E-B Eigen Mode Module

Manual

The **E**-**B** Eigen Mode Module is a Mathworks MATLAB Graphical User Interface for the Finite Element Method computational solution of 3D electromagnetic periodic structure problems. The **E**-**B** Eigen Mode Module utilizes the *Field-Flux Bloch Floquet* formulation of the Maxwell's set of equations, which re-formulates the classic equations in frequency domain (assuming $e^{j\omega t}$ dependence), under the assumption of propagation of a periodic Bloch-Floquet wave.

Physical Problem

The basis of the *Field-Flux Bloch Floquet* formulation of Maxwell's equations lies in the decomposition of electromagnetic waves propagating in periodic media in the form of a plane wave $e^{-j\boldsymbol{k}\cdot\boldsymbol{r}}$, where $\boldsymbol{k} = k\hat{\boldsymbol{k}}$ is the Bloch-Floquet wavevector of prescribed propagation direction $\hat{\mathbf{k}}$, modulated by a three dimensional periodic envelope, i.e. a three dimensional vectorial function defined inside the periodic unit cell. These Bloch Floquet forms of the two electromagnetic fields (the intensity of the electric field **E** and the magnetic flux density **B**), which constitute the degrees of freedom of the Field Flux formulation are:

$$\boldsymbol{E} = \boldsymbol{e}e^{-j\boldsymbol{k}\cdot\boldsymbol{r}}$$

$$\boldsymbol{B} = \boldsymbol{b}e^{-j\boldsymbol{k}\cdot\boldsymbol{r}}$$

Substituting the above into the Ampère and Faraday equations results into the modified maxwell's equations:

$$\nabla \times \boldsymbol{e} - jk\hat{\boldsymbol{k}} \times \boldsymbol{e} = -j\omega\boldsymbol{b},$$
$$\nabla \times \bar{\bar{\mu}}_r^{-1}\boldsymbol{b} - jk\hat{\boldsymbol{k}} \times \bar{\bar{\mu}}_r^{-1}\boldsymbol{b} = j\omega\varepsilon_0\mu_0\bar{\bar{\varepsilon}}_r\boldsymbol{e}$$

Figure 1. E-B Eigen Mode Module Entry Form. Left and Central Panels Visible.

The above system of differential equations can be perceived as an eigenvalue problem, where the norm of the Bloch Floquet wavevector $k = \beta - j\alpha$ is the eigenvalue and the electric and magnetic periodic field envelopes comprise the eigenvector. The solution of this eigenvalue problem set in a periodic structure for a given angular frequency ω and prescribed propagation direction $\hat{\mathbf{k}}$ produces the fundamental eigenmodes that can exist in the periodic structure under these assumptions (given frequency and propagation direction).

Any electromagnetic wave that can support propagation on a periodic structure, even if it is totally evanescent, can be described as a linear combination of the fundamental eigen modes, while the dispersion diagram of the periodic structure can be constructed by the couples of given frequency ω and resulting eigenvalue $k(\omega)$.

A.The **E-B** Eigen Mode Module

The **E**-**B** Eigen Mode module utilizes the finite element implementation of Field Flux Bloch Floquet formulation [1],[4],[5] and its extensions to periodic structures which incorporate bianisotropic media [2],[5] and 2-dimensional graphene components [3],[5]. As the finite element degrees of freedom of the electric field periodic envelope lie in the edges of the finite element tetrahedral mesh, while those of the magnetic field periodic envelope lie in the facets of the tetrahedral mesh, the terms edges and facets that appear in both the remaining of this documentation as well as the **E**-**B** Eigen Mode module are linked to the finite element representations of the **e** and **b** field quantities respectively.

The **E**-**B** Eigen Mode module's Graphical User Interface (Figure 1) is divided into three areas, the left panel area, the central panel area and the right panel area (not visible in Figure 1). The computational modeling of a 3D periodic structure in the in the **E**-**B** Eigen Mode module is controlled by user input inserted sequentially into the left panel area, grouped in terms of the geometric, frequency, material, boundary and assembly (the formation of the finite element algebraic problem) properties of the structure. The lower part of the left panel area also contains a text area, which displays messages (such as the successful completion of a I/O step), and a condition bar which displays the current condition (red for busy and green for available) of the **E**-**B** Eigen Mode module.

All the graphics representing the geometric properties of the model will appear in the center of the central panel, while its upper part contains plot controls for the graphics and the lower part controls for switching between the five user I/O groups of input. The latter controls are enabled, whenever the corresponding group of user input is completed. The solution and export of the results is handled by controls in the right panel area, which becomes visible after all user I/O is completed.

Also visible in Figure 1, are the **Save** (upper left corner) and **Exit** (upper right corner) buttons. Engaging the former prompts a file selection window for the save (in .mat format) of the **ToolboxModel** object (see **E**-**B** Toolbox Classes), which contains all of the model's properties, while clicking on the **Exit** button will result in the immediate exit from the **E**-**B** Eigen Mode module.

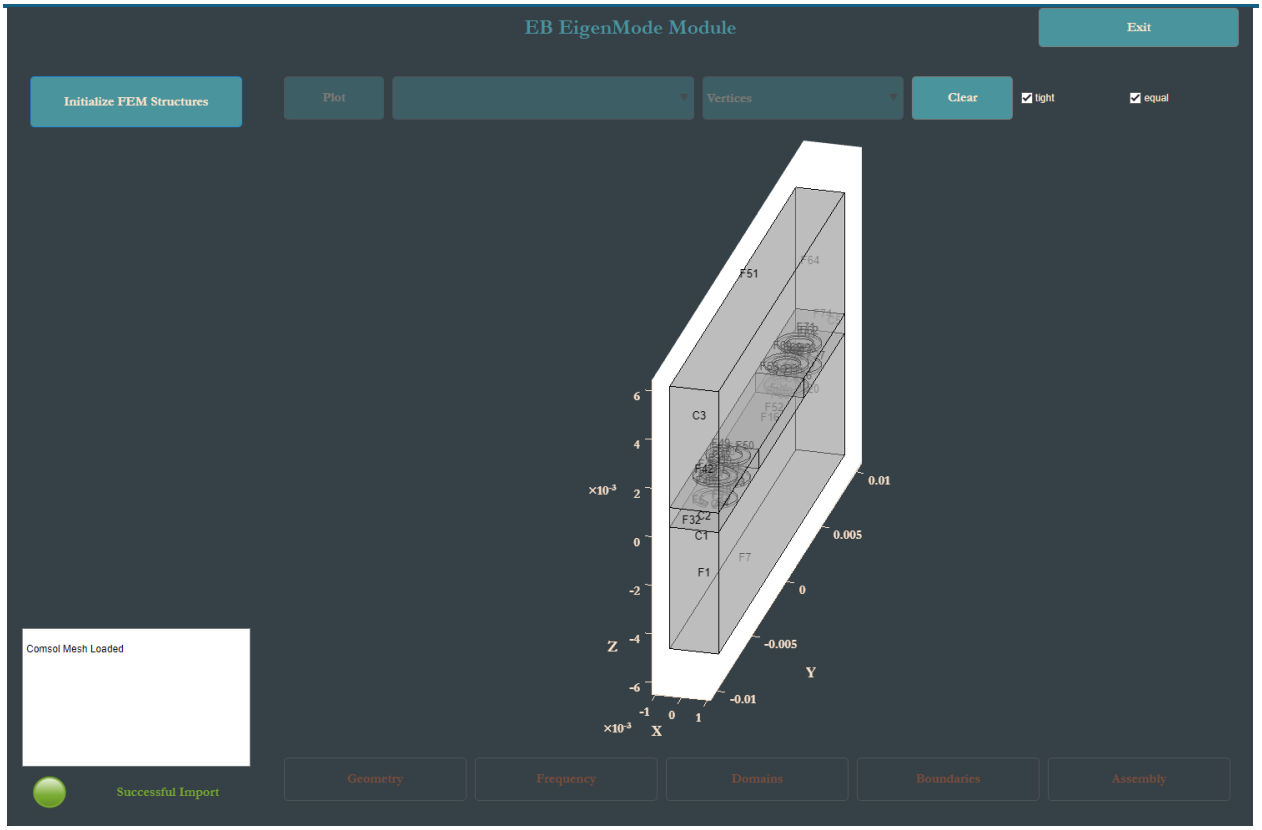

Figure 2. Successful import of tetrahedral mesh.

B. Geometry Panel

The first group of user I/O required for the construction of a **E**-**B** Eigen Mode module model is the determination of the structure's geometry. The geometry panel appears in the entry form of the **E**-**B** Eigen Mode module (see Figure 1) and allows for the import of a tetrahedral mesh or a preexisting Toolbox Model. Selecting the Mesh option and pressing the **Load** button (upper part of the left panel area) will prompt a file selection window, asking for the location of the mesh file. The tetrahedral mesh must be in the form of .mat file, which will contain a matrix p containing the mesh points (tetrahedral mesh vertices), a matrix t containing the mesh elements and a matrix elem, containing domain information for each mesh element (see MATLAB Mathworks geometryFromMesh function [6]). If the tetrahedral mesh is constructed using the COMSOL Multiphysics software, the code in Algorithm I can be used to yield a correct tetrahedral mesh file.

