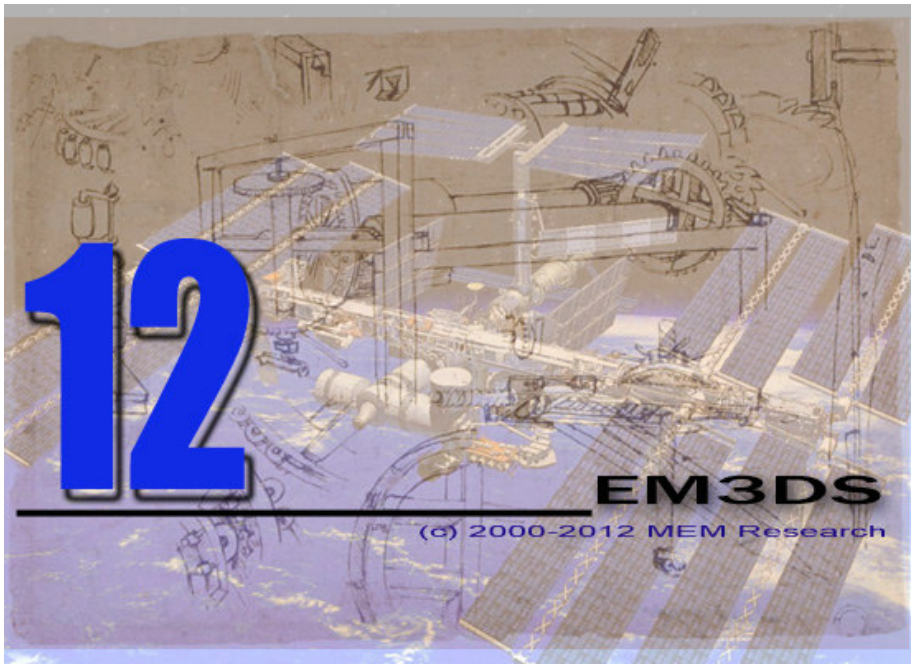


User's Manual

Manual Release October 2011

I What is EM3DS

EM3DS is an integral equation-based full wave simulator, using an approach called *Generalized Transverse Resonance Diffraction* (GTRD). Details about the general technique are available in a number of source, including the book "Advanced Electromagnetic Analysis of Passive and Active Planar Structures", by T. Rozzi and M. Farina (IEE Press 1999) and in the paper "A 3-D Integral Equation-Based Approach to the Analysis of Real Life MMICs: Application to Microelectromechanical Systems", by M. Farina and T. Rozzi (IEEE MTT December 2001).

This simulator accounts for **finite-thickness, finite conductivity real conductors/dielectrics** by using volume currents. The structure is still layered, but current appears to be arbitrarily oriented (J_x, J_y and J_z are always present in the 3D mode; as consequence the total current may have arbitrary orientation even over sides of a thick conductor). This allows EM3DS to fully model also dielectric discontinuities, including Dielectric resonators, which are out of the scope of traditional 2.5D solvers.

Moreover, the ability to handle conductor thickness, which is usually not very important in standard PCB design where substrate losses are dominant and the structures are very large with respect to the metallization thickness, it may fundamentally affect results in MMIC design, where strips may be a few microns thick and wide.

Metal thickness is also important in waveguide filters: consider e.g. that in single cavity WR90 resonator, neglecting the iris thickness may produce a resonant frequency shift of over 5%.

EM3DS is mainly aimed at the analysis of planar structures, but due to its 3D nature it may be used also to address some other problems, like the modeling of waveguide filters with arbitrary thick, possibly lossy, irises.

Actually EM3DS embodies in fact **2 EM Solvers – 2.5D and 3D.** Which solver to use is rested upon the designer's choice and is a simple click to shift the mode of operation. It means that by simple click onto your 3D structure, you obtain 2.5D geometry and you can simulate as 2.5D and compare with your 3D simulation. The speed is increased in 2.5D mode by a factor varying from 50% up to more than 300%: the more the structure is complex, the larger is the time saved. The 2.5D mode comes as a limit of the 3D formulation, keeping several attracting capabilities of the 3D mode.

EM3DS comes with a complete set of tools. The most important is maybe the panel for global parameters/variables: the panel allows you to define variables, and to parameterize the geometrical features of your objects. This means that you can modify and optimize your design without painfully entering new dimensions shape by shape. The optimization can either be manual (the designer does the work) or exploit the embedded optimizer. Variables can be modified also accessing a "tuner", a set of sliders varying those variable indicated as tunable in the optimizer panel. Variables can also be accessed through a pascal "script", namely by writing a pascal code directly in EM3DS.

As to the speed, EM3DS always performed very well thanks to its Asymptotic Estimator, which performs a full simulation only at the first frequency point. Moreover, since version 7 is available a new powerful device, SmartFIT: this algorithm adaptively drives the EM solver, generating a frequency list within the user-selected band. At the end it estimate the response all over that band, in tents or hundreds of points. Of course it is possible to exploit a rational interpolator even in post-processing, but without the same guarantee to get good results.

An important companion to this algorithm, is a wide-band Spice Model extractor: it generates a network, which reproduces the response all over the band, regardless of the electrical length of the EM circuit.

In the release 9 several completely new features were added, such as internal planar ports, antenna functions (namely sense layer to see the E-field and to evaluate radiated far field, both in standard polar diagrams, or in 3D surfaces), object oriented pascal scripting, tuner, enhanced editing etc. In version 10 you will find a first version of a **Macromodel** creator: for a given parameter, EM3DS extracts some features, in order to be able to predict the circuit response in real-time when such a parameter is modified. This is an extremely useful tool for the design. At the moment it is limited to a single parameter but more work is in progress. Additionally the multi-mode calibration algorithm has been enhanced to allow its application in several measurement problems. EM3DS 10 was developed to reduce the time needed to learn how use it, with many on-line animated tutorial.

Version 11 (end 2010) introduces the Time Domain Reflectometry: a set of time-domain functions, including the possibility to analyze the time domain to an arbitrary response, and to see the time domain evolution of current/field distributions. Version 11 also adds the possibility to define your own measurements (custom measurement definition).

Version 12 (fall 2011) enhances the parametric capabilities of EM3DS, allowing to parametrize also the thickness of each substrate, to define a list of sets of parameters, and to automatically perform simulations and comparisons for each set of parameters. Moreover several minor improvements are made available (additional animated tutorials, printing and print preview utilities etc)

Since ver 6.0 EM3DS was the first tool allowing the **full-wave modeling of FETs** in linear regime: this is accomplished through the insertion of distributed controlled current sources. EM3DS 6.0 provides for the materials' constitutive parameters to be frequency dependent, and may further be specified by the user through **analytical expressions (formulas).** Every parameter in EM3DS can conveniently be input as a valid algebraic expression, (for instance: sin(3.5)), and is fully evaluated (calculated) by EM3DS, thus facilitating the overall process and avoiding using calculators.
Unlike some other commercially available tools, EM3DS **does not pose any** requirement on the grid.

The only event when grid is desirable, and in fact EM3DS can make use of it, is during the drawing, where it offers additional flexibility.

As stated above, EM3DS comes integrated with several tools, aimed to ease interfacing with other CAE tools. As to the pre-processing step, filters allowing importing GDSII and DXF (as well as general bitmaps) are available. The post-processor includes a Spice Model Extractor, a simple Linear Circuit Simulator, tools for saving animations (GIF, AVI), link to AVI files via multimedia documents, exporting graphs (plots, Smith Charts), data files (Touchstone), to see the interaction between a user-defined magneto-static field and RF currents flowing across an object etc.

Nonetheless for MEM Research it is imperative to look for partnerships in order to better exploiting the EM3DS power. This is why in the past EM3DS 5 was also fully integrated as Electromagnetic Module in the multi-physic package by Corning Intellisense, Intellisuite®.
Currently, EM3DS is companion of CoventorWare™, by Coventor, leading company in the MEMS software development: EM3DS was carefully selected by Coventor thanks to its remarkable achievements.

By the same token EM3DS has now the ability to be accessed directly from the AWR Microwave Office (since version 6 of MWO) suite. This means that AWR's customers are able to see EM3DS as an electromagnetic engine, still using Microwave Office editor and post-processing capabilities. Refers to the section for details on how to use this feature.

II What's new in Ver. 12 (2011)

- **Extended parametric features**: substrates can have parametric thickness

- **Parametric analysis**: define several sets of parameters and perform group of simulations
- **Now printing of any chart/graphic/text is implemented**
- **Additional tutorials**
- **FreeEM3DS much more powerful!**
- **Bug fixing**

Since version 11 (2010) EM3DS featured

- **Time Domain Reflectometry TDR**: transient analysis in time domain of the signals
- **Time Domain animation of currents and fields**: analysis of current and field distributions in time
- **Custom Measurements**: ability to define and add your own new measurements

Moreover in the editor automatic align of rectangular shapes

Since version 10 (2008) EM3DS featured

- **Macromodeling**: creating macromodels which allow to see how the response of circuit is modified in real-time for a modification of a parameter
- **Macromodel optimization**: the optimization can either run on a macro-model or a native electromagnetic structure
- **Macromodel tuning**: move the tuner slider and see how the response changes
- **Updated help**: migrated into HTML format and reviewed.
- **Animated online tutorials:** reduces the time needed to exploit full EM3DS functionality**.**
- **Multimode calibration algorithm:** now it is possible to import externally defined termination conditions

Since Version 9 (2007) EM3DS featured:

- **Internal Planar Ports**: Ports can now be added everywhere, also off-ground
- **Tuner:** Variables can now be tuned for parametric simulations
- **Scripting:** An Object Oriented Pascal (OOP) Editor/Compiler is embedded in EM3DS and allows to customize the program; some special functions allows to read EM3DS results, modify and create variables, run simulations and optimizations. Complex variables are available as well as functions to handle them. Scripting functions are updated in minor releases of 9.x
- **Sense Layer:** A sense layer can be created and handled automatically to see E-fields in a plane. The sense layer also allows to calculate radiation features of an antenna, such as far field and radiation patterns
- **New charts:** A polar plot for antenna parameters; far field is available also in 2D rectangular plot, and in a 3D surface plot with rendering
- **A general calibration algorithm for multi-mode waveguide calibration:** This is an advanced feature added for scientific purposes, enabling the calculation of the Generalized Admittance Matrix (GAM) of a given waveguide structure. This calculation can be used to split a structure by cutting it everywhere.
- **New measurements** for antenna applications
- **Multimedia documents**: link to multimedia file into a project
- **Enhanced Editor:** Most Functions now available in the toolbar; port numbering; highlight of the active layer in the 3D view and much more

Version 8 (2006) already included several breakthrough features, some of them summarized here:

- **Magnetic walls**: top and bottom cover may be magnetic wall (namely tangential H vanishing); by the same token two of the 4 side walls may be magnetic
- **Symmetry wizard:** magnetic walls may be used to simulate half a symmetric structure, so saving large amount of time. When there are ports, by replacing -in a two-step procedure- the symmetry plane by means of a magnetic and then an electric wall, the response of the whole device may be obtained by suitably superimposing the two half-structures, each one featuring one half of the total number of ports. The symmetry wizard is a simple post-processor generating the correct superposition of the two simulations, recreating the full-port solution.
- **Batch planner:** it allows creating a queue of simulations/optimizations to be automatically ran and saved
- **Differential ports:** a new post-processor allows to renumber ports, connect ports together (and creation of response to "even" excitations), evaluate the differential (balanced, or odd) response of the circuit etc.
- **Enhanced editor:** now you can select multiple objects and change their properties all at once (resistivity, mesh, move by arrows etc). Each layer can be labeled (e.g. "Gallium Arsenide" "copper" etc) so that it becomes easier to identify the layer being edited. Ports can be selected by mouse, edited, deleted etc. Charts can be zoomed, panned etc.
- **Enhanced mesh:** new options allow greater flexibility to control mesh, in particular for non-Manhattan objects
- **License server as service:** now license manager can either work as a program or as a standard Windows service, running in background.
- **Parameters:** layers may have parametric materials, even when no frequency dependence is required.
- **Measurements:** the fraction of power lost in a component for conductive or radiation losses is now evaluated

III How to read this manual

This reference manual is divided in paragraphs. The sequence of paragraphs is selected according to steps and problems that have to be addressed in order to correctly setting up a typical electromagnetic (em) model of your structure. At the end of each paragraph a set of FAQ (Frequently Asked Questions) discuss possible questions that are likely to arise in the reader. Some sub-paragraphs discuss more deeply technical questions (*To Probe Further*), and can be neglected in a first reading. Bold characters are generally used to indicate menu items or buttons.

IV Licensing

When EM3DS runs for the first time, it generates an installation code, that you have to send to MEM Research in order to have an enabling user code. In order to directly generate an e-mail containing the code, just click where indicated.

The installation code is hardware sensitive, so that any new computer needs a new license.

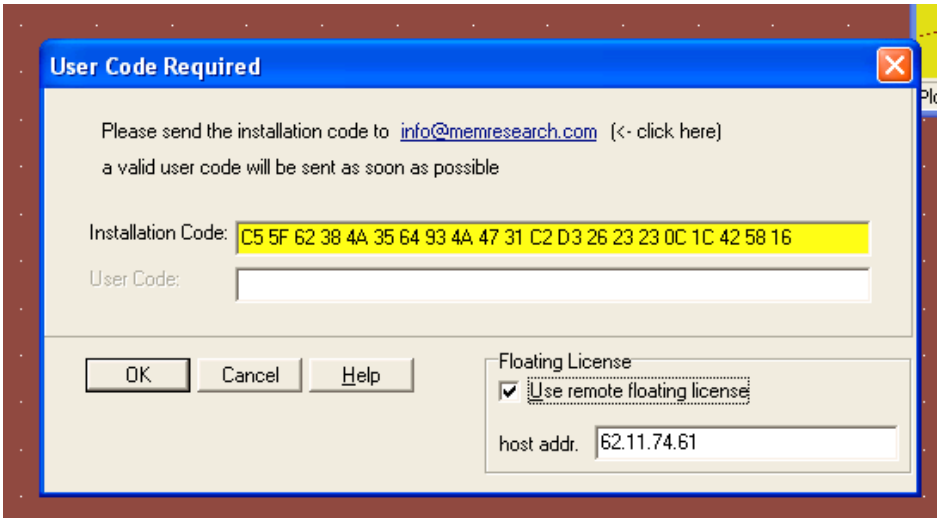


Fig.1 Request for code

EM3DS also supports an alternative licensing scheme, namely floating licenses. This means that you can run a **license manager** in a computer, where EM3DS has been installed and enabled by the user code, and use such a license from a second computer, connected via network.

The License Manager is a program that is installed along with EM3DS, and that you have to run in the license server.

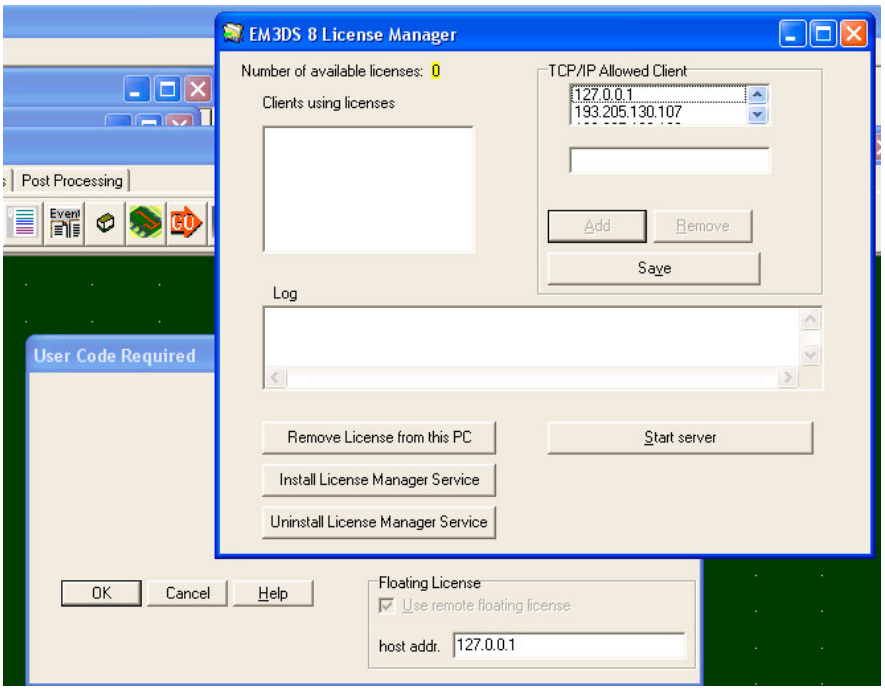


Fig.2 License Manager

Note that License Manager coming with versions 12 will also work with versions 7 to 11, while EM3DS 12 needs License Manager version 12. Moreover, licenses generated for version 12 will work also for versions 7-11, but the contrary is not true. Hence, owners of EM3DS 7-11 will need to update their license.

The first thing to do is to insert the TCP/IP address or its name in the network in the list of allowed clients; otherwise the license server will refuse the license and the event registered in the Log panel. Then you have to start the server: be sure that you allows the software to access the network if, for example, you a have firewall. License Manager will use port 8090.

If License Manager is running in the server, and you try to run EM3DS on the server itself, EM3DS will get license from the license manager, and you have to indicate as Local Host (or IP

127.0.0.1) the server. Otherwise, if you don't need floating license, simply do not start the License Manager, and EM3DS will directly use the available license(s).

To move the license into a new computer, remove the license in the old one, and send both the code generated in the old computer and the one in the new; this way MEM Research will be able to generate the code to move you license.

License Manager is a standard program, that needs to run in the server computer. Alternatively EM3DS let you install License Manager as Windows Service. In order to do that, just click on the button **Install License Manager Service.**

When computer is re-started, the service is automatically launched, and will work in background. Only, be sure that your firewall does not block its access to port 8090. The list of allowed clients is loaded by the service when it starts; the list can only be created and handled by the License manager**. Do not keep License Manager running** if a License Manager Service is installed, to avoid conflict between the two licensing schemes.

You can monitor, start and stop the service from the Windows Setup panel (Control Panel/Administration Tools). The Service does not provide direct feedback to the user: information and messages about its activity can be retrieved via the Windows Events Viewer.

V EM3DS Design Environment

In order to start EM3DS click on the **Start** button on your desktop and choose EM3DS 12. If EM3DS has not been configured by the installer, you can start the application by double clicking over the file **EM3DS 12.exe**, found in the folder where you have unpacked the program files. The EM3DS main window appears as shown below.

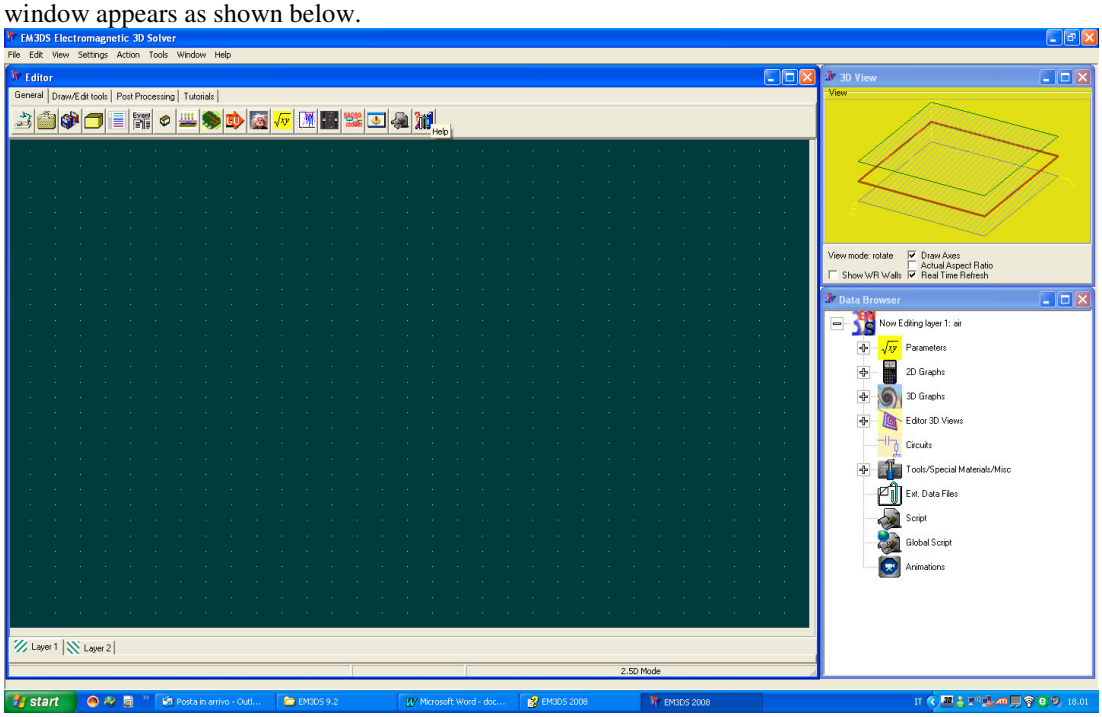


Fig.3 EM3DS main window.

A new system tray icon will appear in the Windows toolbar, where the user can request to hide/show EM3DS while running.

The workspace is the main window of the simulator. There are 4 operational modes, conveniently selected by buttons on the toolbar: General, Draw/Edit, Post-Processing and Tutorial. In General mode the user sets up the entire simulation, defines the structure of the dielectric stack, layers, frequencies, if adding a sense layer for antenna simulation, project notes etc. It is very important the users pay close attention to the appropriate set-up in this mode to properly model the structure of interest. In Draw/Edit mode users draw the structure by placing conductors and dielectrics on various layers. The default view is top-down, however, the auxiliary 3**D View window** displays the complete overview of the structure, and is real time updated. The active layer is highlighted in red. In addition to this **3D**

View, EM3DS features a **3D Viewer with Rendering** supporting OpenGL. This is an indispensable tool for getting design details in structures of a big aspect ratio or where the density is growing up quickly. This viewer is easily accessible by the dropdown menu View and selecting **3D view with Rendering,** or by the popup menu in the **Data Browser.** If in the "preferences" panel (menu **Settings/Preferences**) one selects the checkbox **Create 3D View with Rendering**, by default one of those view will be available for a new project. ***It should be noted that this feature is hardware dependent and may not work well by all graphic cards.***

The editor also shows in the lower status bar coordinates of the cursor, size of a shape being drawn, suggestions and indication of the solver mode, either 2.5D (now default) or 3D.

Data Browser in its root, shows which layer is being edited, if the user has assigned a name/label to such a layer.

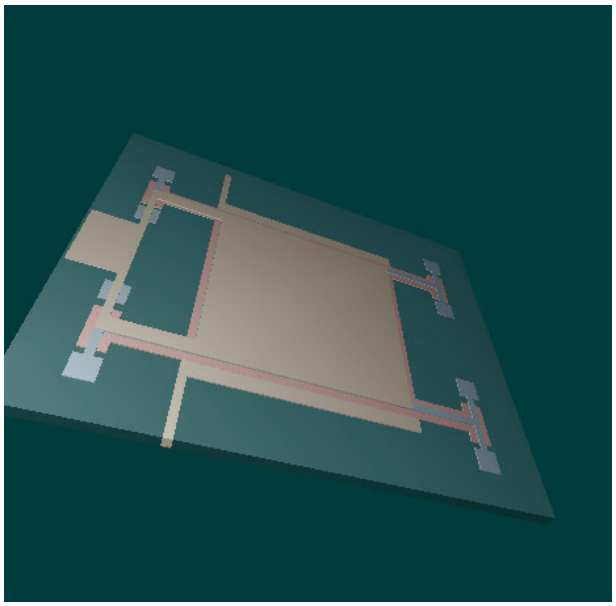


Fig.4 EM3DS 3D Views with perspective and rendering

Many advanced features like zoom in, zoom out, or internal view of closed structures are allowed. Thus, the user may carefully inspect and/or correct the design before starting simulation. In the same manner, this mode is fully real-time updated when changes in the design occur. In this same view you can see current densities plotted as pseudo-colors or as vectors. Windows may easily be removed or accessed by the **Data Browser**. The **Data Browser** is a tree-like structure to facilitate the access of multiple graphs, where all results are contained and to simplify the manipulation of external data files (e.g. provided by a network analyzer or by some previous simulation). Other than through the Data Browser, these windows can easily be accessed through the **View** drop-down menu or through buttons on the toolbar.

Every accessible toolbar button is duplicated within the main menu. A short **Hint** is provided upon mouse being pointed to the buttons.

All buttons are grouped in categories (General, Draw/Edit, Post-Processing) and are accessed by selecting items in the upper panel.

Pressing F1 invokes a context help. Every window has its context help detailing the available functions. Every window has its context help detailing the available functions.

Moreover nearly everywhere, by right-clicking an additional popup menu is made available.

Quite often an **Event** window is displayed. It is where EM3DS places notes, remarks, warnings and hints so as to help you properly set up the structure to be simulated (by its popup menu you can clean that window, so as to avoid to have old messages displayed). You can find more about these messages by searching in the main help topic. In addition, there is an appendix at the end of this manual, describing the most important and frequent messages.

A very useful newly added feature (as of ver.6) is the following: in all instances, where the user is requested to enter a numerical value, a valid expression is allowed that is numerically evaluated by the solver to yield the correct input data. One may specify dimensions or material parameters directly through expressions, e.g. if the available data is the resistivity of a conductor (say 3e-8) yet the conductivity is required, simply enter 1/3e-8 in the conductivity edit box. Material parameters (permittivity, conductivity etc.) may also be frequency dependent and/or parametric: frequency is accessed by the global parameter F or f; f is a reference to the frequency in Hertz. Hence, if for example you want to describe a resistivity that increases as the square root of the frequency, and the

first frequency of analysis is 1 GHz, you can simply assign as resistivity ro*sqrt(f/1e9). Built-in functions and syntax are discussed in the Appendix.

Since version 7, a panel with parameters is available

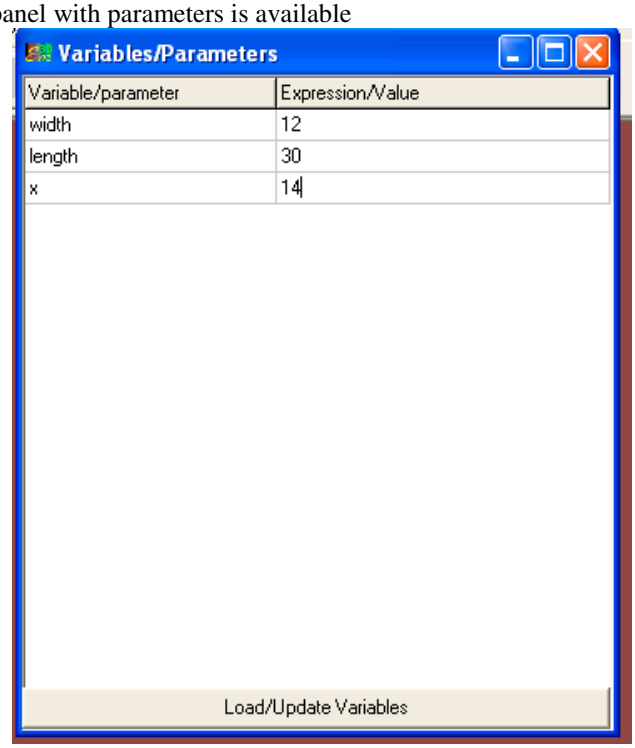


Fig.5 Variables

There you can define global parameters. You can use them to define the geometry and the position of your objects. Reference to parameters is saved when defining geometrical features and properties of materials, while elsewhere EM3DS substitutes the parameter with its numerical value automatically. The parametric definition is enabled only after clicking on **Load/Update Variables**. As an important note: imagine that you define a coordinate of a rectangle to be a parameter **x**. If you change x, the rectangle changes its position. However if you force the position of the rectangle either by mouse or keyboard (arrows) the reference to the parameter is lost. This is important when defining complex shapes, such as spirals or bends: they may be completely parametric, but it is necessary to specify their position via coordinates, not by mouse, or parameters are converted to numbers

VI Getting Started

Using the EM3DS is rather simple. The purpose of this section is to describe the most important steps you have to follow in order to edit and to run a standard simulation. Consider to use one of the tutorials to quickly learn how to set up a simulation.

Choose the box dimensions: your structure is enclosed in a metallic box- or more properly a rectangular waveguide-, the walls forming the four sides of your workspace. The first step is hence to set the size of this waveguide. To this aim either click over **Settings/Box Dimensions** menu, or over the **Box Size** button in the toolbar. The window shown below appears. Here you also select the units used to draw the structure.

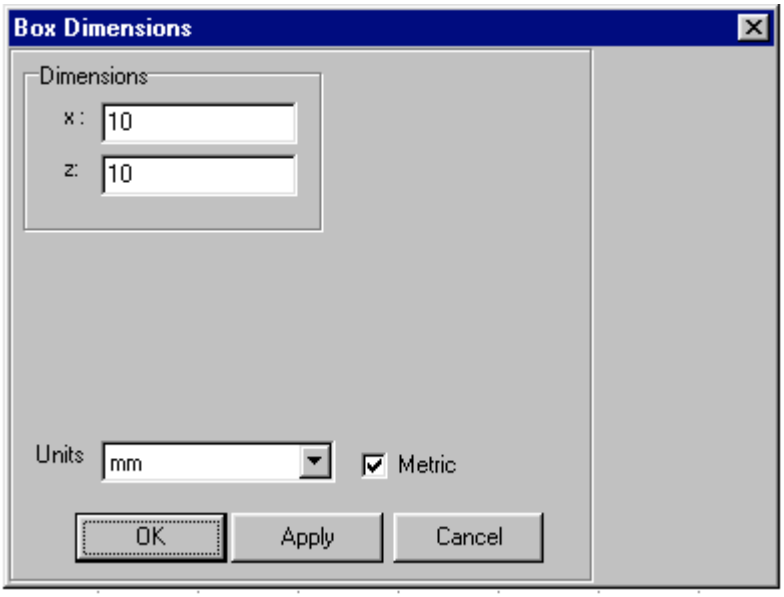


Fig.6 Box dimensions.

FAQ 1 *How do I select the size of the enclosure?*

There are a few rules to be respected when setting up the size of the enclosure. Certainly, the box must be larger than the structure itself. However a major factor affecting the computational time (and load, i.e. required memory) is the ratio between the size of enclosure and the size of the smaller part of the structure. Large circuits should be broken down to smaller circuits, and the results combined at a network level. As a rule of thumb, to minimize the interaction between the circuit analyzed and the waveguide walls, lateral walls should be placed at a distance twice the substrate thickness. It is approximately the same rule that may be applied to evaluate the distance at which two parts of a circuit may be considered standalone. If you analyze devices supposed to radiate in an open environment, like antennas, you should follow the guidelines laid-off in detail in the paragraph "Antenna modeling".

Select the composition of the substrate: the substrate is a multilayered, possibly lossy, dielectric stack. By clicking over the **Analysis and Substrate Settings** button, or over the menu **Settings/Analysis and Subst. Settings** the Substrate Information window is displayed: here you define the cross section of your structure, namely number and composition of the dielectric layers filling the waveguide where object are drawn. Items (e.g. conductors) defined within a layer have the *same thickness of the embedding (hosting) layer*.

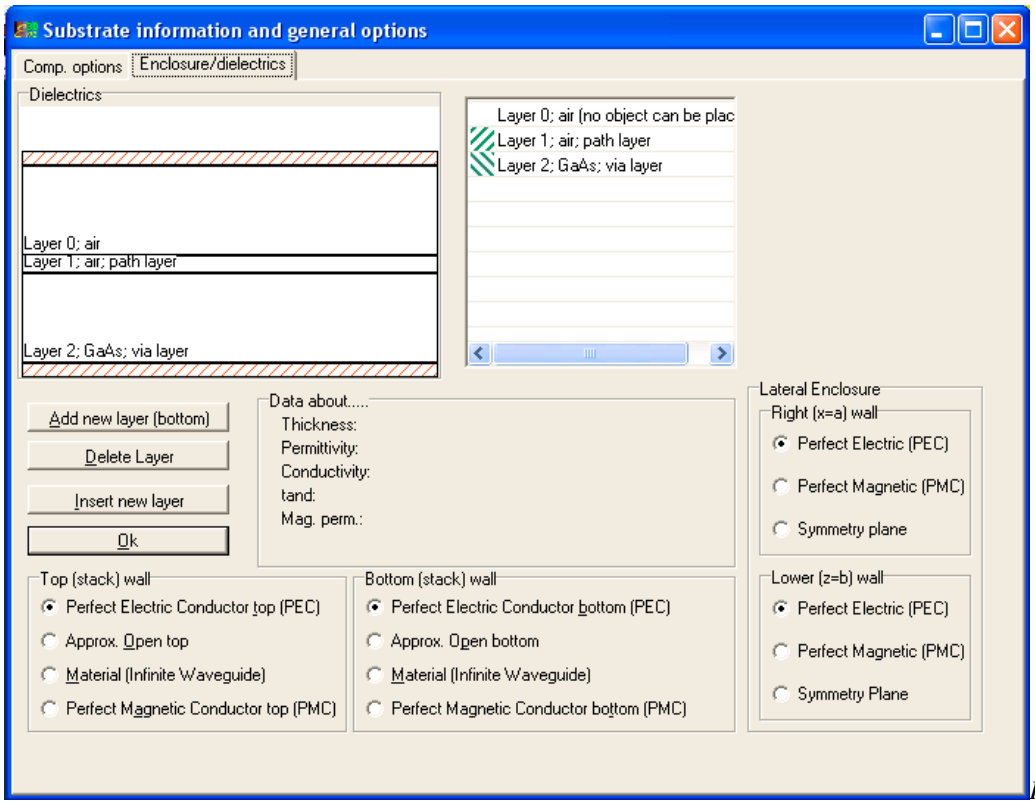


Fig.7: "Substrate Information" window: Enclosure/Dielectrics

Therefore, by assigning the thickness of a layer, one automatically assigns the thickness of each conductor defined in that layer. Figure *7* shows a simple case of a 2-layer structure (default): you have layer 2 (as labeled) and layer 1 for drawing. For example, in a microstrip circuit layer 1 could be the path (layout) layer, while layer 2 is the substrate one, where via conductors could be placed. The layer immediately ***above layer 1***, layer 0, **cannot** be accessed for conductors placing. However this layer is the **only one** that could assume thickness 0 (it is equivalent to removing that layer). It is exactly why we have 3 layers visually presented and not 2!!

