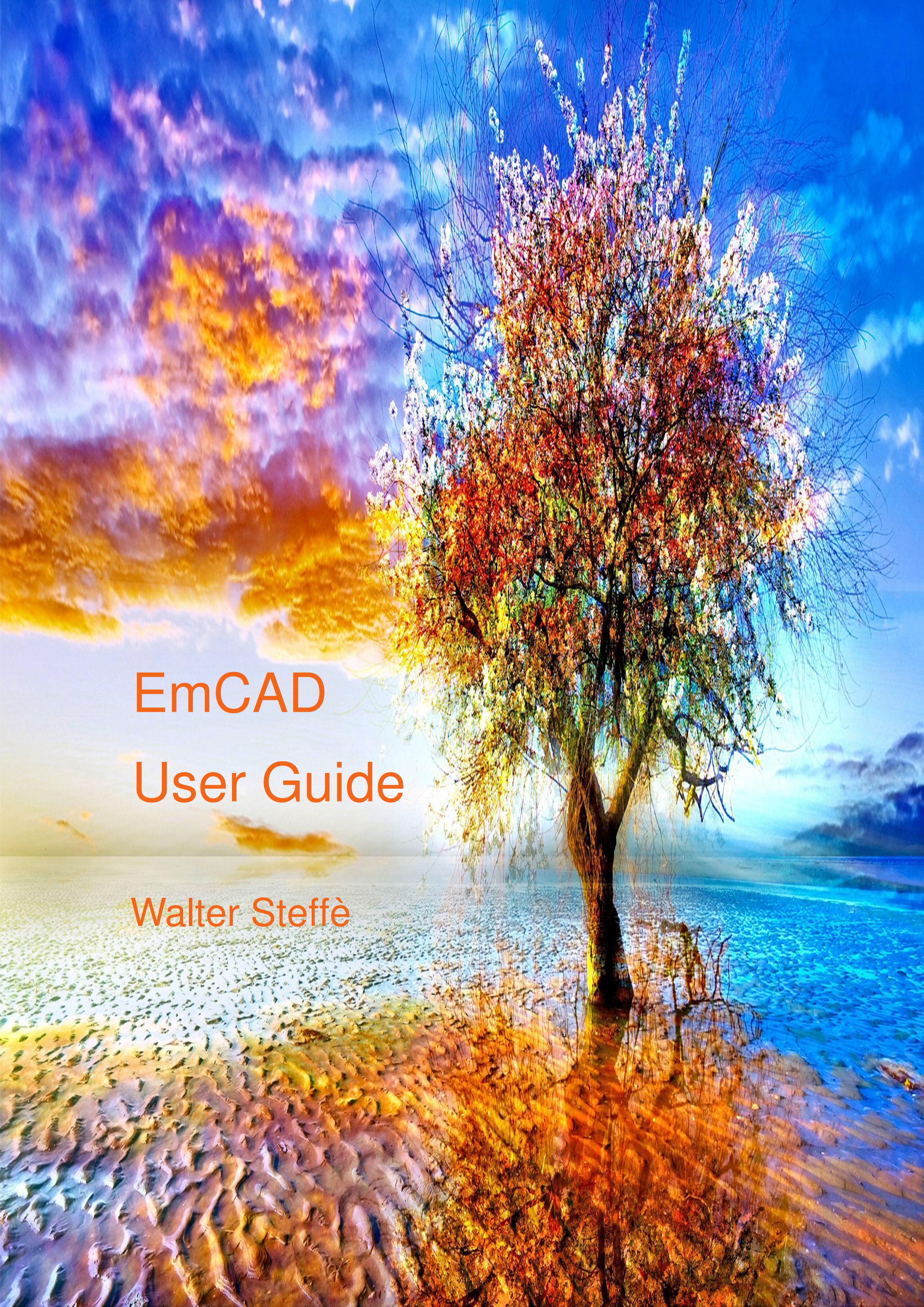


EmCAD User Guide

Walter Steffè

Hierarchical
Electromagnetics

FEEDBACK
Please direct any comments or suggestions about this document to:
walter.steffe@hierarchical-electromagnetics.com

PUBLICATION DATE
First printing, July 2015

Contents

1. Introduction

1.1 What is EmCAD ?

EmCAD is the graphical front-end of a client-server application for the numerical modelization of electromagnetic problems. *EmCAD* is Copyright © 2015 Walter Steffè and is distributed under the terms of the GNU General Public License, Version 2 or later.

The actual modelization work is performed by means of the *Hierarchical Electromagnetic Modeler*. This is a proprietary code (owned by Walter Steffè) which runs on a cloud platform and will be delivered as a service on demand. *EmCAD* provides the graphical environment for the definition of the electromagnetic problem. The problem data are then enclosed in a file document which is sent to the remote server to be processed by the *Hierarchical Electromagnetic Modeler*.

The *Hierarchical Electromagnetic Modeler* is based on a new numerical method which allows the generation of an equivalent circuit that represents the electrical response of a generic electromagnetic problem (microwave, optical ..). The circuit is generated in a direct path without going trough an electromagnetic simulation and a subsequent circuit extraction process as it happens with the macro-modeling techniques based on the zero-pole fitting of response curves. The new method is more robust than the zero-pole fitting which does not scale well when the number of ports becomes large. The new method can in fact handle very large electromagnetic structures excited by a large number of input ports.

The generated circuit is described in the standard Spice format assuring the interoperability with most circuit simulators. It preserves the fundamental properties (such as passivity, causality, reciprocity..) of the actual device so that it can be exploited for reliable and stable time domain simulations.

The *Hierarchical Electromagnetic Modeler* is designed to achieve the maximum efficiency when executed in a distributed environment and to exploit the full power offered by the modern cloud computing architectures. In this way it is able to tackle the complex electromagnetic problems which arise in the design flow of modern microwave/electronic industry and to overcome the constraints imposed by limited local resources.

Because of the steady increase of system performance, operating frequencies and miniaturization, the need of accurate modeling of electromagnetic interactions occurring inside the electronic circuits is becoming always more important. Higher frequencies and smaller spaces lead indeed to an increase of the electromagnetic interactions causing a degradation of the quality of electrical signals (noise effects, crosstalk..).

These adverse effects need to be properly simulated at system level in order to appreciate their global effect on the system performances. The *Hierarchical Electromagnetic Modeler* responds to these needs trough the generation of circuit models that capture all the relevant electromagnetic effects and are well suited to be exploited in any kind of simulation in the time or in the frequency domain.

The *Hierarchical Electromagnetic Modeler* is able to deal with dispersive materials which can be defined using several linear models (Lorentz, Debye, Drude . . .).

The equivalent circuit can be integrated inside a complex system (which may include non linear devices) providing a better interoperability than the simulated responses computed with traditional Electromagnetic Solvers. These responses are indeed typically given in the frequency domain and their utilization in the time domain requires the application of heavy numerical algorithms (domain transforms and convolutions).

The circuit generated by the *Hierarchical Electromagnetic Modeler* represents the electrical response of the actual device over a large frequency band which starts at the zero frequency and goes up to a maximum frequency that is controlled trough the proper setting of the mesh size. The capability of handling, at the same time, the high frequency response and the DC response can, in example, be exploited to analyze the impact on the system performance of the noise effects (such as the simultaneous switching noise) which are generated trough the interaction between the “Signal Delivery Network” and the “Power Delivery Network”. This is a challenging Signal Integrity problem with a strong impact on the performances of a mixed analog-digital devices that is requested to operate at high frequencies.

The most distinguishing property which puts the *Hierarchical Electromagnetic Modeler* aside of competition is nevertheless given by the adoption of a very efficient domain decomposition strategy. This technique allows to overcome the fundamental drawback of all other current general purpose full-wave simulators which, by going against the hierarchical (top-down) design methodologies that the electronics industry is accustomed to, treat the electromagnetic device as an unstructured black box.

The *Hierarchical Electromagnetic Modeler* decomposes the device in its constituent parts (sub-domains) and these parts are subjected to independent modelizations which produce the related sub-circuits. As a result there is a one to one relationship between the hierarchical structure of the sub-components and the hierarchical structure of the related equivalent circuits. This property allows a dramatic improvement in the computational speed of the repetitive runs which are typically required in the design/optimization phase. In fact, thanks to the preservation of the hierarchical structure of the EM problem, a local change in the geometrical or physical parameters of a given subcomponent affects only the related sub-circuit and there is no need to recompute the sub-circuits associated with the other (unchanged) parts.

Actually it is felt that the inability of the present EM tools to cope with a hierarchical design methodology has been, up to know, the principal barrier for their wider adoption. The utilization of a traditional EM solver for the analysis of a complex circuit is a heavy task which requires large computational resources and long run times. Because of that, in many practical situation, this kind of tool is relegated into the final post layout verification phase and can not be fully exploited in the design/optimization phase. We believe that the introduction of the *Hierarchical Electromagnetic Modeler* will fill this gap and will allow a proper handling of the electromagnetic effects since the initial phases of the design flow.

1.2 The Design Flow

EmCAD and the *Hierarchical Electromagnetic Modeler* are not self standing applications and have to be used in cooperation with other software tools. The design flow should include, at least, a CAD system for the definition of the geometrical structure of the electromagnetic problem. The CAD geometry is transferred into the EmCAD envirinment using a STEP file[1].

Depending on the particular application, the design flow may involve other software tools. In example it may include an external simulator used for custom (application dependent) circuital analyses. A typical design flow which makes use of EmCAD and *Hierarchical Electromagnetic Modeler* (*HEM*) is illustrated in the Table 1.1.

In the frame of a design/optimization activity the design flow may be iterated several times by changing, in example, some of the geometrical parameters defined inside the CAD environment. In this situation EmCAD will assure that all the data which were associated to the initial geometry (material properties, waveguide ports, boundary conditions) are preserved and re-associated to the updated geometry.

[1]STEP-File is a widely used data exchange format. It is an ISO standard defined in the document ISO 10303-21 “ Industrial automation systems and integration – Product data representation and exchange – Part 21: Clear Text Encoding of the Exchange Structure“.