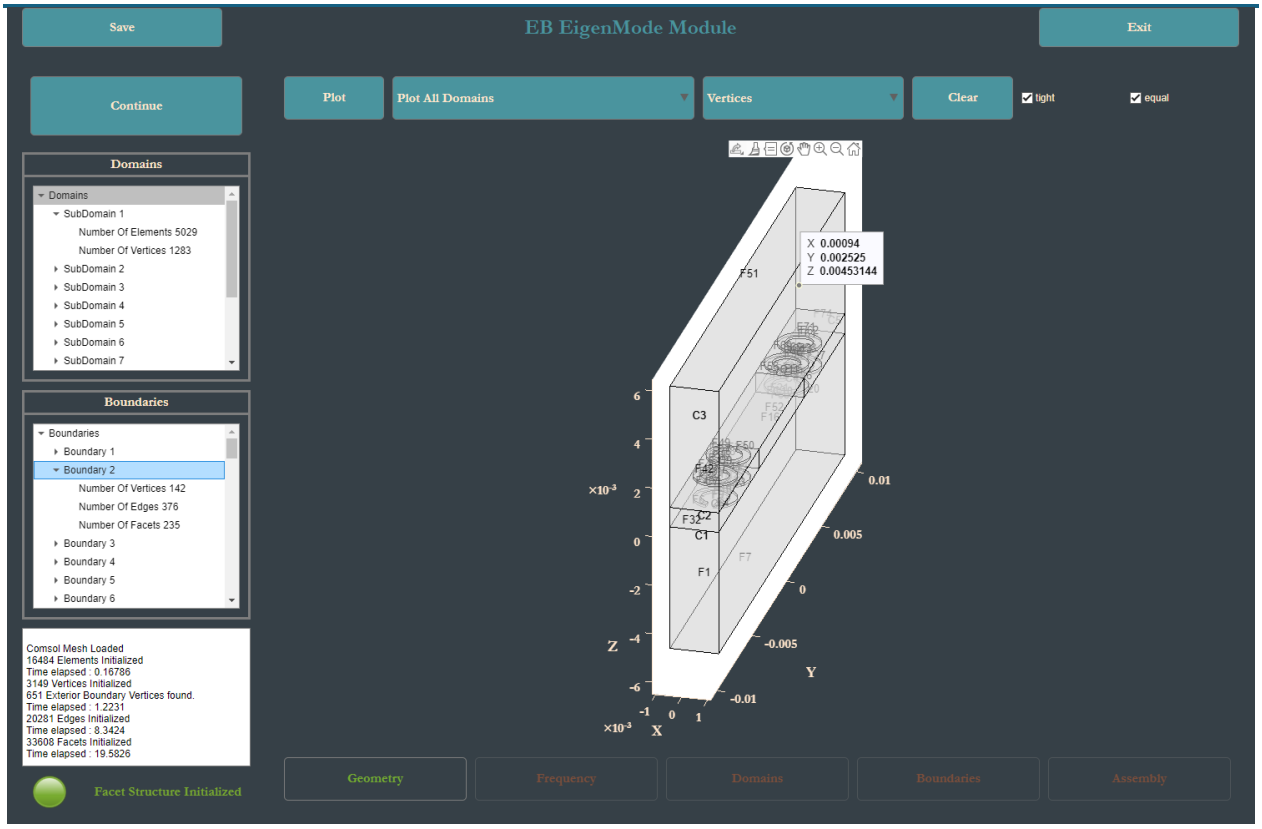

Figure 3 Initialization of Finite Element Structures completed.

```
import com.comsol.model.*;          import com.comsol.model.util.*;

model = mphopen('Comsol_Model_Filename);

[meshstats,meshdata] = mphmeshstats(model);

p=model.mesh('mesh1').getVertex();

t= meshdata.elem{2}+1;

elem=meshdata.elementity{2};
```

Algorithm I. COMSOL Multiphysics with MATLAB code, for the correct export of a tetrahedral mesh file.

Due to limitation set by the MATLAB Mathworks platform, any two-dimensional structures on the interior of the geometry (such as the etched rings of a metamaterial on printed circuit board) must be given a vertical dimension (designed as 3D structures), otherwise the **E-B** Eigen Mode module will fail to recognize these features.

If the import of the mesh file is successful (Figure 2), the geometry of the model as described by the tetrahedral mesh appears in the central panel area. The next step is the initialization of the finite element Elements, Vertices, Edges, Facets structures (see Appendix **E-B** Toolbox Classes) triggered by the **Initialize Finite Element Structures** button. Upon the successful initialization of the finite element structures (Figure 3), the preprocessing time is displayed in the message text area, while the subdomains and boundaries (both interior and exterior) are displayed in the center of the left panel area along with information about the number of tetrahedral mesh vertices, elements, edges and facets that belong in\on them. The plot controls enabled on the top of the central panel area allow for the plot of vertices, edges and facets inside/on the individual subdomains and boundaries of the tetrahedral mesh. Clicking on the **Continue** button

C. Frequency

The input of the predetermined frequency is controlled by the Frequency panel, which supports single frequency operation as well as a frequency range option. The preset default for the Eigen Mode module is that of the single frequency operation, while switching between the single frequency mode and the multiple frequencies mode is achieved by selecting and deselecting the **Single Frequency** check box on the top of the Frequency panel.

In the case of the single frequency mode (Figure 4), the user inputs the frequency in the text field area in the upper left corner of the panel and selects the appropriate unit from the adjacent drop down selection, while moving to the next step of the computational modeling is achieved by clicking on the **Done** button on the bottom of the Frequency Panel completes the import of the model's geometry and switches the module to the frequency group of user I/O.

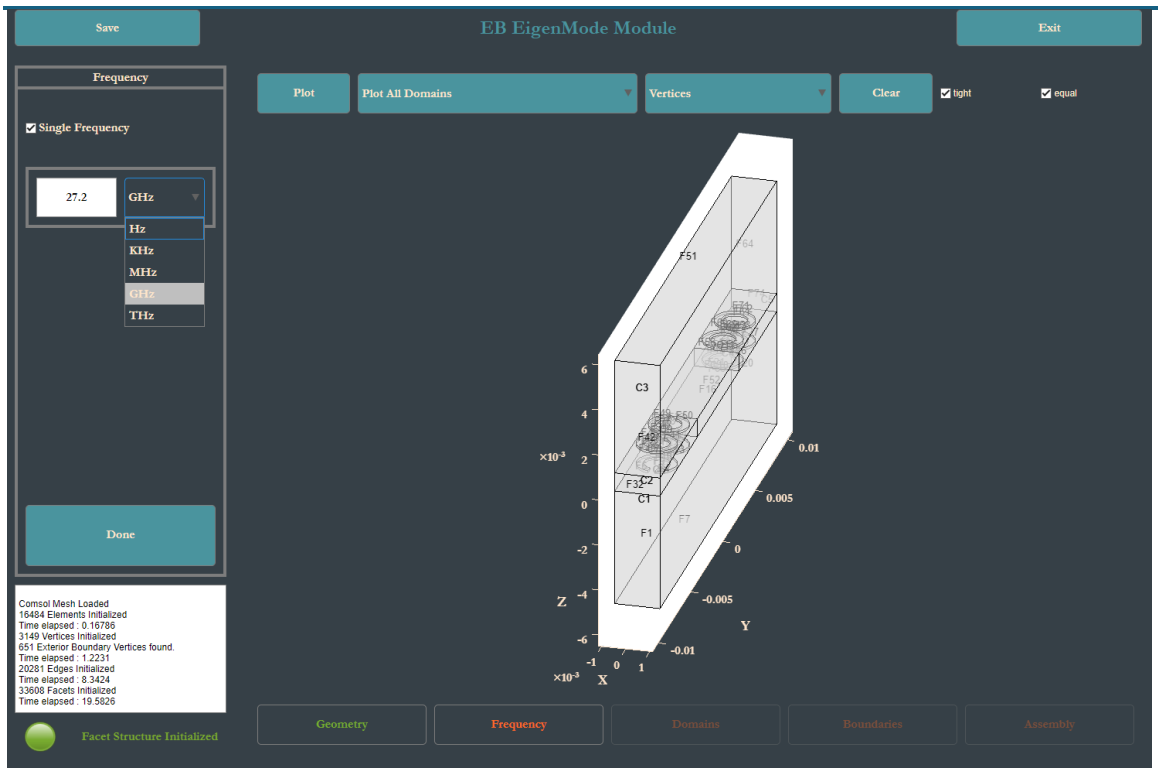

Figure 4. Single Frequency mode

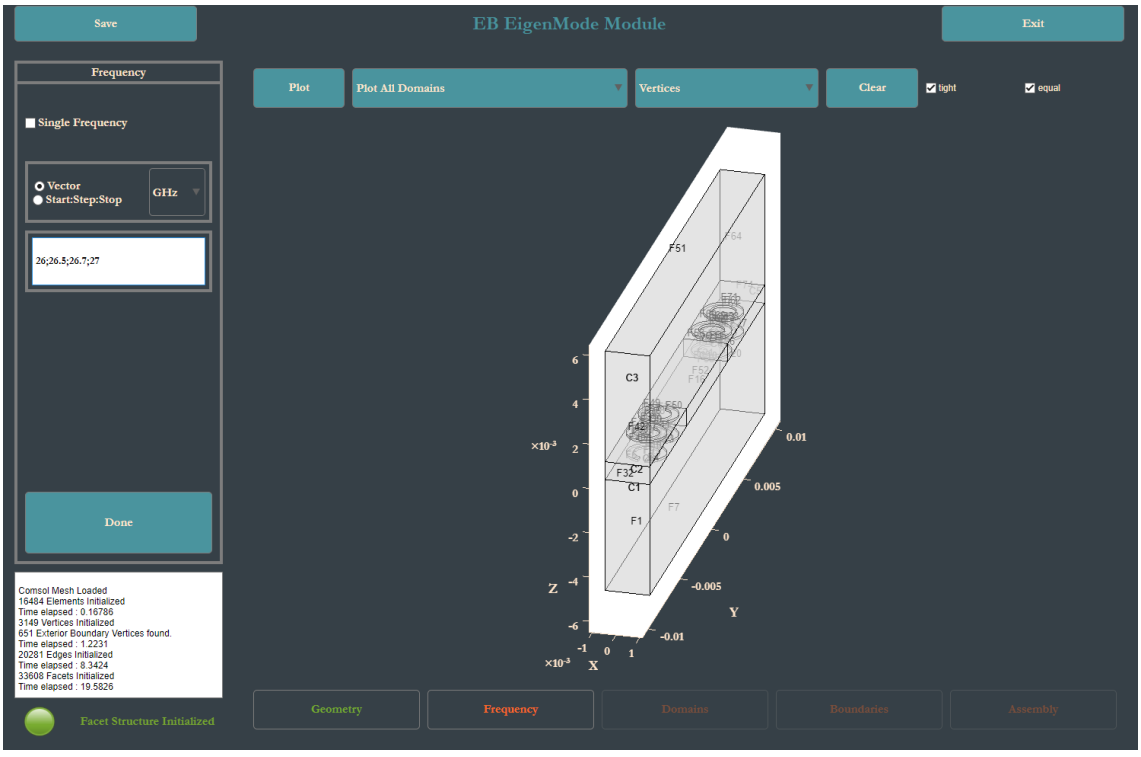

Figure 5. Multiple frequencies mode, vector input.

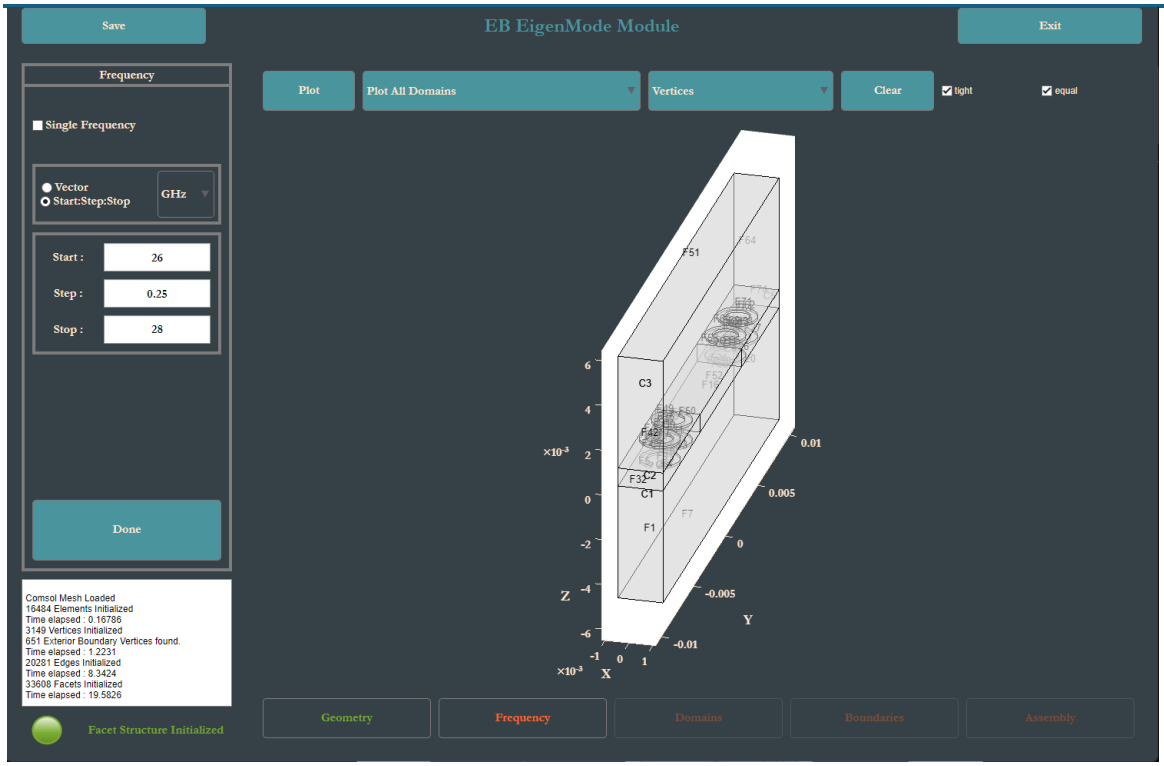

Figure 6 Multiple frequencies mode, Start:Step:Stop input.

