy − ny*thx
		Hy	0
		Hz	–hz
PEC	$n \times E = 0$	Hx	0
		Hy	0
		Hz	0

'*cporder*' is the number of points in an element where a constraint is set, see page 3–47 in the FEMLAB® Reference Manual.[1]

4. The boundary conditions are specified in the *Boundary Settings* dialog box. For a ***H***-vector formulation, an electric wall boundary condition (PEC) corresponds to a Neumann boundary and a magnetic wall (PMC) to a Dirichlet boundary. All the field components in the '*weak*' and '*dweak*' tabs are set to zero and the PMC boundary conditions are specified in the '*constr*' tab. The variables thx and thy are used (instead of hx and hy) for the boundary constraints because the vector elements specified in step 3 use edge variables defined parallel to the element edges. For PEC boundary, all '*constr*' terms are set to zero. These points are summarised in Table 3.
5. The default matrix solver, setup under *Solve Parameters*, is used to solve the problem. To help convergence, an eigenvalue search range should be specified.