The top and bottom walls may assume either perfect conductor (PEC, namely zero E tangential field), magnetic conductor (PMC, namely zero H tangential field) absorbing boundaries or replaced with semi-infinite waveguides filled by certain materials. You can also modify two of the four side-walls of the enclosure to be PMC or symmetry plane: refer to the specific section about PMC an symmetry planes.

By traversing the mouse over cross-section schematic, the **Data About...** panel displays the main properties of the tipped layer. Left-mouse click will provide access to and modify the properties of the selected layer. You can also click in layer list. A window similar to the one shown in figure *9* comes into view.

A few additional considerations should be kept in mind, according the selected working mode, either *2.5D or 3D*. The working mode is either selected from the main menu Settings (see figure below) or from the Project Data window.

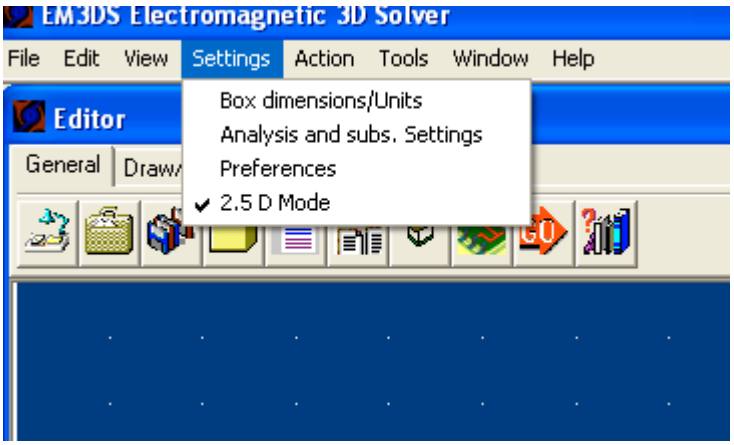


Fig.8: Selecting the working mode

The 2.5D mode is currently the default mode, as generally the 2.5D is enough, while being simpler to use, not to mention its efficiency. While many solvers consider via conductors as special entities with respect to standard conductors, EM3DS, due to its 3D nature, does not distinguish between the two. Hence, in order to make easier to convert a 3D structure into 2.5D counterpart the following assumption is made:

-in 2.5D mode layers having odd index (layer 1, 3, 5 etc) are where planar conductors are placed; they are considered *infinitely thin*, and their inserted thickness is only used to assign a value for the calculation of conductor losses. Planar conductors support only x and z-directed currents (surface currents). An empty odd layer has absolutely no effect on the calculation, regardless its thickness or composition. Hence odd layers may also be used as "dummy layers", layers not having any specific application if not making possible to use the 2.5D convention.

- by the same token, in 2.5D mode layers having even index (layer 2, 4, 6 etc) are where vias are placed. Vias have only vertical (y) currents, so that they cannot be used to model e.g. a wall of a cavity or general 3D features. If you need these features consider to switch to the standard 3D mode. A substrate **must** have even layer, or it will be neglected.

When switching from 3D to 2.5D mode, all 3D views are updated in order to make apparent this set of conventions: this makes possible to identify problems. Also, when defining layers, a red note will remind if the layer being defined has an actual dimension in space, or it is just a dummy layer used to define planar conductors. When switching from a 2.5D mode to a 3D mode it may be necessary to remove empty dummy layers, and for thick metals contacting in different layers (so producing vertical discontinuities) to introduce a slicing (Ny>1, possibly non-uniform; see below). Ver 6.3 may automatically do the job, but it will do it in the safer, yet possibly time expensive, way: EM3DS, after requesting confirmation, will set Ny=3 (non uniform, in odd layers) in any case where even layers (via layers) contained some objects.

If EM3DS is used as server for AWR Microwave Office, that embeds a 2.5D editor, layers are automatically added in order to preserve the correct meaning in the 2.5D mode, so that the user has not to be concerned with the translation.

Fig. *7* displays a simpler case of a 2-layer structure (default): you have layer 2 (as labeled) and layer 1 for drawing. For example, in a microstrip circuit layer 1 could be the path layer, while layer 2 the substrate, where via conductors could be placed.

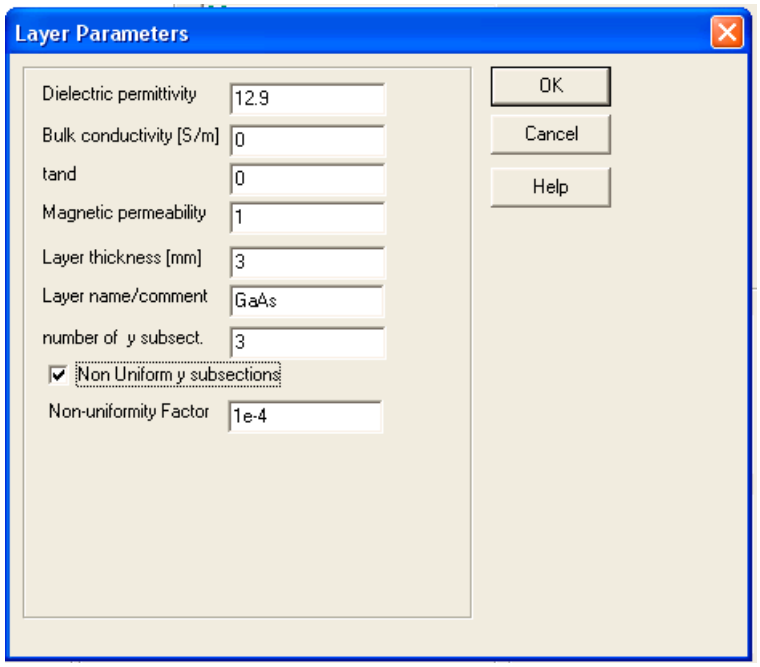


Fig.9: Setting the properties of a dielectric layer

This same window can be accessed from the main popup menu of the editor (**Edit Properties of the current Layer**) The meaning of the items appearing in that window is straightforward:

Dielectric Permittivity: relative permittivity of the selected layer (ε_r)

Bulk Conductivity (S/m): conductivity of the selected layer (S/m)

tand: dielectric frequency-independent losses

Magnetic Permeability: relative permeability of the selected layer (μ_r)

In the 2.5D mode those parameters are not used for layers having odd index, and they appear gray; nonetheless setting sound values is recommended in order to simplify a possible switching to the 3D mode

Note: the above parameters (dielectric permittivity and conductivity) refer to the host dielectric layer and NOT to the conductors or the dielectric bricks defined **within** that layer. The properties of the latter are accessed by first selecting the object (via mouse) and selecting **Resistivity/Material** specifications from the popup menu. If thick conductors are defined in that layer, the dielectric layer properties specify the properties of the dielectric *outside* the conductors.

Any parameter may involve expressions. Reference to frequency may also be inserted in material definitions by using symbols "f" or "F" in the expressions. F is in Hertz. For a list of built in functions to be used in expressions see the list of the Appendix.

Layer thickness: the thickness of the selected layer. Note that any conductor drawn in the layer has its thickness. In 2.5D mode the layer thickness of odd layers is only used to evaluate the ohmic losses of planar conductors in that layers.

Layer name/comments: insert here a string to label the layer. A remainder of the layer name will appear in the **Data Browser** when an object in it is being edited.

Number of y subsections: (only 3D mode) Conductors and dielectric bricks in EM3DS are modeled by means of volume currents. In the vertical (y: thickness direction) they are modeled as piecewise constant. This number (say Ny) sets the number of vertical pieces used to model currents flowing within conductors and dielectric bricks possibly placed in this layer. (see figure). This slicing has no meaning in the 2.5D mode, and so it is not accessible in that mode.

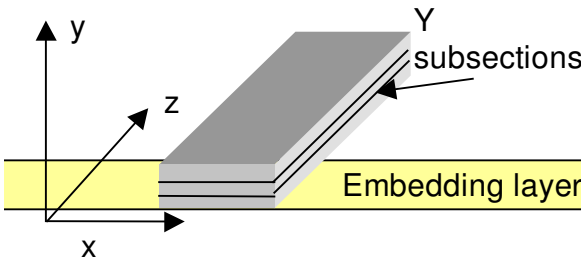


*Fig.10: Meaning of parameter **number of y subsect.***

Non Uniform y-subsections: (only 3D mode) the first and the last slices may be selected to be thinner than the others (see figure below) by checking Non-Uniform... In this case the number of slices is minimum 3, and the Non-uniformity factor allows to select the ratio between the size of such slices and the total thickness. For example Non-uniformity factor set to 1e-3 for a layer having 1 um thickness implies that the first and the last slices are

1um x1e-3= 1nm thick

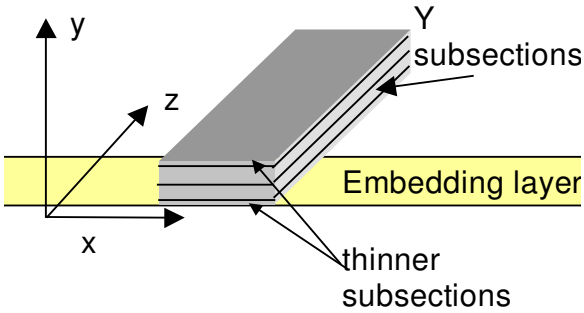


*Fig.11: Meaning of parameter **Non-Uniform y subsect.***

Using non-uniform subsections is recommended when a finite thickness conductor in a layer is in contact with one conductor in other layer. The sharp vertical discontinuity is correctly modelled with a minimum value of Ny (3), providing a powerful additional device to address problems related to such a discontinuity. Moreover in ideal conductors the current should be nearly completely distributed over the surface of the conductors: the non-uniform vertical mesh allows to describe the current more physically with a reduced set of expanding functions.

However in lossy conductors such a choice may constraint currents to flow in an unnatural manner: for low frequencies the current is evenly distributed across the conductor section, and adopting the non-uniform vertical sub-sectioning is equivalent to do a possibly wrong assumption about the current distribution. An internal device automatically compensates this related phenomenon so as to provide accurate results as well, but the user should keep in mind that the compensation relies on a set of hypothesis. Non-uniform slicing should not be used if the layer hosts for example a dielectric resonator. Uniform y-subsectioning allows to get the physical behaviour of current density as a function of the frequency, describing the skin effect and every related phenomenon, provided that Ny is enough. There is a tutorial about using the 3D mode, and one about meshing also describing and demonstrating the issue here described.

FAQ 2 *How do I select the number of subsections?*

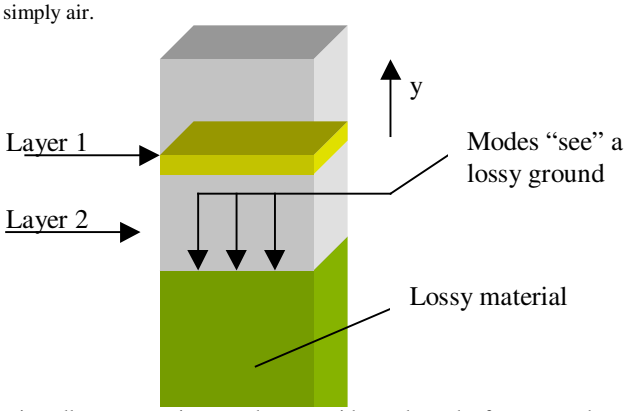
First of all, this parameter has only to be settled in the 3D working mode: in the 2.5D mode currents are either transversal (x,z) or vertical, and transversal currents are only surface currents: conditions requiring slicing do not occur at all.
In 3D mode, when selecting a number greater than 1, EM3DS acts much like the conductor being "sliced" in thinner parts. Generally just once slice (hence 1: default) is necessary, owing to some variational properties of the solver. However more slices are needed whenever **thick** conductors (where thick may simply be a few microns or even 1 micron!) are in contact with conductors in other layers (like e.g. with vias) producing a sharp vertical discontinuity -see the following To Probe Further 2 section. Sometimes more slices are needed for coplanar structures having very small interelectrode spacing, in order to account for the vertical current gradient. Finally slices are needed in order to correctly modeling losses at higher frequencies (skin effect): in this case rule of thumb is that ratio t/Ny, where t is the layer thickness and Ny the number of subsections, be roughly equal to the skin depth.

Note that rising this number the computational load grows very quickly. The best idea is to try with Ny=1 and then raise Ny in order to see how results are affected. There exists always a value after which no improvement is obtained.

If slicing is used where a quasi-planar (but thick) conductor touches a via, in order to allow for the correct modeling of current vertical bending, generally Ny *non uniform* should be used, as uniform Ny may result in a too large number.

To Probe Further. *1: setting top and bottom wall properties*

EM3DS uses the modes of the enclosing rectangular waveguide in order to analyze the user-defined structure. Setting the top (or bottom wall) as "infinite waveguide" means that the enclosing waveguide is infinitely long in the upper (or lower) direction, its modes being terminated over their characteristic impedance. The user can select how the waveguide is filled. This way a lossy ground could e.g. simulated by setting the bottom wall to **material/Infinite waveguide** and by specifying the filling dielectric having a bulk conductivity 3 10^7 S/m (see figure below). The infinite waveguide may also be used so as to simulate an open environment, provided that the rectangular waveguide is large enough to avoid direct interaction with lateral walls and to allow over cut-off modes that could model power leaking in the vertical (y) direction. In this case the top layer could be simply air.

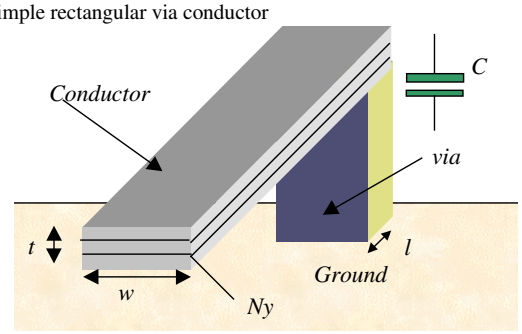


The **Approx. Open** option allows to terminate each waveguide mode to the free space characteristic impedance (roughly 377 Ohms). This is only an approximation, that may be helpful to model open environment but that should be handled with some care, owing to the approximation involved (that could lead to unphysical results). The top and/or bottom cover could be thought in this case as *absorbing* boundaries, that should be kept far enough from the circuit in order to avoid direct interaction by near-field (usually half a wavelength is ok, even if also lower values may provide satisfactory results).

To Probe Further. *2: Modeling sharp vertical discontinuities (e.g. vias)*

When placing overlapping conductors on two adjacent layers (e.g. vias in microstrip circuit) in the *3D mode*, some care has to be used. Connection across different layers causes a sharp vertical discontinuity. While this is not a problem for infinitely thin conductors (2.5D), in the 3D simulation this has to be correctly accounted for (see figures below).

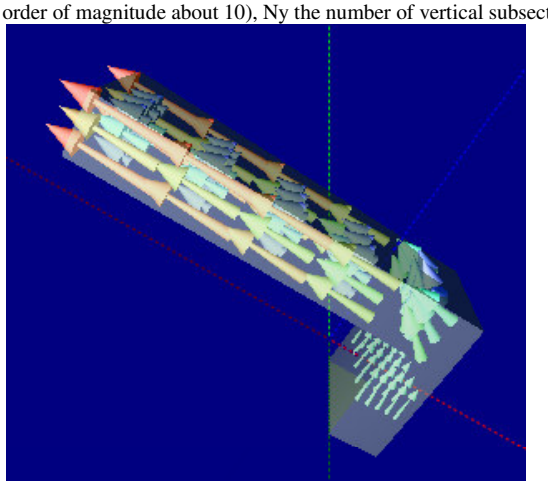
As said above, in EM3DS conductors are subdivided into vertical "slices", the first one being closest to the bottom of the dielectric: the current distribution along the vertical direction is approximated by piece-wise constant functions. A sharp vertical (in the thickness direction y) discontinuity would require a large number of vertical subsections. Generally speaking, the incorrectly accounting for the discontinuity leads to a large parasitic serie capacitor in the network representation: depending on the frequency range and on the overall topology, such a capacitance may play a significant role.
Let us consider as example a simple rectangular via conductor



The "junction" between the two conductors acts as an unphysical capacitance C given by

$$C \approx \varepsilon_r \varepsilon_0 \, l \, w \, Ny \, k / t$$

k being an empirical factor (of order of magnitude about 10), Ny the number of vertical subsections of the upper conductor.

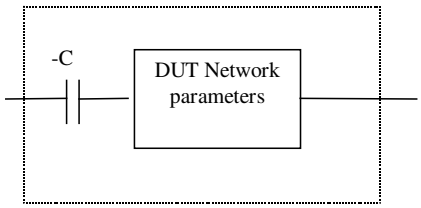


If the DUT is the via, behaving as a few nH inductor, while the feeding line is some microns thick, the resulting capacitance may well lead to a resonance in the lower microwave range.
It appears that the capacitance is related to the thickness of the single “slice”: a thinner slice means a higher capacitance. Such capacitance may be made arbitrarily large by raising the number of y-subsections (Ny), or by reducing the conductor thickness or by raising the dielectric permittivity of the embedding material. Ny is selected with the "substrate parameters" as "number of y subsect.". *Actually in version 5.2 a powerful device has been added*, namely the possibility of using non-uniform slicing: the first and the last slices may be selected to be much thinner than the remaining ones, allowing to circumvent the capacitance issue by just Ny=3, while keeping the actual conductor thickness (see previous section). However this device forces currents to flow in a way similar to the currents you would have if you connect 2 thin conductors (the first and the last thin slices) by via (the middle slice): upper and lower conductors experience most of the horizontal current flow, while the middle one supports a great vertical (y) component. In ideal conductors this is generally ok, as we know that currents have to flow across the surfaces, but in lossy conductors this lead to a wrong resistance of a conductor. EM3DS implements an internal correction for this case (Ny non-uniform and lossy conductors) but one should be aware this is only an approximation. This is why, when using lossy conductors with non-uniform slicing, a reminder in the Message window is displayed. By the way, this expedient works remarkably well.
Coming back to the standard uniform Ny, generally speaking, Ny in lossy conductors has to be chosen in order to allow to correctly model the "skin effect" with increasing frequency, hence Ny is the parameter controlling the accuracy in the evaluation of losses for high frequencies (one "slice" should be thinner than the skin depth at the upper frequency). Still it is possible to keep Ny=1 and, when defining the conductor resistivity, to select the check box **Force Skin-Effect**. In this case the resistivity is assumed to have frequency dependence accounting for skin effect. Note that the frequency variation is evaluated by assuming that the currents penetrate only the lower faces of a strip. This assumption is generally good for microstrips, but may be poor for other structures (e.g striplines). If this is the case one should either resort to Ny>1, or write his own frequency dependent resistivity (remind: EM3DS allows to write parametric expressions when defining the structure).

Finally we have seen that there are cases, such as the via contact, which may require Ny>1 even at low frequencies. As a general rule, Ny should be greater than unity whenever variations of the current densities along the conductor thickness are expected.

The problem about the parasitic capacitance may be solved by using 3 possible approaches:

1- By raising Ny: the number of y-subsections is available when defining the composition of the dielectric stack. The possibility of defining non-uniform vertical slices provides one more tool to circumvent such a limitation
2- By evaluating the parasitic capacitance and by manually removing (de-embedding) such a capacitance: the user should build up a structure involving only the junction and try to evaluate the parasitic capacitance. Once the capacitance is known, a serie negative capacitance may be added to the network in order to cancel out the above effect (see fig. below)



Calibrated net

If the whole via has to be manually calibrated, the user can analyse the via and connect the resulting network as a "Negate network" to the network resulting from the simulation of the whole structure by using the embedded linear circuit simulator of EM3DS

3- [mostly obsolete] An approximate procedure is included, and it is the most economical one: the user can try to make the upper conductor as thin as possible (e.g. 0.0001 um), while scaling the resistivity so as to simulate a thicker conductor, namely to keep invariant the resistivity. This is done automatically by selecting **Scale Resistivity** in the **Resistivity and Material specifications** dialog box (select a conductor, then right click in order to see the popup menu), and by entering in "**Scale to thickness**" the desired (hence the real physical) thickness the conductor should have.

While approaches 1 and 2 are rigorous in principle (actually 2 requires some hypothesis and restriction about the possibility of accessing the physical point where the parasitic capacitance appears), approach 3 is based on a number of assumptions and basically bears resemblance to switching from a full 3D point of view to a classical 2.5D. *Nonetheless conductors are not infinitely thin, and vertical currents are evaluated as well*. Hence, instead of using approach 3, **consider to switch your project to 2.5D** working mode. This would save a significant amount of time and memory space, while raising the robustness. Approach 3 should only be considered where some kind of mixed 2.5D/3D approach should be used (e.g. planar structure still involving some true 3D detail).

In standard microstrip design approach the 2.5D working mode is recommended (as well as for a large class of CPW), while in MMIC and MEMS design, approach 1 is generally needed. Some tests over benchmark structures will give you the necessary sensitivity to select the best way.

Select the Frequency Range: EM3DS is a frequency domain tool, basically providing network parameters over a given set of frequency points. Nonetheless it provides several time-domain tools, that will be affected by your choice of frequency points: see the section about Time Domain for more information.

In order to set up the frequency range over which the analysis will be performed, you have to access once more to the **Substrate Information and General options** window, by clicking over the **Analysis and Substrate Settings** button, or over the menu **Settings/Analysis and Subst. Settings.** This time we select the **Comp. Options** panel (default). The window appearing is shown in figure *13*. There you can set the first and the last frequency points to be analyzed. The parameter **Num. Points indicates** how many points in the given frequency range are calculated; the step is calculated and displayed on the right upper corner of the window.

Actually there is an additional device, **SmartFIT**: SmartFIT is an adaptive rational interpolation. If you select to invoke SmartFIT, EM3DS will decide the frequency points to be evaluated, and finally will generate a complete response, in a number of points decided in the SmartFIT Dialog (menu **Tools/SmartFIT**)

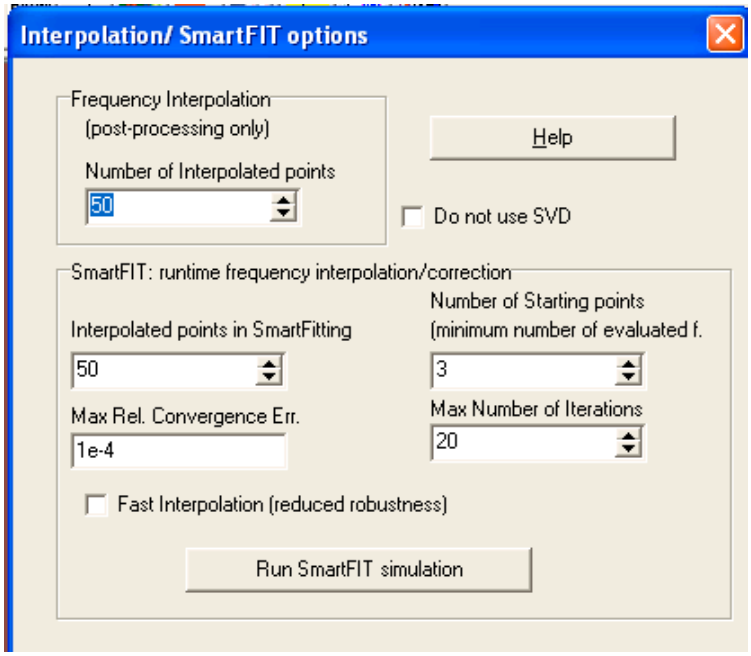


Fig.12: SmartFIT dialog

The number of generated points is the one indicated in **Interpolated points in SmartFitting**. Other parameters include the **maximum Relative Convergence error**, the **Number of Starting Points** (usually 3) and the **maximum number of Iterations**. See the section about SmartFIT for more information.

Note that you can perform several analyses spanning different frequency ranges. Results are cumulated until the structure *or its mesh* are modified. This way analysis can be performed also for unevenly spaced frequencies. Only one care is needed: if you change the upper frequency of analysis, EM3DS ask you if updating the mesh for the new selected frequency band: if you select "yes", old results are discarded (as well as asymptotic estimations [see *to Probe Further 3*]). Hence if you want to cumulate results, you have to select "no". This is also true if you want EM3DS to keep in memory the calculated Asymptotic Estimation (so as to speed up the calculation over the new range).

To Probe Further. *3: Additional options*

There are also several other items that can be entered by the same dialog window. These are

Num. Modes in the x direction: as said before, in order to build up the electromagnetic solution, EM3DS uses the modes of the waveguide enclosure. More properly, modes are used to build up a Green function, a preliminary step toward the solution of the problem. This item let you select how many modes have to be considered. More modes generally mean more accuracy, of course paying with longer running time (at least at the first frequency point). What determine the number of modes strictly required for a sound solution is the ratio between the enclosure dimensions and the smaller discretization cell, namely **mesh cell**. Mesh cells are individuated by vertical and horizontal mesh lines, hence the ratio to be considered is the ratio between one of the enclosure dimensions (e.g. the horizontal one) and the smaller horizontal interval individuated from vertical mesh lines.

The "**Suggestion**" box indicated the number of modes according to Nyquist' theorem: the values are copied in the **Num. Modes** boxes by clicking over the **use suggested values** button. If the checkbox **Always use suggested values** (default) is used, EM3DS will automatically load suggested values when running a simulation. Mesh lines requiring more than 1000 modes are automatically suppressed. By selecting the **Suppress wide-band mesh** check box, and by entering the new value in **limit num. Modes** may control this suppression. *However the use of this option is strongly discouraged, as the resulting mesh may not correctly account for the structure being analyzed.*

Output Touchstone file: this is the name of the standard output function (no extension) the EM3DS uses for the non de-embedded (raw) network result, automatically created for backup

If **Save EM computation** is selected, the current density distribution is stored in a binary ".DAT" file. You will need this ".DAT" file in order to access to the calculated results.

If **automatic de-embed** is selected, EM3DS will enable specifying non zero reference planes for the resulting network parameters: solution will be referred to these planes, while any discontinuity due the ports and the lateral walls is removed by an automatic calibration procedure. Whichever is the reference plane selected for calibration, EM3DS will also make available results for reference at the port plane (feature of ver 6.3)

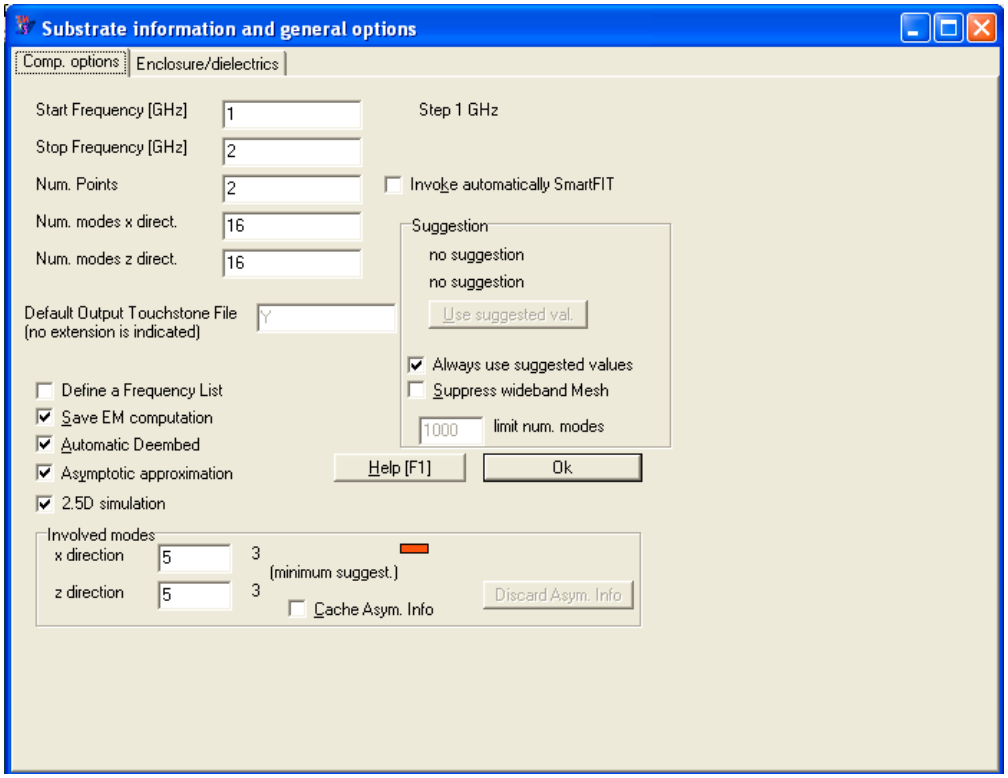


Fig.13: "Substrate Information and General Options" window: Frequency and computation options

Define frequency list allows to create a list of frequencies where the device has to be simulated.
If the check box **Asymptotic Approximation** is selected, EM3DS computes and stores information about the frequency behavior of the Green Function needed to build the solution, and uses this information in order to *dramatically* speed up the computation over the remaining frequency range. In fact only the modes indicated in the "**Involved modes**" panel are used over the remaining frequencies. The lower the "involved modes" the lower the computational time. More modes means more accuracy; however generally no more than 10 modes (both in x and z direction) are needed to obtain excellent results. Generally 3-5 modes are enough! The computation may be really fast for several frequency points. The Asymptotic information is available until the user modifies the structure or the mesh settings: if you change the **Stop Frequency**, you will be asked if updating the mesh according to the new wavelength or not. *By updating the mesh the Asymptotic Information is discarded*. If you believe that the actual mesh is good for the new **Stop Frequency**, it is better to leave the mesh unchanged, so as to use the stored asymptotic information. The only structural modification that you can make without loosing the "Asymptotic information" is changing the resistivity of conductors or the dielectric permittivity of dielectric bricks (NOT of the substrate).

Note: the suggested value for the minimum number of "Involved modes" is usually very reliable. However, if a conducting path (e.g. a microstrip) is made over a strongly lossy layer (e.g. semiconductor substrate), the indicated value may be rather underestimated. The underestimation may also occur if parameters of the dielectric stack are frequency-dependent.

If the **cache Asymptotic Info** check box is selected (and **Save EM Computation** is selected), data about the Asymptotic approximation will be saved in a ".~mp" file and will be available for future use [EM3DS always looks for that files, and if files exist it tries to load them]: note that "~mp" *files are usually very large* and saving or opening them may require a while. A red light in the "**Involved modes**" panel means that no Asymptotic Information is available (hence a full computation is required at the first frequency point). A green light indicates that asymptotic information is available, while a yellow light means that a file containing asymptotic information has been found and it is currently being loaded.
Actually nothing comes for free: using the "Asymptotic Approximation" requires more memory. Moreover a feature of EM3DS is that currents are modeled by piece-wise sinusoidal (PWS) functions with frequency-dependent argument: this results in physically sound results even with very coarse mesh. Whenever the "Asymptotic Approximation" is used, the argument of these functions is fixed at the lower frequency of analysis, so partially loosing this attractive feature. However the reliability, the speed and the robustness of the Asymptotic estimator largely compensate the above drawback, and its use is strongly recommended.
The **Discard Asymptotic Info** button may be used in order to discard information stored when we believe that the information is unreliable. This may happens e.g. if we start an analysis but we stop the computation while filling the Green matrix (actually its discretized counterpart, the MoM matrix) at the first frequency point.

Draw the structure: The tool provides a simple editor: by selecting the menu **Action/ Add conductor** or by clicking over the corresponding button on the toolbar the user can draw rectangular shapes. Non-Manhattan polygons (namely polygons that cannot be described by drawing rectangles) may be drawn by the **Add Polygonal cond.** menu, while circles are added and handled as polygons by clicking over the **Add circle** button. Circles are approximated by polygons: a better approximation usually means a higher computational load. Consider that there are a few cases where such a high

degree of approximation is actually needed (e.g. dielectric resonators). Also **Circular Spirals**, **Curves**, **Slanting lines** and **Rings** are available.
As to the rectangular shapes, once selected **Action/ Add conductor,** the top left corner is fixed by clicking the first time over the workspace, while after a second click the right bottom corner is settled and the conductor is drawn. The horizontal co-ordinate is referred to "x-direction" while the vertical one is "z-direction": the design is a top view, but a real time 3D view is always available, where the active layer is highlighted in red. The direction normal to the dielectric plane is the "y-direction", namely the direction of the conductor thickness and of the dielectric stack (substrate). The axis origin is the top left corner on the screen. The reference system is displayed whenever attempting to modify conductor or box dimensions
When clicking over **Add Polygonal cond.** Menu, a dialog box will ask you if the polygon has to drawn by mouse or by specifying vertices. In the first case every click will identify a vertex, while pressing "ESC" or by right clicking and selecting **Close Polygon** from the popup menu or by clicking a second time on the Polygon button, *the first and the last vertices will be joined* forming a complete polygon**.** If you draw a polygon that is actually a rectangle, shape is automatically converted into a rectangle, namely sharing all the features of rectangular shapes (as the possibility of adding ports). Polygons can accept ports: the only limitation is that to a single object (shape) no more than one port per side can be added; if more ports are necessary just split the object in two or more shapes. Also, it is possible to add internal port; the limitation is that no more than one internal port per object can be added. If more are needed, split the object. Press shift to select **multiple objects**, or use the main menu **Edit/Select Group**, or from the popup menu, or from the button  to enable selection of multiple objects. Multiple objects can be moved by keyboard arrows, cut, copied, pasted, edited for meshing and material properties.

By double-clicking over an object the **Dimension and Position** dialog window is open. Additionally, once an object or a group is selected you can

- *move* the item(s) using **keyboard arrows**
- *delete* the item(s) using **del key**
- *change* the size of the item using **Shift+the arrow key (only for single rectangular conductors)**
- *right click* with the mouse over a item to get additional (popup) menu
- *double click* to access all the object features (position/size, mesh options, material etc)
- *copy* (**ctrl+c**) or menu Edit/copy or button or popup menu
- *paste* (**ctrl+v**) or menu Edit/Paste or button or popup menu
- *cut* (**ctrl +x**) or menu Edit/Cut or button or popup menu
- *mirror x* **(x)** or menu Action/Mirror x or popup menu
- *mirror z* **(y)** or menu Action/Mirror z or popup menu

When an object is selected, a popup menu allows changing position and dimensions of the selected conductor, changing the mesh options, changing the conductor resistivity, selecting the position of de-embedding planes, checking what shapes are in contact with the selected one. You can also double click on an object in order to get a dialog window where to select position, mesh, material etc.