If the division $\frac{f_{Stop}-f_{Start}}{Step}$ has a non-zero modulo, then the frequency f_{Start} does not exist in the frequency range. The frequency I/O process is completed by pressing the **Done** button on the bottom of the left panel. Beyond this point, any changes in the frequency parameters of the model are not permitted.

The secondary mode of multiple frequencies operation allows for two complementary ways of input, where the frequency range for the computational model can be inserted either in the form of a vector or in the form of equally distanced values between an upper and a lower frequency (Start: Step: Stop mode).

The default input method for the multiple frequencies mode is the vector input, where each of the frequencies in the frequency range is inserted by the user separated by the semicolon character (;) . Switching to the alternative input method is achieved by selecting the **Start: Step: Stop** button on the top of the Frequency panel. In this mode of input (Figure 6), each frequency f_i in the frequency range is calculated as:

$$f_i = f_{Start} + i * Step, i = 0{:}\,(int)\frac{f_{Stop} - f_{Start}}{Step}$$

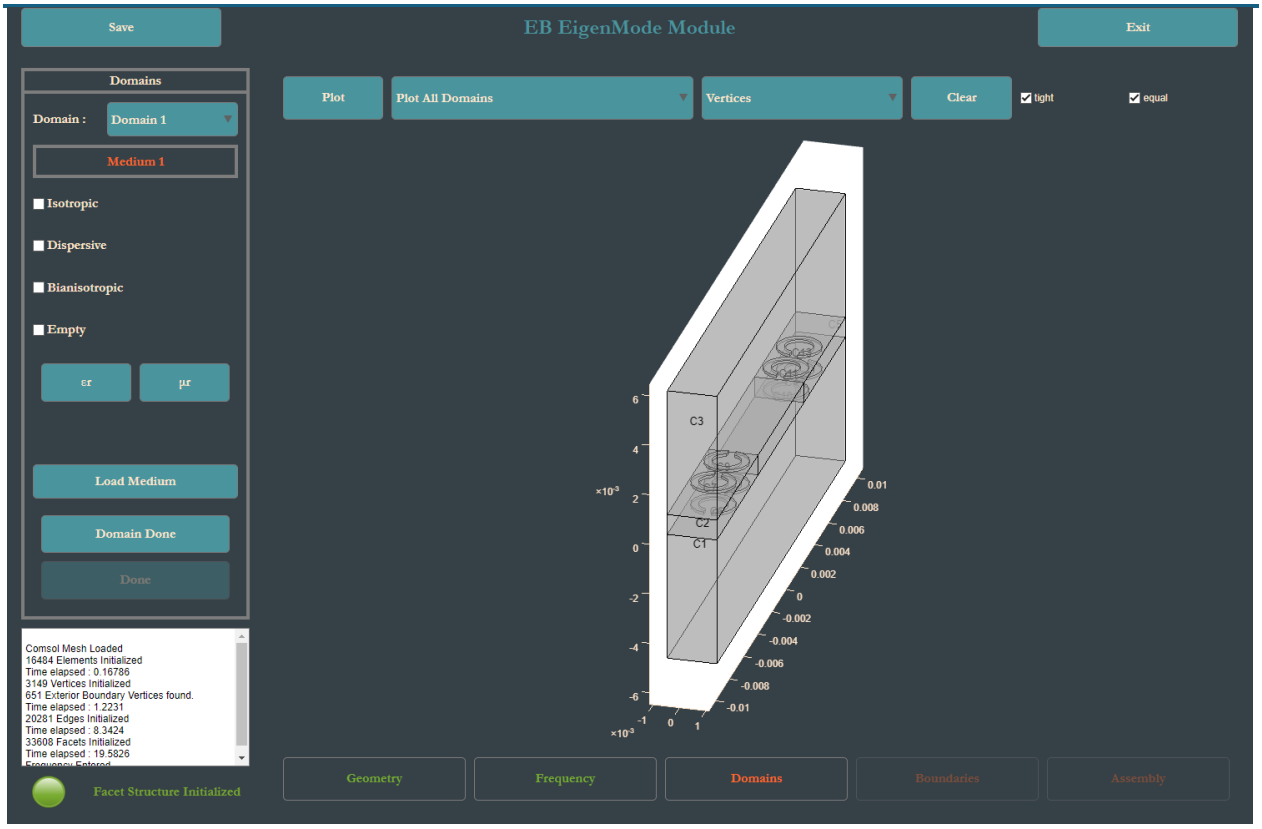

Figure 7. Domains Panel

D. Domains

Following the successful import of the frequency parameters input, is the domains and material group of user I/O which is imported in the Domains Panel and where the user must insert the electromagnetic medium properties for all the domains which form the computational domain of the model.

The **E-B** Eigen Mode module allows for the import of three categories of electromagnetic media. The default preset for every domain is that of a general anisotropic electromagnetic medium, where the relative dielectric permittivity and relative magnetic permeability are in the tensor form:

$$\varepsilon_r = \begin{bmatrix} \varepsilon_{r\,xx} & \varepsilon_{r\,xy} & \varepsilon_{r\,xz} \\ \varepsilon_{r\,yx} & \varepsilon_{r\,yy} & \varepsilon_{r\,yz} \\ \varepsilon_{r\,zx} & \varepsilon_{r\,zy} & \varepsilon_{r\,zz} \end{bmatrix}, \mu_r = \begin{bmatrix} \mu_{r\,xx} & \mu_{r\,xy} & \mu_{r\,xz} \\ \mu_{r\,yx} & \mu_{r\,yy} & \mu_{r\,yz} \\ \mu_{r\,zx} & \mu_{r\,zy} & \mu_{r\,zz} \end{bmatrix}$$

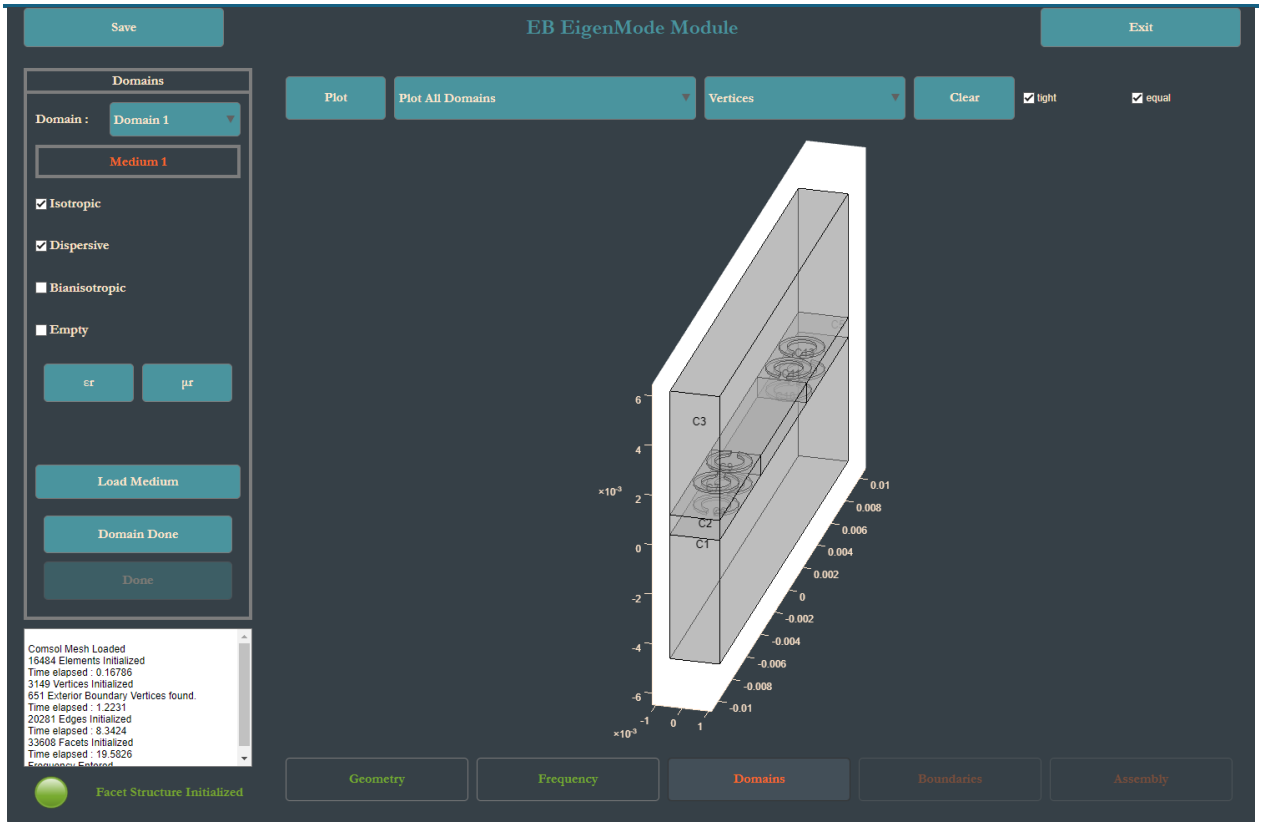

Figure 8. Isotropic Medium Domain

In the isotropic setting, the two tensorial electromagnetic medium properties are substituted by scalar parameters, while the definition of a bianisotropic medium, where the constitutive equations are generalized in the forms:

$$\boldsymbol{D} = \varepsilon_0 \bar{\bar{\varepsilon}}_r \boldsymbol{E} + \bar{\bar{\xi}} \boldsymbol{H},$$
$$\boldsymbol{B} = \mu_0 \bar{\bar{\mu}}_r \boldsymbol{H} + \bar{\bar{\zeta}} \boldsymbol{E},$$

requires the import of two additional electromagnetic coupling tensors

$$\xi = \begin{bmatrix} \xi_{xx} & \xi_{xy} & \xi_{xz} \\ \xi_{yx} & \xi_{yy} & \xi_{yz} \\ \xi_{zx} & \xi_{zy} & \xi_{zz} \end{bmatrix}, \zeta = \begin{bmatrix} \zeta_{xx} & \zeta_{xy} & \zeta_{xz} \\ \zeta_{yx} & \zeta_{yy} & \zeta_{yz} \\ \zeta_{zx} & \zeta_{zy} & \zeta_{zz} \end{bmatrix}$$