Tool	Tasks
CAD	**Geometry Definition:** -The geometrical structure of the electromagnetic problem is an assembly of components which are defined as multi-body parts using a commercial CAD system (in example CATIA V5). -A typical component includes different solids (dielectrics) and surfaces. The latter are used to define boundary conditions, waveguide ports and cutting surfaces. -The CAD data are exported as step files.
EmCAD	**Component Definition:** -The step file associated with a component is imported inside EmCAD. -Different object inside a multi-body part are automatically recognized from a naming convention based on a string prefix. -Electrical properties of materials are imported from a library or created by the user. -Dielectric objects, waveguide ports and boundary surfaces are associated with a material or a boundary condition selected from the material list. -Number of modes are specified at the w.g. ports. -Maximum frequency and mesh-size are specified in the project properties. -Mesh files are generated using “Decompose“ and “Mesh“ commands. -Data are saved into the component EmCAD file.
EmCAD	**Assembly Definition:** -The step file associated with the assembly is imported inside EmCAD. -Sub-component properties are imported from related EmCAD files. -Maximum frequency and mesh-size are specified in the project properties. -Mesh files are generated using “Decompose“ and “Mesh“ commands. -Data are saved into the assembly EmCAD file. -Mesh Files are sent to the remote server starting the modelization.

HEM	**Modeling:** -The modelization request is processed by the server which splits the job in a set of subtasks. -Modelization tasks are executed in parallel on a clusters of computing nodes. -The server sends back a set of circuits which represent the electrical response of assembly components.
EmCAD	**Analysis Definition:** -A frequency band is specified for the Frequency Response Analysis. Analysis band should not exceed that one specified for modelization tasks. -The Analyze command is used to start the computation of the frequency response on the remote server.
HEM	**Analysis Process:** -The frequency domain analysis is preceded by a Model Order Reduction which produces a compact equivalent circuit associated with the electromagnetic structure. -The frequency response is computed on the reduced model. -The generated data (global circuit and related frequency response) are sent back to he EmCAD client.
EmCAD	**Local Processing of Results :** -The the frequency response curves can be plotted on a graphic window or can be exported in a Touchstone file. -The reduced circuit can be exported in a Spice file.

Table 1.1: A typical design flow

1.3 Configuring the connection with the Modeler Service

In order to configure a connection with the *Hierarchical Electromagnetic Modeler* it is necessary to create an account with Amazon Web Services and perform the following actions by means of the IAM management console:

1. Subscribe for the Amazon Storage Service (S3).
2. Subscribe for the Amazon Simple Queue Service (SQS).
3. Create an S3 bucket inside the US-east AWS region. A bucket is the term used in the S3 jargon to indicates a user owned directory which is located inside the S3 storage area. The bucket name is chosen by the user but must be unique across all existing bucket names in Amazon S3. One way to do that is to prefix the bucket name with the user/company's name.
4. Create a bucket policy which allows read/write access on the new bucket to the *Hierarchical Electromagnetic Modeler*. This action can be accomplished using the S3 Management Console provided by AWS. After having selected the bucket press the "Properties", "Permissions" and "Add Bucket Policy" buttons. The latter will open and editable windows where you should paste the following text including the brackets after having replaced BUCKET-NAME with the name you have choosen:

```
{
 "Version": "2012-10-17",
 "Statement":
 [
  {
   "Sid": "BucketAccessToEmCAD_Server",
   "Effect": "Allow",
   "Principal": {"AWS":
   "arn:aws:iam::hierarchical-electromagnetics:user/EmCAD_Server"},
   "Action": ["s3:*"]
   "Resource": [
     "arn:aws:s3:::BUCKET-NAME",
     "arn:aws:s3:::BUCKET-NAME/*"
    ]
  }
 ]
}
```

5. Create a new group of EmCAD users and attach to it a policy which allows the reading and writing on the S3 bucket of your EmCAD projects and the usage of the SQS queue. This is accomplished by pressing the Create Group Policy button inside the IAM section aof the AWS management console. Then you may select the "Custom Policy" option which allows to paste the following text:

```
{
 "Version": "2012-10-17",
 "Statement":
 [
  {
    "Sid": "BucketAccess",
    "Effect": "Allow",
    "Action": "s3:*",
    "Resource": [
```

```
            "arn:aws:s3:::BUCKET-NAME",
            "arn:aws:s3:::BUCKET-NAME/*"
         ]
      },
      {
        "Sid": "EmCAD-queue-Access",
        "Effect": "Allow",
        "Action": "sqs:SendMessage",
        "Resource":
        "arn:aws:sqs:us-east-1:070650497580:EmCAD"
      }
    ]
  }
```

6. Create a new USER and put it inside the EmCAD users group.
7. Create the access keys of the new user. This is accomplished by pressing the Create Access Key button inside the Users menu of the IAM management console. As a result you will get a couple of alphanumeric strings:
   - The *AWS ACCESS KEY ID*.
   - The *AWS SECRET ACCESS KEY*.
8. Download the credentials file which includes the two keys and save it in a secure location.
9. Configure the AWS connection using the "Edit/Configuration" command from the EmCAD environment. This command will open a window like that one shown in the figure 1.1 where you are required to enter:
   - the BUCKET NAME
   - the AWS ACCESS KEY ID
   - the AWS SECRET ACCESS KEY.
10. Send a request to request@hierarchical-electromagnetics.com to insert your user ARN among the authorized users of the *Hierarchical Electromagnetic Modeler*[2]

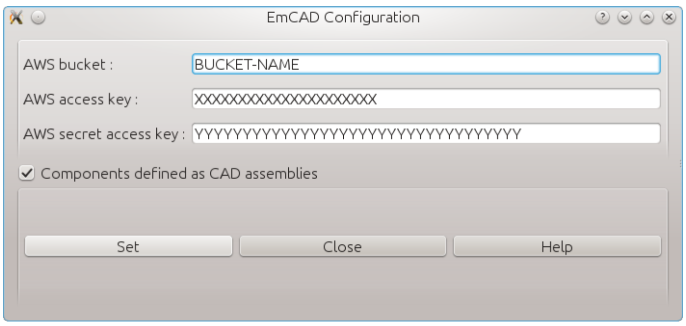


Figure 1.1: Configuration

[2]The *Hierarchical Electromagnetic Modeler* is still in an experimental phase and is not yet available for general usage. At the moment the service is restricted to a small number of partners which are willing to contribute to the development of the EmCAD project.

2. General Usage

2.1 The Graphical User Interface

The figure 2.1 shows the main workspace of the EmCAD Graphical User Interface as it appears after having imported a CAD assembly that describes a microwave filter. This filter is composed of five coaxial resonators described by multi-body parts defined inside the Catia CAD system.

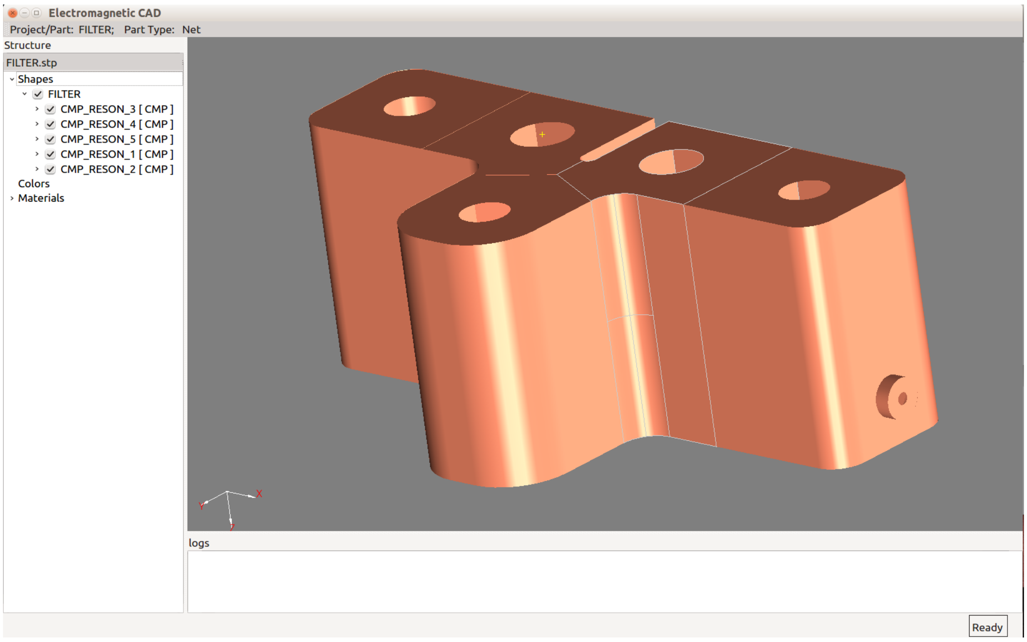


Figure 2.1: The main workspace

2.1.1 The Main Workspace Structure

As shown in Figure 2.1 the main workspace of EmCAD comprises the following items:

The Menu-bar

The menu-bar, which is located across the top of the main window, provides a set of commands. These are distributed among a set of drop-down menus. The utilization of the related commands is explained in full detail in the next sections.

The Geometry Viewer

The geometry viewer is located at the right side area below the menu-bar. This tool displays a 3 dimensional perspective of the imported geometries. The aspect of the displayed objects can be adjusted using the “rotate”, “pan” and “zoom” commands located inside the View Menu. The same commands can also be activated trough the shortcut defined in table 2. The objects displayed in the geometry viewer are mouse sensitive so that the clicking of a given object activates and highlights the corresponding item inside the Structure Tree (see below). This selection mechanism allows an easy identification of all the data (such as the name, the material...) associated with a given geometrical object.

Shortcut	**Effect**
Mouse movement pressing LMB	Object rotation
Mouse movement pressing shift+LMB	Object translation
Rotation of mouse wheel	Zooming

Table 2.1: Shortcuts for Geometry Viewer

The Structure Tree

The structure tree, which is located at the left side of the main window, gives a hierarchal view of the graphical objects (parts, solids, faces, edges) displayed in the viewer window. The tree items can be checked/unchecked by clicking inside a small box. The unchecked objects are hidden (not displayed in the viewer window). In this way it is possible to make visible an internal part/object which otherwise would be obscured by other objects.