A good idea might be to parametrize your structure: define parameters first (see **Parameters and Variables**), and then use them when defining position, size and vertices of your shape. Any update of the variables automatically updates your structure. Moreover this way you have full access to the internal Optimizer and to the Tuner, as well as to the Macromodel builder.

Undo (Alt+backSpace) and Redo (alt+Shift+Backspace) buttons are available.

In the status bar dimensions and position of the object or of its circumscribing rectangle, are displayed.
By clicking over the menu **Action/Drag Mode** or its counterpart in the toolbar, drag & drop is enabled, so that the user can easily displace objects by mouse (not working for groups, only for single objects).

The popup menu allows changing position and dimensions of the selected conductor, changing the mesh options, namely how conductors are discretized, changing the conductor resistivity, selecting the position of the reference (de-embedding) planes, checking what shapes are in contact with the selected one.

Add ports: the structure just drawn has to be excited by some kind of source in order to obtain a network description of its behavior. Excitation is applied by means of Ports. You can add Edge or Via ports. To this aim just select menu **Action/Add Port,** or **Action/Add internal Horizontal (x) port,** or **Action/Add internal Vertical (z) port,** or **Action/Add via port** and then click over the desired conductor. Ports are unit voltage generators used in order to excite your structure and to "measure" currents.

Edge ports are excitations applied with respect to the lateral walls, while via ports are vertical unit voltage generators, applying an impulsive y-directed excitation field. Internal ports are voltage generators, either along x or z directions, applied across a z or x cut in a conductor.
Ports can be selected by mouse, edited or deleted (also you can use DEL key when a port is selected)

Select reference (de-embedding) planes: you can specify a de-embedding plane for each side of the box, even if this is not strictly necessary: these planes are accessed by right clicking over the conductor, by selecting **specify de-embedding planes** from the popup menu (if the conductor has at least one port) –or by double clicking over an edge port- and by specifying the distance from the wall. A de-embedding algorithm removes discontinuities due to the excitations and the part of feeding lines up the defined plane (indicated as a green thick line). In order to perform de-embedding one "standard" structure for each side is automatically built and analyzed. Regardless the position of the reference plane, EM3DS performs the calibration and provides additional results referenced to the port-plane.

Run simulation: just press the **Go** button. You can stop the simulation at any time, still having available partial results. Running may be resumed later (some care is needed if calibration is enabled, and you stop the program after terminating the computation of the structure but before the calibration process is terminated; best to stop the program when a new frequency point is being analyzed). If the selected number of modes for the enclosing waveguide is not equal to the number expected by EM3DS, a warning window is displayed, unless the checkbox **Always use suggested modes** is selected (default) in the "Substrate and General options" window, so that you can either modify according to suggestions or continue ignoring the message.

Alternatively click on the SmartFIT button: EM3DS selects adaptively the frequency points for you and estimates the full band response

Now you have results that you can plot, save and export. In any context, by pressing F1 key, help is made available

VII Perfect Magnetic Conductors (PMCs) and symmetry planes

Since EM3DS 8 PMC planes are implemented. A PMC is a plane where tangential magnetic fields and orthogonal electric fields are zero, namely dual conditions with respect to standard electric conductors.

PMC are useful to handle symmetric structures and hence to save computational time and load. In EM3DS you can set as PMC either top and bottom walls, as well as two side-walls of the conclosure (see the brown walls in the figure).

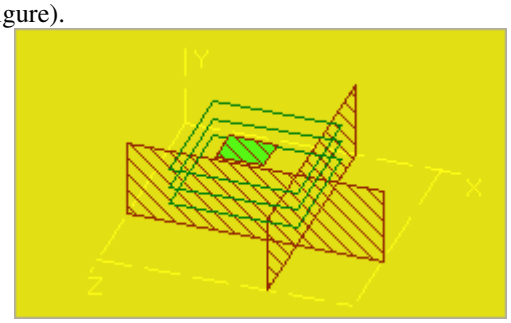


Fig.14: Two side walls are PMC

PMC can be settled in the **Substrate Information and General Options** dialog, accessed via menu **Settings/Analysis and Subs. Settings**.

When a PMC is defined, the structure is somehow duplicated, so that fields are the same that one would obtain if the structure was mirrored with respect to the PMC.

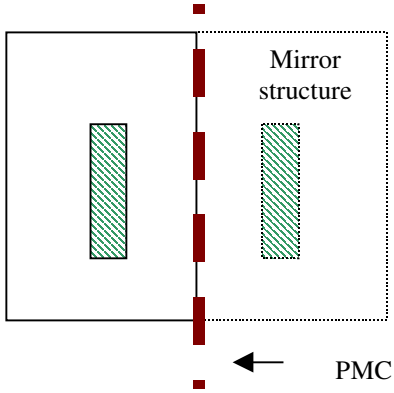


Fig.15: A PMC mirrors a structure

A PMC wall can be hence used to solve half of a symmetric structure. Note that if a port is defined coaxially to a PMC, network parameter Y would have a factor 2 of difference with respect to the parameters of complete structure, owing to the fact that only half a port is considered in evaluating currents flowing. In this case you can choose **Symmetry plane**, instead of PMC, as this will automatically correct such a factor. An example will clarify:

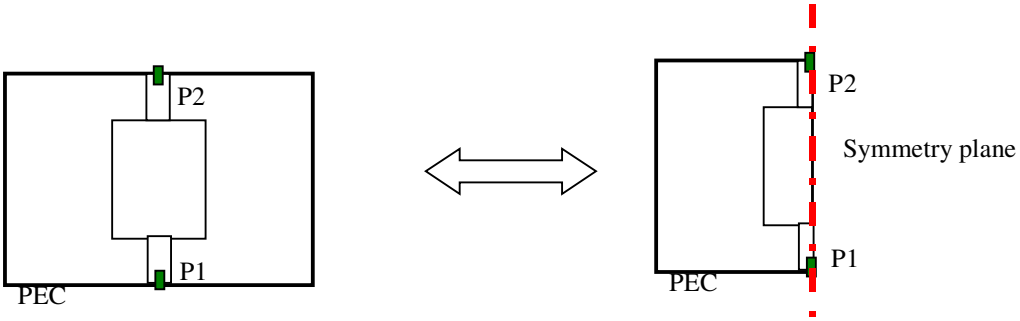


Fig.16: use of Symmetry plane

The example above (top view) shows how the symmetry plane works: the two structures will give exactly the same S-parameters (or Z/Y parameters), while the one in the Right Hand side will require much lower computational effort. If, instead of the symmetry plane the user would have selected a PMC, the two structures would have exactly the same fields, but the network parameters (Z/Y) would differ of a 2 factor, as they would be evaluated over one half of the port sections.

Combination of PMC and PEC simulations can be used to perform effectively simulations of 2N-port symmetric devices by means of 2 simulations of N-port devices. Actually this line of reasoning is valid for more complex symmetries. EM3DS can handle automatically symmetries over one plane, as detailed in the next section.

VIII Combining PMC and PEC (or even/odd) simulations: the Symmetry Wizard

The **Symmetry Wizard** is a useful post-processing tool to recover the complete analysis of symmetric structures from the analysis of two half-structures. This can be employed e.g. to characterize couplers or multiport structures having a symmetry plane.

An example will clarify. Consider e.g. the microstrip coupler below, with 4 ports (top view).

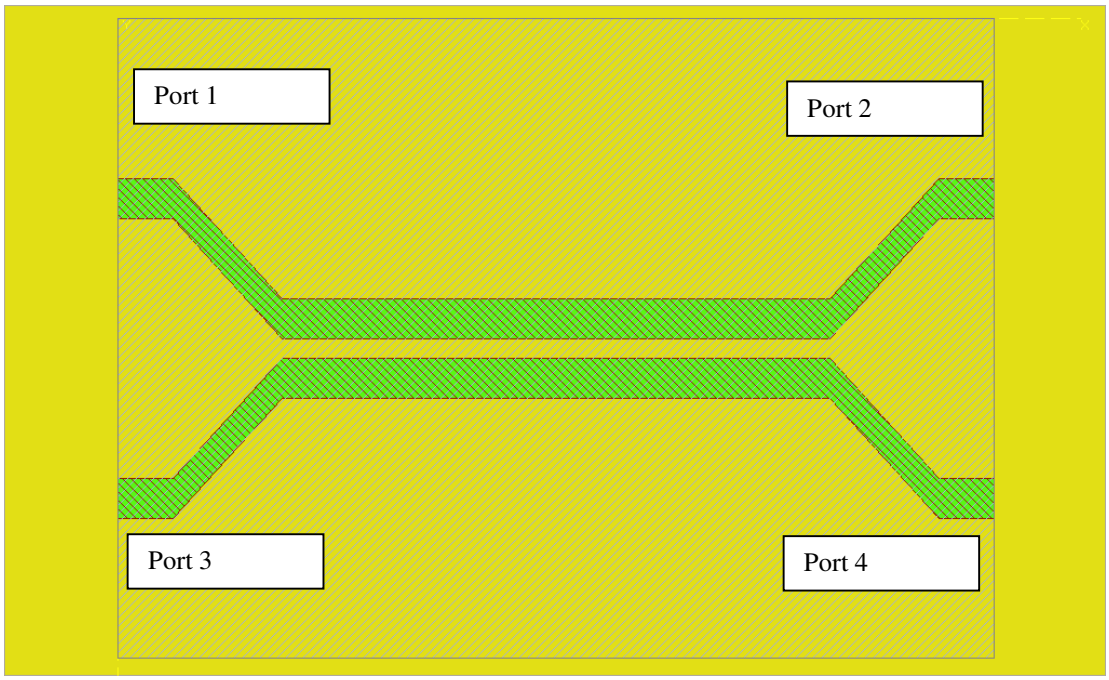


Fig.17: Microstrip coupler

One can actually create two substructures, each one with 2 ports and half of the size of the original one. The first one where the symmetry plane is replaced by the electric wall (or PEC) of the box (see left figure) and the second one where the symmetry plane is replaced by the perfect magnetic wall (PMC) of the enclosure (right image).

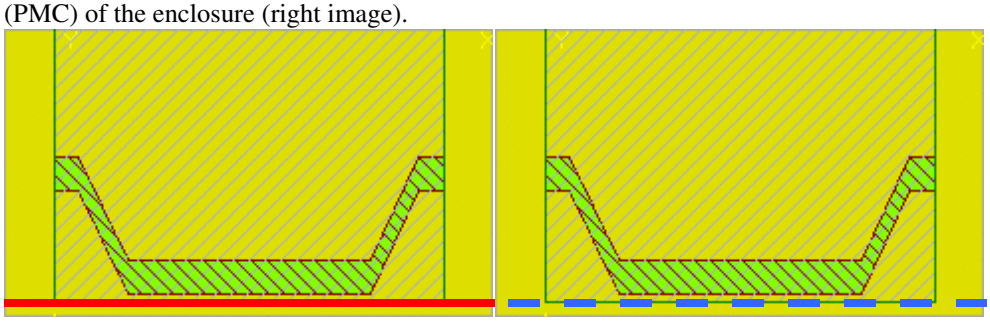


Fig.18: Two half structures, one with electric wall (PEC, odd simulation) and one with magnetic wall (PMC, even simulation)

The user has just to simulate separately the two structures, then invoke the **symmetry wizard**.

In fact if $\mathbf{Y}_e$ and $\mathbf{Y}_o$ are 2-port network parameters for the even and odd mode simulations, $\mathbf{Y}_{LL}$ and $\mathbf{Y}_{LR}$ for the four-port are

$$\mathbf{Y}_{LL\,i,j} = \left(\mathbf{Y}_{e_{0,0}} + \alpha_{i,j}\mathbf{Y}_{o_{0,0}}\right)/2$$
$$\mathbf{Y}_{LR\,i,j} = \left(\mathbf{Y}_{e_{0,1}} + \alpha_{i,j}\mathbf{Y}_{o_{0,1}}\right)/2 \quad for\ i,j = 0,1$$

where

$$\alpha_{i,j} = \begin{cases} +1 & if\ i = j \\ -1 & if\ i \neq j \end{cases}$$

The symmetry wizard will automate this kind of calculation for an arbitrary number of ports. Note that you have to use **PMC** and not **Symmetry Plane** as boundary condition in order to have correct results. If mesh of the original structure and of the two half is identical, results will be completely identical. If you note any difference, this is usually due to the fact that meshing can change according to the different size of the side walls.

In order to invoke the symmetry wizard you one can either click on the main menu "**Tools**" or select the button from the toolbar or from the "tools" icon in the data browser.

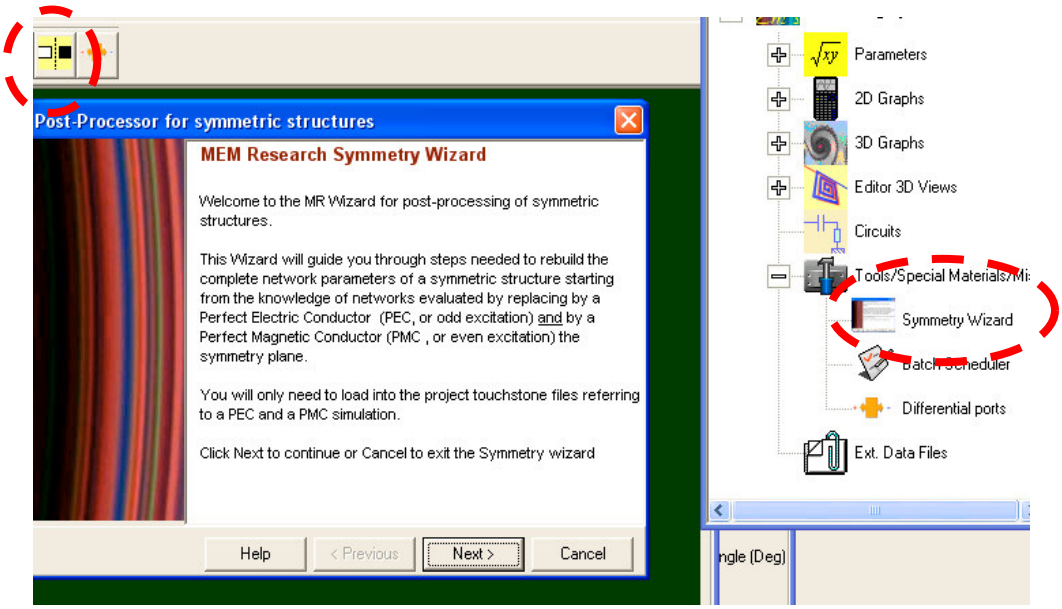


Fig.19:Symmetry Wizard

The results for the PEC and PMC simulations have to be selected from the list appearing in the Data Browser window, in the "External Data Files" folder, by simply clicking on the desired name when prompted by the wizard.

If the desired data files are not there, they can be imported by right-clicking over the Data Browser and by selecting from the popup menu **Add Data Source.**
The resulting 2N port device will appear as a new data file in the Data Browser, automatically saved and available in any chart of EM3DS as Data Source.

Port numbering of the new 2N port is assigned according to the following convention: first ports of the original half structure, then mirrored ports.

IX Mesh options: things to know for an efficient use of EM3DS

In order to obtain network parameters and field properties for the structure being analyzed, EM3DS calculates the volume current distribution within every conductor and dielectric brick defined by the user (not the substrate!). To this end, currents are approximated by a set of known functions defined over rectangular sub-domains, a procedure known as Galerkin method. The unknowns that EM3DS calculates are the (complex) amplitude of this sub-domains currents. A tutorial about meshing will demonstrate some of the issues described in this section.

The process of defining the rectangular sub-domains is usually indicated as **meshing**, while sub-domains are **mesh cells**. Mesh cells are automatically defined in EM3DS, i.e. the meshing procedure is automatic. However it is easy to understand how the meshing procedure is a key process in obtaining an efficient and reliable simulation. Users should anyway understand some basic rules underlying the mesh process.

Meshing is performed automatically over any shape drawn by the user in the workspace. It is a real-time process, so that any time you click in the blank part of the workspace, shapes are meshed. You can disable or enable the real-time mesh by clicking over **Action/Mesh** (in complex structures, especially involving curves, mesh may take time).

Meshing is performed according to the **Mesh Options**: Mesh options are local properties (that is each shape has its mesh options), that are automatically settled by EM3DS but that the user can (and sometimes should) access and modify. To this end, you have only to select a shape, right-click over it in order to see the popup menu, and to select **Mesh Options** as displayed in figure below (fig. *20*). Properly selecting Mesh Options requires some more insight about the mesh algorithm.

Mesh cells, being rectangular domains, are identified by mesh lines (or intervals), namely horizontal (x) and vertical (z) intervals. These intervals are placed according to a few basic rules (see figure *21*), the main ones being

1- there are so called **Border or Edge Mesh Lines**. The main function of edge is to improve the ability to model near singular edge currents, hence to ensure that sharp current gradients near edges are correctly modeled. At low frequencies (low when comparing wavelength and

structure size) edge mesh lines may even be the only ones needed in order to mesh the structure. The user has a nearly complete control over the position of the Border Mesh Lines. The parameters **Edge Mesh Distance** (see fig. *22*) indicate at which distance from the border, as fractions of the box size, edge Mesh Lines are placed. For example, a value of 16 for the **Edge Mesh Distance (x)** parameter says to EM3DS to place vertical lines near the border, at the distance **a/16**, where **a** is the horizontal box (or waveguide) size. Generally values range from 1 to 100-150. Note that if these lines are close to other lines, so probably unnecessary, the current version will automatically suppress them (this may be a drawback if you *want* that lines; it is simply circumvented by raising the Edge Mesh Distance) . Edge Mesh Lines may also be suppressed by the user: in this case deselecting **Add Edge Mesh x**, vertical border lines (namely lines identifying *intervals* over the x axis) are not defined. By deselecting **Add Edge Mesh z** horizontal border lines are not defined. EM3DS usually sets the value of the edge Mesh distance according to the size of the user-selected grid (you can draw either on a grid or not, but the hypothetical grid may be used anyway as reference for edge mesh setting). Edge lines are "static", i.e. not updated with frequency.

2- There are the **Shadow Mesh Lines**: these are static lines produced at the intersection between different shapes, even if shapes are on different layers (in this case a shape seems to project a sort of "shadow"). Vertices of a polygon also produce shadow mesh lines on shapes lying in some other layer. In multilayer structures, using polygons with very large number of vertices may create overmesh. In version 8 **shadow lines produced by objects in different layers, and separated by at least one layer, may be suppressed**, so simplifying the mesh when the two layers involve complex or non-Manhattan shapes. This is done by acting on a local option, to be selected in the mesh options (**no shadow mesh cells**) of the object. In practice, if such an option is selected for a given object, the mesh of the latter will not be affected by the presence of objects placed at least one layer away. Of course, in this case, the user should ensure that mesh, manually settled, be satisfactory for correctly describing the coupling with objects in different layers. Note that if two objects are in adjacent layers, the shadow mesh suppression has no effect, or the current continuity would be violated. This is, for example, the case of vias: polygonal vias should be always approximated with a lower number of vertices

3- There are **frequency-dependent** mesh cells: the higher is the frequency, the higher is gradient of the current density in the conductors. In order to account for that gradients, EM3DS adds mesh lines according to the user-defined upper frequency of analysis. These cells are settled according to a worse case basis, so that sometimes it may be useful to access and modify the size of these cells. The dimension of the cells is entered by the "**Cell Size**" parameter, x for vertical intervals, and z for horizontal ones. When drawing general polygons or circles, this parameter is also tentatively settled by EM3DS so as to correctly discretize in a staircase approximation general shapes. If the generated value is not satisfactory (this sometimes happens with slanting lines) a good idea is try to reduce this size (see next paragraph).

4- The additional mesh option **Uniform Mesh** was added for polygons: mesh cells become independent from the position and the number of vertices, being simply uniformly added so as to fill the selected shape. The size of the cells in this case is specified as fraction of the box size (e.g. if cell x is 16, the size of the cell is *a*/16, *a* being the box width). Using the Uniform Mesh options, problems related with the number of vertices of a polygon are partially fixed

Mesh line should be placed so as to avoid unnecessarily close mesh lines. In fact a dense mesh implies a great number of unknowns in the final system (see **View/Information about Computation**), and a great number of modes (that will affect nearly only the first frequency point if the asymptotic estimator is enabled). If e.g. the box is 1 mm wide and two vertical mesh lines are only 5um apart, building the Green function will require roughly 1mm/2um=500 modes in the x direction!

In order to give an idea of what should be considered sound and what probably wrong:

About unknowns (related to the number of mesh cells): 200 unknowns are a small system, 1000 unknowns are a medium system, 2000 unknowns are a large system, and 5000 unknowns may be, to date, hard to be solved in a PC (the number of unknown affects in the same way the analysis at any frequency point).

About modes (related to density of the mesh cells): 30 modes (in each direction) are a small number, 100 modes are a medium number, 200 are a large number, 400 starts to be hard to be solved (the only advantage being that the computational effort is limited to the first frequency point).

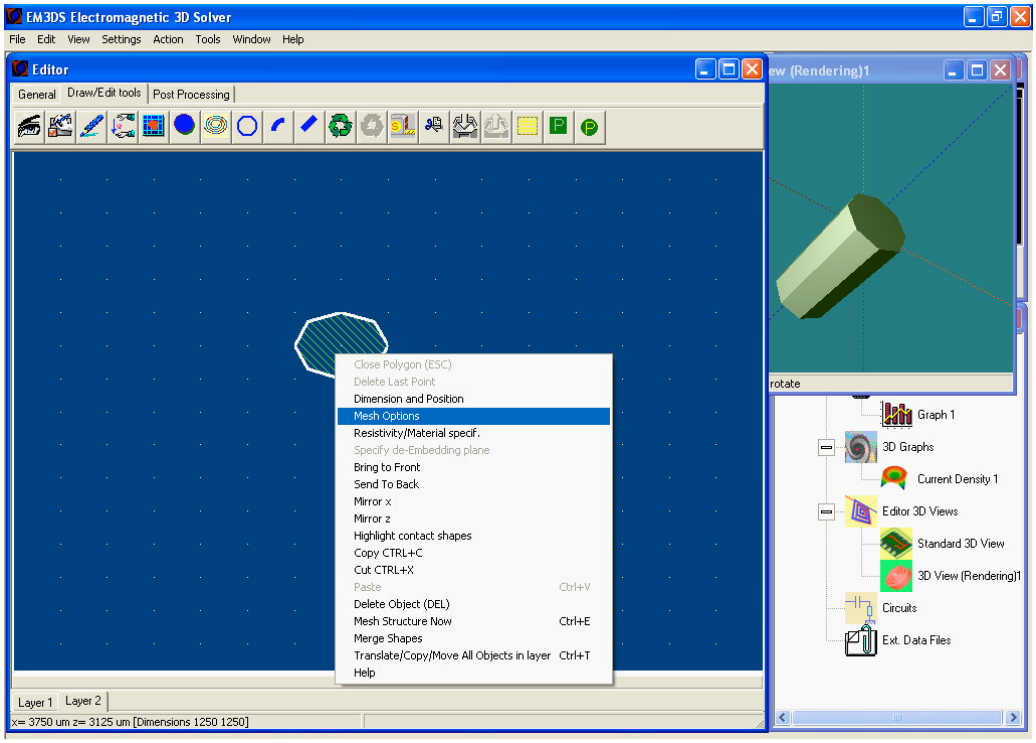


Fig.20: Selecting the "Mesh Options" from the popup menu.

A denser mesh will provide a greater accuracy only if sharp spatial gradients of the currents actually do exists. If this is not the case, most probably denser mesh will only mean a waste of resources (time and memory).

Tips: press CTRL+E or select **Mesh Now** from the popup menu to see Mesh. From the **Action** menu you can also enable Auto-Mesh: in this case any click in the editor with enable refresh of mesh.

When placing a shape, depending on the kind of shape being placed (e.g. spiral, slanting line etc) a set of mesh options is selected as default. The set of default mesh options may be user defined by the menu **Settings/Preferences/Mesh**.

There are several general considerations for requiring a user judgement:

-In several cases the structure may display an "over-mesh", as the automatic mesh is selected adopting a "worst case" approach.
-In some other cases the wavelength may be much shorter than it could be argued by looking at the dielectric stack: this is the case of slow-wave structures like Metal-Insulating-Semiconductor (MIS) structures or, more generally, whenever losses are involved. In this case it is likely your structure require a mesh denser than automatically settled.
-Due to the lack of a grid, mesh lines may be generated that would require a large number of modes
-General polygons or slanting lines are where user insight is absolutely needed (see next paragraph).

FAQ 3 *I have drawn a very simple structure and, in spite of this, it seems to require a large computational time. Where I'm getting wrong?*

The most common error producing a large computational time is a wrong setting of the mesh options (see the previous paragraph). In the present version much work has been devoted so as to reduce this source of error. However the user should be careful as well. Mesh lines may be settled by selecting (click by mouse) a shape, right clicking so as to display the popup menu, and by selecting the **Mesh Options**. The time needed for a simulation rises very fast with the "**number of box modes**" (displayed by clicking over **Settings/Analysis and Subs. Settings** from the main menu). The suggested value for the number of modes is directly related to ratio between the enclosure dimensions and the minimum distance between two mesh lines in the same conductor.

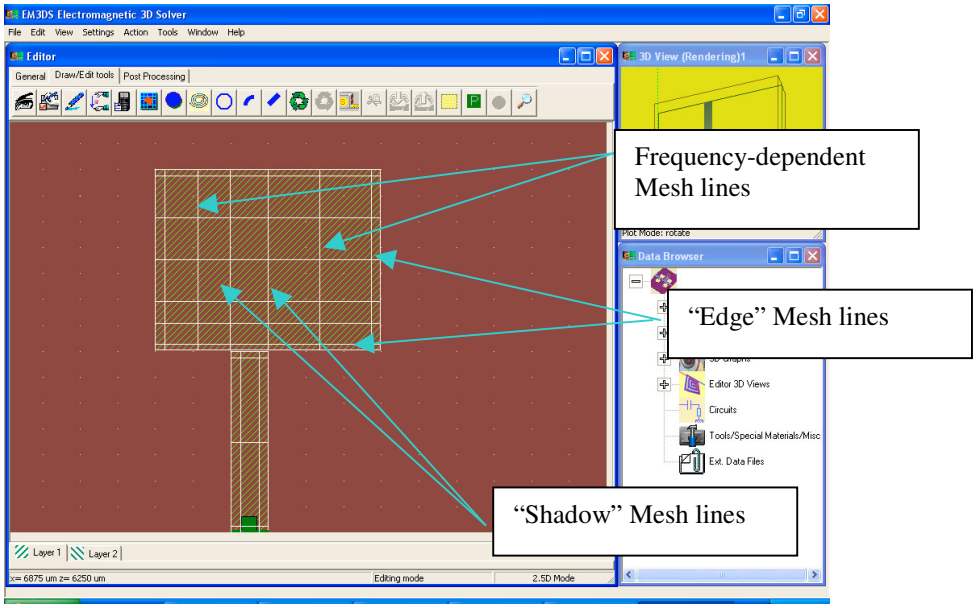


Fig.21:How mesh cells are created

Hence try to set up a more regular mesh by changing the shape Mesh Options. (see also the following *To Probe Further* section). Red lines highlight mesh lines involving a very large number of modes (more than 500!). In this case displacing mesh lines is practically necessary.

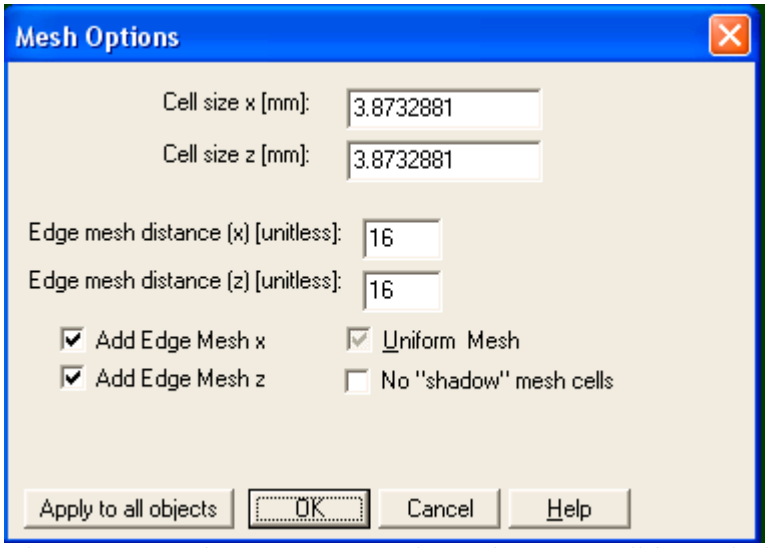


Fig.22: "Mesh Options" window; pressing OK the mesh options will be applied to all selected objects. Pressing Apply to all objects, the selected options will be applied to any object in the project.

FAQ 4 *No signal seems to be transmitted between two ports that are actually connected through a conducting path.*

There are several sources of this kind of error. Let us see some of the most common errors:

1- you have assigned a finite resistivity to your conductor path (e.g. 1.7e-8, copper), but the conductor is placed in a very thin layer (e.g. fraction of a micrometer). As the thickness of the conductor is the same of the embedding layer, the conductor resistance is very high, so not carrying (or at least attenuating) any signal. As a solution you have either to correctly setup the layer thickness (in this case keep in mind the recommendation in "to probe Further 2", p.6), or to select "**scale resistivity**" from the **resistivity/material specif.** Window, and set the desired thickness in the **scale to thickness** entry.

2- You have two shapes in your path that are visually in contact, but actually they are not. In order to verify if two shapes are actually in contact, simply select it, and from the popup menu choose **Highlight contact shapes**. Shapes actually in contact with the selected one are highlighted in a bright blue. Remind that what you see in the monitor is affected by round-off approximation made so as to

fit in the "pixel" grid. An additional possibility is to directly verify the position of the shapes in the path (from the popup menu, select **Dimension and Position**). You can also overlap shapes so as to be sure of the contact, even if usually the number of mesh cells rises slightly. In this case pay attention on where the additional mesh lines are placed due to overlap (see also the next *To Probe Further* section for more details).

3- You have shapes in contact, but in different layers. This situation may add a parasitic hidden capacitance to your structure (only thick conductors, only 3D working mode). Solution and details are indicated in "to probe Further 2", p.6

4- You have assigned the conductor resistivity to its embedding layer. When setting the substrate parameters, dielectric permittivity and conductivity refer to the dielectric layer and NOT to the conductors or the dielectric bricks defined **in** that layer. The properties of the latter are accessed by selecting the object by mouse and by selecting **Resistivity/Material** specifications from the popup menu. If thick conductors are defined in that layer, the dielectric layer properties specify the properties of the dielectric filling the volume *beside* the conductors

5- You have "shorted" your structure to the enclosure. The enclosure acts as ground. The only regions where a shape (not intended to be a ground) should touch the enclosing walls, are where edge ports are attached. In fact edge ports "cut" the conductor where they are defined.

Usually the above errors are easily identified by looking at the current density plot. For example, in case 4 you would see currents vanishing near the excitation port. In case 2 you would see a "cut" where two shapes were supposed to be in contact.

FAQ 5 *Sometimes I get an "Invalid mesh warning" and some shapes become red. Why?*

As a general rule, if a shape has no vertical (z) mesh line, it has no horizontal (x) current component defined, and vice-versa. This is not necessarily source of error, as in some cases you may want to neglect some current component, due to the expected solution, in order to reduce the computational effort (see next *To Probe Further section*). If this is the case you can disable warnings by selecting **Action/Disable Mesh Warnings** from the main menu. If you were importing some structure (e.g. by a GDSII file) you should pay attention on your mapping of layers. If you have incorrectly selected such a mapping, shapes belonging to different layers may be placed in the same layer, and hence completely overlap. In this case your warning is actually a source of computational error.

***To Probe Further**: what mesh cells are used for*

As said before, rectangular mesh cells are used as support for specific expansion functions: the unknown current distributions are approximated by means of a set of known functions (expansion functions) defined over rectangular domains (support). According to the reference system appearing in the 3D view window, the x-oriented current component (Jx) is described by means of piece-wise sinusoidal (PWS) function with respect to x, and piece-wise constant (PWC) function with respect to y and z. Dually the z-oriented current component (Jz) is described by means of piece-wise sinusoidal (PWS) function with respect to z, and piece-wise constant (PWC) function with respect to y and x. Vertical (y) currents are piecewise constant with respect to any direction: they are a sort of cubes.

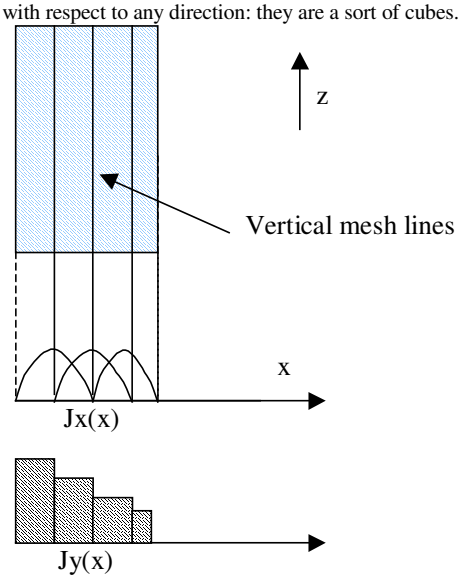


Fig.23: Jx and Jy as function of x: relationship with mesh cells

Figure *23* shows as an example the behavior of Jx and Jy in the x direction (Jz(x) is similar to Jy(x)): the total current for Jx is obtained by superposition of the single PWS(x). A dual rule apply to Jz and its dependence from the z co-ordinate.