To mark a specific domain for the definition of its electromagnetic medium properties, the user must select it from the drop down selection on the top of the left panel,

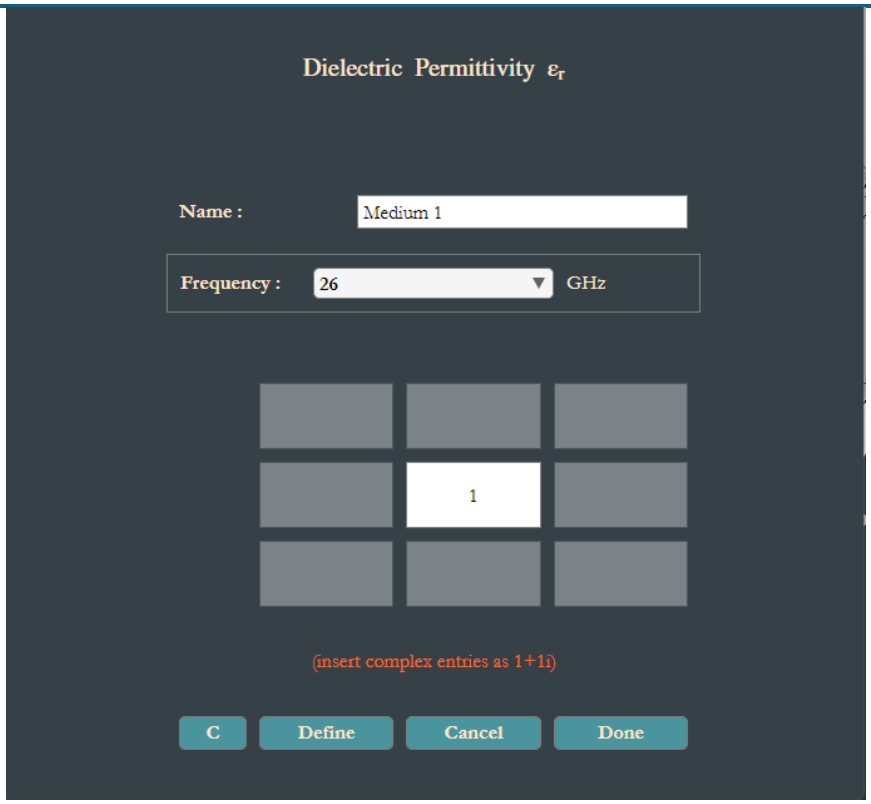

Figure 9. Isotropic Medium Domain Tensor Form

whereas the **E-B** Eigen Mode module updates the stored electromagnetic properties (if no input has been given – to the preset default anisotropic medium). To distinguish between the different domains in the geometry, the user can utilize the plot controls on the central panel area, to plot any of geometric entities (vertices, edges, facets) of the tetrahedral mesh within a specific model domain. When the user has completed the definition of the electromagnetic material properties of a domain, the domain must be marked as defined by pressing the **Domain Done** button on the bottom of the left panel area. Progress to the next group of user I/O is enabled only when all of the model's domain media have been defined.

D.I Isotropic Media

To define an isotropic medium domain, the user selects the **Isotropic** check box (Figure 8). If the medium exhibits dispersive behavior (the electromagnetic properties of the medium are a function of frequency), the user clicks also on the **Dispersive** check box directly below the **Isotropic** check box. The **Dispersive** check box becomes enabled only when a frequency range has been imported in the frequency I/O panel.

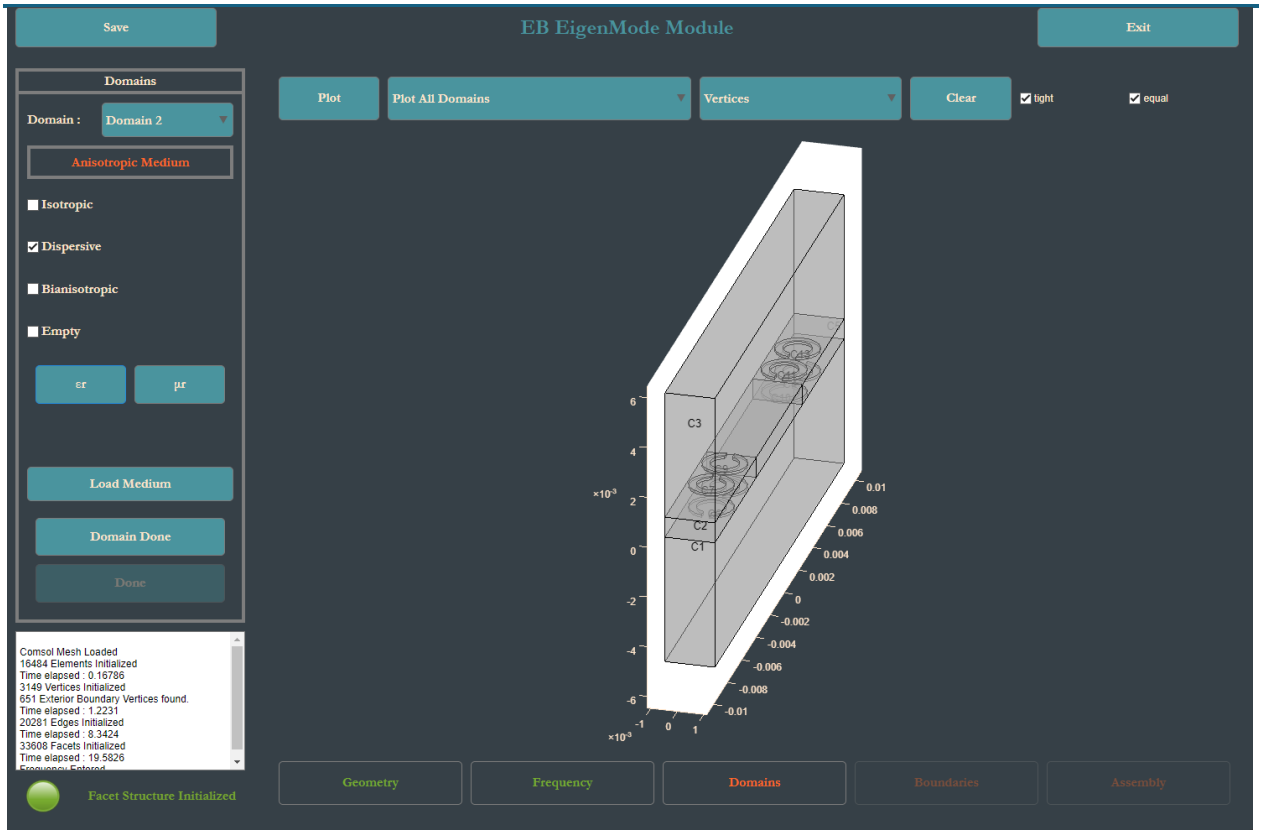

Figure 10. Anisotropic Medium Domain

To insert the scalar values of the scalar relative dielectric permittivity and scalar relative magnetic permeability, the user clicks on the ε_r and μ_r buttons respectively. This action prompts a new GUI form, the domain tensor form for isotropic media. (Figure 9). The electromagnetic scalar property is then inserted into the text area in the center of the form. To store this property in the **E-B** Eigen Mode Module model, the user must then click on the **Define** button located in the lower left part of the form. In the case of a dispersive isotropic medium, this process has to be repeated for each frequency of the frequency range, by selecting it in Frequency drop down selection, inserting the frequency dependent electromagnetic property and clicking on the **Define** button. Failing to do so (click on the **Define** button), the electromagnetic property of the medium in the **E-B** Eigen Mode module will remain the predefined or default one. The form accepts real and complex values as medium property values; however the Domain Tensor form accepts complex values, only when the complex part of the value is denoted by the i character directly after the last digit of the complex part (for example 1+1i).

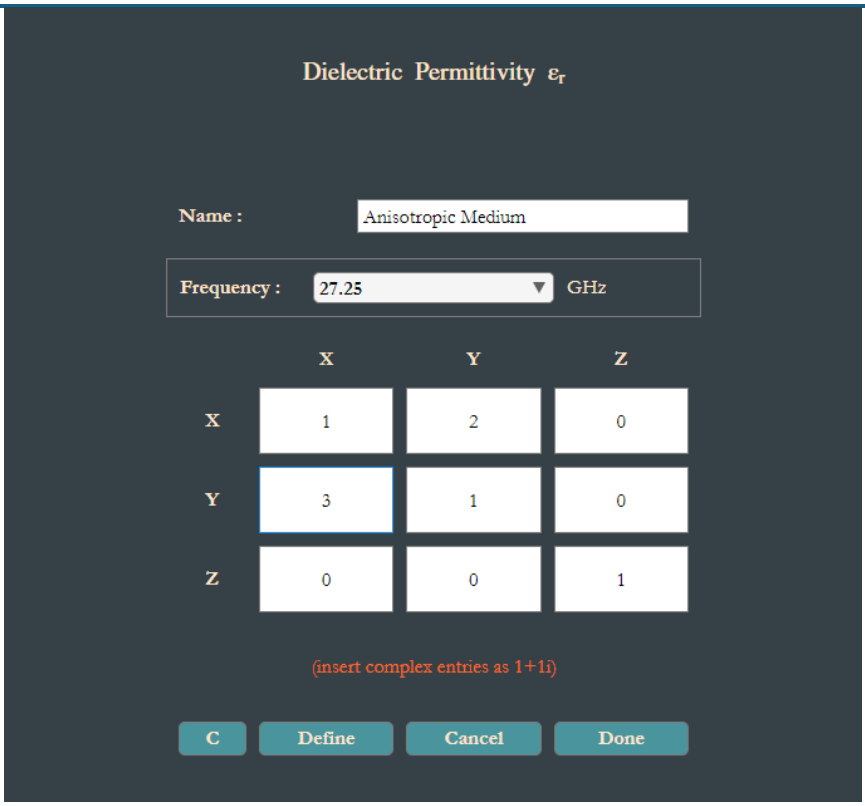

Figure 11. Anisotropic Medium Domain Tensor Form

The Domain Tensor form also allows to name the selected medium by inserting a name tag in the name text field. Clicking on the **C** button, clears any user input and resets the form, while to exit the form after having defined the electromagnetic properties of the medium and return to the **E-B** Eigen Mode module, the user must click on the **Done** button.