Beside of the geometrical objects the structure tree includes also the list of materials imported from the library or defined by the user. All the tree items are mouse sensitive, so that, in example, clicking inside a geometrical item activates and highlights also the corresponding geometrical object in the viewer window. This selection mechanism goes in the opposite direction to that illustrated before allowing an easy identification of the geometrical object associated, in example, with a given part name. The selection of a material item and of a dielectric body (or a boundary conditions) allows an easy association of the two objects.

The Message area

The Message area is located at the lower right side of the main window. It is used to display messages describing the status (or eventually failure) of the running tasks.

2.1.2 Browsing The Assembly Structure

The “Open Sub-Assembly” and “Open Upper Assembly” commands whir are located in the “File” menu allow to navigate inside the assembly structure associated with the imported CAD geometry. By entering the “Open Sub-Assembly” it is possible to enter into the data structure associated with the sub-component selected in the Structure Tree. As a consequence of this action the geometry viewer window displays the geometry associated with the sub-component instead of that one associated with the assembly. In a similar way the tree-structure window displays the data structure associated with the sub-component instead of that one associated with the assembly. The sub-component view can be closed going back to the assembly view by entering “Open Upper Assembly”. The “Open Sub-Assembly” command can also be entered a context menu which drops down when pressing the Right Mouse Button on the selected component.

2.2 Defining a Material

A new material is created and added ti the material list using the “Material/Create Modify” command. This action opens the window shown in fig 2 where it is possible to define the name, color and the main electrical properties of the new material.

2.2.1 Non dispersive materials

The fig 2.2 shows the definition of a material which is (or may be regarded as) non dispersive so that it is fully defined by specifying the relative electric and magnetic constants.

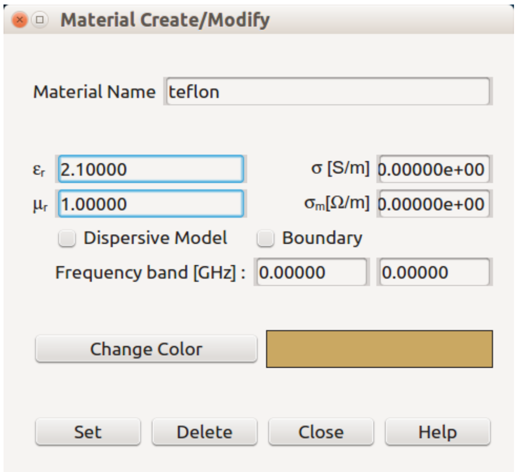


Figure 2.2: Non dispersive material

2.2.2 Dispersive materials

A dispersive material can be defined by checking the Dispersive Model flag inside the “Material Create/Modify” window. The window is then expanded with additional fields (as shown in fig 2.3).

To assure the physical realizability and the passivity of the equivalent circuits, the electric and magnetic constants $\varepsilon(\omega)$, $\mu(\omega)$ can not be arbitrary complex functions of frequency. The realizability condition is enforced by imposing that $\varepsilon(\omega)$ and $\mu(\omega)$ are rational functions expressed by a proper pole/residue expansion. An expansion term associated with a couple of conjugate poles generates a dispersion law denoted as a Lorentz model (defined by eq 2.2) while a simple (real) pole generates a Debye model (defined by eq 2.1.

$$\varepsilon_D(\omega) = \frac{\varepsilon_0 \Delta\varepsilon_r}{1 + j\omega/\omega_r} \qquad \omega_r = \text{Relaxation Frequency} \tag{2.1}$$

$$\varepsilon_L(\omega) = \frac{\varepsilon_0 \Delta\varepsilon_r}{1 + j\omega/\omega_r - (\omega/\omega_0)^2} \qquad \omega_0 = \text{Resonant Frequency} \tag{2.2}$$

The *Hierarchical Electromagnetic Modeler* can handle dispersive materials which are described as a superposition of Lorentz, Debye, constant and conductive terms as expressed by the following equation:

$$\varepsilon(\omega) = \varepsilon_0 \varepsilon_{r\infty} - j\sigma/\omega + \sum_i \varepsilon_{Di}(\omega) + \sum_i \varepsilon_{Li}(\omega) \tag{2.3}$$

This kind of representation allows an accurate modelization of all practical linear materials. The example given in fig 3 refers to the FR4 material which, as shown in [2]; can be well approximated as the superposition of a conductive term and a single Lorentz term characterized by a relaxation frequency of 7.801 GHz and a resonant frequency of 39.5 GHz. By changing the value of “Terms Number” field in the “Material Create/Modify” window it is possible to spcify an arbitrary number of Lorentz/Debye terms. The terms are selected by entering a “0” value in the last column (resonant frequency=0). In a similar way it is possible to specify the dispersion low of the magnetic constant $\mu(\omega)$. The Lorentz and Debye models associated with $\mu(\omega)$ are defined by a couple of equations which are identical to equations 2.2 and 2.1.

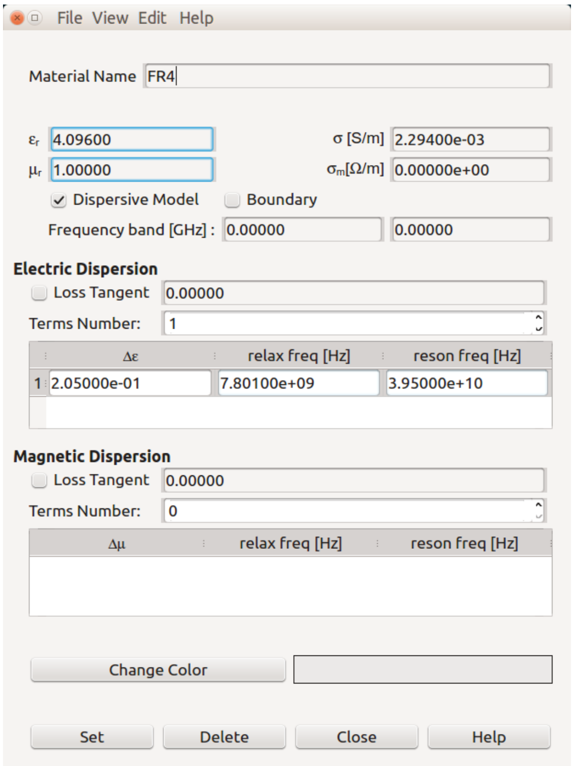


Figure 2.3: Dispersive material

2.2.3 Constant Loss Tangent

Quite often the electrical losses of a material are specified by giving the loss tangent (or $tan\,\delta$) which is defined by equation 2.4.

$$tan\,\delta = -Imag\,(\varepsilon)\,/Real\,(\varepsilon) \tag{2.4}$$

The specified loss tangent is often regarded as a constant value over the whole frequency band of interest. A strictly constant loss tangent is not physically realizable but can be approximated over a finite bandwidth by a proper superposition of a set of Debye terms.

The EmCAD tool allows to compute in an automatic way this approximate material model by checking the Loss Tangent box inside the “Material Create/Modify” window. The example shown in fig 2.4 refers to a lossy Teflon material which has been defined by specifying an (approximate) constant loss tangent of $3 * 10^{-4}$ over the frequency band from 10 to 15 GHz.

The number of Debye terms used in this approximation has been set to 3. The approximation improves by increasing this number.

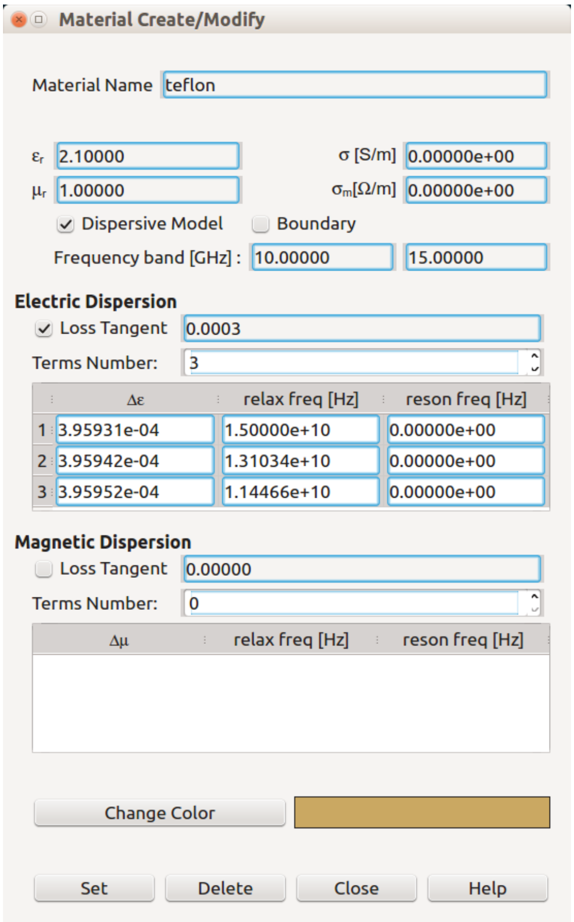


Figure 2.4: Constant Loss Tangent

2.2.4 Resistive surface

Beside of the definition of bulk materials, the “Material Create/Modify” window can be used also to define a Boundary Condition. The boundary conditions are displayed in the material list (under the structure tree) but must be used in a different way than the ordinary materials. The boundary conditions can in fact be associated only with surface objects (with associated BC type) while the ordinary materials can be associated only with solid objects (with associated DIEL type).

Two boundary conditions are already predefined and are the Perfect Electric Conductor (“PEC”) and the Perfect Magnetic Conductor (“PMC”). Additional boundary conditions can created by checking the Boundary flag inside the “Material Create/Modify” window.

The window is the expanded with an additional field (as shown in fig 3) which allows to define a surface resistance.

At the moment the resistive surface is the only type of boundary condition handles by the *Hierarchical Electromagnetic Modeler*. In the next future the modeler will be upgraded with the introduction of new boundary conditions like the Perfect Absorbing Boundary which will allow tocharacterize an open (radiating) boundary.

2.3 Defining a Component

2.3.1 Defining the Component Geometry

The component geometry is defined in a CAD system as multi-body part that may include different Solid objects (dielectrics), surface objects and line objects. The surface objects can be used to define boundary conditions, waveguide ports and cutting surfaces. The splitting surfaces are used by the modeler to decompose the geometry in a number of elementary cells. The line objects can be used to define line ports. The component geometry is then saved in a step file.