Figure *23* is very interesting, as it display a fundamental characteristic: *in order to define Jx(x), at least one intermediate vertical meshing line should be defined. In fact a PWS is defined so as to have its maximum in correspondence to a vertical mesh line.* Hence if no vertical mesh lines were defined for the displayed conductor (e.g. because **add edge mesh x** is not selected and, due to the low operation frequency and the whole structure, no shadow mesh lines, nor frequency-dependent mesh cells do exist), no Jx component would be defined over that conductor. In this case EM3DS would issue an "Invalid Mesh Warning", highlighting in red the conductor were no Jx is defined. The same would happen for the dual case of Jz. This circumstance does not necessarily indicate an error: sometimes one current component may be reasonably neglected, due to the particular sizes of the object. For example a narrow line, oriented along z, will probably have a small x component of its current, so the user can fruitfully disable the mesh of x border and suppress Jx. The same happens sometimes for vias: many em (2.5D) simulators model them just as vertical (Jy) currents, as the current flows mainly in the y-direction (at least if rather thin conductors are connected to the via). The Jy current is sufficient to connect two conductors in two layers. This is the case shown in figure 24.

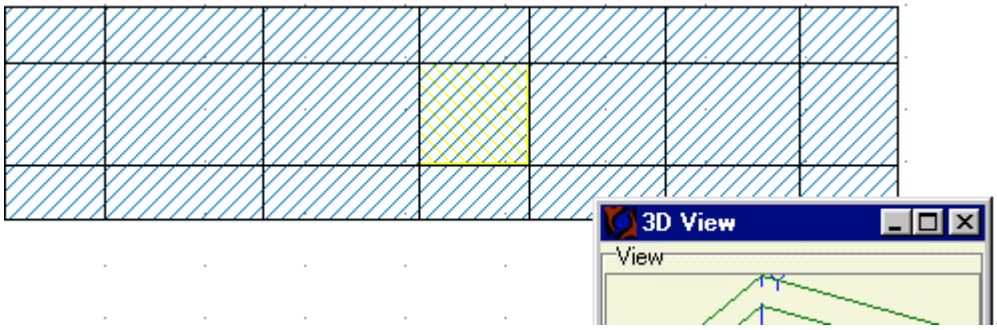


Fig.24: By disabling "Add Edge Mesh x" and "Add Edge Mesh z" options of the via conductor, no mesh lines are displayed in the via and the program issues an "Invalid mesh warning". In fact in this condition no current in the x direction, neither in the z-direction, is defined. However if the via is "small" the only vertical (y) current is sufficient in order to correctly simulate the structure, reducing both the number of needed modes and the number of unknowns (dimension of the system)

The computational load is reduced for two reasons. On the one hand there is a reduction of the number of current elements to be calculated (size of the system) and on the other hand mesh lines may be more spaced so requiring a smaller number of box modes when building the needed Green's function (MoM matrix).
If you are voluntarily suppressing some current components to reduce the computational load, you can avoid that the warning window appears during editing by selecting **Action/Disable Mesh Warnings.**
Be careful if the warning is issued after importing some DXF or GDS file: positioning entities in wrong layers may produce *completely overlapping object that the mesh algorithm is unable to handle.*
According to figure *23* it can be understood that, when two shapes are in contact (adjacent along x in this example) Jx should not vanish at the extremes where the two shapes are in contact. In fact vanishing of Jx would brake the current continuity and, as result, no or limited signal would be transmitted by the structure in this direction. Up to ver. 4.2 of EM3DS it was important to ensure this continuity by directly overlapping items in contact, and EM3DS automatically resized contact shapes so as to guarantee overlapping. In the present version this is no longer necessary, as the mesh algorithm will automatically recognize this situation and generate the right expansion functions ensuring continuity. This simplifies greatly drawing structures, especially structures involving general polygons. However it is ever possible to directly overlap contact shapes, even if this usually results to a slightly higher number of unknowns. Even the "automatic resize" procedure may be enabled by selecting **Settings/preferences/Editor** from the main menu and by selecting the check box **Auto resize rect. for overlap**. However it is very important that a project be developed "ab initio" either with the "auto resize" enabled or not (the mesh algorithm, mostly for backward compatibility, produces a different mesh!). This setting is embedded in the EMS file (command //AUTORESIZE=). EM3DS may be unable to reopen results for a structure developed partly with auto-resize enabled and partly not.

Note: whenever Jx or Jz are not defined over one shape, shapes adjacent (in contact) to the conductor/dielectric for which Jx/Jz are not defined should overlap such a conductor/dielectric. In fact *only in this case* automatic mesh algorithm is unable to set up the right expansion functions. Continuity violation has very evident effects (nearly no signal transmitted [e.g. S12 nearly zero], a null over the contact plane in the graphic of currents is apparent etc.) so that you can easily locate and fix the problem.

About the previous note: it is easy to explain why the problem may occur. Let us consider the following figure.

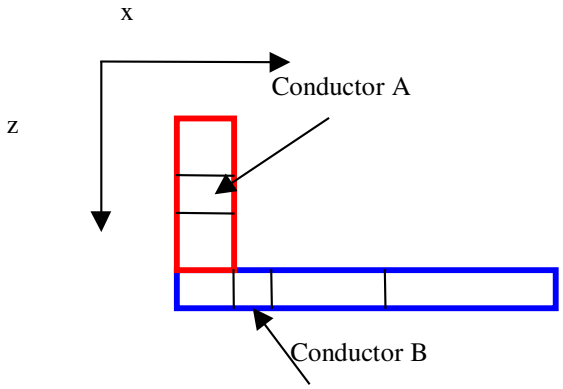


Fig.25: Two conductors are in contact (no overlap): in spite of this results seem to indicate no contact. This happens as conductor B has only Jx while conductor A has only Jz.

Let us suppose that conductor B has no z-edge (and no other z mesh line) and dually for conductor A. This may happen e.g. because either you have not checked **Add Edge Mesh z** in the **Mesh Options** of conductor B, or its **Distance z** value is too small (so specifying a distance that can not fit into the shape. Up to a certain extent the mesh algorithm inserts any way the mesh line in the middle, but if the conductor is very narrow with respect to the specified border ratio, the border mesh line is suppressed!). According to the previous explanation, conductor B has no Jz, while conductor A has no Jx. Generally this fact would be acceptable as conductor B is "narrow" in z, while A is narrow in x. However this way no continuity is enforced and the conductors behave as being close but not in contact: the above mesh is unable to model current "turning" at the edge.
A solution is to force B to have its z-edge mesh line and A to have its x-edge mesh line, for example in the middle (to this aim be sure that " **Add Edge Mesh** " options are checked and raise the value of the edge mesh distance at least up to the ratio between the z box dimension and the width of conductor B; dual for A).
However it is apparent as, in this case, the number of required modes to build the solution is doubled in both directions (as the minimum distance between close mesh lines, including the rectangle sides, is reduced to one half). A more economic solution is simply to completely overlap conductor A to conductor B, as this way there is at least one cell having contemporarily both Jx and Jz so as to guarantee the correct boundary condition

Figure *26* displays one possible case where optimization of the mesh by the user will save a large amount of time. One "Shadow" mesh line in the middle conductor due to the conductor on the left hand side is very close to the one due to the conductor on the right. Most probably displacing vertically one of the two conductors up to merge the shadow mesh lines will not affect the accuracy (as, in order to be a problem, the two positions have to be very close), while substantially reducing the "number of required modes" in the z direction. Remind that this number is directly proportional to the ratio between the enclosure dimension along z and the vertical distance between the closer horizontal mesh lines.
As it is apparent, the need of the user insight is mainly due to one of the main features of EM3DS: the designer has no grid constraining placement of objects. Hence, when the user becomes familiar with this sort of tricks, he may take strong advantage of the complete freedom that EM3DS offers.

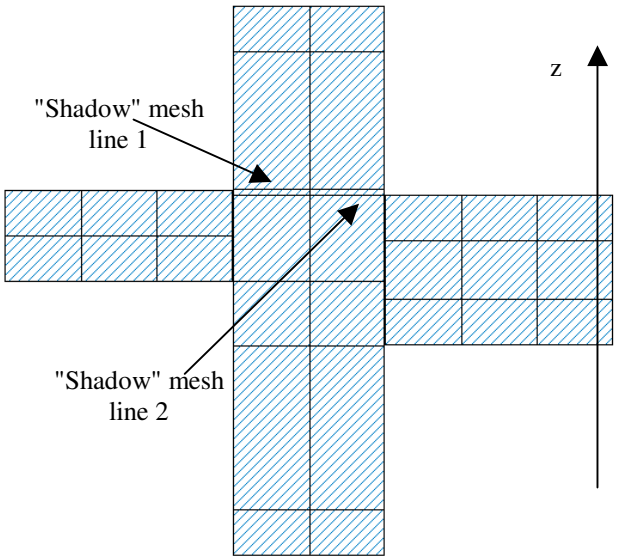


Fig.26: One "Shadow" mesh line in the middle conductor due to the conductor on the left hand side is very close to the one due to the conductor on the right. Most probably displacing vertically one of the two conductors up to merge the shadow mesh lines will not affect the accuracy, while substantially reducing the "number of required modes" in the z direction. Remind that this number is directly proportional to the ratio between the enclosure dimension along z and the distance between the closer horizontal mesh lines.

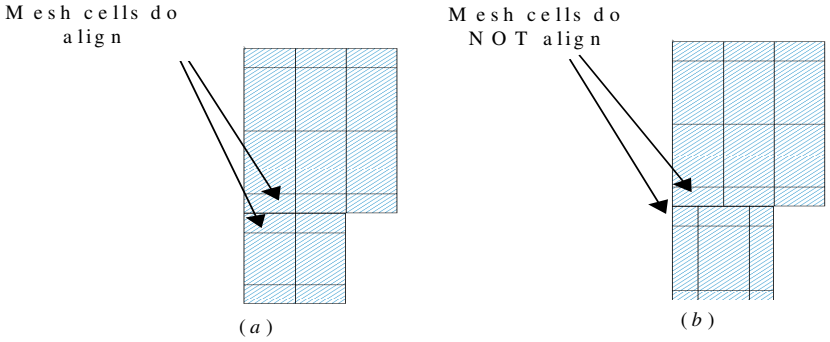


Fig.27: When possible, setting (a) should be preferred. To this aim shapes adjacent along z (as in this case) should have the same "Edge Mesh Distance x" parameter. Dual for adjacency along x.

One final note, just a matter of numerical convenience: greater accuracy is obtained when mesh cells align. This is not a "must" and may be not possible to respect this rule in a complex structure, but when possible it should be respected. As shown in figure *27*, case (*a*) where mesh cells between adjacent shapes align should be preferred.

Generally, in order to respect this rule you have only to set the "Edge Mesh Distance x" in the mesh options to the same value for shapes adjacent along z, and dually for adjacency along x.

X Working with non-Manhattan polygons

When clicking over **Action/Add Polygonal Cond.** a dialog box asks if the polygon has to drawn by mouse or by specifying vertices. In the first case every click identifies a vertex, *while pressing "ESC" or by right clicking and selecting* **Close Polygon** *or by clicking a second time on the* **Polygon button**, the first and the last vertices are joined forming a complete polygon.

In the second case a dialog box let you specify the vertex co-ordinates, adding, removing and modifying the existing co-ordinates. The position refers to the Top left corner, but the position of the bottom right corner is automatically calculated and shown.

When editing/modifying any general polygon, a red circle highlights the point being edited. You can also select in which MKS or imperial unit information is displayed.

The **Delete Points Closer than** panel allows to remove points closer than the specified distance. This may be useful in order to reduce the complexity of a polygon used to approximate, e.g., a curve. To this aim, once settled the distance, press the **Delete Pts** button. Changes will take effect only after pressing **Apply** or **Ok**.

The **Round Dim** button allows rounding of the vertex positions "all at once". **Displace** allows moving the whole object of a given amount. In any case changes are only effective when pressing **Apply** or **Ok**.

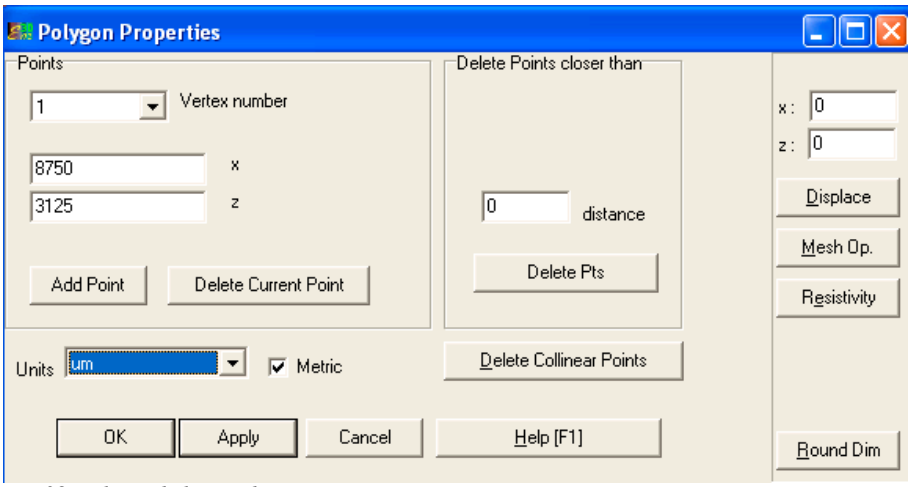


Fig.28: Polygon dialog window

Note that in any co-ordinate you can enter **parameters**, or expressions involving parameters (not the frequency!). This allows you to create a parametric geometry.

Manhattan polygons are those polygons that you can directly draw by combining rectangles: you can use the **Add Polygon** button for both Manhattan and non-Manhattan Polygons. However shapes drawn this way are subjected to a few restrictions: ports may only be added to rectangular conductors, so if you have to add a port, you have to add to your structure a rectangular shape. Polygons that are actually rectangles are automatically converted in rectangles, so that ports may be added.

In the present version, a number or rectangular domains model slanting lines, namely by applying a staircase approximation. It is possible that the number of intervals added by the automated mesh be not the best choice, so the user should try by changing the size of the mesh cells in the **Mesh Options** dialog box, in order to get a better trade-off between accuracy and simulation time (see previous section). The experienced user gets very accurate results with a low computational effort. See figures.

Raw (ineffective) mesh Better mesh

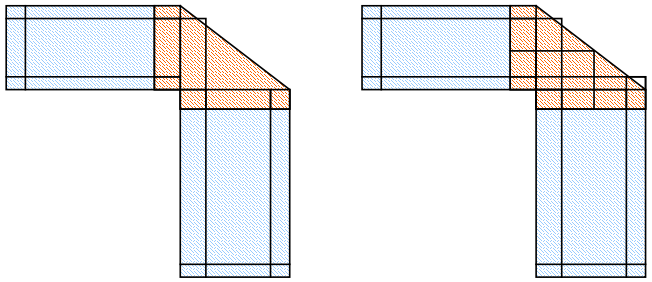


Fig.29: Mesh polygons: in both cases mesh is coarse, but the right example would provide sound results.

Slanting narrow lines are probably where the currently used algorithm finds the worse difficulties (currently the algorithm is being updated in order to automatically perform the steps indicated in the following).

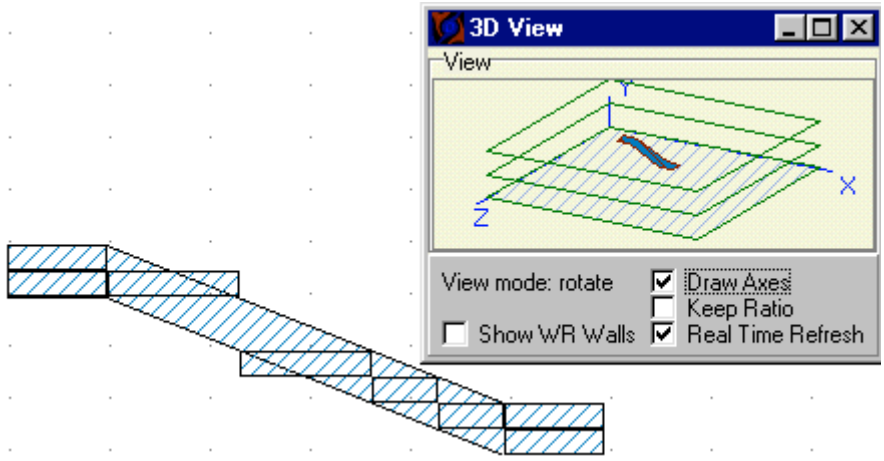


Fig.30: Slanting line where the automatic mesh algorithm is unable to provide a useful mesh: continuity over the structure is violated.

If cells are not surrounded by other cells, conductor may be not "connected", as shown in fig.30. In fact there is not continuous current flow along the line. By suitably changing the mesh cell size of the polygon one obtains what shown in figure 31.

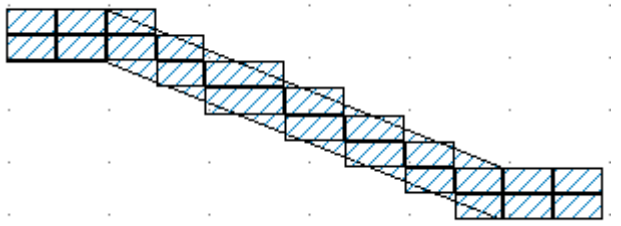


Fig.31: Mesh polygons: By changing the mesh cell size, one obtains an acceptable mesh.

Usually slanting lines are better meshed by using the “uniform mesh” option (default for **add slanting line**).

By recalling what we have said in the previous section (*To probe Further*: a horizontal current is only defined over a rectangle if at least one horizontal division [vertical mesh line] of the rectangle appears. Dual reasoning for vertical currents) it is straightforward understanding as this is the simpler mesh accounting for the correct continuity of the currents. The schematic path of the current is displayed in fig. *32*.

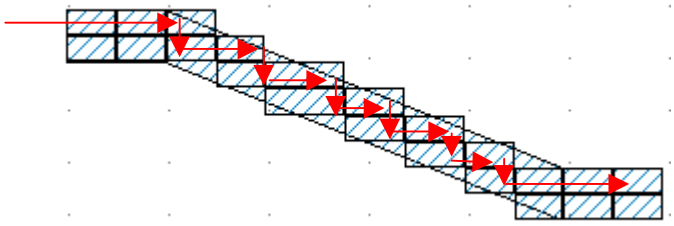


Fig.32: Schematic of the current path

***To Probe Further** : How polygons are meshed*

Meshing rule for polygons are slightly different (actually only a "super-set") than those described in the previous section. Mesh lines, if **uniform mesh** is not selected, are in fact associated to each polygon vertex. This explains why sometimes, reducing the number of vertices (e.g. convert a circle into a polygon with a reasonable number of vertices) may be necessary. Also, consider that each vertex touching an other shape, produces a "shadow mesh line" (see previous section). Additionally each vertex belonging to polygons in other layers will also produce shadow mesh. However the latter can be avoided if in the mesh options the checkbox **no shadow mesh cell** is selected.

The frequency-dependent mesh cells are in this case also settled by the EM3DS by considering the possibility of slanting lines (hence by considering the relative position of vertices). However, as we have seen, this is the weak point of the currently implemented mesh algorithm, and curved or slanting lines should be checked by the user (more work is being devoted to this problem). Playing with the Mesh Options would fix several troubles, as it has to be ensured that mesh cells fill the whole shape. "edge" assumes the more general meaning of "line between vertices" in general polygons, so that the rectangular shapes are just a particular case of polygon.

XI Circles

By clicking over **Action/Add Circular Cond.** circles, approximated by a polygon, are added. EM3DS displays the window reported below.

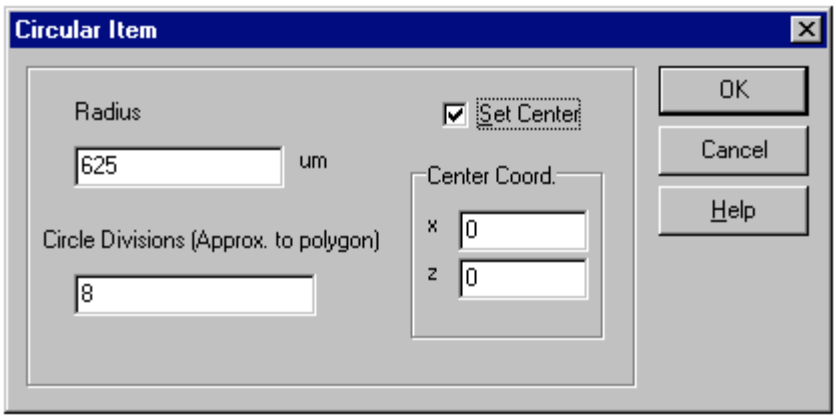


Fig.33: The "Add circle" window

If **Set Center** is selected, you have to enter in the **Center Coord** panel the center coordinates. Otherwise you are enabled to place the circle by mouse. **Set Center** should be selected if you want to create a parametric circle; otherwise parameters are converted to their assigned values. Actually what is inserted is a polygon, that, as discussed in the previous section, is modeled by staircase approximation. **Circle Divisions** let you specify how many vertices are used in order to approximate the circle. While of course more vertices enable to a better approximation of the circle, the mesh density raise rapidly. Hence consider that generally a low order polygon provides fast and reliable results, and that there are a few cases where the number of vertices is worth to be raised (e.g. circular dielectric resonators)

XII Rings

By clicking over **Action/Add Ring Cond.** rings, approximated by a polygon, are added. EM3DS displays the window reported below.

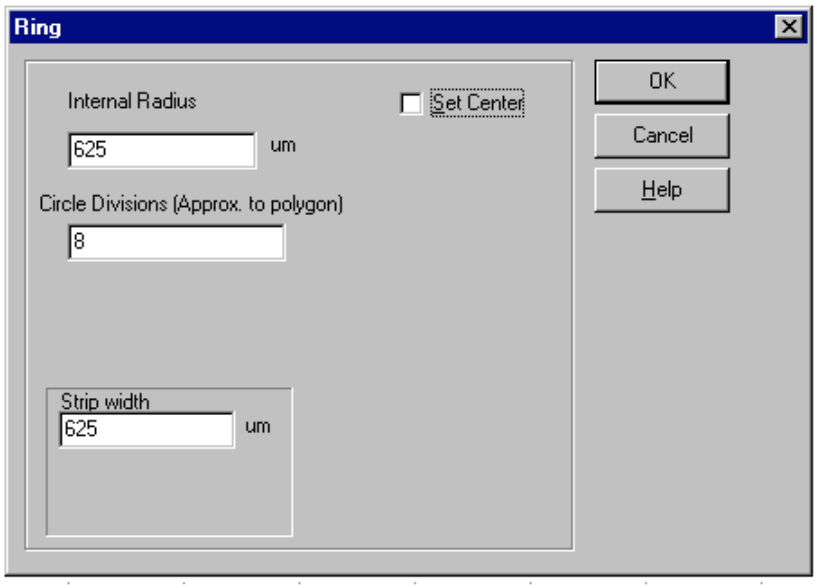


Fig.34: The "Add Ring" window

It works as for circular conductors (see previous section). Internal radius and strip width have to be specified.

XIII Circular spiral

By clicking over **Action/Add Circular Spiral Cond.** the user can add spirals.

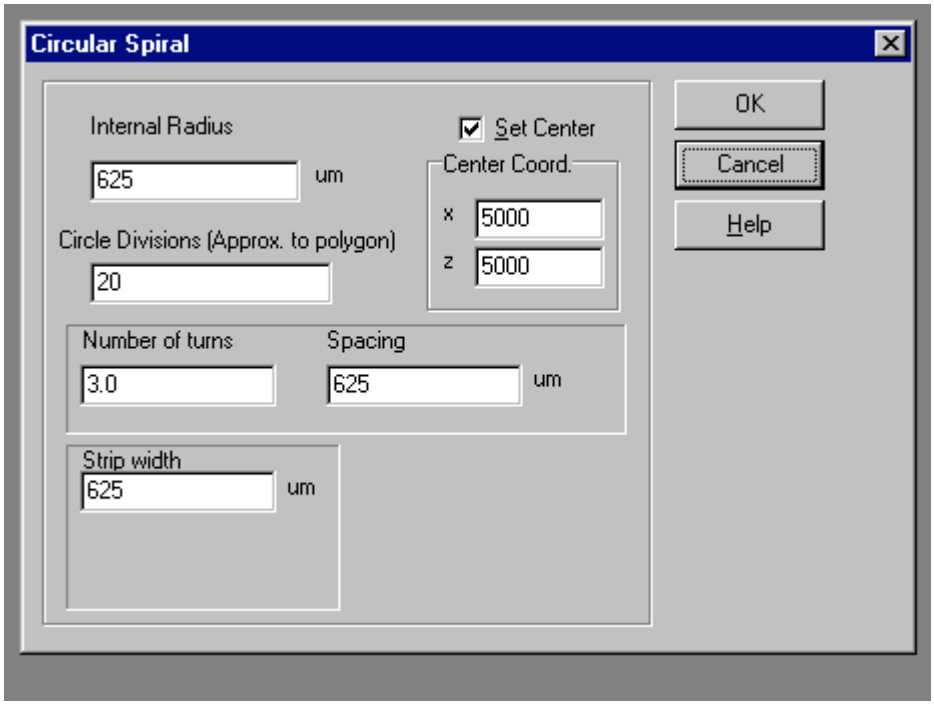


Fig.35: The "Add Spiral" window

Several items of the dialog box are common to those appearing when adding a circular conductor. Additionally the user has to select the spacing, the strip width and the number of turns, that is not necessarily integer.

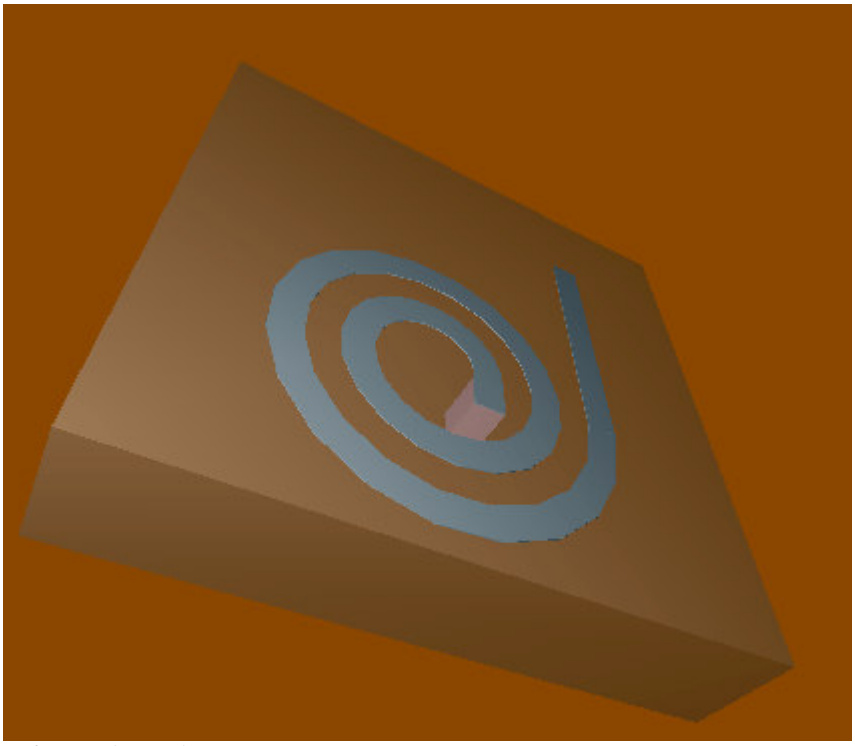


Fig.36:Round spiral in EM3DS

XIV Curve

By clicking over **Action/Add Curved Cond.** the user can add a curvilinear bend to a conductor/dielectric. Several items of the dialog box are common to those appearing when adding a circular conductor. Additionally the user has to select the start and stop angles of the curve with respect to the x axis. Mirroring, available after placing the item, simplify drawing arbitrary structures

XV Slanting line

By clicking over **Action/Add Slanting line** the user can add a rotated line; the amount of rotation is defined by the angle with respect to the x-axis. Dimensions and position are specified as in standard rectangular shapes.

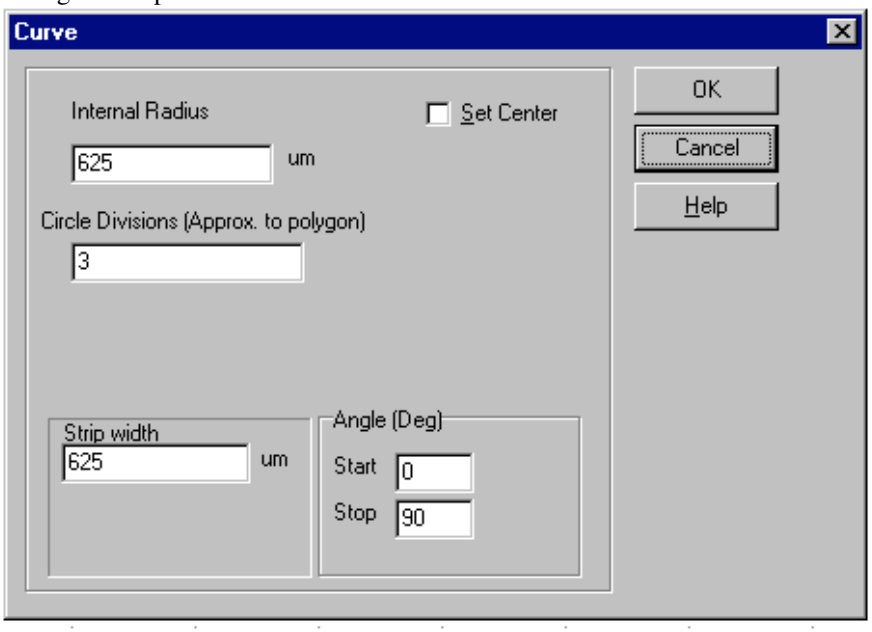


Fig.37: The "Add Curve" window

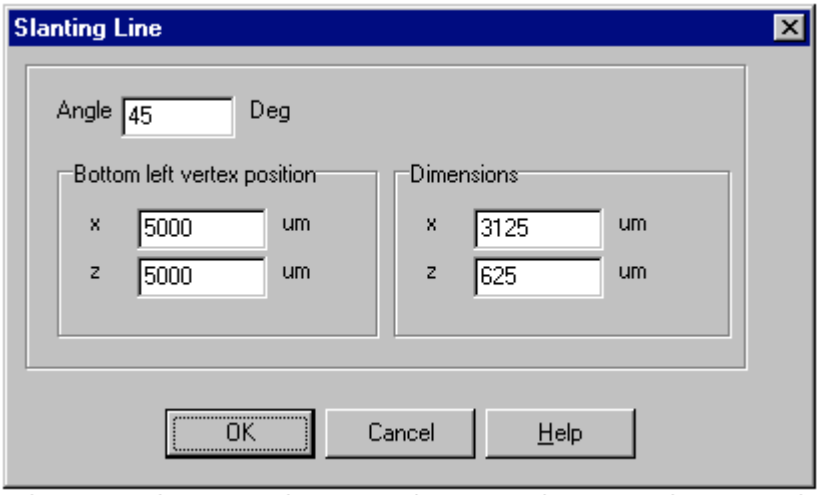


Fig.38: The "Add Slanting line" window

XVI Merging shapes

As we have seen previously, where two shapes intersect, intersection mesh lines appear. Sometimes this may produce overmesh in the resulting structure. By merging the two shapes and by using a uniform mesh, the user may obtain simple and effective mesh. Merging is performed by selecting the **Merge** command from the popup menu or from the **Action** Menu: if an object is selected, the user is required to select one more object to be merged. Rectangles including ports cannot be merged. Note that the algorithm may work not properly if vertices of a polygon are exactly over sides of the other, In such a case it is sufficient to displace by a very small amount one of the two shapes so as to guarantee overlap. If Merge fails and resulting polygons are corrupted, use the "undo" button. Even polygons including holes may be merged (e.g. rings), but overlap should not be over the line cutting the ring (every polygon with holes has a cutting line, has polygons are defined by connecting first and last vertex).

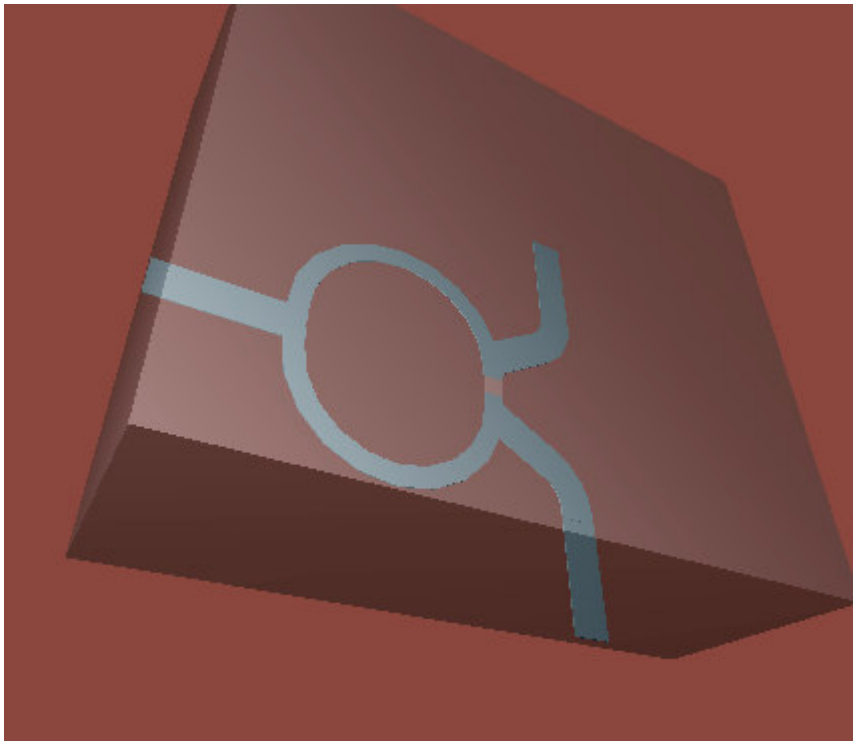


Fig.39: Example of circular Wilkinson power divider in EM3DS: the structure has been obtained by merging curves and slanting lines.