D.II Anisotropic Media

A general anisotropic medium for domain in the **E-B** Eigen Mode module model is defined, when apart from the **Dispersive** check box, all other check box controls in the left panel area are de-selected (Figure 10). To insert the tensor values of the tensorial dielectric permittivity and tensorial relative magnetic permeability, the user clicks on the ε_r and μ_r buttons respectively. This action prompts the domain tensor form for anisotropic media. (Figure 11), where the nine values of the electromagnetic tensor are inserted in the nine cells in the center of the form. To store the tensorial property in the **E-B** Eigen Mode Module model, the user must then click on the **Define** button located in the lower left part of the form. In the case of a dispersive isotropic medium,

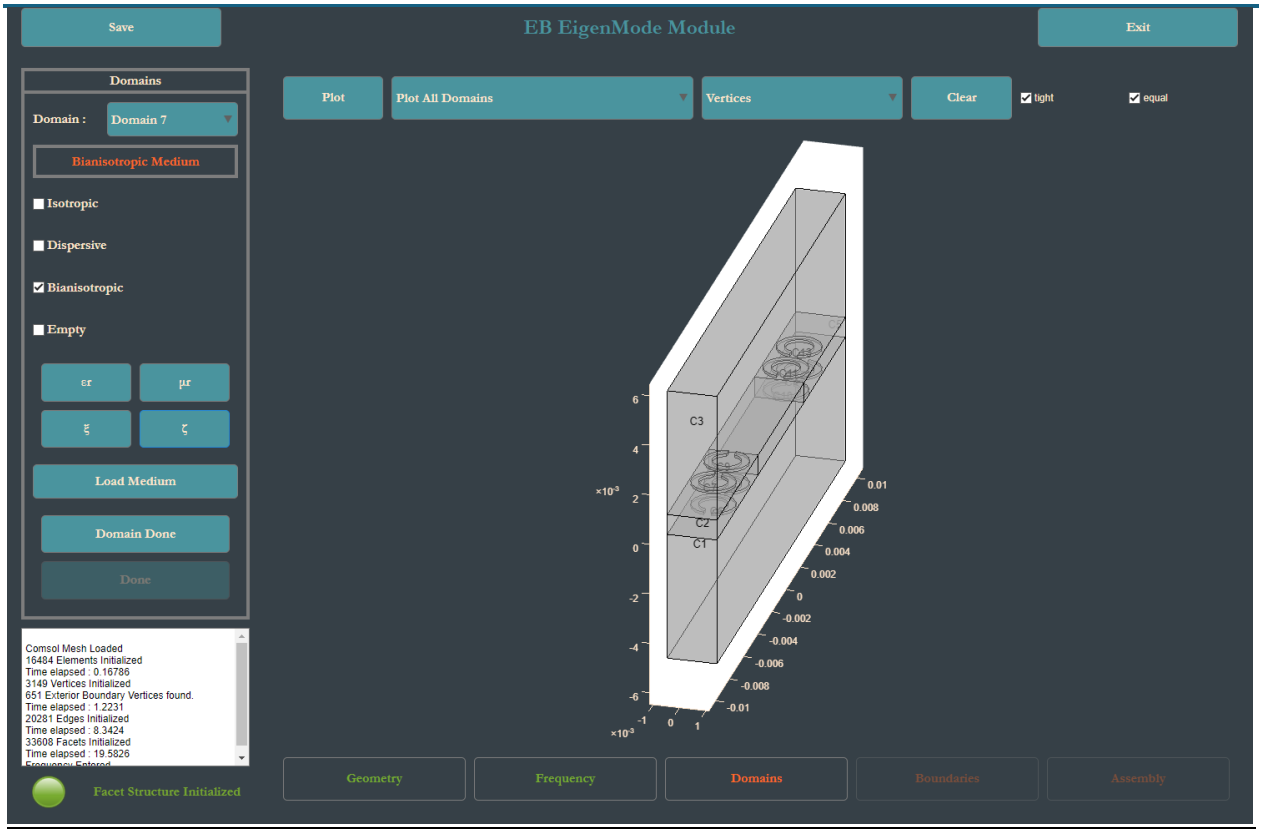

Figure 12. Bianisotropic Medium Domain

this process has to be repeated for each frequency of the frequency range, by selecting it in Frequency drop down selection, inserting the frequency dependent electromagnetic tensor and clicking on the **Define** button.

D.III Bianisotropic Media

A general bianisotropic medium for domain in the **E**-**B** Eigen Mode module model is defined, when the **Bianisotropic** check box in the left panel is selected (Figure 12). To insert the tensor values of the tensorial dielectric permittivity, relative magnetic permeability and magneto electric coupling parameters ξ and ζ, the user clicks on the ε_r, μ_r , ξ and ζ buttons respectively. This action prompts the domain tensor form for anisotropic media. (Figure 11), where the nine values of the electromagnetic tensor are inserted in the nine cells in the center of the form. To store the tensorial property in the **E-B** Eigen Mode Module model, the user must then click on the **Define** button located in the lower left part of the form. In the case of a dispersive isotropic medium, this process has to be repeated for each frequency of the frequency range, by selecting

it in Frequency drop down selection, inserting the frequency dependent electromagnetic tensor and clicking on the **Define** button.

D.IV Empty Medium

Selecting the **Empty** check box, regardless of any other user input, renders the corresponding domain external to the E-B Eigen Mode Module model (any degrees of freedom within this domain will not be accounted for in the finite element assembly process). This type of domain is reserved for the implementation of the impedance boundary condition.

D.V Load Medium function

The **E-B** Eigen Mode Module allows the user to circumvent the process of defining the material properties of the computational model, by importing a predefined medium in the form of .mat file. When using the Load Medium function of the **E-B** Eigen Mode module no further action, apart from finalizing the medium definition by clicking the **Domain Done** button, is required of the user.The required variables in the .mat file with the corresponding type of electromagnetic medium are listed in Table 1.

Table 1 Load Medium File Variables

Variable	*Isotropic Medium*	*Dispersive Isotropic Medium*	*Anisotropic Medium*	*Dispersive Anisotropic Medium*	*Bianisotropic Medium*	*Dispersive Bianisotropic Medium*
epsilon	Scalar	Vector	3x3 matrix	Cell with 3x3 matrix entries	3x3 matrix	Cell with 3x3 matrix entries
mu	Scalar	vector	3x3 matrix	Cell with 3x3 matrix entries	3x3 matrix	Cell with 3x3 matrix entries
ksi	-	-	-	-	3x3 matrix	Cell with 3x3 matrix entries
zita	-	-	-	-	3x3 matrix	Cell with 3x3 matrix entries
tag	optional	optional	optional	optional	optional	optional

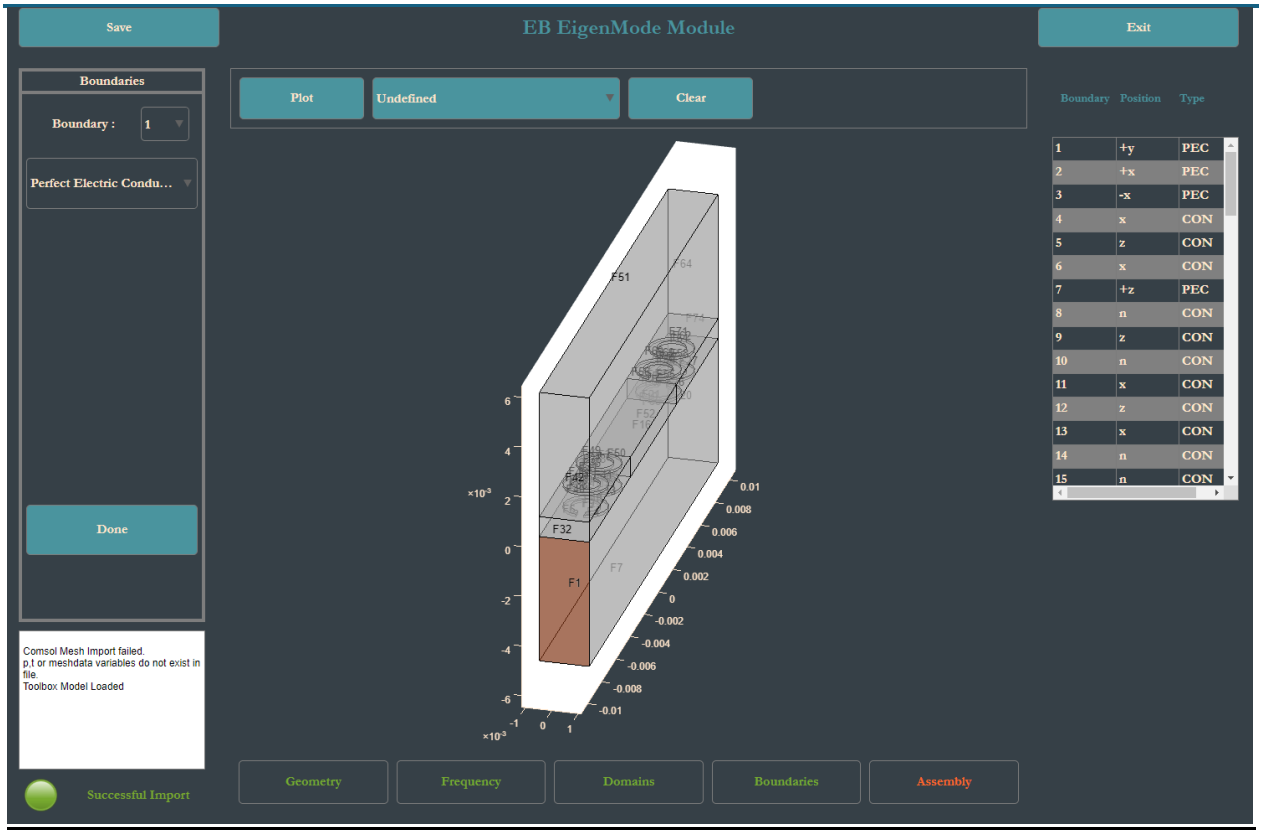

Figure 13. E-B Eigen Mode module Boundaries Panel

For the dispersive isotropic, anisotropic and bianisotropic media, the dimensions of the cells and vectors must be equal to number of frequencies in the frequency range.

E. Boundary Conditions

As with any implementation of Finite Element method, the definition of the boundary conditions truncating and interfacing the domains of the computational model is of outmost significance to the correct modeling of the problem under investigation. The main controls implicated in the definition of boundaries appear in the Boundaries Panel in the left panel area.

On the top of the Boundaries Panel, a drop down selection allows switching between the different model boundaries, while the type of the boundary condition applied onto a boundary is selected from the drop down selection directly below. The **E-B** Eigen Mode module allows for the implementation of seven different boundary conditions, *Perfect Electric Conductor*, *Perfect Magnetic Conductor*, *Continuity*, *Absorbing*

Boundary Condition, *Absorbing Boundary Condition Z*, *Absorbing Boundary Condition B*, *Graphene, Impedance Boundary Condition* and *Periodic Boundary Condition.*

Every time a new boundary is selected from the boundary drop down selection, its area is highlighted in the center of the central panel area, while the user is further aided by the table of current boundary condition on all of the boundaries displayed on the right panel area. At any time, the user can highlight all of the boundaries that are marked to be modeled as a certain boundary condition using the plot controls on the top of the central panel area.