R *The cells generated by the cutting surfaces must be simply connected (no holes) and their sizes should be small in term of wavelength but larger than the mesh-size (in example cell-size=wavelength/8, mesh-size=wavelength/20).*

R *A line ports must always be placed on the boundary of the elementary cells. This can be assured by placing the line object on the boundary of a solid object or on a cutting surface.*

A new Component is created with with the “File/New” command and the step file is then imported in this component using the “File/Import Geometry” command.

Different object in the multi-body part are automatically recognized by EmCAD thanks to a naming convention (to be followed in the preparation of the CAD data) that is defined in the table 2.2.

Name Prefix	**Object Type**
DIEL_	Dielectric
BC_	Boundary Condition
WG_	Waveguide Port
LP_	Line Port
SPLIT_	Partitioning surface
GRID_	Grid of partitioning surfaces

Table 2.2: CAD Naming Convention

2.3.2 Associating Materials and Boundary Conditions

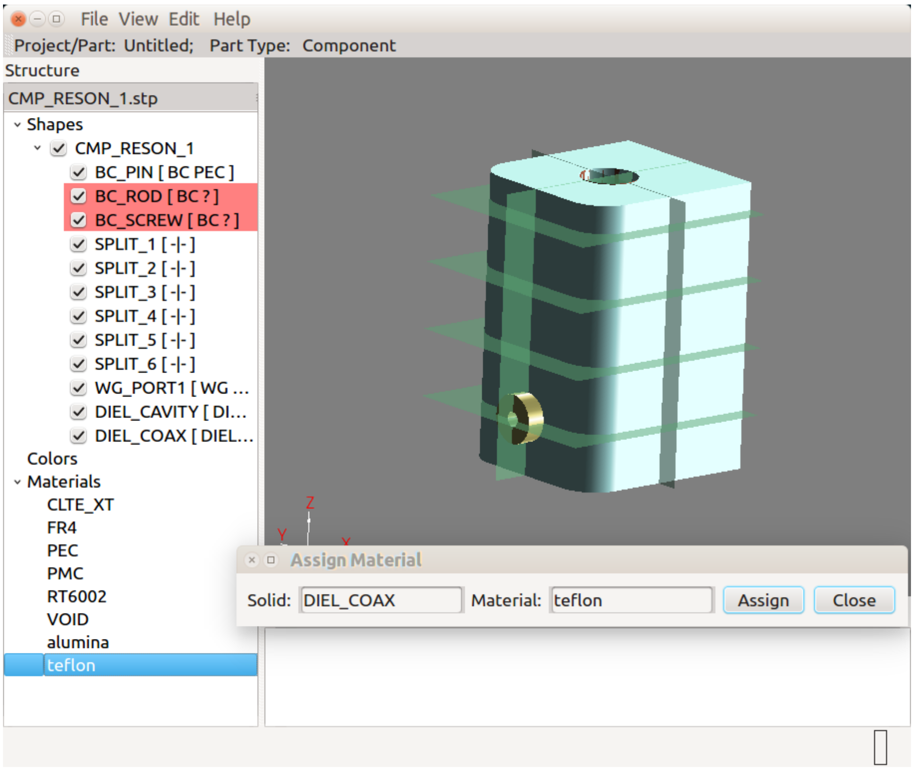


Figure 2.5: Material Assignment

The objects which represent dielectrics, boundary conditions and waveguide ports must always be associated with a material property. The material association is done using the “Set Material” command in a context menu that drops down when pressing the Right Mouse Button on the selected object in the Structure Tree. If this association is not yet established, for a given tree item, that item is highlighted with a red color to warn the user that some action is still needed to fully define the component.

The "WG_PORT1" item of the structure tree shown in fig 2.5 refers to a surface object that represents a waveguide port (the coaxial port of the component "RESON_1") and has been associated with the Teflon material.

The structure tree of fig 2.5 includes other three surface objects "BC_PIN", "BC_ROD", "BC_SCREW" which represents specific boundary condition zones. These surfaces identify different areas of the component boundary which are subjected to specific boundary conditions or specific local refinements of the mesh size.

A boundary condition ("PEC") has already been associated with the first of the three objects ("BC_PIN") but is still undefined for the other two which are therefore highlighted in red.

2.3.3 Part Properties

Other properties (different from associated material) of component parts can be defined using the “Set part properties” command in a context menu that drops down when pressing the Right Mouse Button on the selected object in the Structure Tree.

The “Set part properties” command opens a new window which allows to define a local mesh refinement.

For a part whose type is Waveguide Port the “Set part properties” window presents additional fields (see figure 2.6) associated with the numbers of modes (TEM, TE and TM). The number of TE and TM modes are specified by the user while number of TEM modes is automatically evaluated from the surface topology and the related field is not editable.

For a part whose type is Line Port the “Set part properties” window presents an additional field for the specification of a Load Impedance. This parameter defines a reference impedance used in the computation of the frequency responses associated with the scattering matrices.

The local refinement expresses an increasing factor which is applied to the number of mesh elements per wavelength associated with the selected part (vs the global value defined under the "Project Options"). Local Refinement=1 means no refinement.

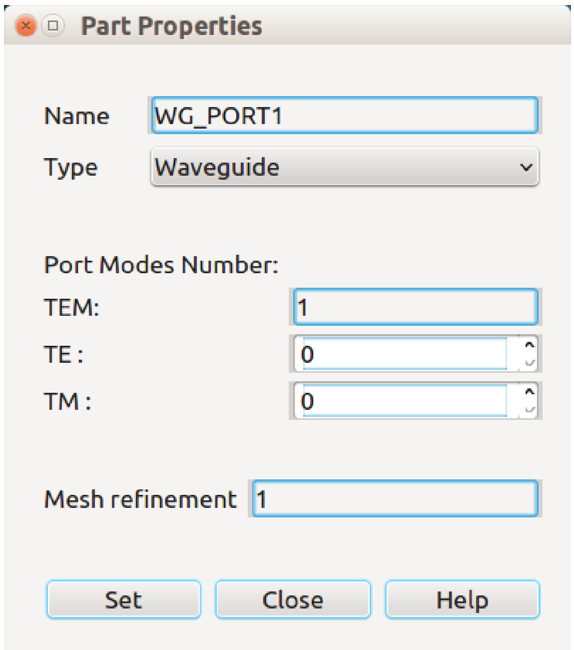


Figure 2.6: Waveguide part properties

2.3.4 Geometric decomposition

The dielectric parts of the electromagnetic structure must be partitioned in a set of simply connected cells (without trough holes). The elementary cells have to be small in terms of wavelength (let say cell diam $< \lambda/4$) so that their electrical response can not resonate within the considered frequency band. The circuital realization is based on the “Rational Padé approximations“ of the admittance matrices associated with these cells. Thanks to the not resonant behaviour the cell responses are well approximated by rational functions characterized by small order (N<=4).

The cell decomposition is specified by the designer which may directly draw the splitting surfaces (entity name prefixed with SPLIT_) or may use the splitting grids (entity name prefixed with GRID_). The former allow to control the exact position of each splitting surface while the latter are a convenient mean for the definition of regular decompositions.

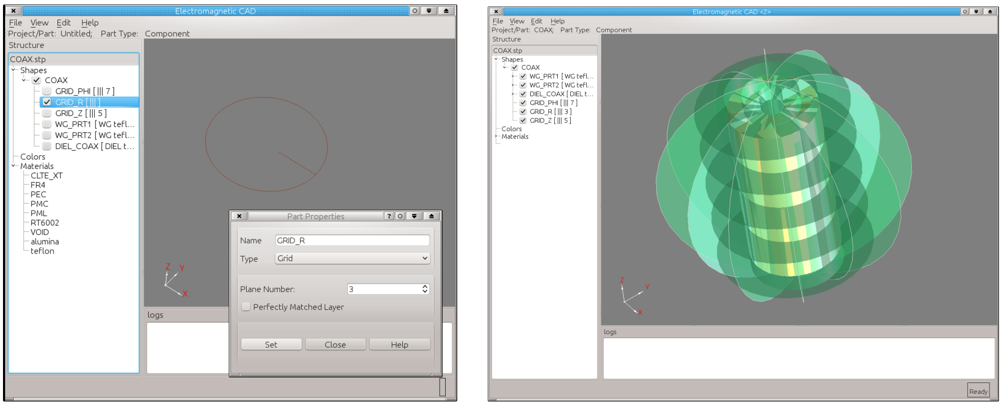


Figure 2.7: Regular Grid

A splitting grid consists of a set of equispaced planes which are defined by a curve segment drawn inside of the CAD environment. The curve segment can be linear or circular and is orthogonal to the splitting planes. The first and last planes pass trough the segment vertices. The total number of equispaced planes is specified inside of the EmCAD environment after having selected the imported GRID object.

A cylindrical grid is specified by drawing a linear segment and a circle (or a circle arc). The circle is used to define the common axis of the cylindrical surfaces. The linear segment is normal to the cylindrical surfaces (it is directed along the radius) and its vertices define the first and last cylindrical surface.

The figure 2.7 shows a coaxial dielectric which has been partitioned by means of three orthogonal grids: a planar grid orthogonal to the z coordinate (cylinder axis), a planar grid orthogonal to the angular coordinate and a cylindrical grid orthogonal to the radial coordinate.

A regular object (like a brick or a cylinder) which is split by regular grids generates several identical cells (which differ only for a rigid transformation). The EmCAD code will produce identical meshes for identical cells and than the *Hierarchical Electromagnetic Modeler* will reuse the same sub-circuit for the identical meshed cells. This strategy allows a great improvement in the computational efficiency associated with the modelization of large regular and homogeneous parts.

2.3.5 Global Properties

The global project options are defined using the “Project Options” command which opens the graphical window shown in fig 2.8. In this windows it is possible to set the units (length and frequency) and the control parameters of the component modelization. The latter consist of the Frequency Band, the Mesh Size and the MOR frequency number.

The equivalent circuit of a component is initially created by assembling a set of equivalent circuits which are associated the cells created by the decomposition. Depending on the mesh size, the component circuit may contain a large number of lumped elements. A Model Order Reduction is therefore applied to this circuit before inserting it in the global circuit. The result of this algorithm is a compact circuit which interpolates the frequency response associated with the original (large) circuit on a set of points distributed over the frequency band.