XVII A simple step-by-step example: a MEMS Switch

There are several way to learn about using EM3DS: the most effective are the animated tutorials. In the following we will describe step by step how to enter a MEMS switch.

Start EM3DS by clicking over EM3DS 12.exe icon. By moving the mouse over the upper button toolbar, hints are displayed about the button functions. In the following we will assume these hints to be the name of the buttons

1) The first step is selecting the enclosure dimensions. Your structure is enclosed in a metallic box, the walls being the four sides (the upper and the lower planes may be actually open) of the windows where you draw the structure (work area). To this aim select the "**General**" panel, and the button "**Box size**", or by the main menu "**Settings**", "**Box Dimensions**".

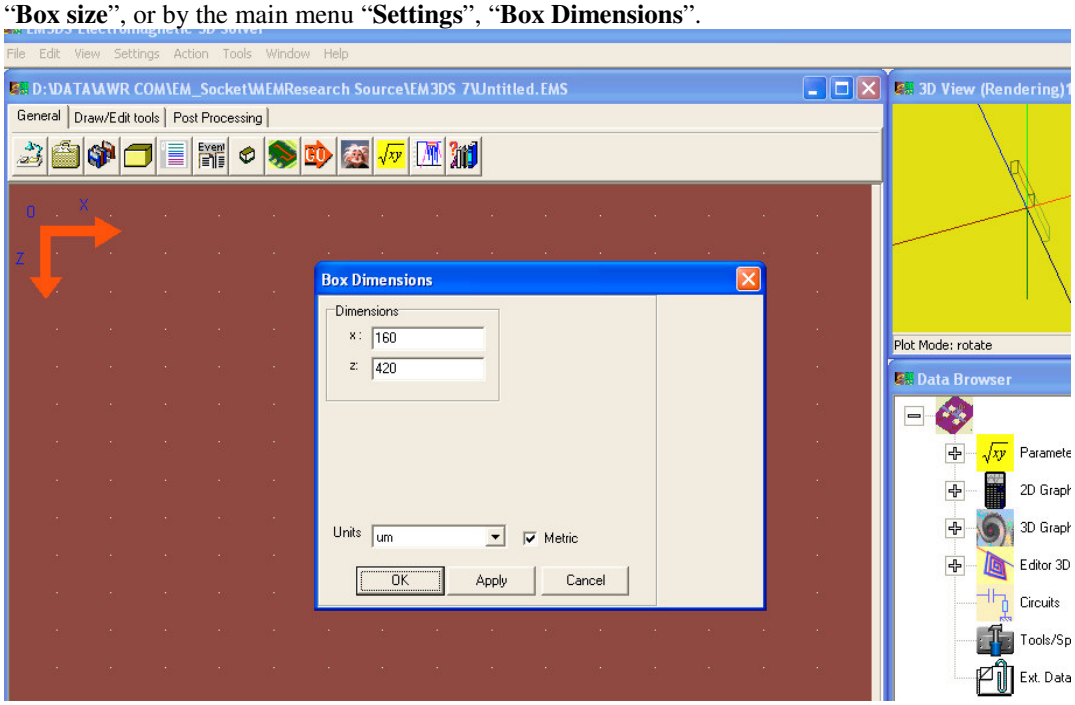


Fig.40: Size of the box

Check the "**Metric**" checkbox, Select units as μm, enter the horizontal (x) dimension 160, the vertical (z) 420. Note that in any point you can enter expressions instead of numbers (e.g. in x 280-160), where needed. If this was an antenna, and we were interested to the antenna plots, we should have selected **Add Sense Layer** from the main menu **Action**, but this is not the case. You can find more information in the Antenna section.

2) While EM3DS does not require the design to fit into a grid, a grid is available in order to simplify entering the structure. Hence select the "**Drawing/Edit tools**" panel and click over the button "**Fit in Grid**"

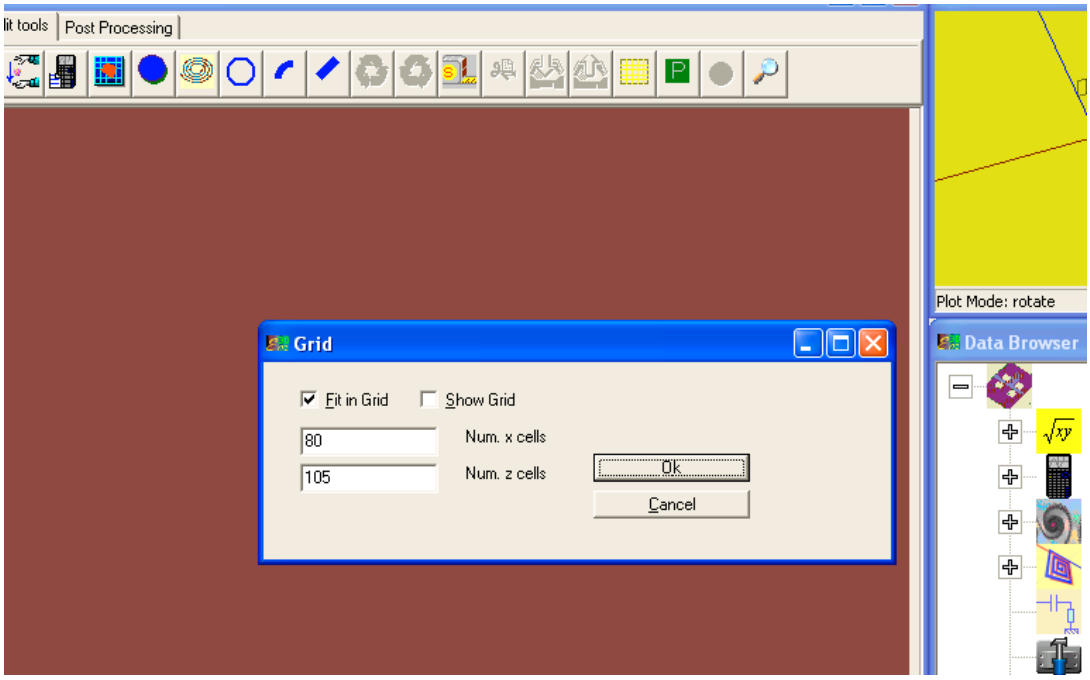


Fig.41: Grid

Enter e.g. 80 cells for x and 105 for z; this will only affect editing and you can change these parameters as you want during the editing process.

3) EM3DS supports two working modes: the full 3D mode and the 2.5D mode (default). In the remainder of the example we will first consider the full 3D mode, and then see what has to be modified in order to correctly run the example in 2.5D mode. The 3D mode assumes that each conductor has its thickness, which is the same of the layer in which the conductor is placed. Hence there is no distinction between normal "planar" conductors and via conductors, its role only depending on the geometry. It is somehow a "context definition". In the 3D mode volume currents are used to define the conductors, the dielectric bricks and the active regions. On the other hand in 2.5D mode, the distinction between two class of conductors is assumed. The first is the one of normal, thin planar conductors and the second is that of the via thick conductors. Normal thin conductors may only be placed in layers having odd index that will be so assumed to be infinitely thin for all purposes. Actually the user will define the thickness also for those layers: such a thickness will only be used for the correct evaluation of the conductor losses. Via conductors and substrates, on the other hand, may only be place in layers having even index. These conductors are defined by surface currents (Jx and Jz) and via conductors by vertical volume currents (Jy). We will see at the end of the section what to do in order to create the 2.5D example. The 2.5D mode is much faster, and easier to use, especially at relatively low frequencies.

In order to set the 3D mode just go in the menu "**Settings**" and deselect the item "**2.5D mode**". This item will be also accessible from the Project Data Window (see next step). Immediately any 3D view is updated according to the new selection (any shape has the thickness of its host layer). The lower status bar, on the right side, indicates that the current mode is "**3D mode**".

4) The next step is to specify the number of layers and the materials composing your switch. Select the panel tool "General" and then click over the **button "Analysis and Substrate Settings**"; alternatively click over the main menu "**Settings**" and then "**Analysis and Substrate Settings**". The "**Substrate Information**" window appears.

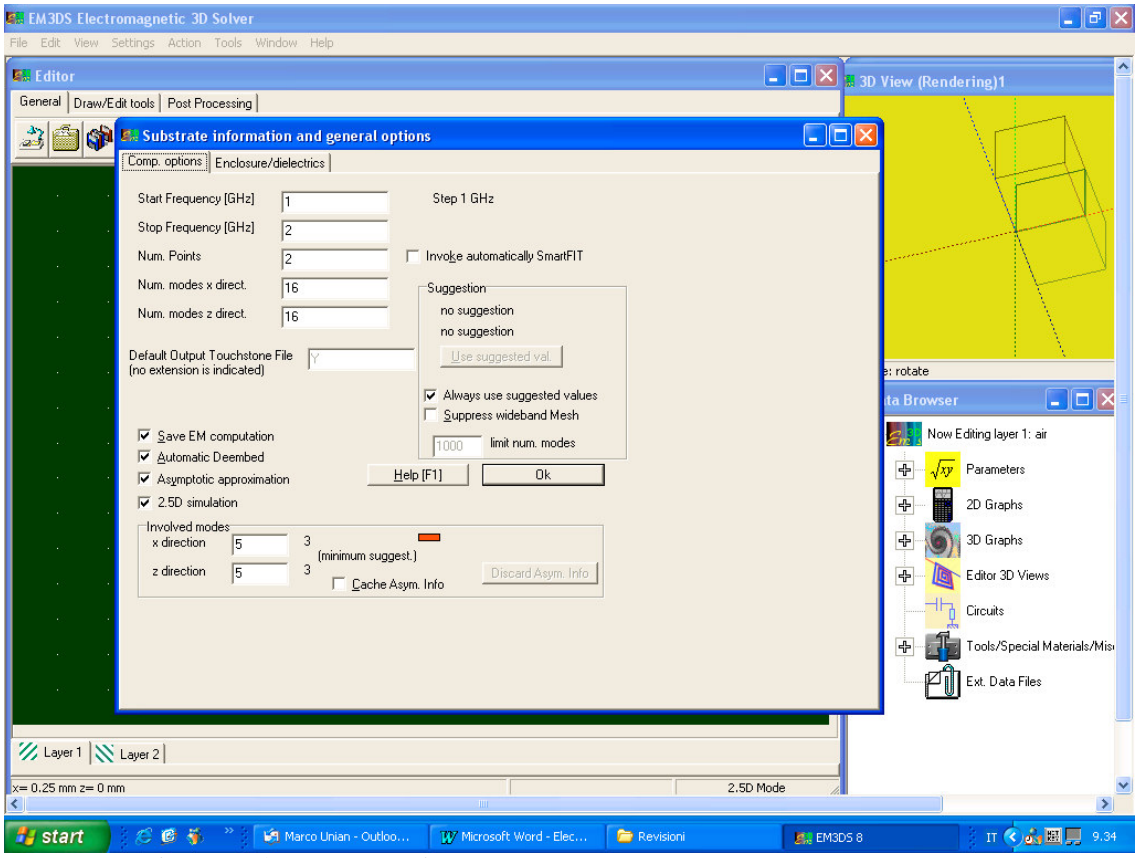


Fig.42: Substrate Information Window

Here you can also

Enter as start frequency 2 GHz, stop frequency 40 GHz and num. Points 20: at the upper right corner a small message informs you that the selected frequency step is 2 GHz; there are other parameters to be selected in this window, but you'll need to change those parameters after editing the structure. If you want, you can also select **Invoke automatically SmartFIT**: in this case EM3DS will use SmartFIT, namely an adaptive calculation of the frequency points; the final number of points is the one selected in SmartFIT (so that, in such a case, the Num. Points info is not used).

In the same window select the panel "**Enclosure/Dielectrics**."

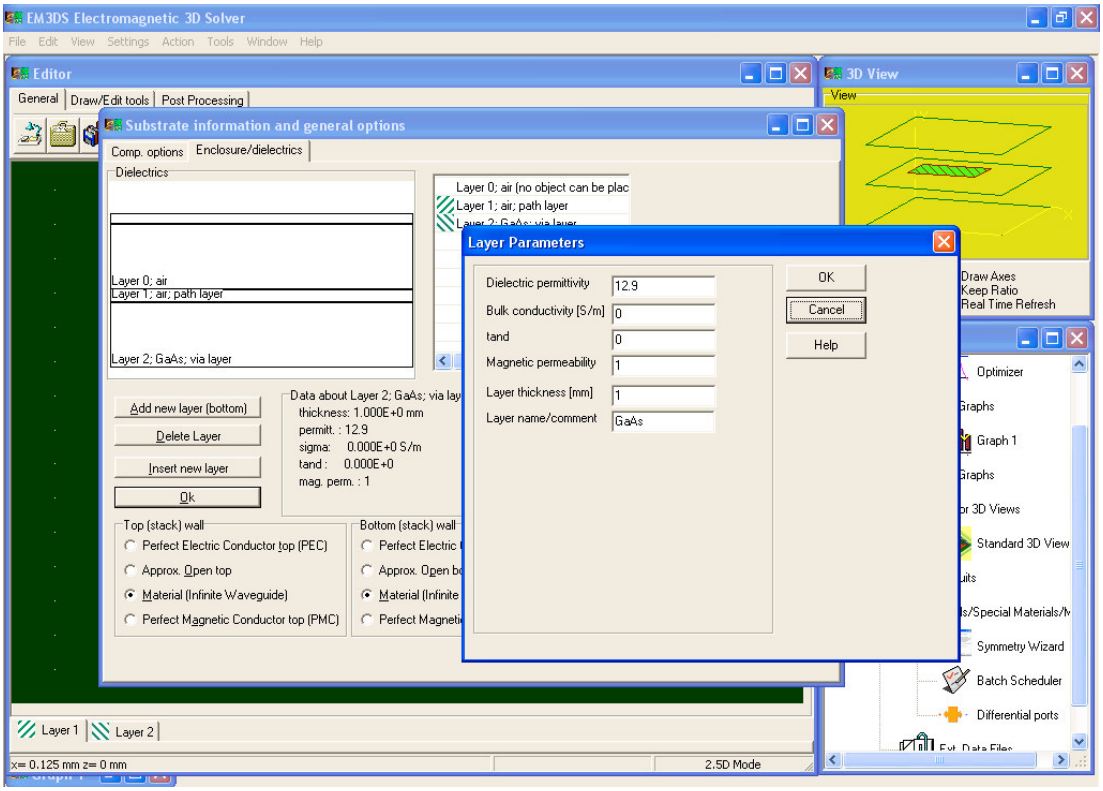


Fig.43: Dielectric stack

Select check boxes "**Material (infinite Waveguide)**" for both top and lower walls. This way you specify that top and bottom covers are removed and that the lateral walls are infinitely long in the y direction.

By moving the mouse over the "**dielectrics**" window, information about the composition of layers is displayed. By clicking over a section you can edit layer parameters. Alternatively you can click in one of the items in the list beside. Later, you'll also have access to any active layer directly from the Editor's popup menu.

Every shape that will be entered later will have the thickness of its embedding layer. Layer 1 is by default very thin, and in this case we will leave it very thin, as layer 1 contains the switch bridge, whose thickness is not very important in the current structure. In the **3D mode**, owing to the need of correctly accounting for the y directed current gradients, if objects in a layer are connected to objects in different layers, sometimes it is necessary to require a "slicing" of the current layer (the number of "slices" is indicated by Ny): as described in the manual this raises quickly the computational effort; hence if thickness of a set of objects is believed to have a limited effect, the best choice is to neglect it. If more in general thickness is not believed to be important for any of the elements of the geometry but vias, the best choice is to switch to the **2.5D mode.**

Click now over layer 2 and edit the layer parameters by setting the relative permittivity to 1 and the layer thickness to 1.35 µm. Layer 2 contains the bridge anchors. Hence add one further layer after layer 2, by selecting "**add New Layer (Bottom)**" button. Specify the relative permittivity to be 7.6 and the thickness 0.15 µm. It is the layer that will model the Silicon Nitride passivation. By this choice the silicon nitride is assumed to cover and embed the entire lower electrode; this is not necessary in EM3DS, as you can also enter dielectric discontinuities, but our choice simplifies the structure drawing. Whenever you want to specify a silicon nitride brick to exist only over a portion of the structure, leave the relative permittivity of the current layer to 1, and when entering conductors, you will have to enter a dielectric brick having 7.6 relative permittivity in layer 3.

Add one more layer having relative permittivity 7.6 and the thickness 0.8 µm: this is the lower electrode layer and we assuming that it will be embedded in silicon nitride, namely spaces besides conductors are filled by silicon nitride.

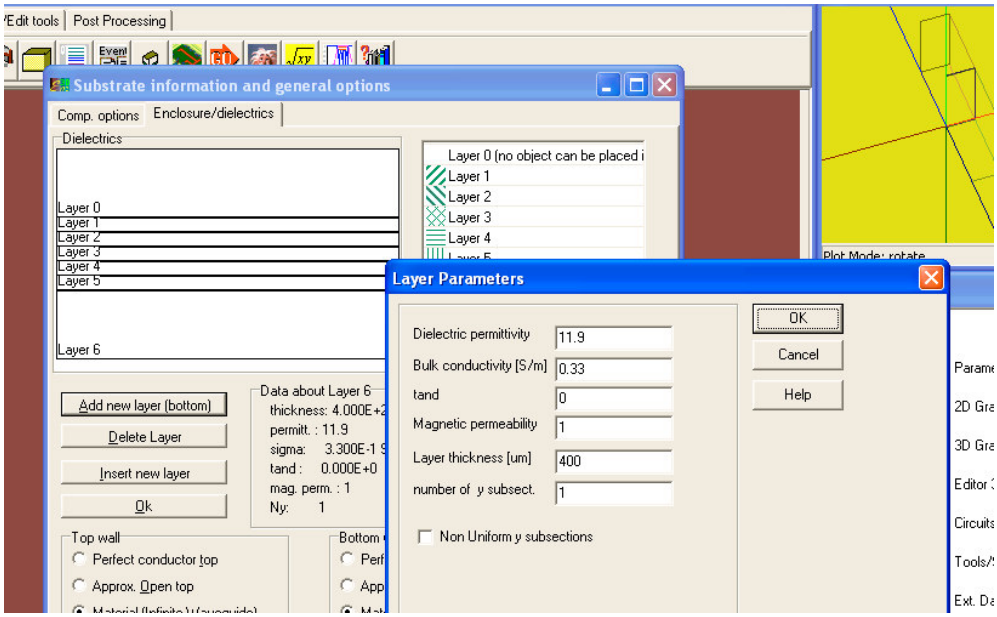


Fig.44: Layer setting

Add another layer having relative permittivity 3.9 and the thickness 0.4 μm: this is the silicon oxide layer added to avoid contact with the silicon buffer. Hence add the last layer: permittivity 11.9, bulk conductivity 0.033 S/m and thickness 400 μm: this is the silicon buffer.

The dielectric stack is now completely specified. You can close the project data window. You should have now 6 layers appearing (plus layer 0: layer 0 is over your structure; you cannot place objects in it, but it is the only one that can safely assume 0 as value for the thickness, which means it is removed).

4) Now we have to enter the structure, by specifying position and size of the conductors. Let us select layer 1 from the lower left panel. Hence select the "**Draw Edit tools**" panel, and select the **button "Draw Rectangular Conductor/Dielectric Brick**". Alternatively select the menu "**Action"/"Add conductor**". By the first click over the working area you set the upper left corner; release the mouse button and move the mouse up to click where you want the lower right corner. In the lower bar, information about the position of the mouse and the size of the rectangle being drawn is displayed. This is not very important as we are going to specify exactly the position and the size of the rectangle. By the way, you can click after over the conductor: this way the shape is selected and you can displace it by the keypad arrows. If "shift" if kept pressed you can also change the rectangle dimension. Moving the rectangle by "drag-and-drop" may also be enabled by selecting the button "move conductors" and then by dragging any of the available shapes.

Right click over your rectangle and select from the popup menu "Dimension and Position", or simply double click over the rectangle to access directly this window.

Actually, a very good idea is to parametrize circuits; by defining a set literal of parameters; and then using them to define size and positions (see Variables and Parameters). In the latter case, remember that any modification you make moving directly or resizing the shape, will delete reference to the related parameter. By the way, let's click on the "**parameters**" button (or directly in the XY icon appearing in the data browser) and define some variables:

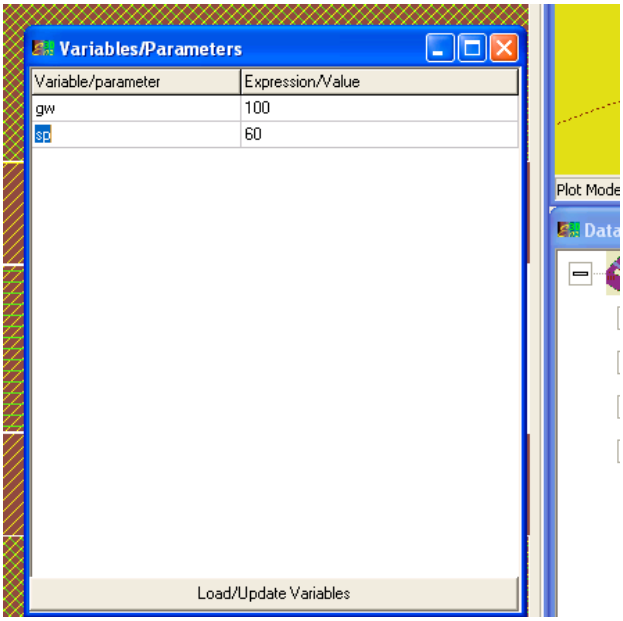


Fig.45:parameters: use the keyboard arrow to navigate in, and add rows for new parameters.

If parameters are used, you will later be able to use either the **Optimizer** or the **Tuner** (menu **Tools**). Just as example we have only defined 2 variables, gw (ground width), sp (line spacing), but consider to parametrize every geometrical property of your objects.

Now draw a rectangle and set in the panel 160 and gw µm, while the top left corner to (0,0); click over apply button. The structure is updated and the right bottom vertex is also calculated. Click over "Ok".

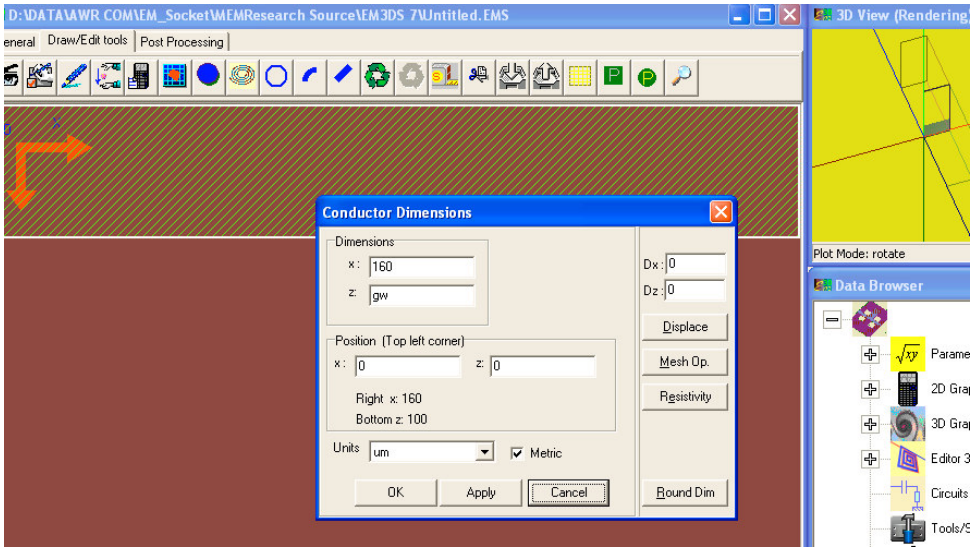


Fig.46: Parametric definition of the width

Note that you can reference to variables like "gw" not only when defining geometry, but also anywhere in the editor. If the specific property does not support parameters (e.g. the box size, in the plane direction, in the present version) the parameter is immediately resolved for, and only its current numeric value is used. Note also that parameters will be the mean to modify your structure via your own **script** (see the Script section for more information)

Coming back to our structure, the rectangle is placed in layer 1 and has thickness of layer 1, as you were in layer 1 when entering the shape. We have not yet specified the material composing such a shape. To this aim select from the popup menu available when right-clicking over the rectangle **resistivity-Material Specifications**, or simply "**Resistivity**" from the same dialog where you have entered the coordinates.

Set the resistivity to 2.5e-8 Ohm m (remember, this is resistivity, not conductivity, in contrast to the specifications of the embedding dielectrics!). However we have deliberately left layer 1 and any of its object to have a very reduced thickness (fraction of microns). Hence any conductors having non-

zero resistivity, would have a very high total resistance! In order to avoid this problem the resistivity has to be scaled to the actual (wanted) conductor thickness, namely 2 microns: hence select “scale resistivity” and enter into the “scale to thickness” 2 microns. We can also select “Force skin effect”: as we are considering very thin conductors in this layer, no skin effect would appear; by selecting “Force skin effect” we require an approximate accounting for such an effect. The above resistivity scaling will be needed only by objects in layer 1.

We need to add other two rectangles: we can simply copy and paste the previous rectangle so as to avoid to specify every time its parameters. To this aim simply click over the rectangle, select “copy” (or CTRL+C) then “paste” (or CTRL+V) and place where you want the rectangle. You will note that, even if you displace shapes either by the keyboard arrows or by mouse, your reference to the variable is not lost: this is only because the variable is used for the size and not the position. Reference to variable is only lost if your editing conflicts with requirement imposed by your parameter. Edit its dimensions and set them to 80 (x) and 220 (z) while selecting as top corner 40 in x and 100 in z. Repeat those steps once more, editing one more rectangle with 160(x) and gw(z) microns as dimension and 0(x) 320(z) as position. This way you have drawn the bridge with its upper support.

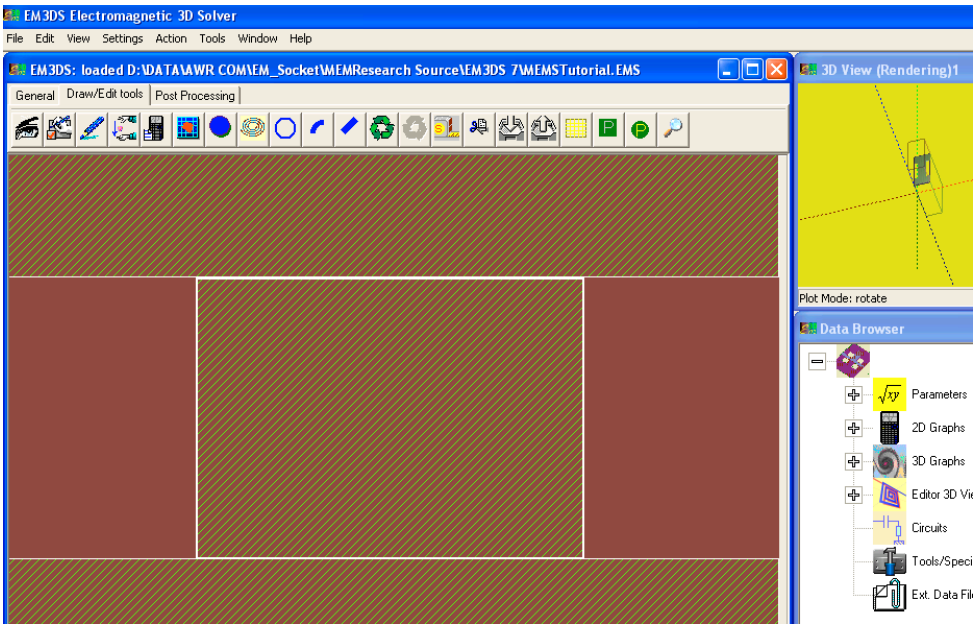


Fig.47: The bridge

Now select layer 2. The bridge structure appears yellow if you require to show all layers (CTRL+A). In layer 2 you should enter two more rectangles: size and positions have to be the same of the first and last rectangles entered in layer 1. So we can avoid to repeat the same steps by right clicking and selecting from the popup menu “**Translate/Move/Copy All objects in layer**”.

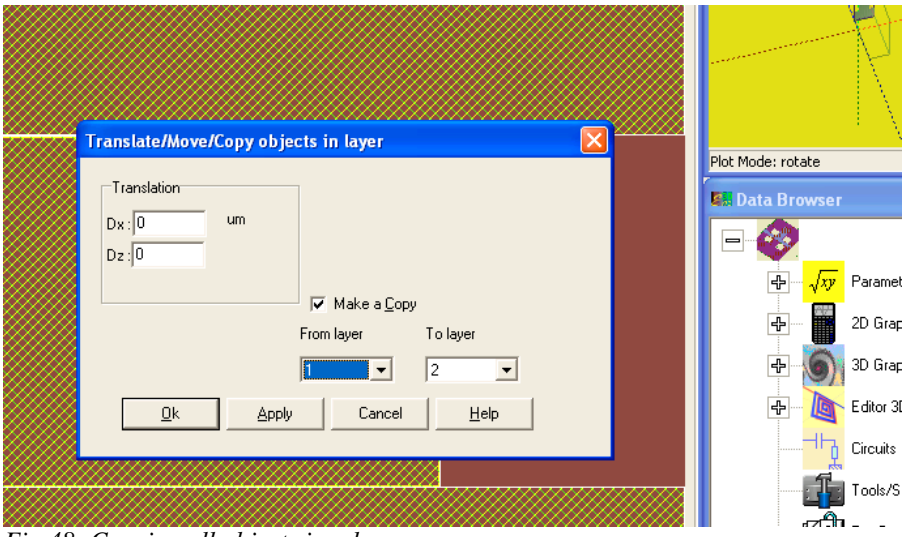


Fig.48: Copying all objects in a layer

Require to make a copy of objects from layer 1 to layer 2. Then press ok.

This way the position of the rectangles is exactly the same. Otherwise you'd meet problems with the mesh, as any item in one layer produces some mesh in items over other layers, and minimum displacement may produce very close mesh cells, so as to involve large computer resources. Now delete the center electrode: select it by mouse and press **Del** on the keyboard. Incidentally, note that since EM3DS 8 you can select multiple objects, by keeping pressed SHIFT key and clicking over the objects, or by enabling **Select Group** from menu **Edit**, or from the popup menu. If using the Select Group option, you can click by mouse to set the upper left corner of a rectangle and then click to set the lower right corner: all objects included in the selection rectangle will be selected. Once selected objects can be deleted (**Del** key), moved by arrows key on the keyboard, copied, cut, pasted etc. **Mesh options** and Resistivity in the menu **resistivity-Material Specifications** are applied to all selected objects. Note that buttons **Apply to all objects** will apply the setting to any item in the project.

Coming back to our editing, resistivity for items in this and the next layers has not to be scaled, as these layers have their actual thickness. Hence just specify 2.5e-8 Ohm m as resistivity (deselect "**Scale Resistivity**")

Same identical steps have to be followed for layer 3, placing two rectangles with same dimensions and same resistivity, working as anchors for the bridge and lateral ground planes for the CPW. The same expedient used above to save time may be employed. Note that these conductors are also in contact with the lateral box walls, that is an ideal ground.

Same identical steps have to be followed for layer 4. Here we have also to add the central CPW conductor, the signal line. Dimensions are 280 (x) and 420-2*(gw+sp) (z) while position is 0 (x) (gw+sp) (z): this ensures that the line is placed in the middle if you change spacing.

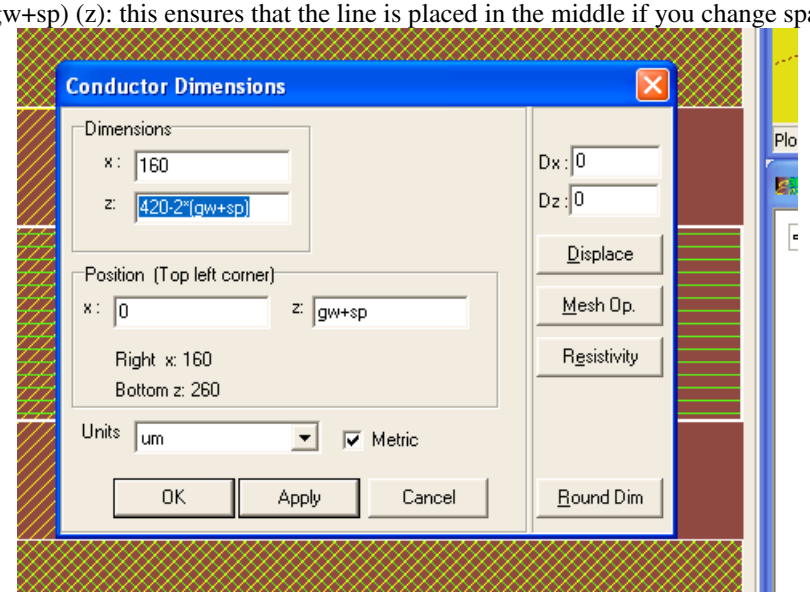


Fig.49: Position and size of the middle electrode: expressions allow to automatically place in the middle of the box the conductor

The structure is virtually ready to be simulated. However ports have to be specified.

5) Click over the "**Add edge port**" button then click over the center CPW conductor near the left side.