The preset default settings for all external boundaries is the *Perfect Electric Conductor*, while for all internal boundaries the *Continuity* boundary condition. As the **E-B** Eigen Mode module is built around the Field Flux Bloch Floquet formulation, at least one pair of boundaries must be marked to model the *Periodic Boundary Condition.*

E.I Perfect Electric Conductor

The *Perfect Electric Conductor* boundary condition sets the tangential electric field components and the corresponding tangential components of the periodic electric field envelope to zero. It also set the normal component of the magnetic flux density and periodic field envelope to zero.

$$-\hat{\boldsymbol{n}} \times \hat{\boldsymbol{n}} \times \boldsymbol{E} = \boldsymbol{0} \quad \rightarrow -\hat{\boldsymbol{n}} \times \hat{\boldsymbol{n}} \times \boldsymbol{e} = \boldsymbol{0}$$

$$\hat{\boldsymbol{n}} \cdot \boldsymbol{B} = \boldsymbol{0} \quad \rightarrow -\hat{\boldsymbol{n}} \cdot \boldsymbol{b} = \boldsymbol{0}$$

As such, the *Perfect Electric Conductor* is treated as a homogeneous Dirichlet Boundary condition and the corresponding electric and magnetic field degrees of freedom are not accounted for in the finite element assembly. To mark a boundary modeled as *Perfect Electric Conductor* boundary condition, the user must simply select the Perfect Electric Boundary Condition form the boundary condition drop down selection.

E.II Perfect Magnetic Conductor

The *Perfect Magnetic Conductor* boundary condition sets the tangential magnetic field components to zero and is not directly enforced on the periodic envelope of the intensity of the electric field and the vectorially normal to the boundary's periodic envelope of the magnetic flux density. This choice of boundary condition is equivalent to the *Continuity* boundary condition. To mark a boundary modeled as *Perfect Magnetic*

Conductor boundary condition, the user must simply select the Perfect Magnetic Conductor entry in the boundary condition drop down selection.

E.III Continuity

The *Continuity* boundary condition is the default boundary condition for all interior boundaries in the computational model. This boundary condition simply retains the tangential continuity of the electric field and the normal continuity of the magnetic flux density field on the interface of adjacent elements. To mark a boundary to be modeled as *Continuity* boundary condition, the user must simply select the Continuity entry in the boundary condition drop down selection.

E.IV Absorbing Boundary Condition

The *Absorbing Boundary Condition* is the boundary condition that is used to truncate the computational domain in the open space.

$$\hat{\boldsymbol{n}} \times (\mu_r^{-1}\boldsymbol{b}) + \frac{1}{Z_w}(\hat{\boldsymbol{n}} \times \hat{\boldsymbol{n}} \times \boldsymbol{e}) = \boldsymbol{0}$$

The first order ABC implemented on the **E-B** Eigen Mode module guarantees the reflectionless truncation of the computational domain, when the incident to the ABC boundary electric field is strictly tangential. As the implementation of the ABC boundary condition for media with tensorial electromagnetic properties requires the precise calculation of the complex Wave Impedance of the medium directly adjacent to the boundary, selection of the *Absorbing Boundary Condition*, when truncating bianisotropic media is not advised. To mark a boundary to be modeled as *Absorbing Boundary Condition*, the user must select the Absorbing Boundary Condition entry in the boundary condition drop down selection.

E.V Absorbing Boundary Condition Z

To truncate bianisotropic media with an *Absorbing Boundary Condition* boundary, the user is advised to use the Absorbing Boundary Condition – Impedance, by selecting it in the drop-down selection. For this boundary condition the user is required to input the complex and/or tensorial wave impedance of the bianisotropic medium.

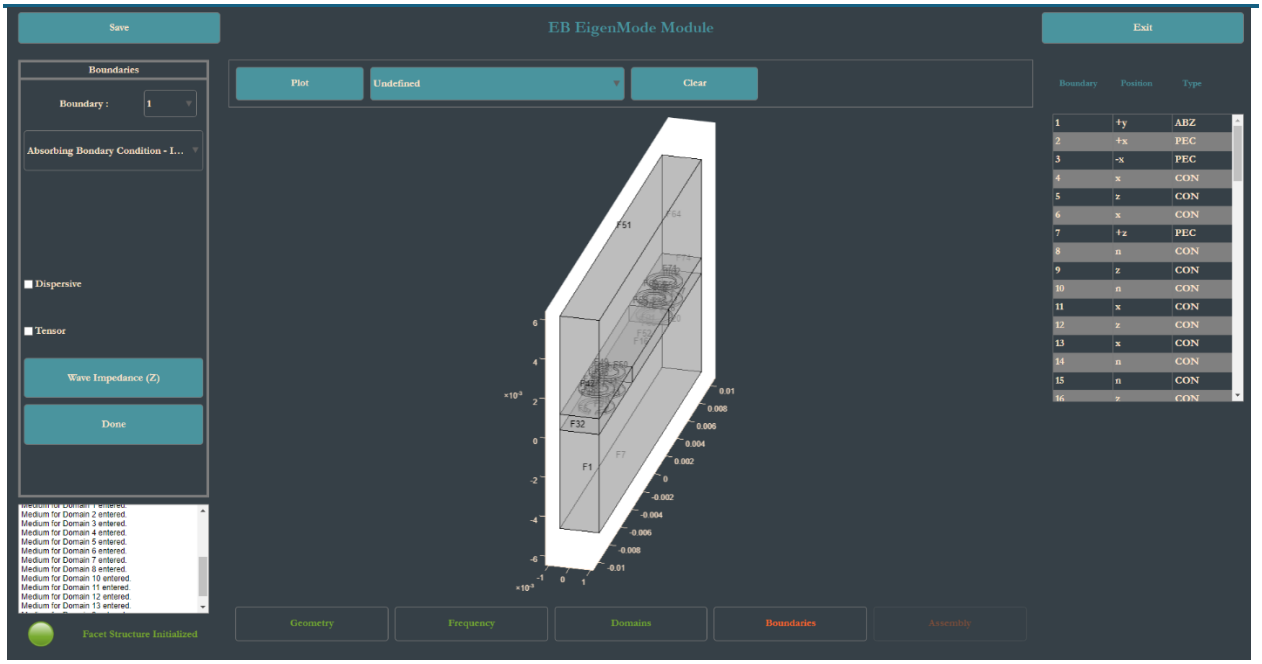

Figure 14. The Absorbing Boundary Condition – Impedance.

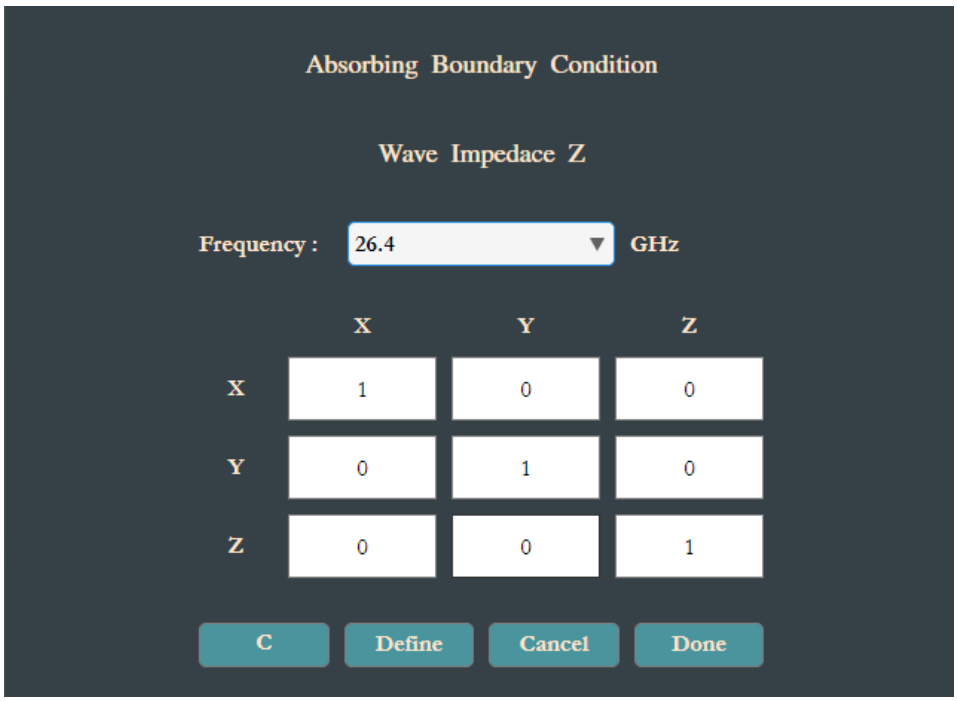

Figure 15 Boundary Tensor Form – Dispersive Tensorial Absorbing Boundary Condition.

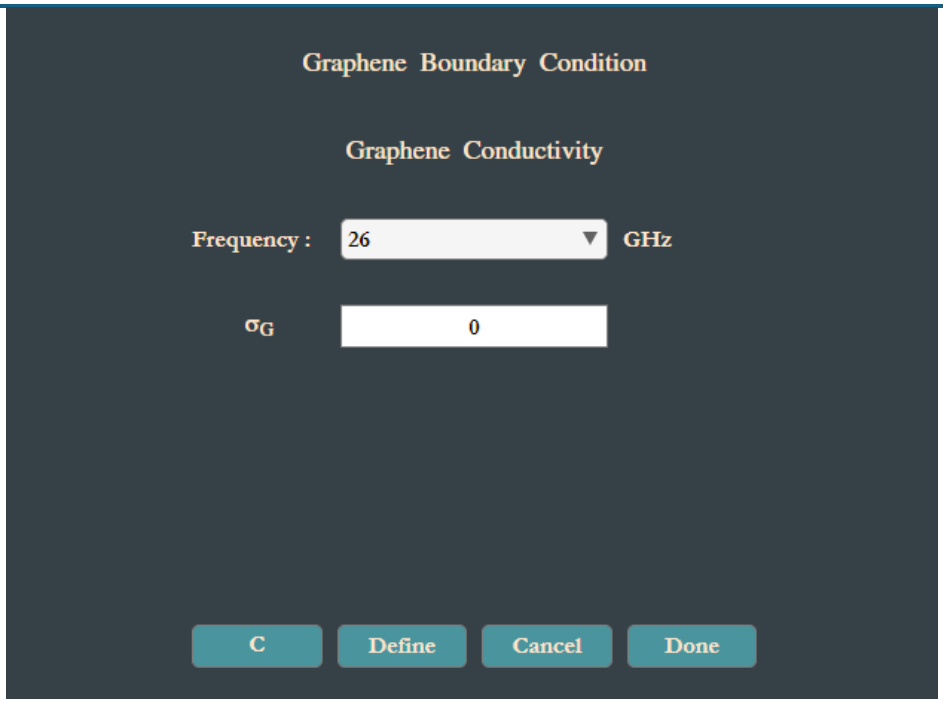

Figure 16 Boundary Tensor Form – Dispersive Graphene Boundary Condition.