The *Hierarchical Electromagnetic Modeler* achieves the maximum efficiency when the

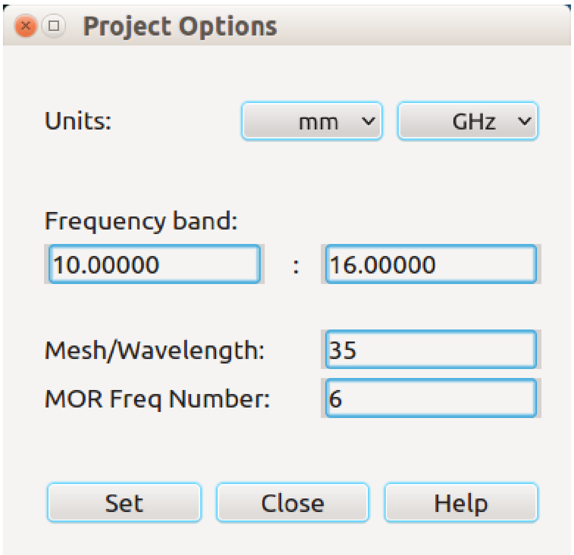


Figure 2.8: Project Options

electromagnetic problem, which can be very large, is decomposed in a set of components that are quite small (less than a half wavelength).

The frequency response of such small components can not change too fast in the frequency band and can be represented with a good accuracy using a low number of interpolation points. The default value of the MOR Freq Number is 5. With this choice the MOR algorithm generates, for each component, a reduced circuit that matches, at 5 frequency points, the frequency responses of the original circuit. This approximation is sufficient in most practical cases.

2.3.6 Decomposition Command

The “Decompose” command is disabled until all the component objects have been properly defined. The “Decompose” command performs a decomposition of all solid objects in a set of small cells which defined by the splitting surfaces. The result of this action can be visualized in geometry viewer by entering the “Open Sub-Assembly” command which allows enter into the partitioned geometry of the sub-component.

The fig 2.9 shows the partitioned geometry associated with the resonator of fig 2.5. From the structure tree on the left area of fig 2.9 it can be seen that the solid body named “DIEL_CAVITY” has been partitioned in 20 cells. The first four cells are unchecked in the structure tree and for this reason they are not displayed in the geometric window. This makes it possible to visualize the surface objects BC_PIN (boundary of the coaxial pin) and BC_ROD (boundary of the resonator rod) which are associated with the PEC boundary condition.

Beside of partitioning the solid objects in the elementary cells, the “Decompose” command performs also the “imprinting” and “sharing” of the adjacent geometrical entities. In example if two solids touch on two adjacent faces, these faces are split by separating their common part (the intersection) which is assigned to both solids.

The face sharing generated by the decomposition can be visualized in the structure tree. So in example, by opening the items named “Vol7” and “Vol8” of the structure tree (see fig 2.9) it is possible to visualize the faces associated with the two cells and to verify that the face “F108” is in both faces (is a shared face).

It is important to check the component decomposition and face sharing has been done correctly. Sometimes, for certain configurations of the component objects (solids and splitting surfaces), the decomposition algorithm can fail or can generate very small faces/edges. These situations can lead to failure of the meshing algorithms and must be avoided trough a proper definition of the CAD geometries.

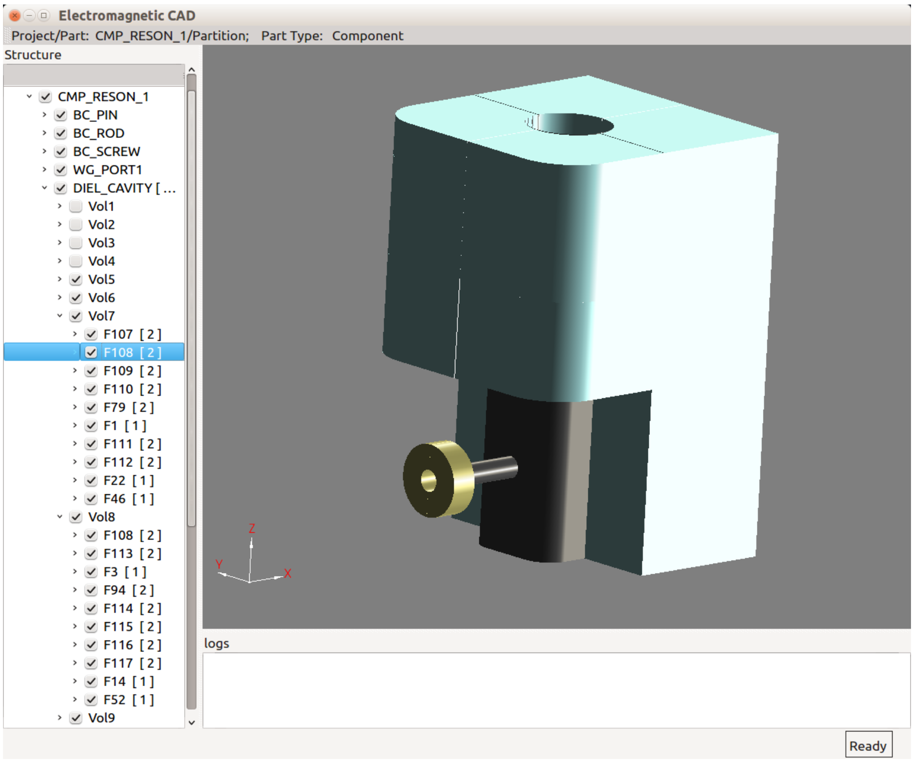


Figure 2.9: Component Partition

2.3.7 Component Meshing

The “Mesh” command is disabled until all the component objects have been properly defined and the frequency band and mesh size have been set. The “Mesh” command performs the triangulation of all the surface objects associated with the partitioned component. The meshing must be always preceded by the decomposition but the latter is automatically invoked by the “Mesh” command if not yet done.

The meshing algorithm assures the congruence of mesh data on all the faces and edges shared by adjacent objects. This property assures the compatibility of the electrical ports associated with the equivalent circuits of adjacent cells. The surface mesh associated with the resonator of fig 2.5 is shown in fig 2.10. This image is generated by the “Mesh View” command which makes a call to the Gmsh software [1] passing to it the mesh file associated with the component.

Gmsh is an open source code for the generation and visualization of mesh data. In particular Gmsh provides a tool for the the clipping of the meshed geometry so that it is possible to visualize the triangulations associated with internal faces as shown in the figure.

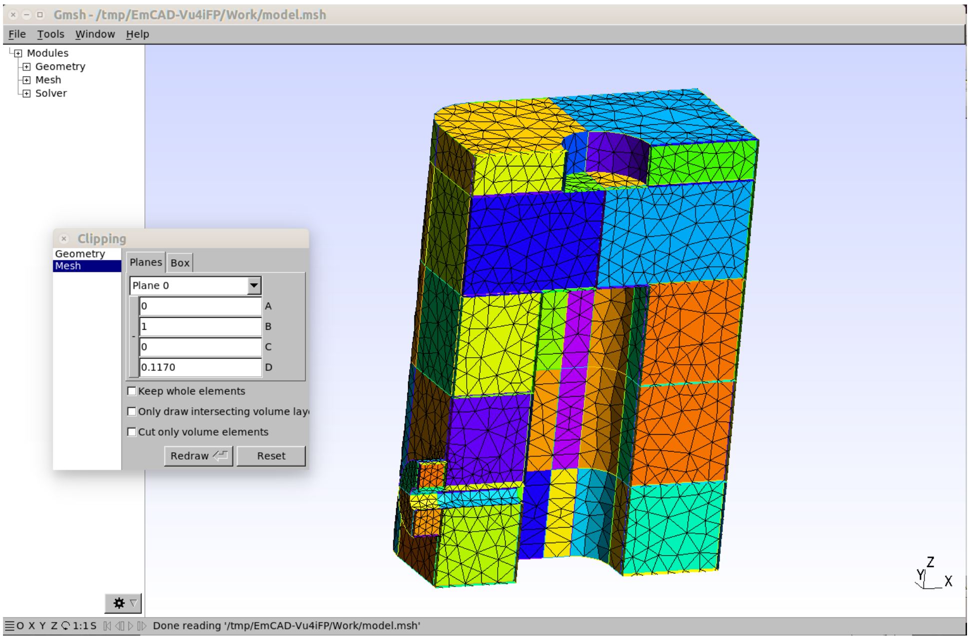


Figure 2.10: Visualization of the Component Mesh

2.3.8 Saving the Component Data

All the resonator data are than saved into an EmCAD data file using the "Save As" command.

2.4 Defining an Assembly

2.4.1 Defining the Assembly Geometry

The assembly geometry is defined in a CAD system and saved into a step file. A new EmCAD Assembly is created using with the “File/New” Command. The step file is then imported into the EmCAD Assembly using the “File/Import Geometry” command.

The fig 2.11 shows the geometrical structure of the filter assembly as it appears after having imported the related step file.

2.4.2 Defining the Sub-Components Properties

The electrical properties of the components were already defined according to the procedure described in the previous section. These data can be associated with the assembly components using the “Import Part Properties”. This command opens a new window which allows to associate an component, selected in the structure teee, with an EmCAD data file selected in the file browser.

The figure 2.11 shows the establishment of an association between the tree item named "COMP_RESON_5" and the data file "CMP_RESON_5.emc". Some of the items in the structure trees are highlighted with a red color indicating that the corresponding components ("COMP_RESON_1", and "COMP_RESON_2") have not yet been defined.

It may happen that an assembly makes use of a set of similar components which are related by small changes of some parametric dimensions in the same basic design. In this situation it will be sufficient to prepare a single EmCAD file which describes the electromagnetic properties of one representative component.