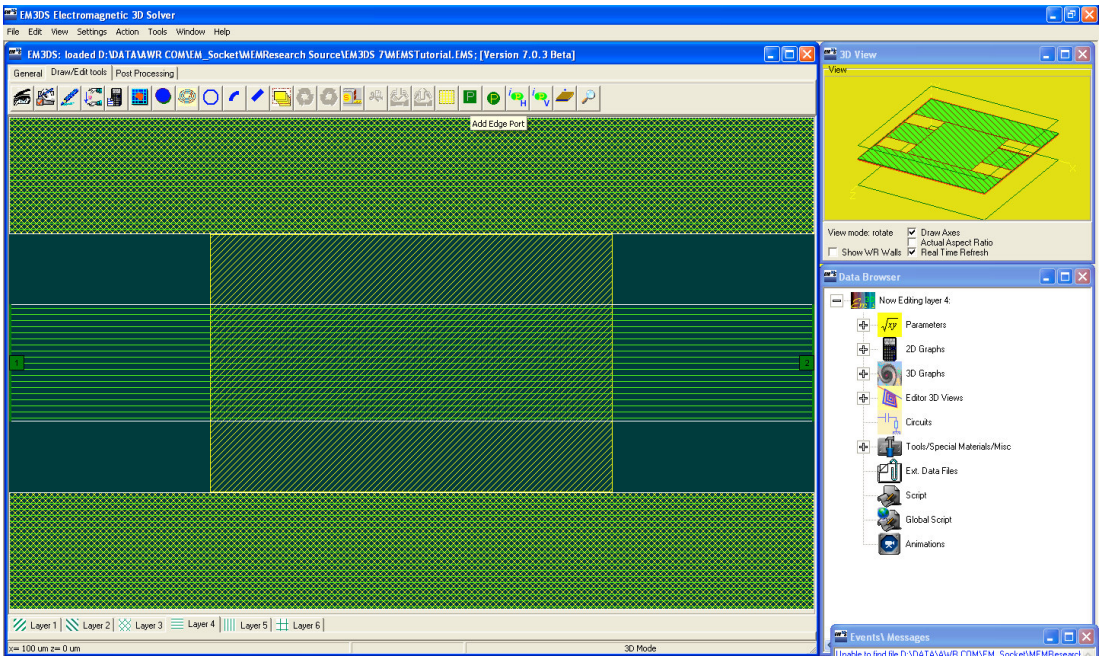


Fig.50: Edge Ports

A green box will indicate that the left port has been added. Repeat the same steps to add one more port at the right side. Port numbering is automatic and appears when moving the mouse over the port symbol.

This is enough to start the simulation. However we have not yet specified the reference planes. If no reference plane is specified, the default planes are the ones of the ports. Results calibrated at port plane will ever be available along with the ones at the specified plane. Calibration is necessary to remove the effect of port discontinuity. In a 50 Ohm microstrip line this is roughly 0.1 pF shunt capacitance. While such a capacitance may not be important for some electrically large structures, such as filters of switches in actuated position, in our case it may be even larger than the bridge capacitance. Hence calibration is mandatory. If you want to displace calibration planes (appearing as green lines) you can either double click over the port or access the calibration plane from the popup menu. In our example we'll leave the reference at the port plane. Note that via ports and internal ports are not calibrated in this version.

The structure is now ready. You can see in the 3D window how it appears. By dragging the mouse over the 3D window you can rotate the structure and, by accessing to the popup menu, you can zoom in and out the 3D window.

All you have to do now is to select the "**General**" panel and click over the "**Go**" button (Or menu **"Action"/"Computation"** or simply press F9).

In versions up to 7, a warning message is displayed: the number of required modes to build the solution has not yet been selected. In version 8 and above a checkbox in the "Substrate Information and General Options", namely "Always use suggested values" allows to skip this message, and EM3DS will automatically perform the following steps. Choose "**yes**" from the dialog, and the project data window is once more displayed.

Select the button "**Use Suggested Val.**". An additional recommendation: there is the panel "**Number of involved modes**". These are the modes used to build up the asymptotic part of the Green's function. Strictly speaking, the engine will use the number of modes selected above only at the first frequency point, while exploiting the number entered in the "Number of Involved modes" at any other frequency. This is what makes EM3DS so fast. This number is usually very small, independently from the aspect ratio. There is a suggestion also for those value, and you have to manually enter a number slightly greater than the minimum suggested (e.g. if suggested are 3 for x and 3 for z, enter 5 and 5). If however the structure involves quite lossy substrates, the suggestions may be currently underestimated, and a good rule to get broadband results is to with a value significantly larger than the suggested one (e.g. for a suggested minimum value of 3 use 8 or 10). Typically this correction is necessary for substrate with losses larger than 0.01 S/m.

In our case we have a substrate with conductivity 0.033S/m, so well in the range where some care has to be used. Set the involved modes to 8 in x and 8 in z directions.

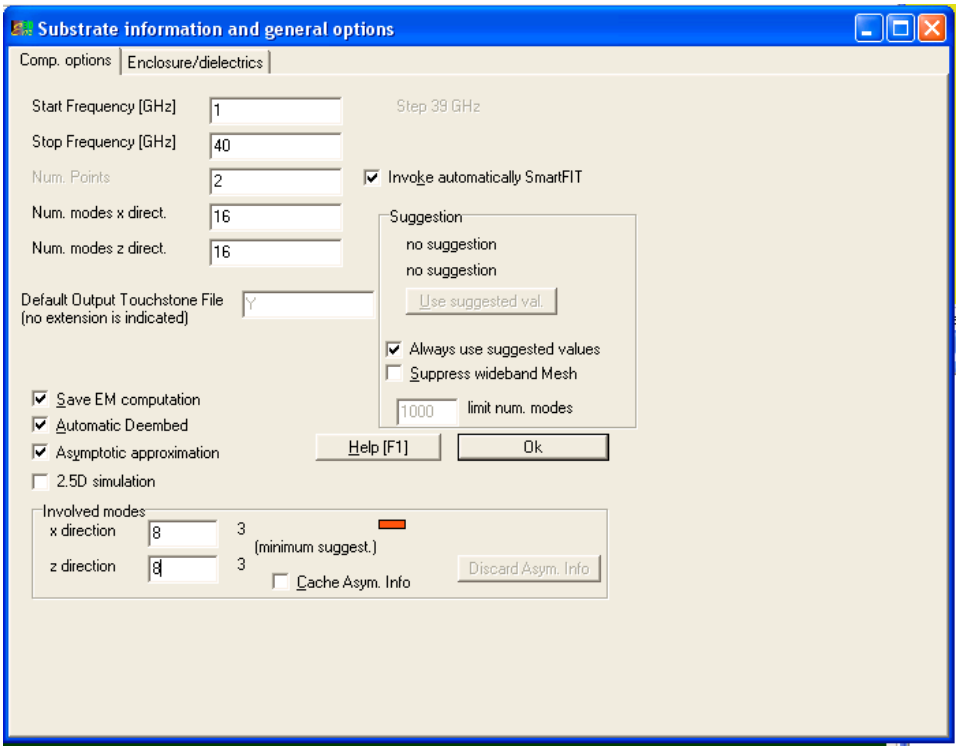


Fig.51: Selecting the number of modes

In any case a remainder of this danger appears in the Event window. At this point everything is ready. Then close the window, and try again with the "**Go**" button.

Calculation starts. It takes something more than the enclosed example HISSwitchOn.ems in the MEMS gallery. This is due to the mesh, that has not been optimized. For any shape you can select from the popup menu the mesh options. Tips and tricks about mesh, as well as the meaning of the parameters, are described before in the manual, and we recommend to read those sections carefully, as very high efficiency and accuracy may be obtained by some practice.

The first frequency point requires substantially more time than the others, as the asymptotic estimator is invoked starting from the second point. Asymptotic estimation is available for refining analysis by adding frequency points if neither the structure nor the mesh have been changed (hence it is often useful avoid the automatic mesh update when changing the upper frequency of analysis; this way no additional time for the first frequency re-computation is needed).

Simulation may be stopped at any time: the best moment to stop is when a frequency point is completed with its calibration. All partial data are available and if calibration and analysis points match, the analysis may be resumed without loss of information.

If you use **SmartFIT**, at the end you will have two data sources, the actual spot frequency calculation and the interpolated data set (usually an additional trace, whose name contains symbol ~); in this particular case, standard settings of SmartFIT do not show the impressive performances of this tool (in terms of time reduction). Usually SmartFIT performs much better if inside your band the response has pole and/or zero (such as in filters). The Lowpass.ems example clearly shows this.

You can display the response while being calculated by selecting from the "**Post Processing**" panel the "**Network Parameters**" button, and by selecting "**Refresh during Data Processing**" from the popup menu.

7) The best way to use the post-processing features, is to use the "**Data Browser**", (Menu "**View"/"Data Browser**") namely a tree where charts and data are displayed. By right clicking over the data browser you can add Smith Charts, rectangular plots, current plots etc, as well as to use post processor devices such as a circuit simulator and a spice-model extractor.

If you want to compare your results with experimental ones, simply select the rectangular network plot, and from the popup menu select "**Add Data Source**"; select the file HisSWOnMis.s2p.

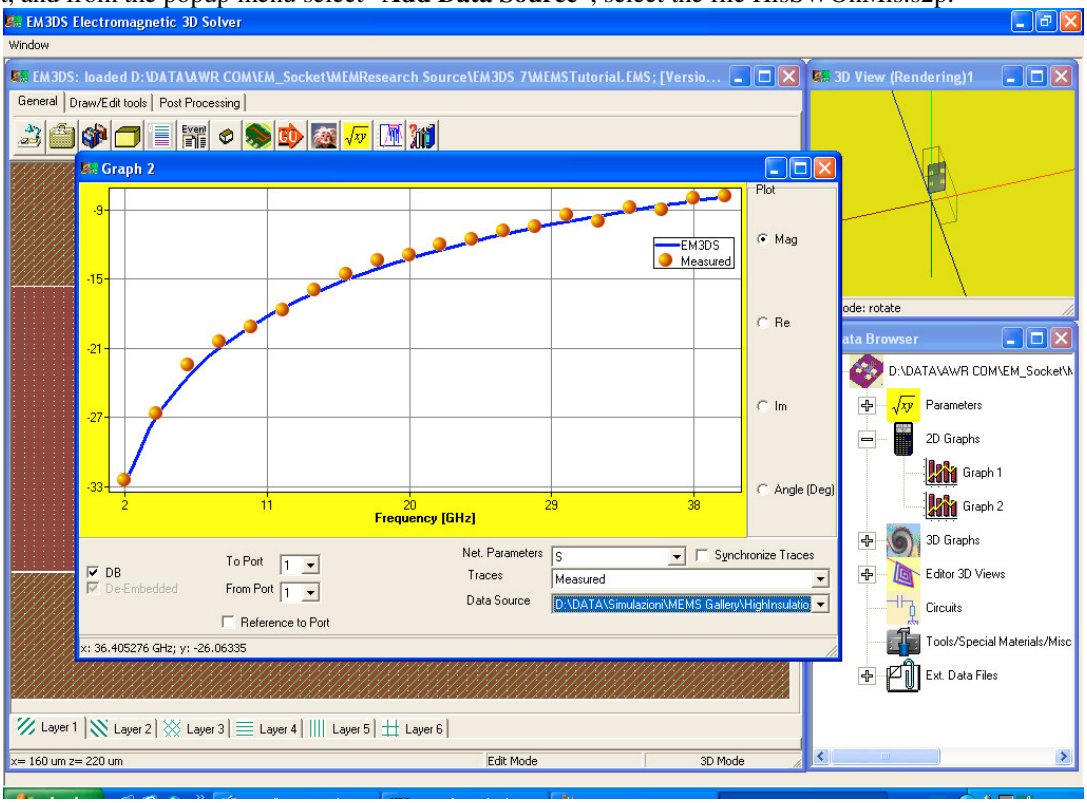


Fig.52:Standard result chart

If you want to change the kind of displayed parameters, e.g. if you want to display s21, remember to change "**To Port**" parameter for every one of available "**Traces**". Note that in the file HisSWOnMis.s2p no data about phase is available, hence comparison for other parameters than magnitude of S is not meaningful.

By accessing to the popup menu, you can select **Zoom by mouse** or **Pan by mouse**: in this case, by dragging the mouse you can either zoom or pan your plot.

If you want to see a more realistic view of the structure, go on the Data Browser, right click and select from the popup menu "**Add 3D View (Rendering)**": a window showing a 3D view with lights and impressive visual effects that you can rotate, zoom, copy etc appears. You can also zoom into stacked structures to see internal details! Also currents may be displayed using this powerful post-processor: by the Data Browser select "**Add Current 3D Graph (Rendering)**". You can add other 3D views and Graphs and to access them by the Data Browser. When creating or reading a new file Graphs are discarded, but the 3D Views are still available (you can delete them by "**Del**"). Note that not all functionality of these windows are guaranteed to work in any computer, as they are hardware-dependent, and your Graphic Card may not support all features. You can also plot vector currents on these view.

Design of the actuated switch follows similar lines: only now the bridge should be moved from layer to layer 2, so as to be over the silicon nitride layer.

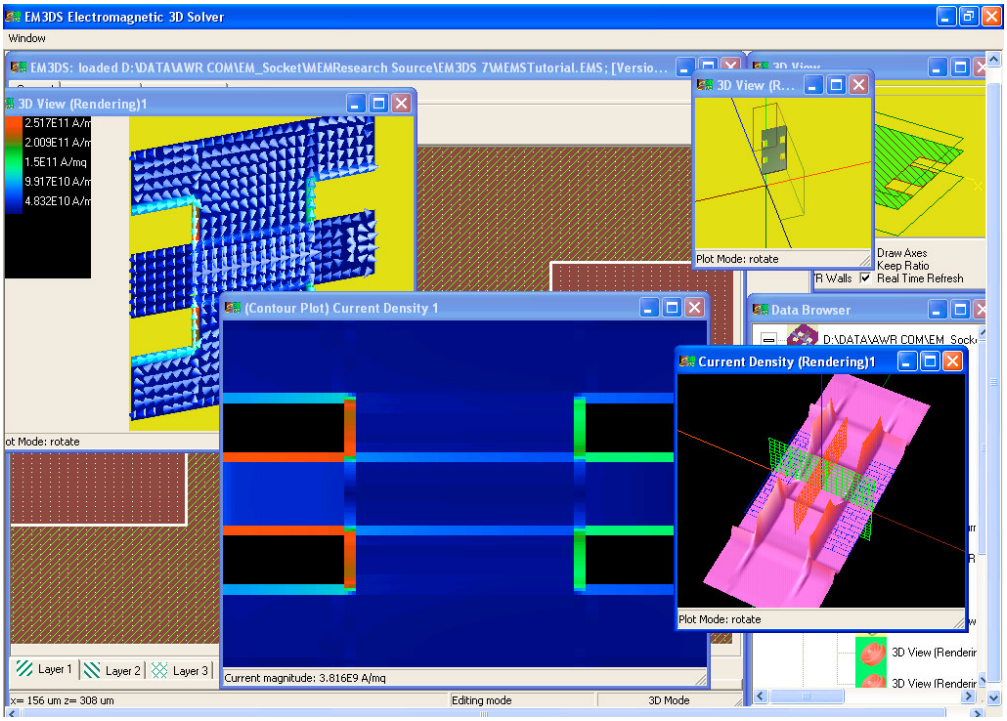


Fig.53: Screen-shot: several kind of current visualisation

Now you have correctly simulated and processed your switch in the 3D mode. Yet we have seen that most of the 3D features were not used (as we have left thin most of the layers) witnessing that in this case a 2.5D mode would be ok. We can run the simulation in 2.5D mode very quickly and just a few modifications are necessary to fit into the conventional assumption than odd layers are infinitely thin. So we have to add some dummy empty and thin layer here and there to ensure that the geometry is the one desired.

As first step let us check "**Settings**"/"**2.5D mode**". Immediately any 3D window is updated to highlight that all odd layers have virtually disappeared. Now layer 1 (odd) is where is the bridge, so it is ok: it is a thin conducting path. The same for layer 2, which is a via layer, and there we find the anchors. The problem is with layer 3, where the passivation substrate is: being a substrate, whose thickness is very important, it must be a "via layer", hence an even index layer. Consequently we open the Substrate Information window, and we add before layer 3 a dummy empty layer, say 0.01um thick (actually, being an odd layer, its value will be zero, so for any practical purpose it does not play a role in the simulation, as far as we do not introduce objects in it).

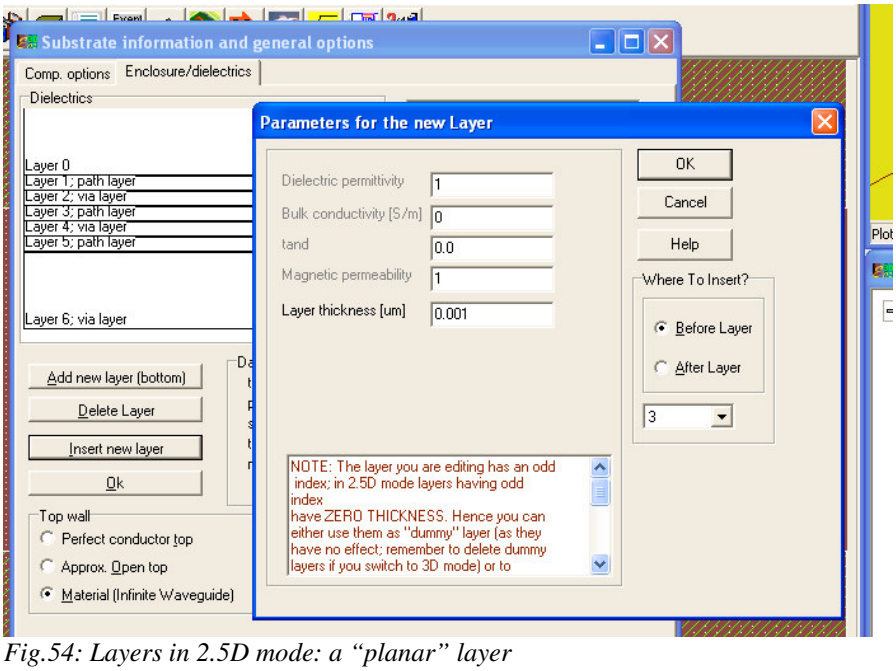
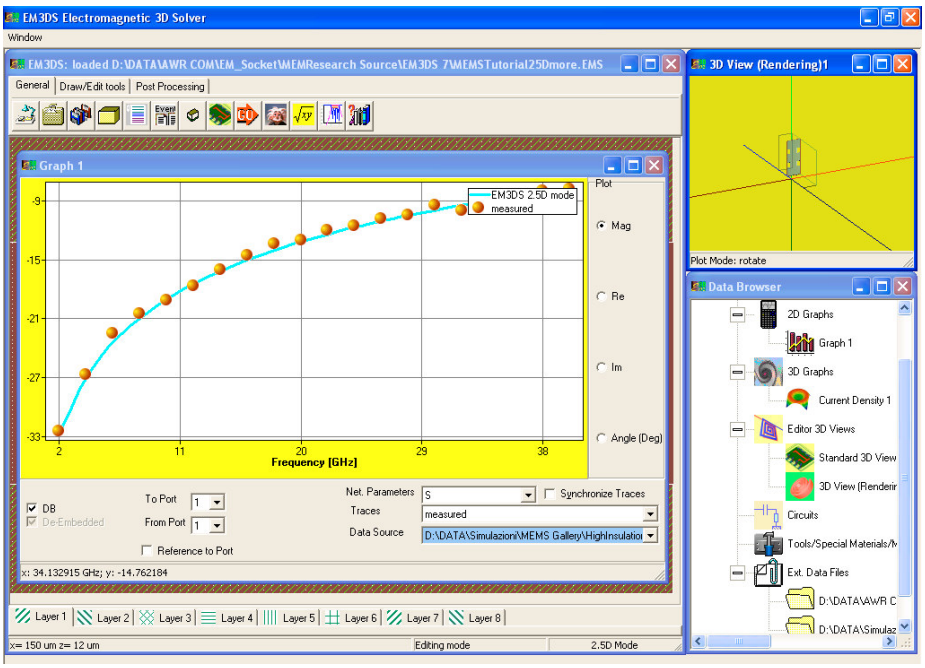


Fig.54: Layers in 2.5D mode: a "planar" layer

Being a dummy layer, its permittivity and physical parameters are not relevant, but we can assign values (e.g equal to the ones of the passivation layer) so that, whenever we revert to the 3D mode, we have not necessarily delete it (of course in that case we should add lateral CPW conductors, to maintain the current continuity when layer has finite thickness).

Now edge ports are in layer 5, which is odd, then a standard conductor layer. Hence ok. Layer 6, the oxide substrate, is an even layer, hence hosting vias and substrate: even in this case no modification is necessary. The problem is with layer 7, that was supposed to be the silicon substrate; but substrate must be in an even layer. So before layer 7 we will add one more dummy layer. Even in this case we can use the physical properties of layer 7, while not strictly necessary. We usually recommend doing that in order to avoid confusion.

Now the 2.5D project is ready (you should have 8 layers), and you can run the whole simulation in a time which is a fraction of the one needed by the 3D simulation



You can find this example in MEMSTutorial25Dmode.EMS.

XVIII Dielectric Bricks and resonators/ material specifications

Dielectric bricks are dielectric regions not extending up the enclosure boundaries. Hence dielectric bricks allow to model dielectric discontinuities.

EM3DS does not distinguish conductor and dielectric bricks: if a dielectric is being specified in place of a conductor, the obtained currents are mainly displacement currents. In order to specify a dielectric brick one has to add a conductor (rectangular, polygonal or circular), right click over the conductor and select **Resistivity/Material Specifications** (either from the main menu **Action** or from the popup menu). In the dialog box one has to select the check box **is a dielectric**, specifying the relative permittivity and changing the default resistivity (Note that the relative permittivity cannot be the same of the embedding medium: a waning is issued and, when trying to perform computation, also a fatal error will stop calculation).

The user should remind that if information about the **Asymptotic behavior** (see p. 8) is available from a previous simulation, it is not discarded when changing the parameters of this window, so that simulations by varying resistivity or permittivity of the brick can be performed very quickly. *Note that any constitutive parameter may involve frequency-dependent expressions as well as parametric expressions.*

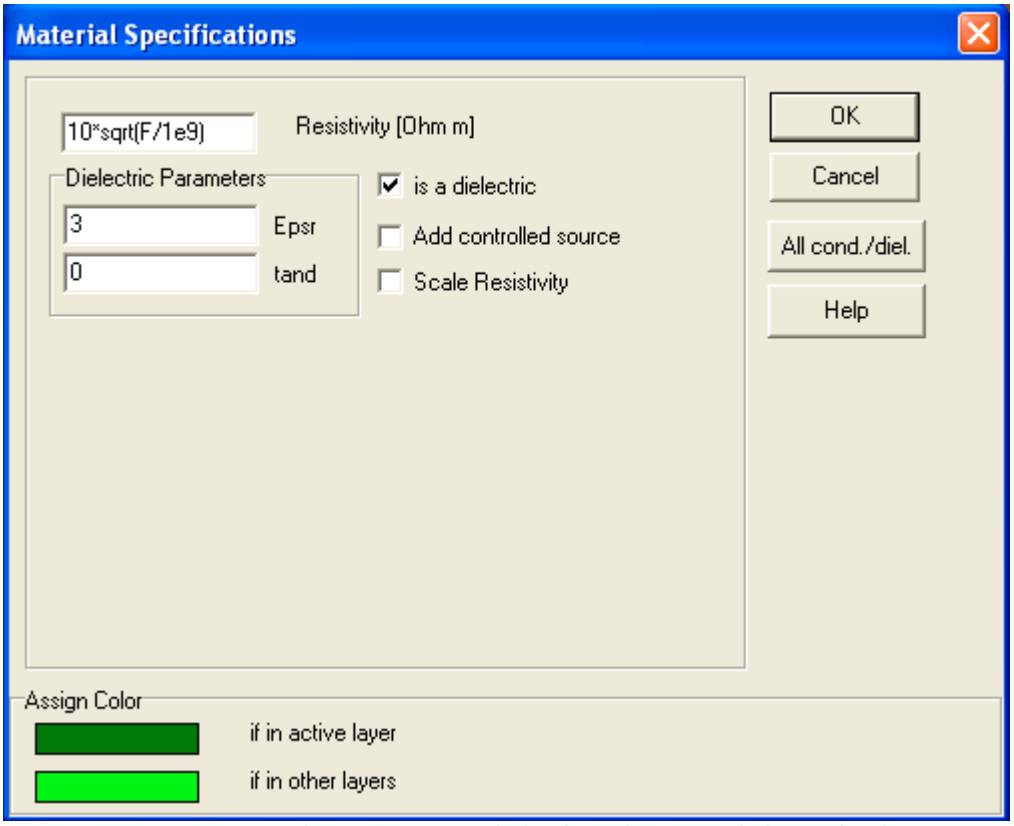


Fig.55: "Resistivity/Material Specifications" dialog box: defining a dielectric brick

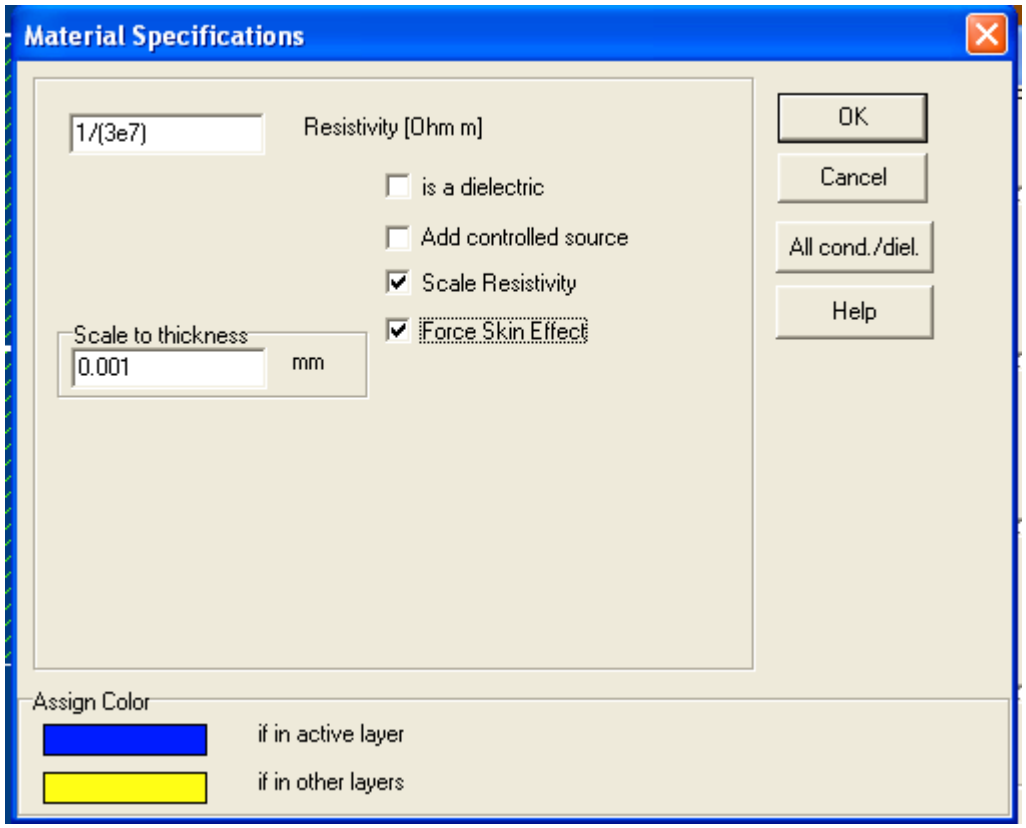


Fig.56: "Resistivity/Material Specifications" dialog box:Setting "Scale Resistivity" for a thin conductor

Use dielectric bricks even in order to remove some dielectric area in the substrate (e.g. by specifying a unitary relative permittivity) or to model integrated capacitances. (Notes: Avoid using

dielectric brick to remove conductive substrates. The magnetic permeability is the same of the embedding substrate).
Dielectric bricks are used in the same way as conductors.

A controlled current source may be superimposed by selecting **Add controlled source**: this is used to simulate active linear devices like MESFETs, HEMTS, etc.

The user has to select the **direction** of the controlled source (-x or -z). The source will be proportional to the voltage calculated across a given layer, (usually the one defining a Shottky contact) and selected by the **Control layer list**. The proportionality factor is the transconductance gm (mS/mm or S/m). The control voltage will be taken along the co-ordinate of the **control plane** in the control layer. The control plane is displayed as a cyan line.

Modeling active devices requires an in-depth knowledge of the working principles of FETs: for more information see next section (Modeling Active Devices).

Notes and restrictions: layer including controlled sources *cannot* have non-uniform slicing. Scaling resistivity not allowed. The controlled source cannot flow across a dielectric brick. Controlled source cannot flow across perfect conductors (zero resistivity not allowed). An object including a controlled source cannot touch other similar object. Modifications of the parameters pertaining to the controlled source produce automatic discarding of the asymptotic information (unlike what happens when modifying conductor properties or converting conductors in dielectric bricks). Using controlled sources has the effect of making the device non-reciprocal: Gauss Solver is invoked regardless the user selection, resulting in slower simulation and higher memory requirements

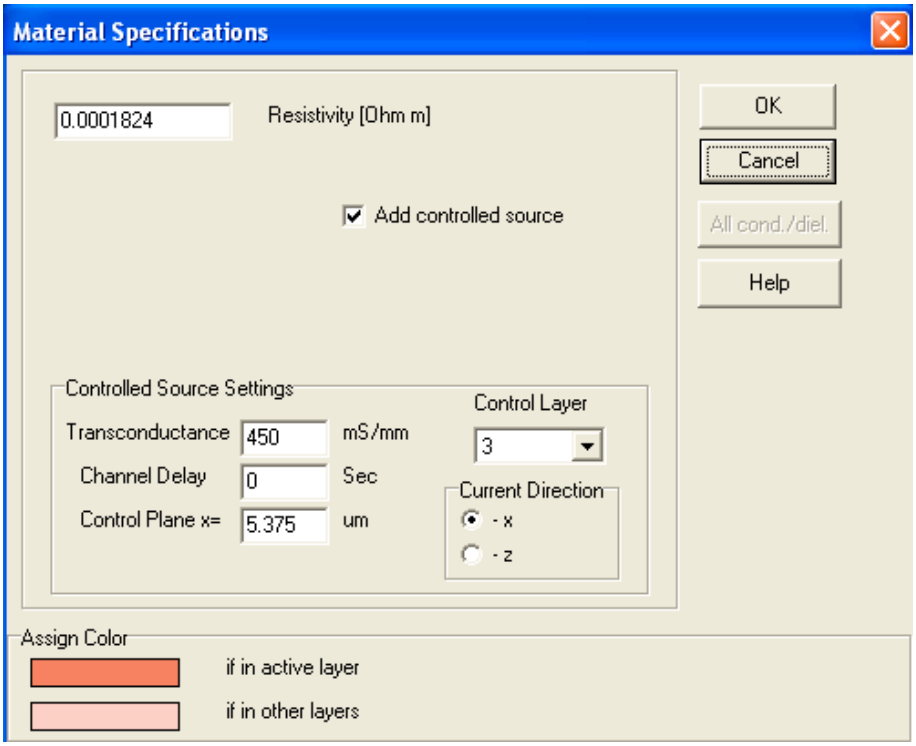


Fig.57: "Resistivity/Material Specifications" dialog box: Adding controlled sources

In the same window there are a few additional options.

As discussed in the previous section, by selecting "**Scale Resistivity**" check box you can require that a thin conductor have a finite conductivity: this option may be useful to evaluate conductor losses while keeping a conductor thin, so as to avoid the additional computational load that a thick conductor would require. In this case the conductor resistivity is replaced according to

$$\rho \to \rho \frac{t_{desired}}{t_{actual}}$$

where $t_{desired}$ is the thickness we want to simulate (and is the value entered in **Scale to thickness**) and t_{actual} is the actual thickness of the conductor (i.e. the one of the embedding layer). This is an

approximation that may be useful to reduce the computational effort whenever finite thickness is believed to play a limited role: this reduces the analysis to the one commonly adopted by 2.5D approaches. Due to the above formula, do not apply the **Scale resistivity option** to via conductors (scaling is made over the wrong dimension!). While in the standard 3D mode skin effect is modeled by subsectioning the conductor thickness (see p.6), in the 2.5D mode the only way to add skin effect is by checking the "force skin effect" check box: the resistivity is scaled according to the well known skin depth rule. The problem is that such a rule assumes the current density flowing mainly at one side of the conductor: this hypothesis is usually barely acceptable for microstrips, but it fails e.g. in stripline structures. The user should keep in mind this set of limitations when using the above options.

Note: a typical error (see FAQ 4) is assigning a non-zero resistivity to thin conductors; in this case, of course, the resulting conductor resistance becomes very large and no signal is carried by the conductor*! Do not assign non-zero resistivity to conductors lying in thin layers*, unless **Scale Resistivity** is selected and **Scale to thickness** parameter is properly selected.

Assign Color provides a tool for assign a color to the material: "**if in active layer**" specifies the color of the material when it is in the active layer, and its color on the 3D window, while "**if in other layers**" specifies the color in the editor if the shape lies over a non active layer.