If the wave impedance is tensorial the user must select the **Tensor** check box, while if the wave impedance is dispersive, the user must select the **Dispersive** check box. (Figure 14).

To proceed to the input of the tensorial or scalar wave impedance the user must click on the **Wave Impedance (Z)** button, which produces the Boundary Tensor form (Figure 15). As with the Domain Tensor form, any input must be validated by clicking the **Define** button before returning to the **E-B** Eigen Mode module.

E.VI Graphene

The Field Flux Bloch Floquet formulation of the **E-B** Eigen Mode module allows the analysis of periodic structures incorporating 2D Graphene components.

$$\hat{\boldsymbol{n}}_{\boldsymbol{g}} \times \left(\mu_{\boldsymbol{0}}^{-\boldsymbol{1}}\mu_{\boldsymbol{r+}}^{-\boldsymbol{1}}\boldsymbol{b}^{+} - \mu_{\boldsymbol{0}}^{-\boldsymbol{1}}\mu_{\boldsymbol{r-}}^{-\boldsymbol{1}}\boldsymbol{b}^{-}\right) = \sigma_{g}(\hat{\boldsymbol{n}} \times \hat{\boldsymbol{n}} \times \boldsymbol{e})$$

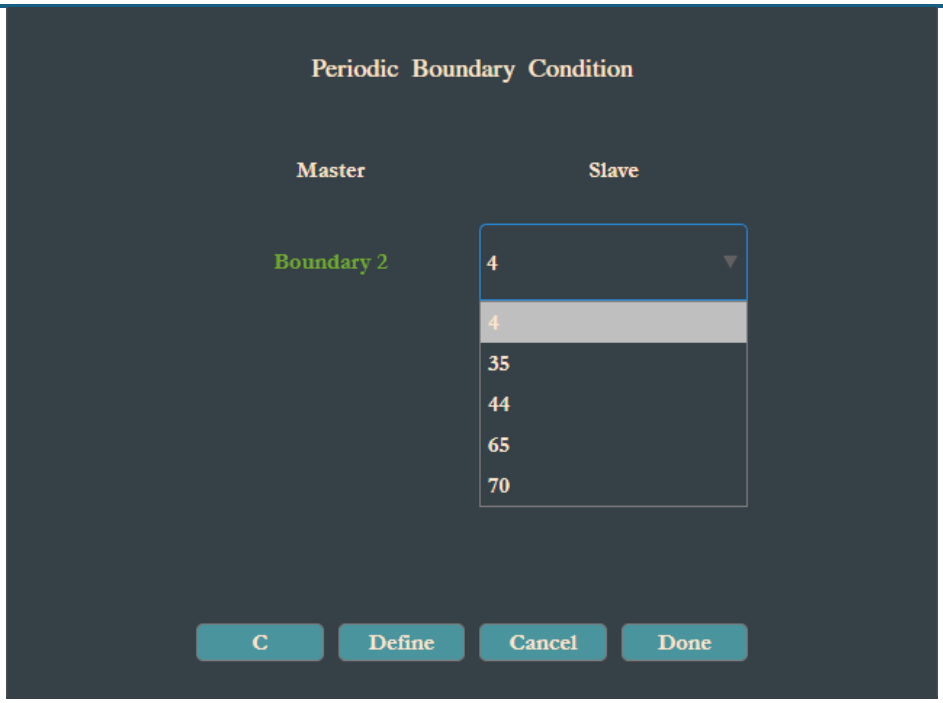

Figure 17 Boundary Tensor Form – Periodic Boundary Condition.

A boundary is modeled as a Graphene boundary condition, when the boundary drop down selection is set to Graphene. The user imports the current boundary's graphene conductivity (a dispersive parameter, when the **Dispersive** check box is selected) by pressing the **Conductivity** button, which prompts the Boundary Tensor Form for the Graphene Boundary Condition (Figure 16). Any input in the Boundary Tensor form must be validated the **Define** button before returning to the **E-B** Eigen Mode Module.

E.VII Periodic Boundary Condition

The *Periodic Boundary Condition* is implemented on the boundaries that truncate the computational domain along an axis of periodicity. Field quantities on a pair of periodic boundaries are equaled to indirectly model the behavior of electromagnetic propagation in periodic media. Implementation of the *Periodic Boundary Condition*, requires identical triangulation on the pair of periodic boundaries and enforcing it to a tetrahedral mesh that does not satisfy this condition, will lead to errors.

To mark the periodic pair of boundaries for *Periodic Boundary Condition*, the user must first locate one of the two periodic boundaries and select it as a Periodic Boundary Condition boundary in the drop-down selection. The boundary's pair is then selected by clicking on the **Periodic Pair** button, which prompts the Boundary Tensor Form for periodic boundaries. The computational domain's external boundaries that exist on the opposite side of the periodicity axis are listed in the drop-down selection. The user selects the one that satisfies the periodic boundary requirements, validates the selection by clicking on the Define button and exits to the **E-B** Eigen Mode Module.

E.VIII Impedance Boundary Condition

The Impedance Boundary Condition can be utilized to produce more realistic models by substituting the lossless *Perfect Electric Conductor* boundary condition, when modeling metallic surfaces. The Impedance Boundary Condition introduces the Ohmic losses present when a finite conductivity surface is present in the computational domain.

$$\hat{\boldsymbol{n}} \times (\mu_r^{-1}\boldsymbol{b}) + \frac{1}{Z_s}(\hat{\boldsymbol{n}} \times \hat{\boldsymbol{n}} \times \boldsymbol{e}) = \boldsymbol{0}$$

In order to treat a metallic surface with the *Impedance Boundary Condition*, the surface must be designed with thickness (the geometry of the metallic surface must be a volume and not a two-dimensional surface) and all boundaries enclosing the volume marked to be modeled as a Impedance Boundary Condition. The domain of the metallic volume must be set as an empty domain in the Domains panel.

To assign a boundary to the *Impedance Boundary Condition*, the Impedance Boundary Condition is selected in the drop-down selection. The surface impedance of the boundary is set via the Boundary Tensor form by clicking the **Surface Impedance** button.

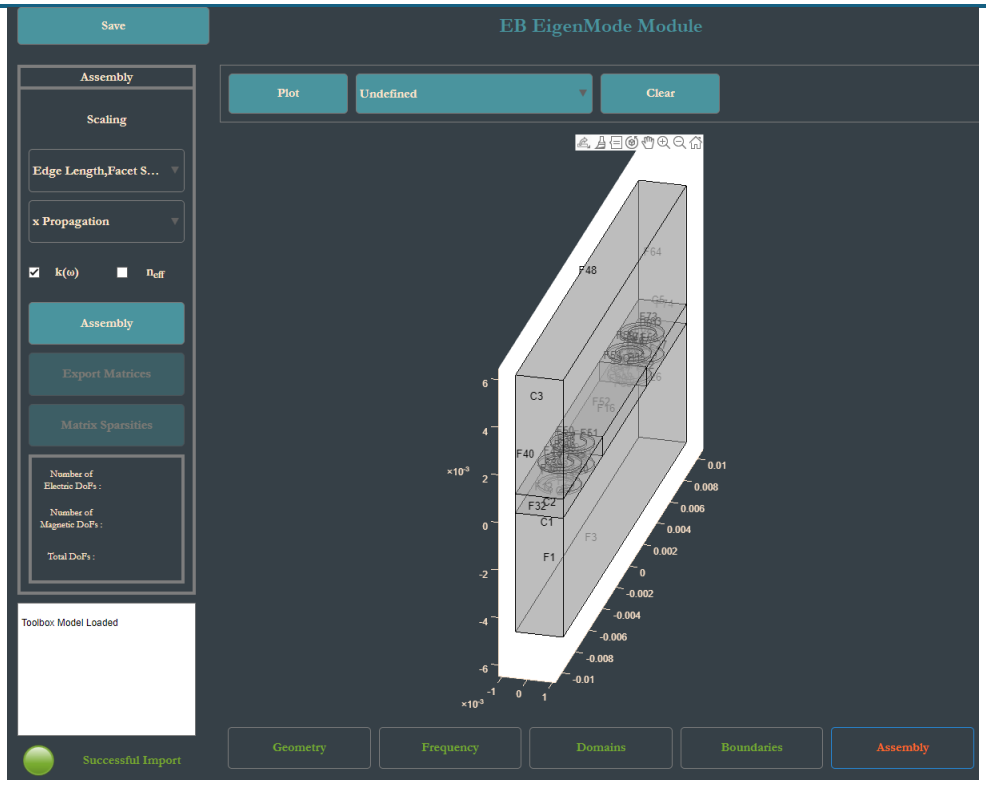

Figure 18 Assembly Panel

F.Assembly

The final preprocessing step in the **E-B** Eigen Mode module is the construction of the finite element matrices via the assembly process. The **E-B** Eigen Mode handles all user input regarding the assembly process in the Assembly panel (Figure 18) located in the left panel area. The **E-B** Eigen Mode module allows for four different basis function scaling strategies (the basis functions discretizing the periodic electric field and magnetic field envelopes are weighted with the tetrahedral mesh edge's length and facet's surface). These scaling strategies can be utilized to improve the algebraic characteristics of the eigen problem and hasten the convergence. The user can select any of the four strategies in the upper drop-down selection and the corresponding effects are listed in Table II.

The axis aligned to the periodic electromagnetic propagation is selected in the lower drop-down selection. The user can select propagation along the $\hat{\boldsymbol{x}}$, $\hat{\boldsymbol{y}}$ or $\hat{\boldsymbol{z}}$ axis, provided that *Periodic Boundary Condition* boundaries truncate the computational domain along the axis.

Table II E-B Eigen Mode Module Strategies

Scaling Strategy	*Effect on basis function for* **e**	*Effect on basis function for* **b**
Edge Length, Facet Surface	Edge Length multiplication	Facet Length multiplication
Edge Length	Edge Length multiplication	none
Facet Surface	none	Facet Length multiplication
None	none	none

The user can also select on the type of the problem's eigenvalue, with the two alternatives being the norm of the Bloch Floquet wave vector k and the effective refractive index $n_{eff.}$ The latter option can be utilized to improve the search for eigenvalues, when the algebraic eigen problem corresponds to a spectrum with very large wave vectors. To switch between the two types of eigenvalues the user selects or deselects the **k** and $\mathbf{n_{eff}}$ checks. The assembly process is initiated by engaging the **Assemble** button.

Any change to the assembly parameters, after the initiation of the assembly process is void and in order to be valid the **Assemble** button must be re-engaged, restarting the assembly process. When the assembly process is initiated, the dimension of the algebraic problem and the number of the electric and magnetic degrees of freedom is displayed in the Assembly panel, while upon the successful conclusion of the assembly process, the **Export Matrices** button, which triggers the export (in .mat format) of the finite element matrices along with the finite element structures of the tetrahedral mesh edges and facets, and the **Matrix Sparsities** button, which produces visualizations of the finite element matrices sparsity, are enabled. Finally, the solution panel on the left panel area of the **E**-**B** Eigen Mode module is enabled and ready to be used.