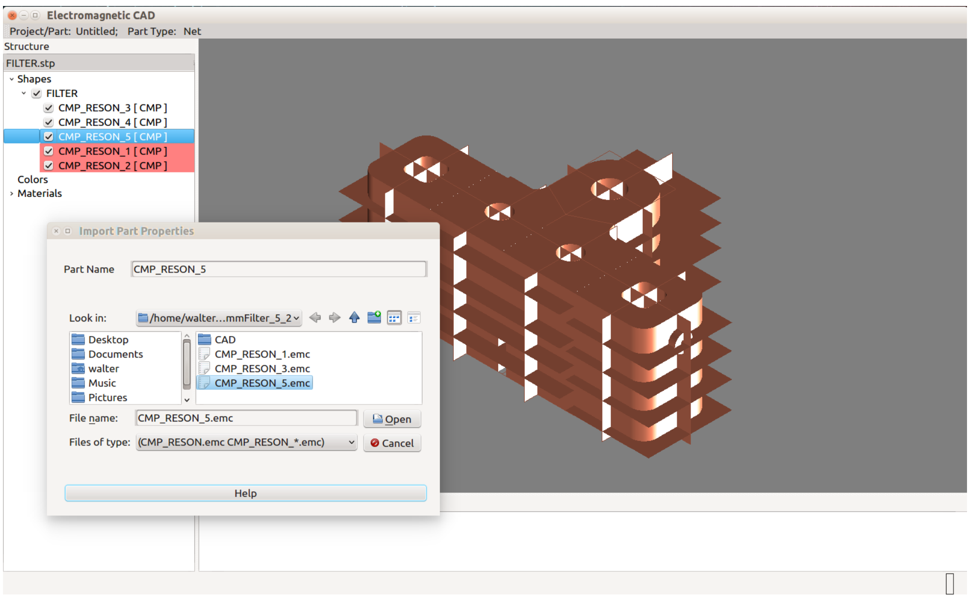


Figure 2.11: Importing sub-component properties

The representative component can then be used to define, trough the “Import Part Properties” action, the electromagnetic properties of all the variations included in the assembly. This is possible because the “Import Part Properties” action affects only the the non-geometrical data (material properties, number of ports..) associated with the selected component. These kind of data will of course be defined in the same way for all the variations of the given component.

2.4.3 Global Properties

The global project options are not imported from the component data and must be defined by using the "Project Options" command. Entering this command at the assembly level opens a window that is identical to that one opened at the component level (see fig 2.8). In this case the parameters entered in the window are applied to all the sub-components included in the assembly.

It is so assured that the global parameters are aligned with the projects specifications for all the constituent parts. If of one these parameters need to be changed after the initial setting, the variation is automatically propagated to all the subcomponents.

2.4.4 Assembly Decomposition

The "Decompose" command is disabled until all the components have been properly defined. At assembly level this command is automatically applied to all the components and generates, for each one of them, a decomposition similar to that one shown in the figure 2.9.

This result can be visualized by opening the selected sub-component by means of the "Open Sub-Assembly" command. Then, a second call of the "Open Sub-Assembly" allows to visualize the partitioned geometry of that sub-component. The *EmCAD* GUI can be brought back to the assembly view by entering two times the "Open Upper Assembly" command.

2.4.5 Assembly Meshing

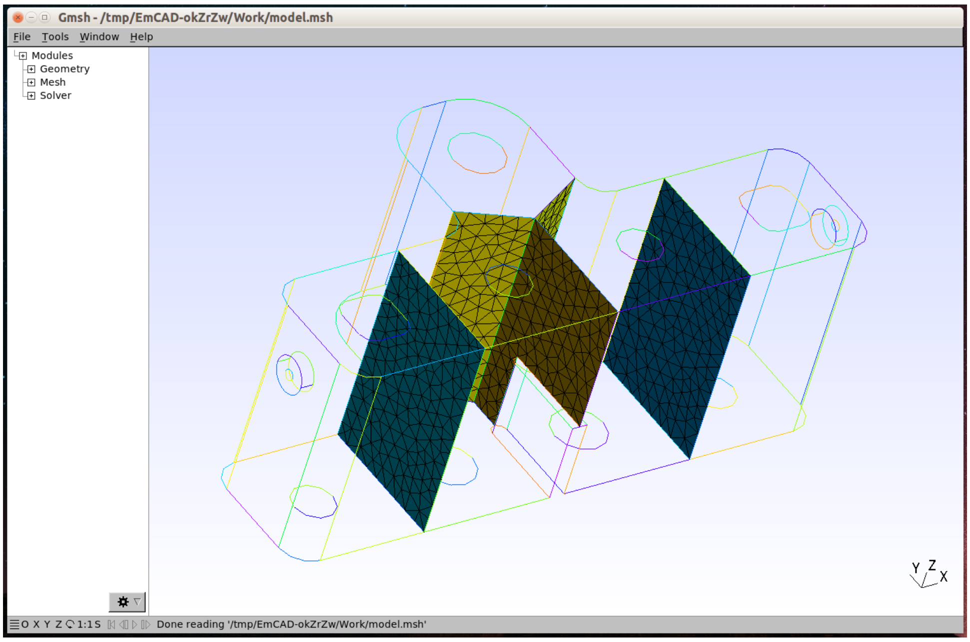


Figure 2.12: Meshed Interfaces

The "Mesh" command is disabled until all the components have been properly defined and the frequency band and mesh size have been set. At assembly level this command is automatically applied to all the components and generates, for each one of them, a triangulation similar to that one shown in the figure 2.10.

The meshing must be always preceded by the decomposition but the latter is automatically invoked by the "Mesh" command if not yet done. A sub-component mesh can be visualized by entering the "Mesh View" command after having opened the selected sub-component with the "Open Sub-Assembly" command.

Besides of the meshing of all the sub-components, at the assembly level the Mesh Command generates some additional surface meshes which represent the electrical interfaces between adjacent sub-components. These triangulations can be visualized (as shown in figure 2.12) by entering the "Mesh View" command when the geometrical window is positioned at the assembly level.

The electrical circuits associated with the subcomponents represent their electrical responses versus a set of port modes which are defined on these interfaces. The number of port modes distributed on a given interface depends on the surface area and on the mesh size. This number is automatically chosen by the *Hierarchical Electromagnetic Modeler* as a result of a trade-off which takes into account the accuracy of field representation and the compactness of the generated equivalent circuits.

2.5 Modelization and Analyses

2.5.1 Modelization

The “Modelize” command can be given on a component or on an assembly and in both cases it is disabled until the meshing tasks are completed.

The “Modelize” command copies the mesh file (or files) associated with that structure (component or assembly) into a directory on a remote storage area, it sends a "modelization" request to the server and then it starts polling for a result.

The storage area is owned by the local user but it must be configured so that it is accessible also by the server.

After having loaded and processed the mesh files, the remote server generates a set of equivalent circuits and saves this result in the storage area under the same directory of the input mesh file. The EmCAD code downloads this circuits into the local machine signalling the user that the modelization task is completed.

When the “Modelize” command operates on an component it sends a single mesh file and the server generates a single equivalent circuit which represents the electrical response of that component.

When the “Modelize” command operates on an assembly it sends the mesh files associated with all the sub-components. The related modelization tasks are run in parallel and the results are downloaded when ready in an asynchronous way. The EmCAD window informs the user, with messages displayed in the message area, about the current status of the modelization activities (see figure 2.13).

The "Modelize" command terminates (with a ready prompt) when all the generated circuits have been retrieved and downloaded to the local machine.

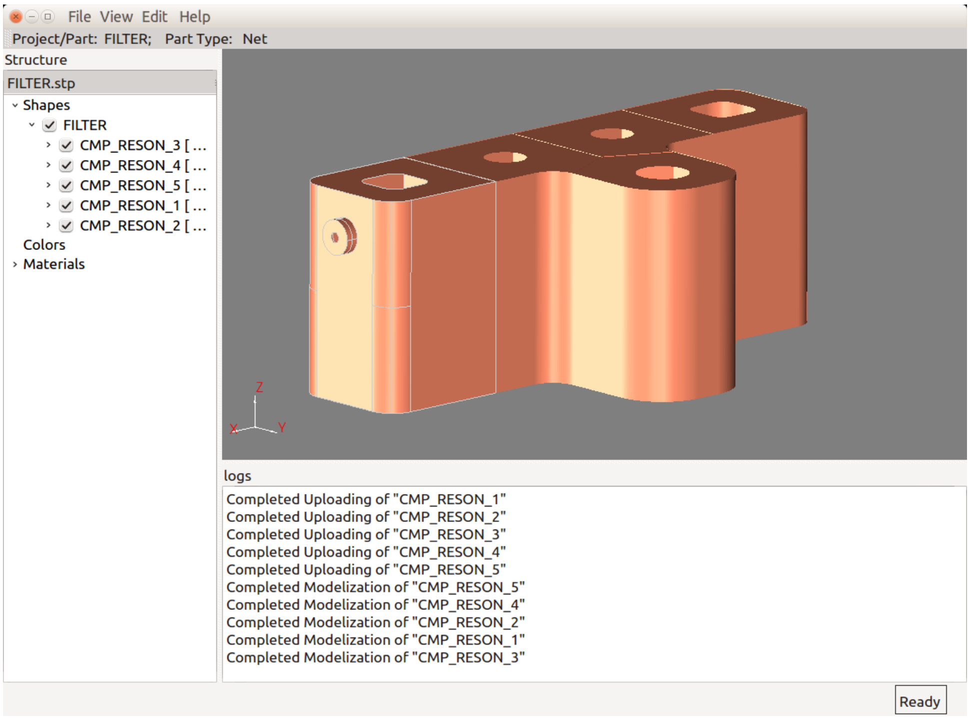


Figure 2.13: Modeling

The generated circuit can be exported into a Spice Netlist file using the command

"File/Export/Export Spice Circuit" as shown in figure 2.14.