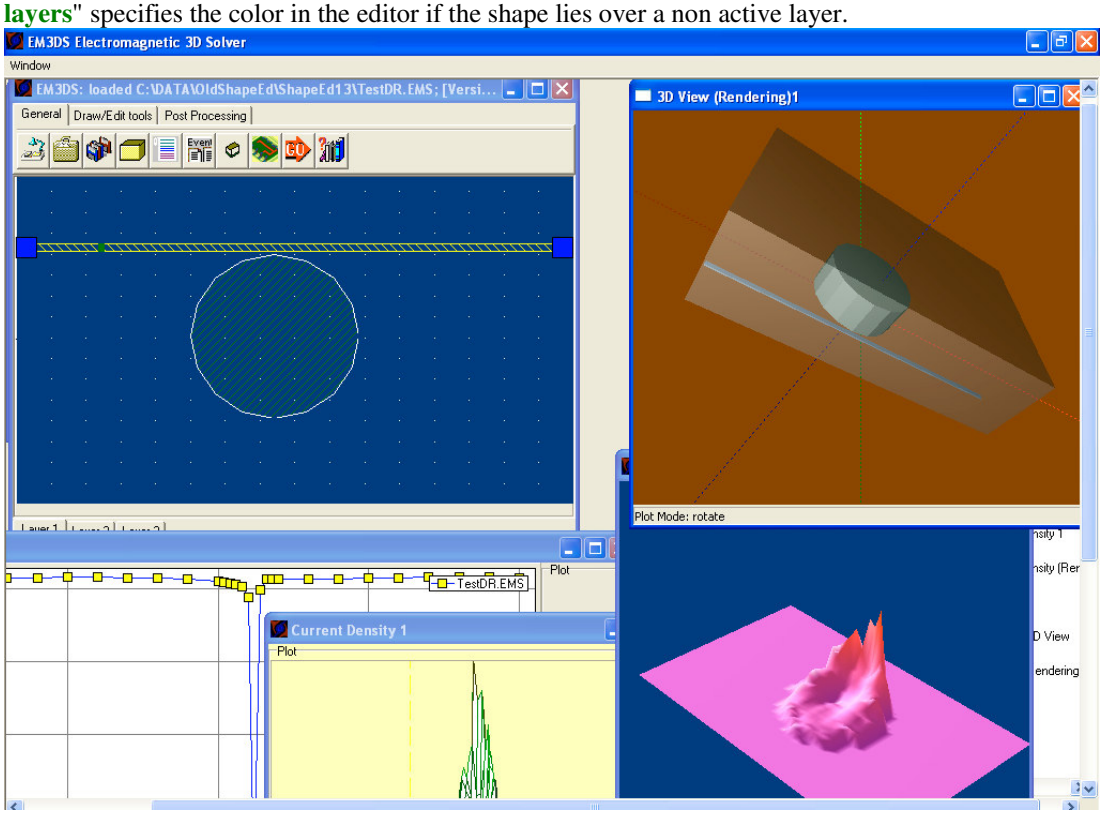


Fig.58: A microstrip-fed dielectric resonator

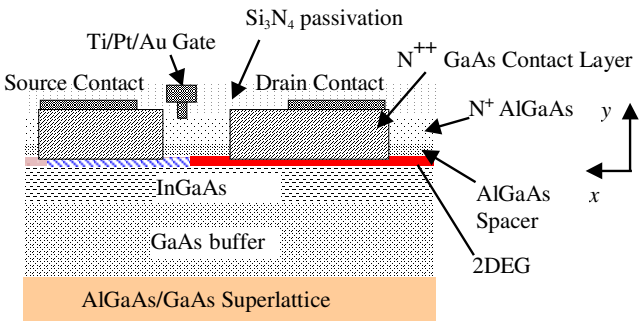


Fig.59: Schematic view of the cross-section of a single "mushroom" AlGaAs-InGaAs pHEMT active element.

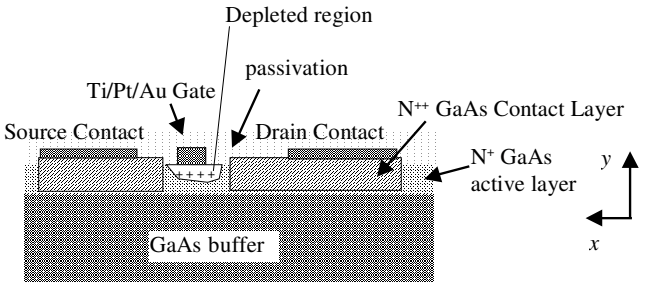


Fig.60: Schematic view of the cross-section of a single MESFET active element.

In EM3DS volume currents model conductors, dielectric bricks and active regions. The very nature of such currents is specified by imposing:

- Ohm's law to be satisfied within the conductors
- the Two Dimensional Electron Gas (2DEG) in HEMTs and the channel current in the epitaxial layer in MESFETs to be linearly controlled by the vertical (*y*-directed) field underneath the gate electrode.

Note that the second constraint yields a fully electromagnetic treatment of the *active* part.

Applying these two constraints yields the integral equation to be solved for the complete current distribution.

In both MESFET and HEMT families the gate electrodes form a Schottky junction with the underlying layer, but their behavior in the normal operating mode is rather different.

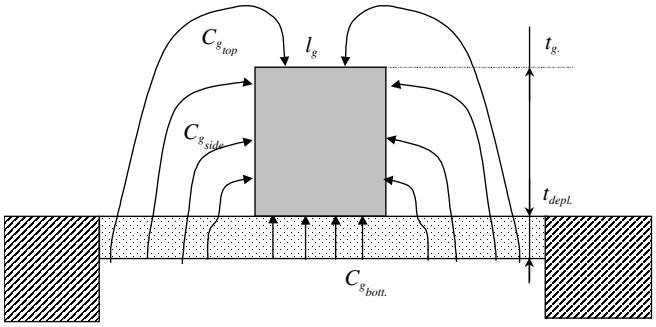
In the pHEMT of figure above, the doped AlGaAs region may be assumed to be completely depleted, as usually the case for normal HEMT operation in order to avoid parasitic paths between source and drain: the incoming electric field over the complete AlGaAs layer controls the density of the 2DEG. In the MESFET, on the other hand, the depletion region controls the width of the channel, modulating the total current flowing from source to drain. Due to the small-signal hypothesis, it is assumed the depletion cross-sectional dimensions to be fixed, while the channel current density to be modulated by the field across the depletion region. An important consideration has to be made as to the shape of the depletion region: its form is important as it strongly affects the static gate capacitance. On the other hand, in as far as dynamic effects are concerned, the cross-sectional dimensions are much smaller than the wavelength up to very high frequencies, far beyond the millimeter range. For this reason the depletion region is modeled as an insulating layer, horizontally limited by the source and drain caps, and having a mean depth in order to accounting for the static gate capacitance.

A more accurate model of the shape of the depletion region is possible indeed, by accepting a severe additional computational load, but in most practical cases it does not result in any accuracy improvement. In fact, a direct correspondence between the differential gate capacitance, due to the charge accumulated in the depletion region, and the capacitance due to the field between the gate electrode and the channel exists only under the hypothesis of an abrupt passage from the ionized region to the channel layer and for vanishing drain-to-source voltage. In this case a well-known property is that the simple parallel plate expression for the gate capacitance -eqn. (1) in the following- is formally the same one obtains by differentiating the charge in the Schottky contact with respect to the gate-to-source voltage. Owing to these considerations, it is generally preferable to assume an effective depth for the depletion layer.

To this aim it is important to estimate the static gate-to-channel capacitance per unit length: for micrometric gate electrodes, the common "parallel plate" formula

$$C_{g_{bott.}} = \frac{\varepsilon_r \varepsilon_0 l_g}{t_{depl}} \qquad (1)$$

holds quite satisfactory. However, for submicrometric FETs the contribution to the total capacitance of the side-walls and eventually of the top surface of the gate electrode is generally not negligible, and equation (1) gives underestimated values.



By simply assuming a uniform circular electric field in the medium embedding the strip and approximating its lines of force as straight lines in the depletion region and by imposing continuity of the electric flux density across the boundary between the two regions, one obtains

$$C_{g_{side}} \approx \frac{2\varepsilon_0 \varepsilon_{r_{emb.}}}{\pi} \ln\left(\frac{\pi \varepsilon_{r_{depl.}} t_g}{2\varepsilon_{r_{emb.}} t_{depl.}} + 1 \right) \qquad (2)$$

The total gate capacitance per unit length is obtained by adding to eqn. (1) twice the value of eqn. (2). For T-shaped gates some further (usually minor) correction may be needed. Its amount is evaluated by calculating the capacitance of the upper part of the T-gate alone.

The needed differential static gate capacitance, as function of the bias point, may obtained by using the non-linear models. Note that this way the present model is able to completely predict performances of a device simply starting from structural parameters - virtually avoiding any fitting. However, to this aim a detailed knowledge of the physical parameters is needed; these parameters are often not available, or at least they are not known to the degree of precision necessary for a reliable simulation. On the other hand, measurements on a given sample may be available: in this case the static gate capacitance is derived by fitting at the bias point measured low frequency S-parameters to the lumped equivalent circuit for the device. The advantage of the latter approach is that one does not have to distinguish between MESFET and HEMT devices, as the equivalent circuits share the same topology and the present method may still be used in order to analyze and predict scaling and topological effects. The same considerations apply to the modeling of SiGe devices.

The differential gate-to-drain feed-back capacitance (C_{gd}) has also to be considered: in the normal (saturation) operation of linear microwave FETs it is nearly independent of the biasing point, reducing to the gate-to-drain interelectrode capacitance, hence no further modeling is needed. Whenever the operating point does not completely satisfy the saturation hypothesis, an additional capacitance between gate and drain electrodes has to be added at the network level

With a view to model velocity saturation of the carriers, the active channel is divided in two regions: a high conductivity area between gate and source and a lower conductivity region (saturation region) between gate and drain, as shown in figure above for the HEMT. The highly conducting region is responsible for the intrinsic gate-to-source resistance R_i. Its resistivity is linked to the latter parameter as follows:

$$\rho_h \approx \frac{R_i h W}{l_g + l_{gs}} \qquad (3)$$

where l_{gs} is the distance between gate electrode and source cap, W the device width, and h is the depth of the undepleted part of the channel for MESFETs and the 2DEG thickness for HEMTs.

The saturated low-conductivity part of the channel is responsible for the differential output resistance of the FET:

$$\rho_l \approx \frac{R_{out} h W}{l_{gd}} \qquad (4)$$

l_{gd} being the distance between gate electrode and drain cap. Values for ρ_l and ρ_h may also be obtained directly from the saturated and the linear values of the carrier mobility, when available from Hall measurements, i.e.

$$\rho_{l,h} = \frac{1}{q n \mu_{l,h}} \qquad (5)$$

where q is the unit charge, n the carrier density and μ the carrier mobility. Note that when using (5) in MESFETs a slight correction may be needed in order to account for the layered model of the depletion region.
The controlled source across the low conductivity region of the channel or of the 2DEG completes the physical picture. In EM3DS such a source is assumed to be

$$\mathbf{J}(\mathbf{r}) = \mathbf{G}(\mathbf{r},\mathbf{r}') \cdot \mathbf{E}(\mathbf{r}') + \sigma(\mathbf{r})\mathbf{E}(\mathbf{r}) \qquad (6)$$

The operator **G** is the one linking the controlled source to the controlling field: its domain is the Schottky contact, being non-vanishing just in the 2DEG or, more generally, in the channel.

An animated tutorial, in the **tutorials** panel (button ), shows how practically set up the simulation of an active device.

XX Asymptotic Estimator: saving large amount of time with limited or no loss of accuracy

Handling 3D structures could cost a rather large computation load. However some devices in EM3DS allow reducing considerably the time needed for simulation. In particular EM3DS implements a new algorithm resulting in a dramatic speed improvement. The algorithm (**Asymptotic estimator**) requires a full (and hence slow) analysis *at just one frequency point (the first one)*; some data, related to the asymptotic behavior of the Green Function, are mathematically manipulated, stored and used to speed up the simulation over the whole band.

For broadband analysis a better choice would be to obtain asymptotic data in the middle frequency. This is possible, by running a simulation from middle to upper frequency and then by changing the frequency band for the remaining part; however you have to select NO when EM3DS asks you if updating mesh as consequence of having changed the upper frequency of analysis (or the asymptotic part is discarded).

This way the time required for a full 3D simulation is dramatically reduced, while preserving the accuracy. Settings about the asymptotic estimator (enabled by default), are available within the **Project Data** window, namely by clicking over the **Settings/Analysis and Substrate settings** menu.
If the check box "**Asymptotic Approximation**" is selected, EM3DS computes and stores information about the frequency behavior of the Green Function, and uses this information to dramatically speed up the simulation over the remaining frequency range where only the modes indicated in the **Involved modes** panel are used. The lower the **Involved modes** the lower the simulation time. More modes mean more accuracy; however generally no more than 10 modes (both in x and z direction) are needed to obtain excellent results. In some cases 3-5 modes may be enough! **If the number of involved modes is lower than the minimum suggested, a warning message is added into the event window. Results may lose precision in the higher frequency range**.

Some care is only needed if a conductor is placed over (direct contact) a highly lossy substrate, like a semiconductor: in this case the suggestion about the involved modes is known to be underestimated. The same consideration applies whenever frequency-dependent parameters *for the substrate* are used (no problem for properties of conductors and dielectric bricks). A warning is issued in the Message Window in the above cases.

The simulation may be really fast for several frequency points. The **Asymptotic information** is available until the user modifies the structure or the mesh settings. Note that if you change the **Stop Frequency**, you are asked if updating the mesh or not. If you select to update the mesh, the **Asymptotic Information** is discarded. If you believe that the present mesh is good enough for the new "stop frequency", it is better to leave the mesh unchanged. Usually a look to the current distribution helps to see if mesh is satisfactory or not: by rising the frequency of the displayed current it may happen that standing waves become more and more coarsely described. This is a symptom of insufficient mesh.

The best way to verify the accuracy of a mesh selection, valid for any em simulator, is performing a new simulation with a denser mesh and evaluate how results are modified.

The only structural modification that you can do without losing the "Asymptotic information" is changing the resistivity of a conductor or the dielectric permittivity of a dielectric brick (or even

changing the nature of a conductor in dielectric brick and vice-versa).

If the **cache Asymptotic Info** check box is selected (and **Save EM Computation** is selected), data about the Asymptotic approximation are saved in a ".~mp" file when discarding the current structure, and are available for future use: note that "~mp" *files are usually large* and saving or opening them may require a while. A red light in the "**Involved modes**" panel means that no Asymptotic Information is available (hence a full simulation is required at the first frequency point). A green light indicates that asymptotic information is available, while a yellow light means that a file containing asymptotic information has been found and is currently being loaded, so that you have to wait until it becomes green.

Actually there is a minor price to pay: using the Asymptotic Approximation requires more memory. Moreover a feature of EM3DS is that currents are modeled by piece-wise sinusoidal (PWS) functions with frequency-dependent argument: as a consequence physically sound results may be obtained even with very coarse mesh. Whenever the Asymptotic Approximation is used, the argument of these functions is fixed at the lower frequency of analysis, so losing this attractive feature.

XXI Even faster: SmartFIT

SmartFIT is an adaptive scheme for evaluating the response within a given band: EM3DS will select automatically points within the range you have specified, and at the end it will create an additional data set, available in the data browser and in any chart, containing the estimated response. The new data set is always saved into the HD with a unique name, possibly containing a progressive number.

You can access it either from the toolbar, or from the menu **Tools/SmartFIT**. Menu **Action/SmartFIT Computation** directly run a simulation driven by SmartFIT.

SmartFIT will ever start at equally spaced frequencies defined by **the "Number of Starting Points**" (usually 3, band extremes and one middle point). Then it will iteratively progress, until the result is within the **Maximum relative error**, or the maximum number of iterations has been exceeded**.** ***Note that if the number of evaluated frequency points is too large, the estimation may be poor and no convergence is obtained***; while quite rare, in that case consider to subsection your frequency band. In any case, the "spot-frequency" response is available.

The prediction is performed over a number of frequency points, according to what entered in "**Interpolated Points in SmartFitting**". You can also leave this number low (to speed up the process) and then tentatively **create a new interpolated data set** in post-processing, from the popup menu in any chart; the interpolated data set will have as many points as specified in **Number Of Interpolated Points**. In this case, one should use as input (Data Source) the current EM simulation, not the interpolated one, as the number of input points should not exceed 40-50. Also note that this post-processing may be applied to any data set, even imported data. *However, while the adaptive fitting will look runtime for significant frequency points, the interpolation made in post-processing phase (eventually on external data) is completely blind, and interpolation may well be not satisfactory, regardless of the number of input points.* The interpolation is, in fact, a standard rational interpolation.

As to SmartFIT, note that, even setting a relative error as high as 1, EM3DS will locate pole and zeros, and may provide a good result with very few points; if relative error is kept high, results are usually poor if no pole/zeros are in the selected band.

The checkbox "**Fast Interpolation**" will reduce to the minimum the controls, and this results into a drastically reduced robustness: this usually only works well for symmetric structure having in-band pole/zeros, but not private poles.

Checkbox **Do Not Use SVD** may be used if for some reason interpolation fails: SmartFIT will usually select automatically an internal solver for best performances; **Do Not Use SVD** forces SmartFIT to avoid SVD solution. SVD, in the way it is used, is very robust, even though sacrificing some precision at the interpolation points. This also affects the way in which the Broadband Spice Model Extractor works.

XXII Automatic De-Embed

Ports introduce a discontinuity, partly due to the presence of the lateral walls. In many cases this discontinuity does exist in real structures, so one has to include its effect. This is e.g. the case if a coaxial connector is used.

Even when such a discontinuity is an undesired effect, it may be neglected as well in many

important cases: in filters, resonators and antennas the resulting additional shunt capacitance (ranging around 0.1pF) produces a negligible effect (at least one the moduli of S parameters). This is not the case of small discontinuities, as e.g. a microstrip bend or cross, and generally of electrically short structures, where the discontinuity effect may heavily affect the results and need to be removed.

Moreover often some additional line length, introducing additional phase shift and possibly losses, is added in order to keep the analyzed structure far away from the box walls: the user may want to characterize the structure by removing the feeding line effect. To this aim a de-embedding (calibration) algorithm is implemented. Note that in the present version only edge ports are de-embedded.

The de-embedding algorithm is a new Short-Open Calibration (SOC) technique presented in *IEEE MTT April 2001* in a paper by M. Farina and T. Rozzi: usually de-embedding requires the analysis of two standards with different enclosures to be computed, while in this approach only one standard is used. If more ports are coupled at the same side, coupling is also removed by de-embedding.

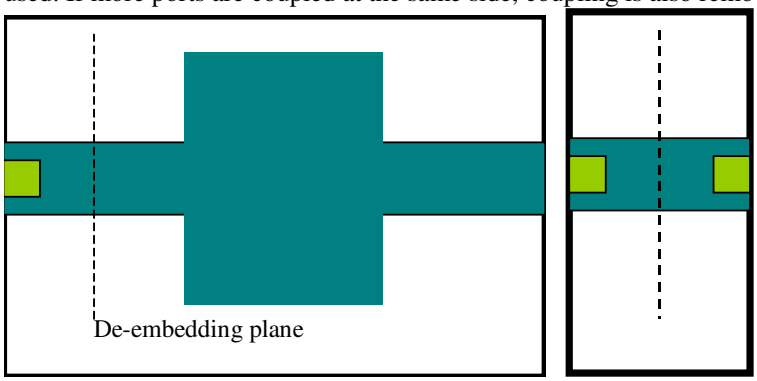


Fig.61: Device under test (DUT) including a feeding line and, on the right hand side, standard structure automatically built by EM3DS and not shown to the user.

If the **automatic de-embedding** option is selected, the standard, not de-embedded output file is produced, and additionally other de-embedding files are saved (namely error-networks representing a negated form of the dicontinuity+feeding lines system), including the overall de-embedded results.

EM3DS directly connects the above networks and the user should not be directly concerned with above files: this is done in the Plot Window (menu "**View/Graph**" or button "**Plot network parameters**", or select a graph from the Data Browser by double clicking over it), where network parameters are shown. De-embedded network, as well as the standard network, may be exported by this window.

What the user should only be aware of is that when opening a pre-calculated project, including de-embedding, EM3DS needs

- file "DLY*.*" "DRY*.*" etc. possibly created during de-embedding procedure
- File ". DAT": the distribution of the current density; this file allow obtaining the non-de-embedded network parameters
- File ".EMS ": native file describing the structure (it is the only one strictly needed to perform a simulation). It is the only one strictly needed to describe the structure
- File ".DES " (description) is an RTF file, created by EM3DS, where design notes may be stored.
- File ".GOS " (Graphic Options/Settings), storing number and formatting of network graphs.

When at least 1 port is attached to a conductor, it is possible to specify the distance of the de-embedding (reference) planes from each side of the box (left, right, top and bottom). The calibration plane is available only if the "automatic de-embedding" check box, in the Project Data window, is selected.

Specifying the distance of the de-embedding planes at each side of the box (left, right, top and bottom) is only possible when at least one port is attached to a conductor. The selected plane is displayed only if the "automatic de-embedding" check box, in the Project data window, is selected

If de-embedding is selected, the additional button "**De-embedding line parameters**" is made available: by clicking over this button characteristic impedance, effective dielectric permittivity etc. of the feeding lines are displayed. If more than one edge port is attached at one side of a box, the eigenvalues of the characteristic impedance matrix are calculated. Those eigenvalues coincide with Z_{oEven} and Z_{oOdd} for a couple of symmetric lines, and are a generalization of these impedances in the general multi-port case (see See the paper *"On the Definition and the Derivation of the Characteristic Parameters of Coupled Lines, and Application to MoM Analysis"* by Marco Farina, Antonio Morini and Tullio Rozzi, in the *2005 IEEE MTT-S Int. Microwave Symp Dig.* For more info.

Also, propagation constant and effective relative permittivity are calculated. Note that in order to have sound parameters, the de-embedding planes should not be placed at more than λ/4 from the enclosure. If this condition is not satisfied the line parameters could be incorrect (as highlighted by a warning); however de-embedding works correctly as well, as it does not rely on these parameters.
If the calibration structure resonate at some frequency point, calibrated results appear to contain some narrow spikes, and at the same frequency the feeding line parameters contain similar irregularity. Usually spikes disappear by reducing the distance of the de-embedding planes.

If the structure is enclosed in an infinite waveguide (top and/or bottom walls not present) or terminated in absorbing boundaries, some times the de-embedding may be not accurate: de-embedding is performed by building up a structure having a length double with respect to the de-embedding plane (calibration structure). The enclosure of the original structure and the one of the calibration structure may have different modes over cut-off leaking power so that the calibrated results may be affected by significant error. To solve this problem you can place the de-embedding plane at right the middle of the box, if possible, or by adding top and bottom cover in order to avoid power leakage (what of course cannot be done when analyzing antennas: see last section).

One more problem can be caused if the calibration plane distance is too short: in this case higher modes may connect ports, so affecting the calibration procedure. As a rule of thumb, the de-embedding plane should be placed at least at one substrate thickness from the enclosure (at least for microstrips). A remainder of violation of such a rule is displayed by the EM3DS when needed. It is generally better to leave the reference plane coincident with the ports than putting it at a very small distance from the port.
Moreover the user should verify that at least one mesh cell is defined between the port and the calibration plane (otherwise a warning is issued).

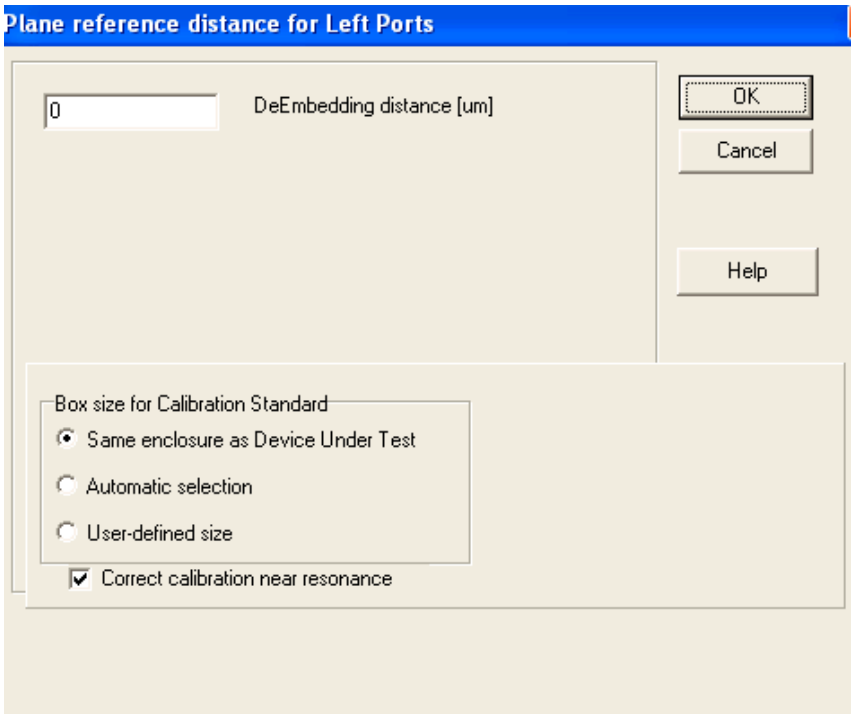


Fig.62:Dialog for reference plane and de-embedding options

Since version 6.3, EM3DS allowed to select reference plane coincident with the port plane. Actually, whenever the users select a non-zero reference plane, both results (namely at the specified reference plane and the port plane) are available.
In order to be able to perform this further calculation, EM3DS *–only to recover response at the port plane-* need some hypothesis about the nature of the discontinuity added by the port(s), and then evaluate its admittance. This evaluation may fail if the enclosure is multiple of the wavelength (actually failure occurs slightly before this condition, over a band). In order to circumvent this limitation, EM3DS (from ver 7.0.5) adds in the post-processing phase a correction algorithm, that is enabled if in the dialog for reference plane (see figure) the checkbox **Correct calibration near resonance** is selected (default). If **correct calibration near resonance** is selected, EM3DS will use

data from a lower frequency of analysis to correct resonance problems. We want to stress that this is usually not a problem when reference plane distance is non-zero, as in that case no hypothesis about the nature of the discontinuity is made.

From a general point of view, we can summarise saying than:

When the user selects a non-zero reference plane, the "standard" is in a box having double size with respect to the reference plane. Hence in this case, the user can control the size of the box by acting on the reference plane. This may useful as in some conditions (as discussed above) the analysis of the standard may fail

When the user selects a zero-reference plane, namely only the port discontinuities are removed, the size of the standard is selected according to the user's selection:

1) ***Same Enclosure as DUT*** (Default)
2) ***Automatic Selection:*** the size is selected according to the upper frequency of analysis, trying to keep the box size below λ: at multiple of the wavelength, resonant conditions deteriorate the accuracy by which the port discontinuity is evaluated. The user can try this option if there is evidence of such a problem.
3) ***User-defined size***: the user selects manually the enclosure size

While selecting above option 1) 2) or 3) will discard data previously calculated, the **correct calibration near resonance** checkbox only acts on the post-processor, so that you can ever see the response obtained with or without the correction (possibly, you have just to update existing charts by any modification in the displayed measurement). Hence this correction may also be enabled for pre-existing simulations.

If the lower frequency of analysis is already too close to a multiple of lambda, the correction will not work, and a warning is issued.

As final note, when using SmartFIT, this correction may force SmartFIT to run quite more simulations.

***To Probe Further**: Meaning of error network file*

Names to the error networks are assigned according to the following conventions:

"DL"+the selected filename for the output file for the de-embedding network for the Left hand side ports, "DR" for the right ones, "DT" for the top ports and "DB" for the bottom ones.

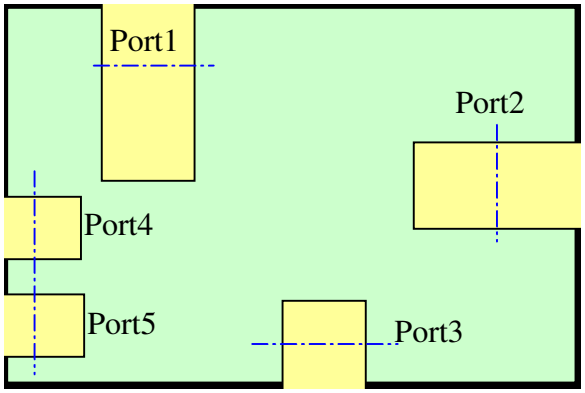
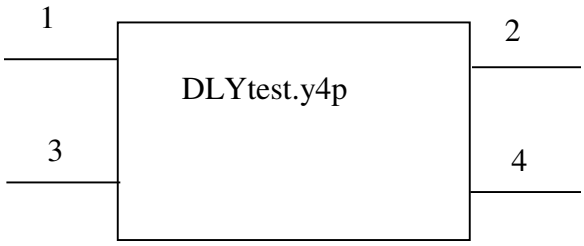


Fig.63: Example about meaning of de-embedding files

Let us consider an example on how these files could be used in principle. If the name of the default raw output file is Ytest.y5p (this is signalled in the Event window at the end of the processing), the de-embedding algorithm produces DLYtest.y4p (de-embedding network for the left ports), DRYtest.y2p (for the right port), DTYtest.y2p ecc.

The numbering of ports for DLYtest.y4p is



that is odd ports on the left-hand side and even ports on the right hand side. By connecting port 1 of DLYtest.y4p to port 4 of the output network Ytest.y5p, and port 3 to port 5, DLYtest.y4p the effects of the embedding is "negate", removing the discontinuity and the lines up to the reference plane, so that looking into ports 2 and 4 of DLYtest.y4p one obtains a de-embedded measure. The same holds for DRYTest.y2p and to DT and DB files. There are a few exceptions to the above numbering, occurring whenever during editing, ports are first inserted and then removed.

XXIII Waveguide components

EM3DS allows modeling a class of 2-port waveguide components. The purpose of this section is to show how to model waveguide components by means of a simple example: a single cavity waveguide resonator with inductive irises.

In order to draw the structure you have to consider direction y, namely the vertical one, as the propagation direction. Irises are drawn in the layers of the structure

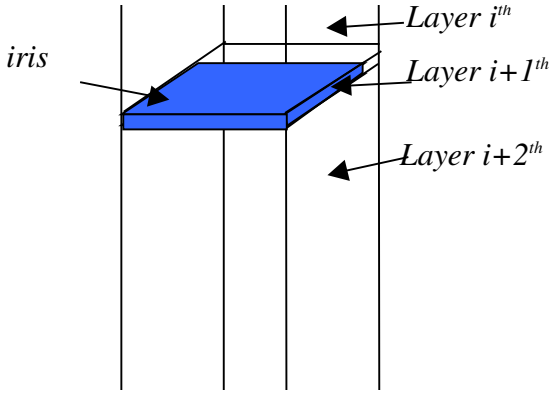


Fig.64: Drawing irises in the multilayer waveguide

The structure has to be excited. A possibility is a couple of strips, extending up to the walls, placed at the bottom and the up sides, respectively, of the structure. The strips mimic the way you probably would do the measurement of such a device: the network analyzer would be connected to the waveguide component by means of a couple of coax-to-waveguide transitions.

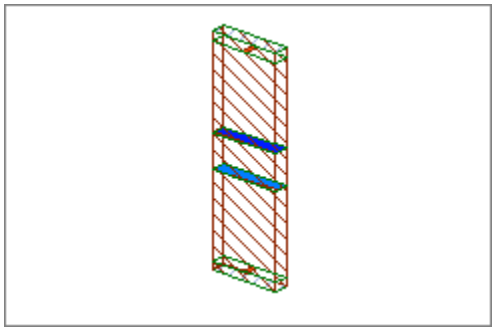


Fig.65: Single-cavity filter and strip excitation

In the example shown in figure there are 8 layers: in layer 1 there is the upper microstrip, with its edge port, layer 2 is empty, layer 3 is the first iris, layer 4 empty etc. Bottom and top walls enclose the structure.

Before (or after) simulating such a structure, you need a calibration procedure, able to remove the effect of the "coax-to-waveguide" transitions. To this aim 2 standards have to be drawn and simulated namely a "thru standard" and a "matched standard" or, as alternative to the matched one, a “short standard”. The "thru" standard is only composed by means of the two strips and one empty layer (total number in EM3DS: four layers). The length of the empty layer defines the reference plane of the "measurement". Hence if the reference plane has to be e.g. 1 cm from the irises, the "thru" standard requires the 2 strips be 2 cm vertically spaced.

An important underlying assumption is that excitations interact only via the fundamental mode; hence keep the planes far enough (e.g. more than half wavelength) from the excitations and the DUT.

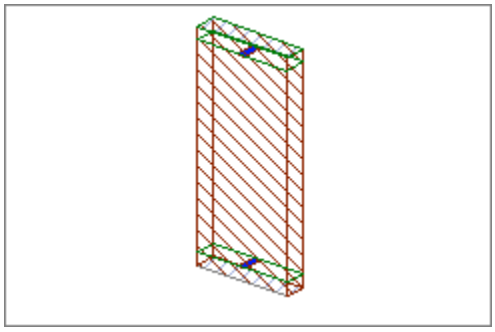


Fig.66: “Thru” standard

You should think about this standard as exactly what you would do in a laboratory: you take two coax-to-waveguide transitions and connect them together; hence you perform the “measurement”.
The "matched standard" is simply one strip having the upper (or the lower) wall settled as **Material/Infinite waveguide**: all the waveguide modes are terminated on their characteristic impedance. In EM3DS you should need only 2 layers.

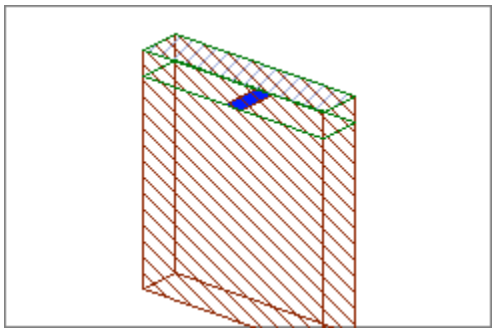


Fig.67: “Matched” Standard

In the above figure in layer 1 is drawn the microstrip, the top wall is perfectly conducting, while bottom wall is “infinite waveguide”

In version 5.2 the “matched standard” may be replaced by a “short standard”: if in the above figure the bottom wall is “perfect conductor” you have a “short standard”. In this case calibration file requires to specify at which distance from the calibration plane the shorting plane is placed: be sure that such a distance is consistent with the one used when defining the thru standard.

Analysis of the structure and of the standards does not require (and it is not recommended) that the standard port de-embedding be enabled: the port discontinuities are automatically removed in the following calibration procedure

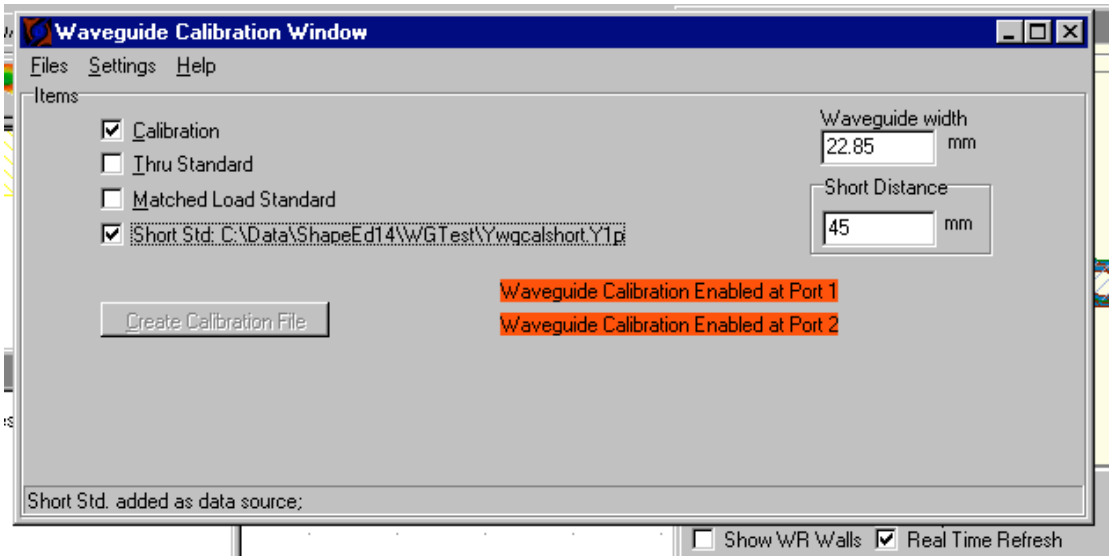


Fig.68: Waveguide Calibration Tool

Once results for the two standards are available you can process them to get a calibration file. Click over **Tools/Waveguide Calibration** menu (only available when two ports are defined in the workspace). You have only to load the touchstone output files by clicking over the **Thru Standards** check box and **Matched Load Standard (or Short Std)**, respectively. Files are loaded and available as data sources in source lists of the **Data Browser** (as folders) and of each graph (both rectangular and Smith Chart plots).