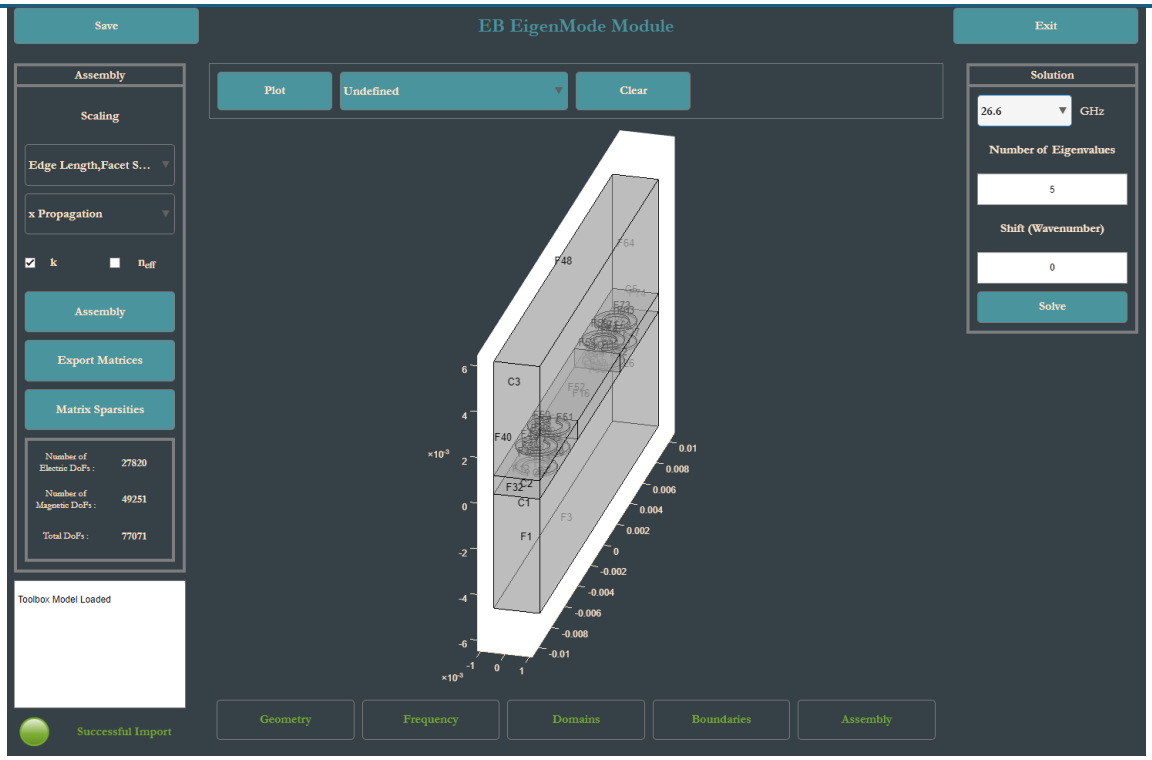

Figure 19 E-B Eigen Mode Module - Solution Panel.

G. Solution

The solution panel occupies the top of the right panel area and controls the parameters of the solution of the generalized eigenvalue problem. The Field Flux Bloch Floquet formulation leads to the pencil matrix eigenvalue problem:

$$\{[A] - \lambda[B]\}\begin{bmatrix} \boldsymbol{e} \\ \boldsymbol{b} \end{bmatrix} = 0,$$

where the matrices $[A]$ and $[B]$ are the finite element matrices formed in the assembly process, $[\boldsymbol{e} \quad \boldsymbol{b}]^t$ is the eigen vector and λ (either the Bloch Floquet wavevector $\boldsymbol{k}$ or the effective refractive index n_{eff}) is the eigenvalue of the problem.

When the **E**-**B** Eigen Mode module is set for single frequency operation, the user can specify the number of eigenvalues and eigenvectors to be computed by the solution and the eigenvalue shift, a scalar value (real or complex). Solving the eigenvalue problem for N eigenvalues with sigma eigenvalue shift will return the N eigenvalues closest to the sigma values and their corresponding eigenvectors. In the case of a frequency range operation, the user can opt for the solution of a single frequency or the

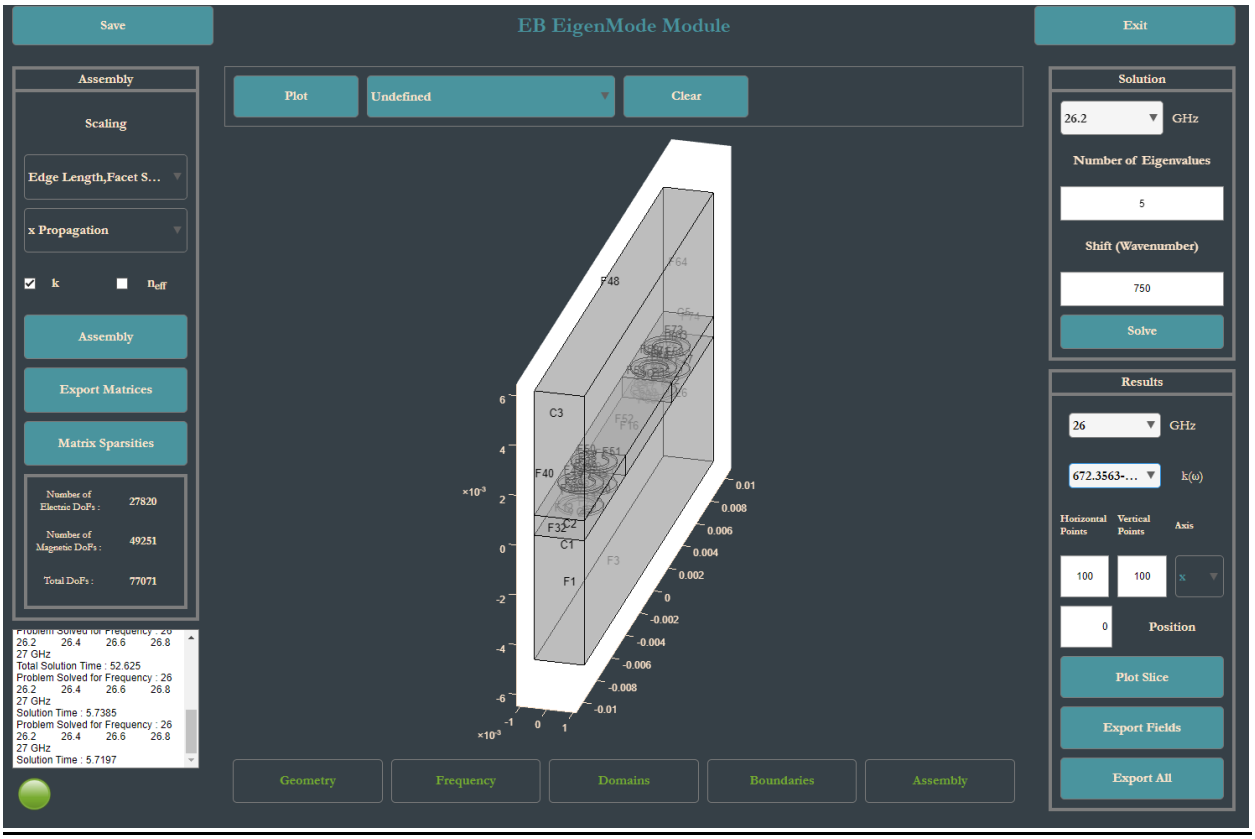

Figure 20. E-B Eigen Mode Module - Results Panel.

entire frequency range (**Solve All Frequencies** entry) by the drop-down selection on the top of the Solution Panel (Figure 19).

H. Results

The results panel occupies the lower part of the right panel area. The top of the results panel contains two drop-down selections. The upper one is enabled when the **E-B** Eigen Mode module operates in the frequency range setting and lists the frequencies of the frequency range, while the lower drop-down selection lists the computed eigenvalues for the drop-down selected frequency in the frequency range. If the **E-B** Eigen Mode module operates in the single frequency setting, the panel's frequency drop down is disabled and the eigenvalue drop-down selection lists all the model's computed eigenvalues. The computed eigenvectors are examined visually in plane slices by selecting the corresponding eigenvalue in the drop-down selection and

engaging the **Plot Slice** button. The real and imaginary components of the complex intensity of the electric field **E** and the magnetic flux density **B** are evaluated in a stencil of N_h by N_v equidistant points located on a cross-section of the unit cell. The number of horizontal points N_h and the number of vertical points N_v are specified in the **Horizontal Points** and **Vertical Points** fields, whereas the orientation of the plane is set by the **Axis** drop-down selection, while the coordinate of the plane on the selected axis by the **Position** field.

The electric and magnetic field components, along with the coordinates of the stencil's points, can be exported by engaging the **Export Field** button, while the **Export All** button exports all eigenvalue and eigenvector pairs in the solution.

I. Saving and Loading a Model

At any point in the model's design process, all progress and model parameters can be saved by engaging the Save button on the top left corner of the **E**-**B** Eigen Mode module. These model files (in. mat format) and all the model's properties can be manipulated in the Mathworks MATLAB environment using the E-B Toolbox Classes (see **E**-**B** Toolbox Classes). Importing a saved model in the **E**-**B** Eigen Mode module is achieved by selecting the **Toolbox Model** button in the Geometry Panel (Figure 1) and engaging the **Load** button.

References

[1] Nitas, M., Salonikios, V., Antonopoulos, C.S. and Yioultsis, T.V., 2019. Numerical calculation of dispersion diagrams and field distributions of waves in 3-D periodic split-ring resonator media. IEEE Transactions on Magnetics, 55(12), pp.1-4.

[2] Nitas, M., Salonikios, V. and Yioultsis, T.V., 2022. Alternative finite element eigenvalue formulations for the simulation of arbitrarily bianisotropic media. IEEE Transactions on Microwave Theory and Techniques, 71(2), pp.570-578.

[3] Salonikios, V., Nitas, M., Raptis, S. and Yioultsis, T.V., 2020. Computational analysis of graphene-based periodic structures via a three-dimensional field-flux eigenmode finite element formulation. Progress In Electromagnetics Research M, 92, pp.157-167.

[4] "Υπολογιστική ανάλυση, σχεδίαση και χαρακτηρισμός περιοδικών δομών μεταϋλικών και σύνθεση διατάξεων κυματοδήγησης και επίπεδων κεραιών για μικροκυματικές συχνότητες και ασύρματες επικοινωνίες 5G", Μιχάλης Δ. Νήτας (Ph.D Thesis) (in Greek)

[5] "Ανάπτυξη Υπολογιστικών Τεχνικών με τη Χρήση Διατυπώσεων Μεικτών Πεπερασμένων Στοιχείων στο Σύστημα Εξισώσεων του Maxwell για Προβλήματα Ηλεκτρομαγνητικής Διάδοσης", Σαλονικιός Ν. Βασίλειος,2024 (Ph.D. Thesis)(in Greek)

[6] https://www.mathworks.com/help/pde/ug/pde.pdemodel.geometryfrommesh.html