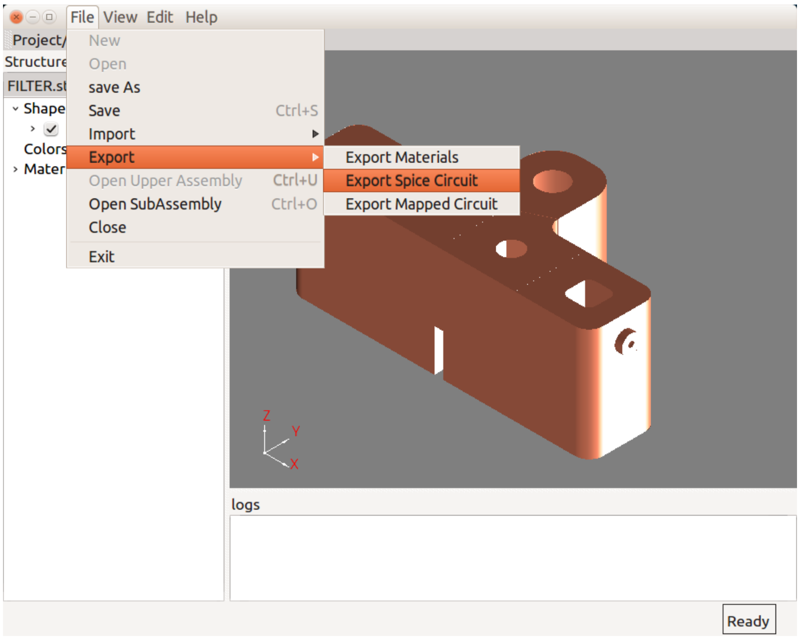


Figure 2.14: exporting the Spice Circuit

2.5.2 Frequency Domain Analysis

Once the Modelization Task has been completed it is possible to perform a Frequency Domain Analysis. This task is activated by the "Edit/Analyses/Freq Domain Analysis" command which opens a window (see figure 2.15) which allows the specification of a few parameters. These are the frequency band used in the analyses, the type of parameter to be computed (Scattering, Admittance, Impedance ...) and the number of frequency points used in the Model Order Reduction.

The reduction is applied to the global circuit obtained by assembling the reduced circuit associated with the sub-components. The frequency response is then computed on the reduced global circuit.

The default value of the number frequency points used in the global circuit reduction is 20 but it should be increased when the electromagnetic problem is large (in terms of wavelengths). Depending on the physical size of the electromagnetic structure and also on the width of the frequency band, the global circuit may be characterized by a frequency response that presents many variations inside the given frequency band. In this situation a large number of MOR points (leading to a large reduced circuit) may be necessary to capture the fast in band variations of the circuit response.

The command "View/Plots/Frequency Response/Electromanetic Model" allows to visualize the frequency response associated with the electromagnetic model. The figure 2.16 shows the frequency response of the S parameter of the filter shown in figure 2.15. The axis scale associated with this plot can be changed by activating the "Edit/Set Scale" command from the plot window.

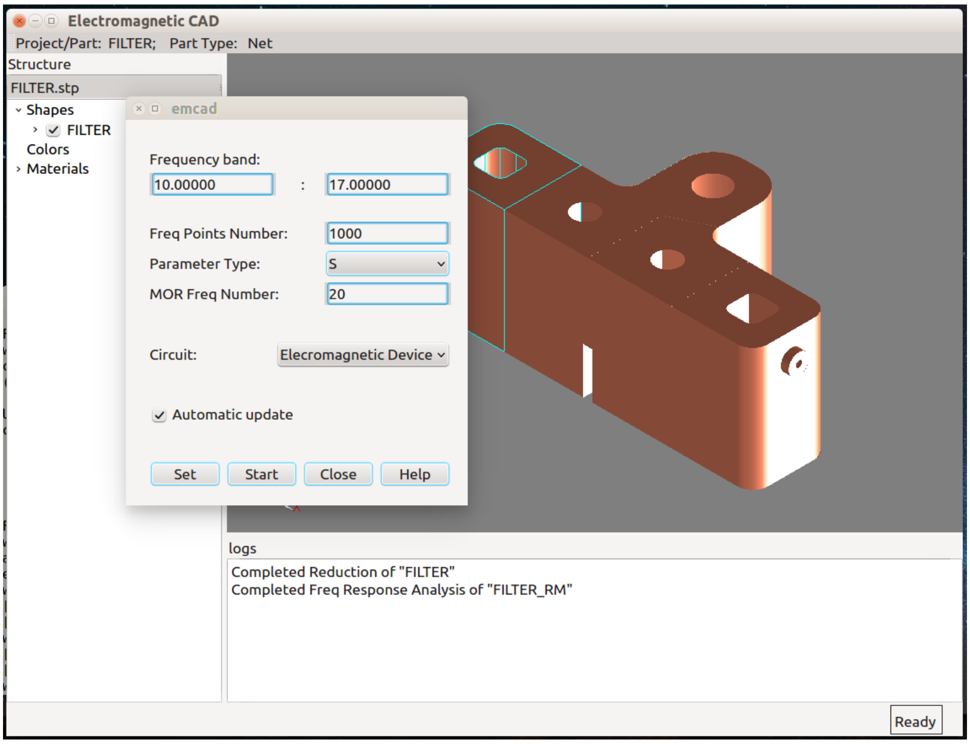


Figure 2.15: Frequency Domain Analysis

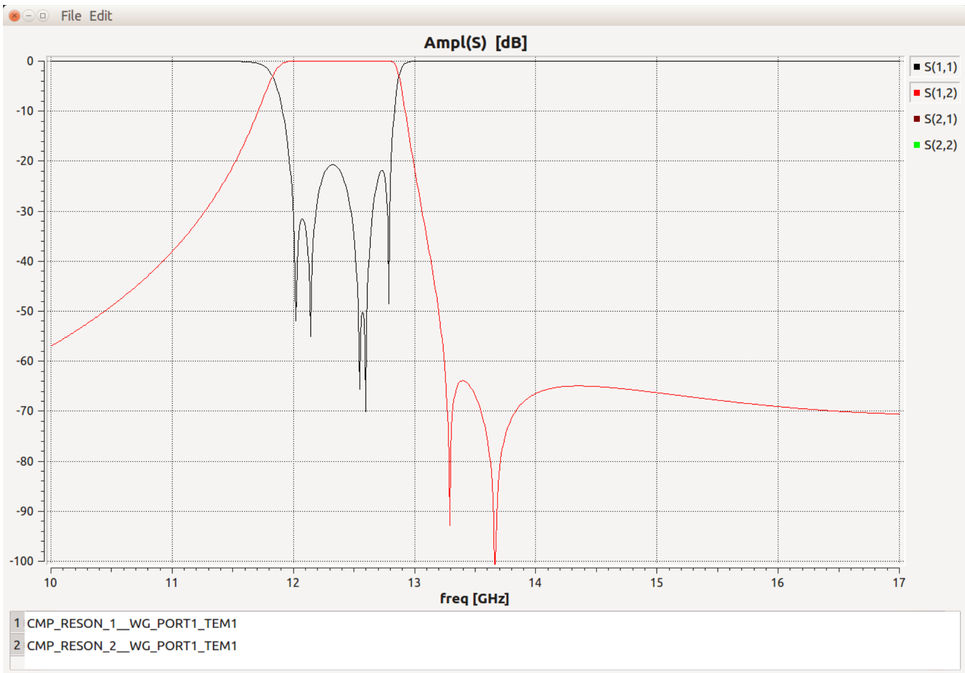


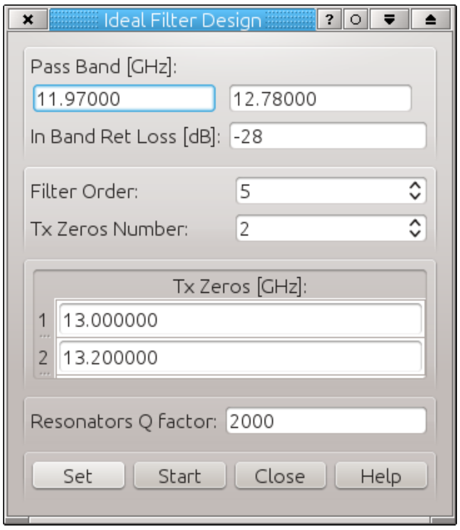
Figure 2.16: Frequency Response Plot

3. Application Cases

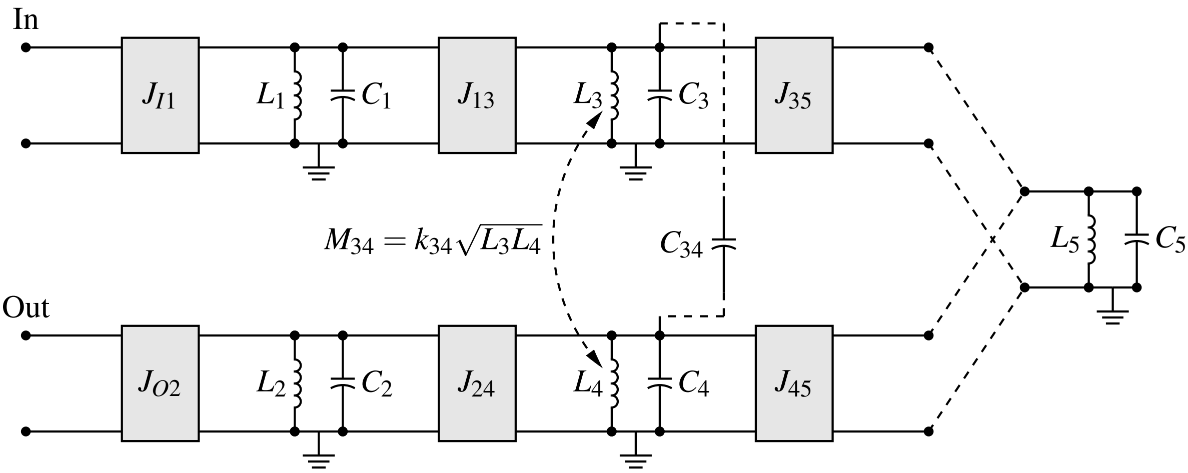
3.1 Filter Design

3.1.1 Ideal circuit

The EmCAD code provides a tool (accessible under the "Edit/Analyses/Design/Ideal Filter" Menu) which can be used to generate a lumped equivalent circuit which realizes a frequency response that meets the given design specifications.



(a) Input Parameters



(b) Generated Circuit

Figure 3.1: Design of an Ideal Filter with 5 resonators and 2 transmission zeros

The design parameters are: 1) The filter passband. 2) The maximum return loss over the pass band. 3) The filter order (number of resonators). 4) The number and the location of the transmission zeros. The number of these zeros may be zero (corresponding to a Chebyschev response) and must not be greater than the filter order.

The input values reported in the figure 3.1a lead to the equivalent circuit depicted in the figure 3.1b. This circuit is composed of 5 shunt L-C resonators and icharacterized by a reflection symmetry in the vertical direction. The resonator couplings is realized by shunt capacitors and mutual inductances in the transversal (vertical) branches and by J inverters in the longitudinal branches. The number of transversal branches increases with the number of transmission zeros and is null for a Chebyschev design.

The J inverters are two port networks which are widely used in the synthesis of ideal resonator circuits and are characterized by admittance matrices with the following structure:

$$Y = \begin{pmatrix} 0 & jJ_{ij} \\ jJ_{ij} & 0 \end{pmatrix}$$

The circuit configuration generated by EmCAD tool is such that its response does not change if all the J inverters are replaced by the corresponding real inverters which are two port networks defined by the fillowing admittance matrix.

$$Y' = \begin{pmatrix} 0 & J_{ij} \\ -J_{ij} & 0 \end{pmatrix}$$

The admittance matrix Y' is real and untisymmetric and therefore not reciprocal. A real inverter can be easely implemented in a spice circuit using a couple of voltage controlled current generators having opposite sign.

The EmCAD command "File/Export/Ideal Spice Circuit" allows to export the spice circuit asociated with the ideal filter which is composed of LC resonators, real inverters, coupling capacitors and mutual inductances.

The frequency response of the generated ideal circuit can be analysed and plotted using the commands "Edit/Analyses/Freq Domain Analysis/Ideal Filter" and "View/Plot/Freq Response/Ideal Filter". The frequency response associated with the previous circuit (specified by input parameters shown in figure 3.1a) is depicted in figure 3.2.

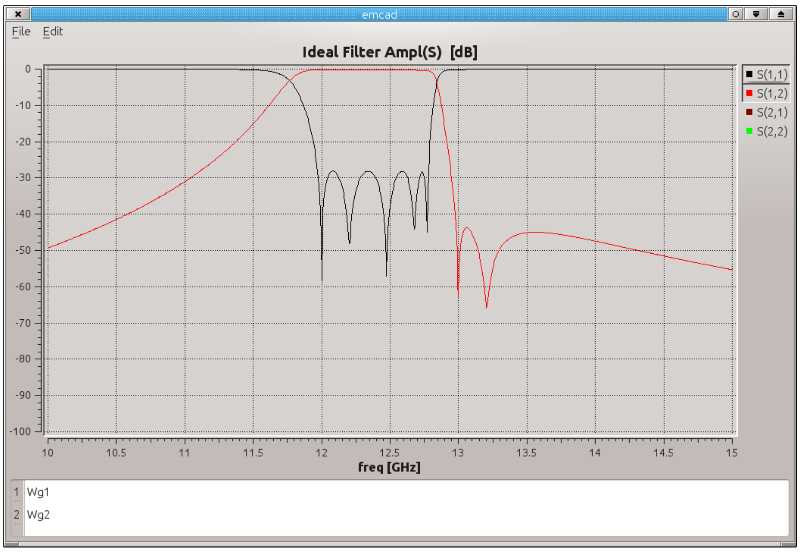


Figure 3.2: Frequency Response of the Ideal Circuit

3.1.2 Microwave Realization

The ideal equivalent must be translated into a microwave structure which will depend on the selected resonator technology (waveguide, coaxial, stripline, . . .). The figure 3.3 shows a possible realization of the previous filter design which makes use of interdigitated coaxial resonators.

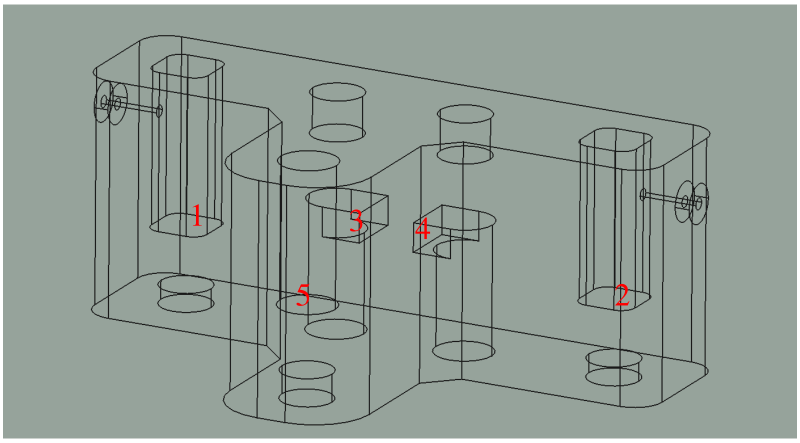


Figure 3.3: Coaxial Realization

In order to implement the equivalent circuit of type shown in figure 3.1b the microwave structure must allow a separate regulation of the capacitive and of the inductive parameters associated with the transversal coupling branches. For the coaxial configuration shown in figure 3.3 these regulations are actuated by changing the distance between resonators 3 and 4 and the gap between the protrusions associated with the same resonators.

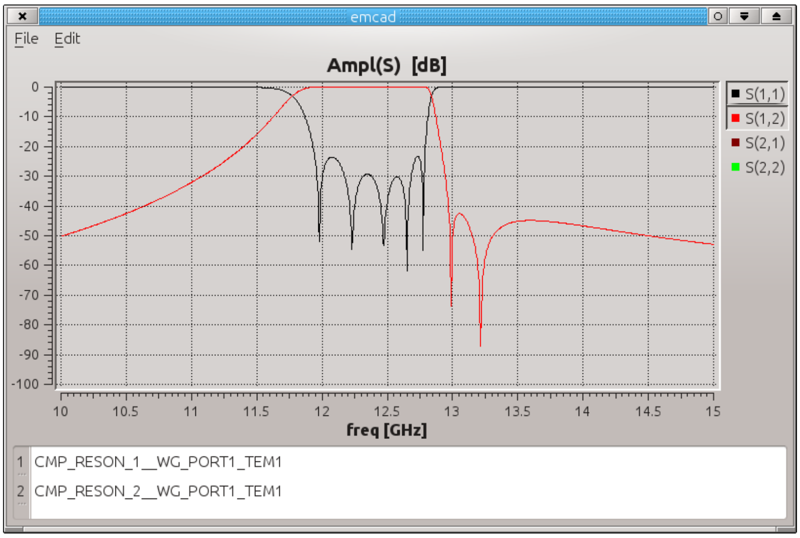


Figure 3.4: Frequency Response of the Coaxial Filter

For the regulation of the inverter parameters it is sufficient to change the related inter resonator distances while the resonant frequencies associated with the five resonators are cntrolled by the coaxial lengths.

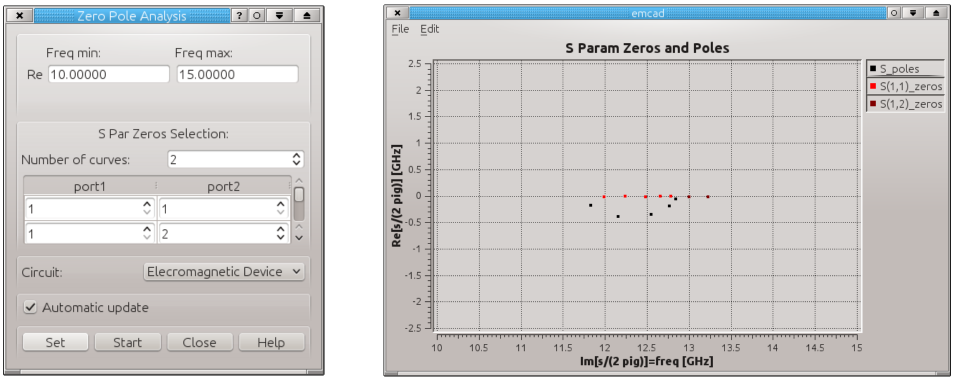
All these degree of freedom can be easily tuned by exploiting the Filer Mapping tool which is available inside of the EmCAD environment (see section 3.1.4). The frequency response associated with the optimized coaxial structure is shown in figure 3.4

3.1.3 Zero Pole Analysis

The Zero Pole Analysis (activated by the command "Edit/Analyses/S Parameter Zero/Poles") is a unique capability of the *Hierarchical Electromagnetic Modeler* which differentiates it from all other full wave electromagnetic solvers. This command opens the window shown in figure 3.5a where it is possible to specify the analysed model (Electromagnetic Device/Ideal Filter or Mapped Circuit), the selected S parameters and a frequency window which limits the search area.

The computed Zeros and Poles pattern can then be visualized over the Laplace Plane using the command "View/Plots/Zeros and Poles". The Laplace Plane is rotated by 90 degree with respect to its usual orientation in such a way that the frequency axis becomes horizontal and the left side of Laplace plane ($Re(s) < 0$) becomes the bottom side of the plot image.

The zero pole pattern associated with the coaxial filter of figure 3.3 is shown in figure 3.5b. It includes five zeros of S_{11}, two zeros of S_{12} and five poles which belong to both scattering parameters. Being related with a passive component these poles must lie on the negative side of the Laplace plane which corresponds with the lower side of the plot plane.



(a) Input Parameters (b) Plot

Figure 3.5: Zero Pole Analysis

The particular arrangement of the zero/pole pattern shown in figure 3.5b (with all zeros on the frequency axis) is characteristic of a well tuned filter. The reflection and transmission zeros associated with a detuned filter may occur in a more general position outside of the frequency axis.

3.1.4 Filter Mapping

The zero pole pattern associated with a microwave filter can be used to evaluate the current values of the circuital parameters. This evaluation is done using the command "Edit/Filter Mapping" which can be activated after having computed the zero-pole pattern. The mapping command produces a new equivalent circuit that is characterized by the same topology of the circuit of figure 3.1b but different values of the circuit elements. This circuit can be exported in the spice format using the command "File/Export/Mapped Spice Circuit".

The comparison between the values of the parameters associated with this circuit and those associated with the ideal circuit provides a precise information about the status of the design of the microwave filter. This information can be exploited for an efficient tuning of the microwave structure.

Bibliography

Articles

[GR08] Christophe Geuzaine and Jean-François Remacle. "Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities". In: (2008) (cited on page 26).

[KROD02] Marina Y. Koledintseva, Konstantin N. Rozanov, Antonio Orlandi, and James L. Drewniak. "Extraction of Lorentzian and Debye parameters of dielectric and magnetic dispersive materials for FDTD modeling". In: 53.9 (2002) (cited on page 18).