Select the right waveguide width, used to calculate parameters of the TE10 mode supported from the waveguide.

By clicking now over the button **Create Calibration File** a Touchstone calibration file is created and loaded.

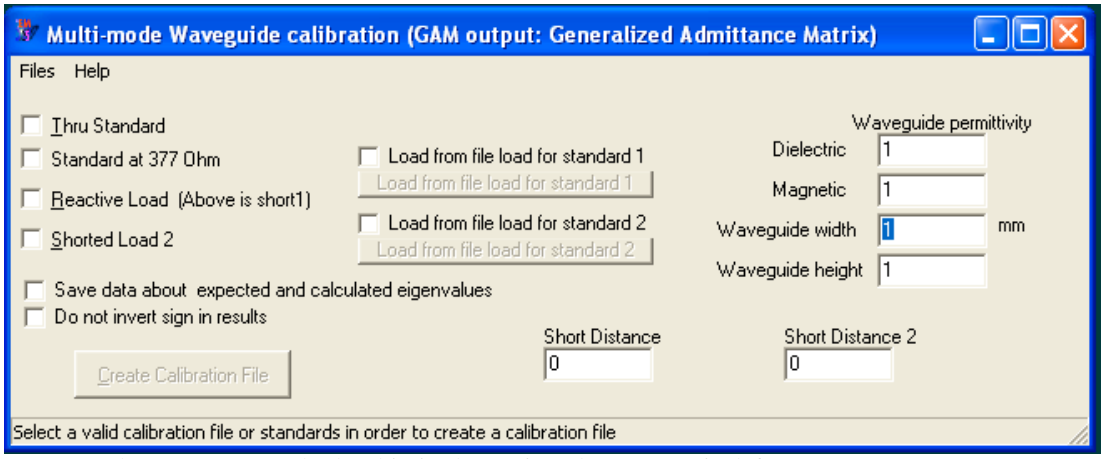
The calibration file will be valid for any subsequent analysis involving the same frequencies and the same excitation structure: you won't need to do the whole procedure if, e.g., you want to analyze a 5-cavity filter in the same waveguide. In this case you have just to analyze your filter and load the previously calculated calibration file by clicking over the calibration check-box. Calibration is enabled by clicking over **Settings** and **Enable Calibration at Port 1** and **Enable Calibration at Port 2**. In this case the plot windows display the calibrated results. By default S parameters are referred to the fundamental TE10 mode; in each graph you can change this option.

The calibration quality is somehow dependent on how you have selected excitations, namely what could mimic the coax-to-waveguide transition. You can verify to some extent the validity of your calibration by e.g. analyzing an empty waveguide having some length more than the one used in creating standard. Calibration should provide you perfect match, namely S11 less –60 dB, often even less –100dB. The phase of S21 may be compared to what analytically expected.

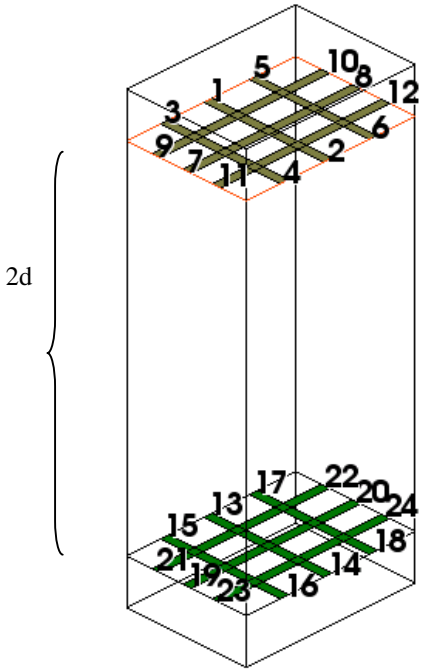
Note that also multiport waveguide component can be analyzed in principle: you should only create the above calibration files for each ports, and then connect them to the multiport structure by means of the embedded circuit simulator.

XXIV Multimode Waveguide calibration

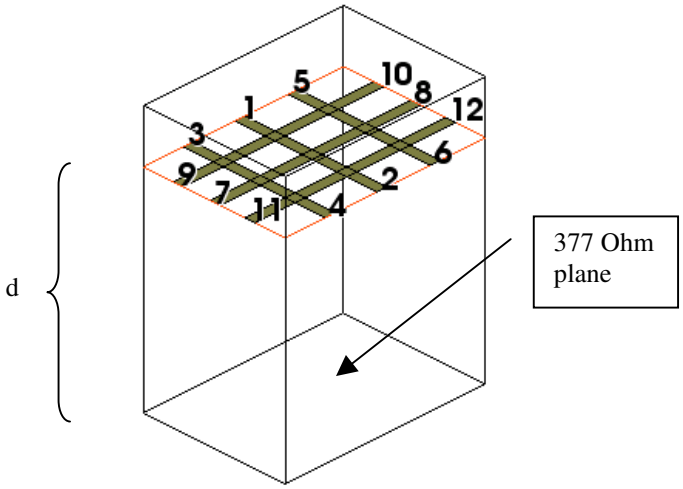
The problem can be formulated as follows: let us consider the problem of getting a the Generalized Admittance Matrix (GAM) of a circuit involving N ports attached to a planar circuit, and N modes (either below or above cut-off); basically you want to characterize a circuit involving both standard gap ports (external ports, attached to strips) and some kind of internal, modal ports, exploiting only measurement made on the external ports. There are two possible procedures. You will need to build two or three *standards*, namely additional simulations, according to the selected calibration procedure.



-Procedure 1: two standards; this is the easiest, but unpractical for measurement purposes: you build a double (symmetric or mirror) standard and a standard involving the "**approx open enclosure**" (termination on 377 Ohm). As example, consider a grid of strips involving 12 ports and exciting a waveguide: the double or symmetrical standard is shown in figure, and represents a mirror of the structure to be characterized.



Hence the structure to be characterized has to be terminated on a 377 Ohm plane as shown below.



Results from the two simulations are loaded by clicking over "**Thru Standard**" (namely the double standard) and "**Standard at 377 Ohm**" (the one involving the absorbing boundary) . Waveguide width and height should be settled to the correct values, and if the waveguide is filled with dielectrics (namely the region defining the reference plane, indicated with the distance d in the figures), sets their values in the respective edit-boxes.

At this point click on the button **Create Calibration File**. The calibration file is a 24-port network, whose ports 1-12 in our example are such that, when connected to ports 1-12 of the structure, remove the structure itself, allowing to access the internal modes (modes 1-12).

> **Note**: when performing such kind of calibration, due to the sensitivity to numerical error, select the menu **Settings/Preferences/Computation** and the checkbox "**show panel for advanced setting**". Hence set the **value Max Att Factor for Evanescent Modes** (default 3) to 9. This modification is not saved, as high values of this factor can compromise the accuracy in evaluating 3D structures

The above procedure only works if the travelling modes at the reference plane are not interacting each other, namely if the whole discontinuity exciting the modes is planar; an example: if the result is a 4 port network, 2 ports connected to strips (say 1 and 2) and two modes (say 3 and 4), the above procedure is valid only if any cross connection Y14 is equal to the cross connection Y23 (the other parameters are arbitrary). Moreover the quality deteriorates when strongly below cut-off modes are considered.

As a drawback, there is no direct way to know the order of numbering in the modal ports.

The above drawback can be avoided by using a 3 standard procedure

-Procedure 2: 3 standards. This is more general, and only reciprocity of the standards is assumed. In this case the first standard is the same of the previous algorithm (double). Hence select **Reactive Load** and the two additional standard are two sections of waveguide closed over a short circuit. The two **Short Distances** have to be settled to the distance of the shorting plane from the reference plane (zero not allowed) used in creating the standard.

*Alternatively you can use any kind of termination: in this case you can select "**load from file load**" to load the touchstone file describing the load used in the calibration. Note, however, that the termination load **must be diagonal**. This is especially useful if the process is used to evaluate multi-port circuit calibration, or in lab for multi-mode calibration from experimental data*

Now load the standard and click once more the button create calibration file.

In this case the order of the modes is the one of the cut-off. Actually, in order to sort the ports, the algorithms use the evaluation of eigenvalues of a matrix: if eigenvalues are too close, the procedure can fail. One can look at the eigenvalue, and also at the error in obtaining the network, if the check box **Save Data About Expected and Calculated Eigenvalues** is checked. In this case a file Eigenvalues.txt is created. Error can be due to a number of reasons, most notably if there are insufficient ports (namely the structure has more accessible ports in the modal side than physical ports), or if there are too many ports (i.e. you are including waveguide modes far in the cut-off region, so that their contribution is nearly zero).

In both cases the results are in terms of "error boxes", namely the "negate" the circuit; if "**do not invert sign in results**" is selected, the actual network is exported.

XXV About Ports

EM3DS allows three kinds of ports: edge, internal planar and via ports (either internal or ground referenced).

Edge ports may only be added to conductors sharing at least one side with the enclosing walls: in order to add an edge port click over corresponding button (or **Action/Add Port** in the main menu) and then click over the conductor, where the conductor touches the lateral wall. Results in terms of network parameters are ground-referenced.

*Hint: if you draw a rectangle larger than the box, EM3DS can automatically cut it in order to fit it into the box: just open the "**Conductor Dimension**" window (right click over the rectangle or menu **Action**) and click over the "**Apply**" button.*

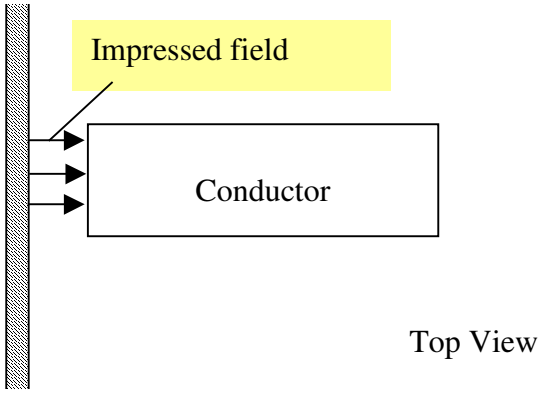


Fig.69: Schematic of Edge port

Ports are actually impulsive impressed fields, lying along one direction and not vanishing over an infinitesimal gap. In 2.5D working mode edge ports can only be added to planar conductors, namely conductors located in layers having odd index.

Internal planar ports are ports that can be added nearly everywhere to any conductor, the only restriction being that only one internal port can be added to a given object; if the circuit needs additional internal ports, split the circuit in additional conductors. In order to add them, you click over the button  for x-directed ports and  for z-directed ports. Either use the main menu **Action**.

Internal ports are the most difficult port to use, and should be used with care, as they are not ground-referenced. Basically, if the user adds an "**Internal Horizontal Port**" to a conductor, the conductor is "cut" vertically, as shown in the figure, and a voltage generator is applied across the resulting gap

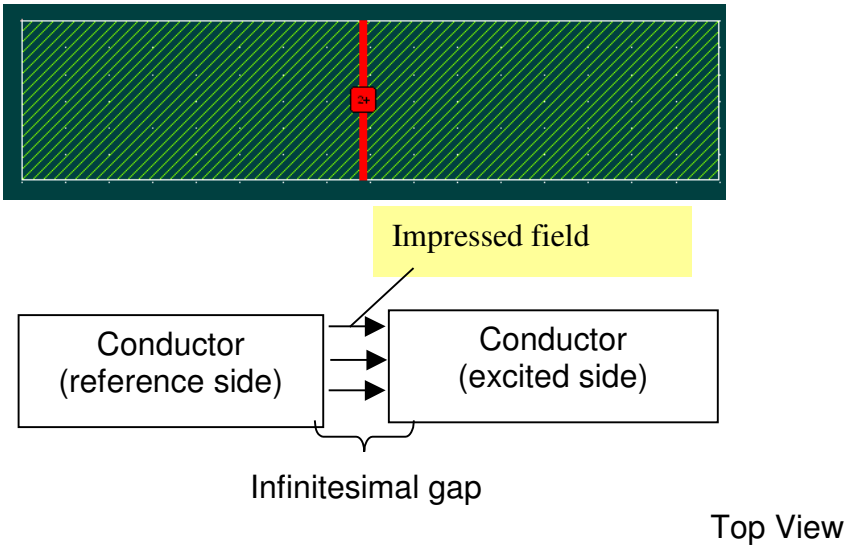


Fig.70: Internal Horizontal Port (namely field along x)

Ports are actually impulsive impressed fields, lying along one direction and not vanishing over an infinitesimal gap.

The current across the gap is continuous, and the short circuit admittance is measured across the gap: consequently one of the two parts of the conductor is ground reference for the other (the latter being indicated by a "+" sign).

Internal port can be reverted (exchange sign) and displaced by double clicking over the port.

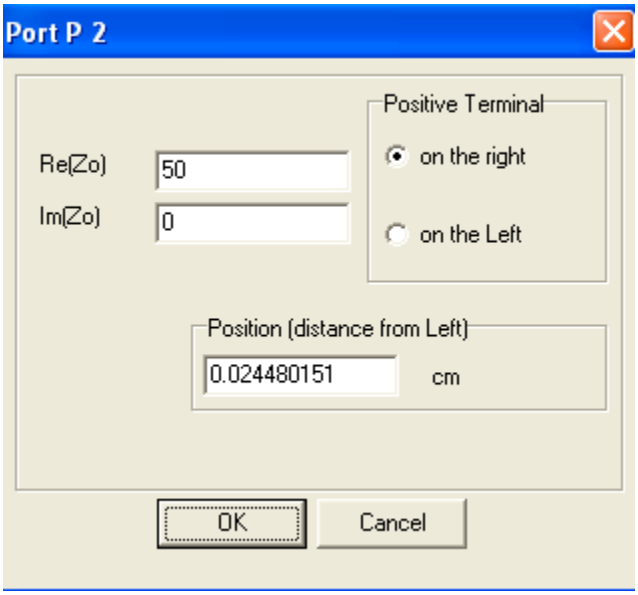


Fig.71: Port Dialog

The position is with respect to the hosting conductor (top left). Refer to the section describing the circuit simulator to have more information on how correctly using results obtained with off-ground ports.

As a restriction, the internal ports are not automatically calibrated in the present version, so that the port discontinuity is included in the results, even though it can be removed manually by using "negate" sub-circuit in the circuit.

Note that in 2.5D mode planar currents are only defined over odd layers. As a consequence planar internal ports may only be defined in odd layers. In 3D mode there is no such a restriction.

Select ports by mouse, then access additional properties by the popup menu (e.g. port impedance, calibration plane). Press DEL to remove the selected port.

Via ports, either ground referenced or "internal" off-ground, are impulsive field generators, applied at the bottom of a given conductor: hence no "via" is actually added and, if needed, the user should add a via conductor. Results in terms of network parameters are either ground-referenced or off-ground (internal) depending on the position of the conductor to which the via port is associated. Refer to the section describing the circuit simulator to have more information on how correctly using results obtained with off-ground ports.

If the conductor is in the lower layer and a perfect conductor plane is selected as bottom plane, results are ground-referenced. No automatic de-embedding is implemented for via ports in the present version.

In order to add via port, simply click over the via port button and then over the desired conductor. In 2.5D working mode via ports can only be added to conductors in layers with even index.

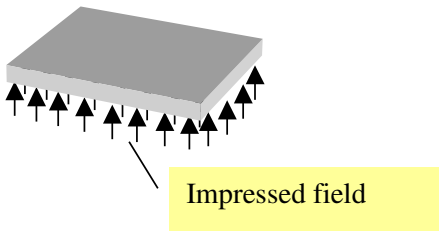


Fig.72: vertical excitation (by via port)

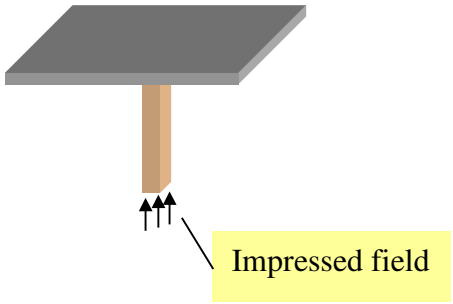


Fig.73: Via port (obtained by applying a via excitation to a via conductor)

Note: Ports cannot be attached to Perfect Magnetic Conductors (PMC): the E field orthogonal to PMC is zero.

Ports can be selected, edited and deleted by mouse (popup menu).

XXVI Post Processing: Differential ports

EM3DS handles differential ports in a post-processing phase: in order to create differential ports, first add standard edge or via ports, perform the simulation, then access the **Differential Ports** dialog, either from the main menu Tools, or the dedicated button in the toolbar or the icon in the Data **Browser**.

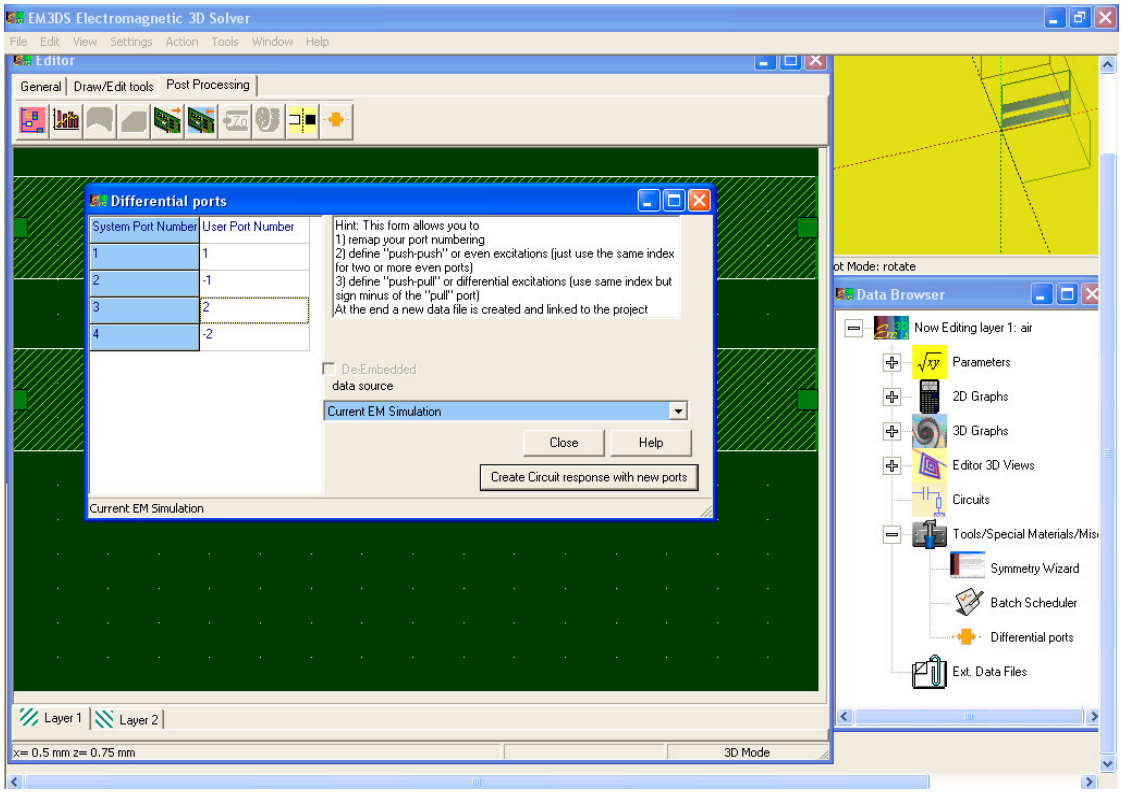


*Fig.74: **Differential ports** form*

Using this post-processor is straightforward, and it can process any network data, including those imported externally and appearing in the Ext. Data File folder.

Hence the first step is to select the desired **Data Source**. If the source is the Current EM Simulation, be sure to select **De-Embedded** if you want to work on calibrated data. At this point the table appearing allows to remap port numbering, define push-push (even) excitations and define push-pull (odd) excitations.

In order to remap port numbering just insert in the **User Port Number** part the desired port number. If two or more ports have the **same index, and same sign**, they are **shorted together (push-push configuration)**, namely, in circuit terms, they have same voltage.

If two groups have **same index but different sign**, they are connected in **differential or push-push configuration**: basically it is imposed that current flowing into the group with sign +, is the same exiting from the group with sign -.

In the example above, the result will be a 2-port, where new port 1 will be the one (differential) having P2 as reference for P1, and by the same token, new port 2 is the one having P4 as reference for P3.

Press button **Create Circuit response with new ports**, and a new touchstone file is automatically created and link made available in the Ext. Data Folder and in any chart as data source.

XXVII Variables

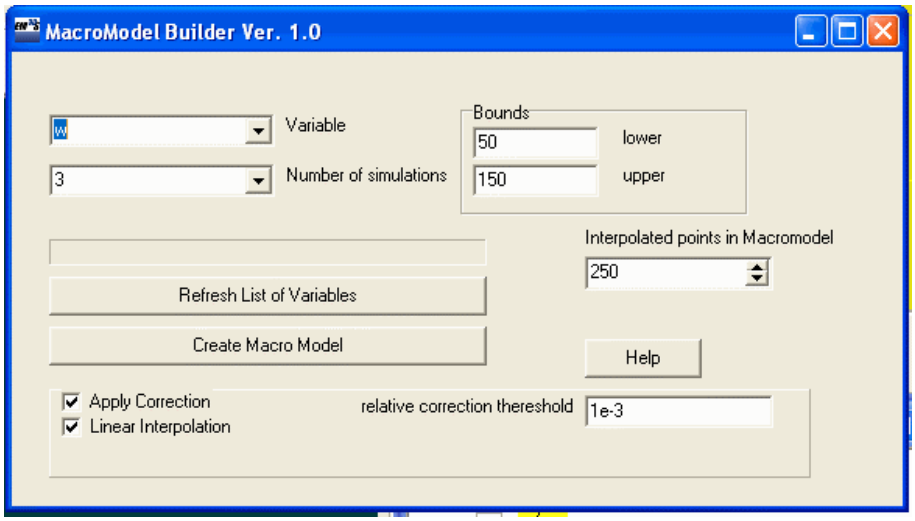
In the **Variable/parameters** panel, accessible either from the toolbar, from the menu **View** and from the **Data Browser** you enter global quantities that can be used anywhere in the EM3DS. Geometry can be parametric, so that coordinates can be expressions involving parameters: modification of the parameters will automatically modify the geometry (click on **Load/Update Variables** to make changes effective). If a quantity cannot be parametric, the parametric value or the expression is resolved immediately, replacing the parameter with its value.

The optimizer uses the defined variables to perform its task.

If a coordinate is entered as parameter, but the user moves the item either by the mouse or the keyboard arrows, parameter is replaced with its numerical value, and that coordinate is no longer parametric

XXVIII Macromodel builder

This form is accessible either by the main menu>Tools>Macromodel builder, or the button  *or the same icon in the Data Browser*



Aim

The Macro-model builder is a tool to create a parametric simulation: EM3DS runs a given number of simulations for a varying parameter, and interpolates the response to predict the circuit response over a continuous range of the selected parameter. Such a macro-model is stored in the project (actually an additional file having .mmd extension is created), and allows to see in real time the response of the circuit when the parameter is modified. The macro-model can be used also for optimization (see Optimizer) purposes

Restrictions

In the present version the macro-model can be built for a single parameter; optimization can only be performed over the macro-model if one single variable is being optimized.

In the present version there is no automatic criterion implemented to check the reliability of the macro-model (namely if the number of EM simulations was sufficient or not). Always check the result of your optimization by a final full-wave simulation of the response.
In the present version, if a macromodel is created, it is not automatically discarded when the structure is manually modified by the user, but only when a new macromodel is created (or if the file .mmd is removed). Pay attention to this fact, as a modified structure can be linked to a meaningless macromodel!

Use
Click "Refresh List Of variables" to have the list of variables updated, and select from the "Variable" list-box the parameter over which to build the macro-model. Set upper and lower bound for the variable. Be careful in setting those bounds: the model assumes that the topology of the circuit is not modified (namely, for example a low-pass circuit is not modified into an high-pass or something like that; the builder extract an equivalent circuit and assumes that only the values of the equivalent components are modified and not their mutual interconnection). For example, if there is a transmission line, avoid those values shorting the line to the lateral walls (ground) creating an abrupt discontinuous modification of the response.
Select the "Number of Simulations": this is the number of actual EM simulations performed by EM3DS over uniformly spaced values of the parameter (variable). Of course, the larger this number, more likely accurate is the obtained macro-model. Consider that in many cases **very few simulations** are needed, and this depends on the kind of structure being simulated, the frequency range etc.
Note that often a broad-band simulation is beneficial: the macro-model exploits information about zero-poles of the response, and their migration. If some pole/zero goes out of the current frequency range for a given parameter value, the resulting macro-model might be useless. This is particularly true when optimizing filters. Hence, if you are creating a macromodel for a pass-band filter, keep the frequency range much larger than the expected filter bandwidth. Some experiments will let you understand this point.

The value of the "Interpolated points in Macromodel" is the number of frequency points where the macromodel is available.
When the above parameters are settled, just click "Create Macro Model". At the end of the simulations the macromodel is available in any chart; if the parameter is selected as "tunable" in the optimization window (see the next paragraph about the Optimizer) you can tune the variable (see Tuner) and see the response evolution in real time. The optimization can be performed over the macro-model.

You still have two additional options:
"Linear Interpolation": if selected the interpolation is linear, otherwise is cubic spline. The modification affects the evaluated macro-model in real-time (no need to re-calculate).
"Apply correction": by default the interpolated macro-model is obtained by Y admittance matrices. If there are poles, around the poles the response can be poorly interpolated, creating unphysical spikes. When "Apply Correction" is selected, EM3DS automatically detects poles, and tries to correct the possible response deterioration, by exploiting results from a Z impedance model. If it is detected a discrepancy between Y and Z interpolation greater than the "relative correction threshold", Z response is assumed around the pole. The correction usually removes spikes around poles, but should not be used if in the selected frequency range there are no poles at all. In any case, even this option has a real time effect (no need to re-calculate) so the best thing to do is usually to try.

XXIX Optimizer

Once geometry is defined by means of Variables, the optimizer can be invoked. Select which variable to optimize, by inserting “Y” in the corresponding position of the variable list, as well as lower and upper bound (pay attention that no geometry is be able to exit the box!). If you select also **Tune?** The variable will appear in the tuner as tunable.

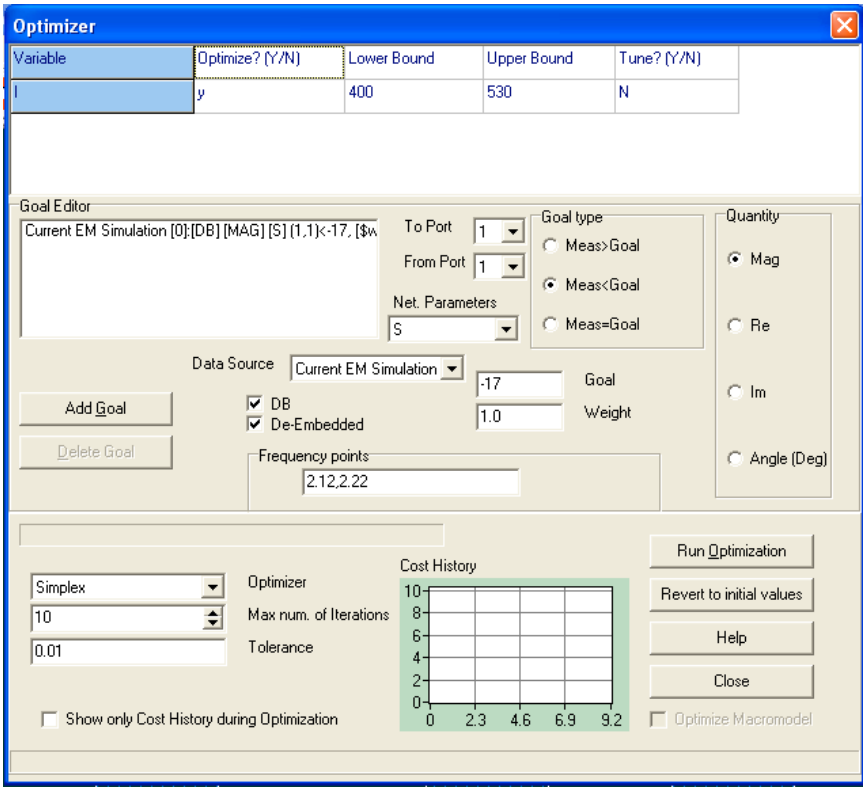


Fig.75: Optimization Panel

In the "**Goal Editor**" select the goals:

Select the parameter to be optimized, insert a list of frequency points where the parameter is optimized then select "**Add Goal**" to have the goal in the Goal List. Goals are relatively weighted according to what specified in the weight factor. By selecting a goal you can see the details in the status bar, and possibly delete it. If you have to modify a goal, create a new goal and delete the old one.

The quantity minimized is the Cost function: cost is defined according to

$$Cost = \sum_{n=1}^{N}\sum_{k=1}^{M} w_n \left| G_n(f_k) - M_n(f_k) \right|^2$$

where n is the number of goals, w_n is the weight of the n-th goal, G is the goal and M the measured quantity.

There are several optimizers available, and cost function is shown runtime. The checkbox "**Show only Cost history during optimization**" allows to reduce the size of the dialog window during optimization. Optimization may be interrupted at any time either by clicking over the stop processing button or the "abort" appearing in the progress window referring to the single EM simulation.

Button "**Revert to Initial Values**" brings back the variables to their value before optimization.

Remember that a good optimization can only start from a good design: avoid to think this is a panacea! Most typically optimization does not bring results if you have defined a set of goals in contrast each other, or simply physically meaningless, or just because your initial design was too poor.

A number of outstanding papers about optimization is available in the literature: among them, several paper by Prof. Antonio Morini, University of Ancona, have focussed on the filter optimization, showing that a filter can be optimized by only optimizing its response at a number of critical points. By the same token, one usually only needs some "hot-spots" to be optimized for whatever circuit, and this is why EM3DS specifies a list of frequencies and not a range, for the optimization.

If a Macro-model (see the previous paragraph) is available, and a single variable is selected in the optimization (coincident with the one over which the macro-model was created), the checkbox "**Optimize Macromodel**" is available: in this case the macro-model is used during optimization without additional EM simulations. The optimization is very fast, but results have to be finally checked with a full wave simulation. Moreover, if the variable is tunable, the results are updated in real time in response to the tuning.

Once geometry is defined by means of Variables, select in the optimizer dialog which variable should be tunable (see previous section). At this point either select **Tools/Tuner** or click to the tuner icon in the data browser or in the toolbar. The following dialog appears.

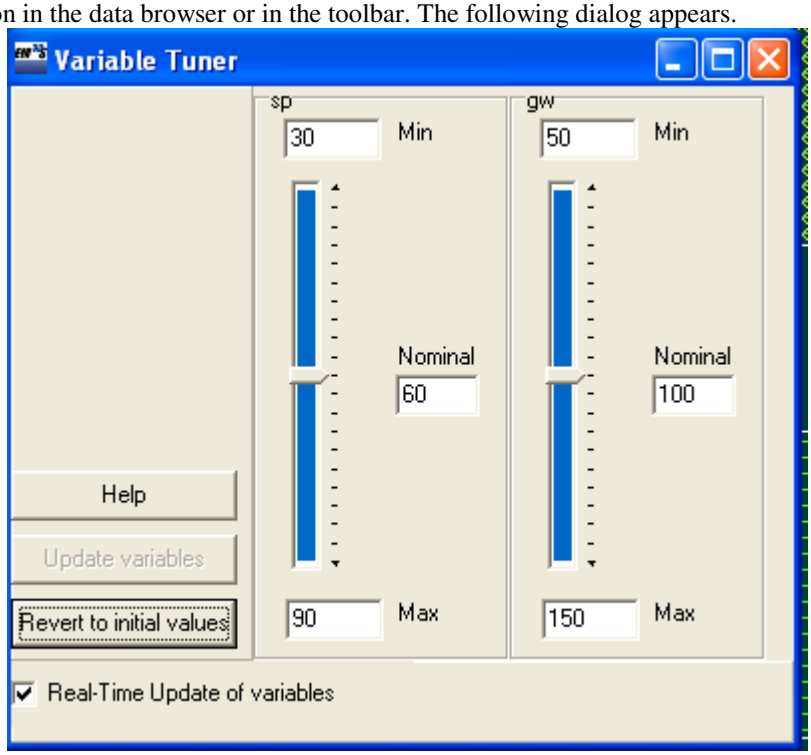


Fig.76: Tuner Panel

By moving the corresponding sliders, variables and hence the structure are modified accordingly, and the geometry updated in real-time, or upon pressing **Update Variables**, according to the selection of the checkbox **Real-Time Update**. Variable can be reset by clicking to **Revert to Initial Values**.
If a macro-model was available and created over one of the tunable variables (see Macromodel) the results are updated in real time in response to the tuning, without additional EM simulations.

XXXI Parametric Simulation

You can access the parametric analysis form either by the main menu**>Tools>Parametric Analysis**, or the button  , or the same icon under the Data Browser
It allows to define a list of parametric setups and to run a set of simulations, one for each parametric setup

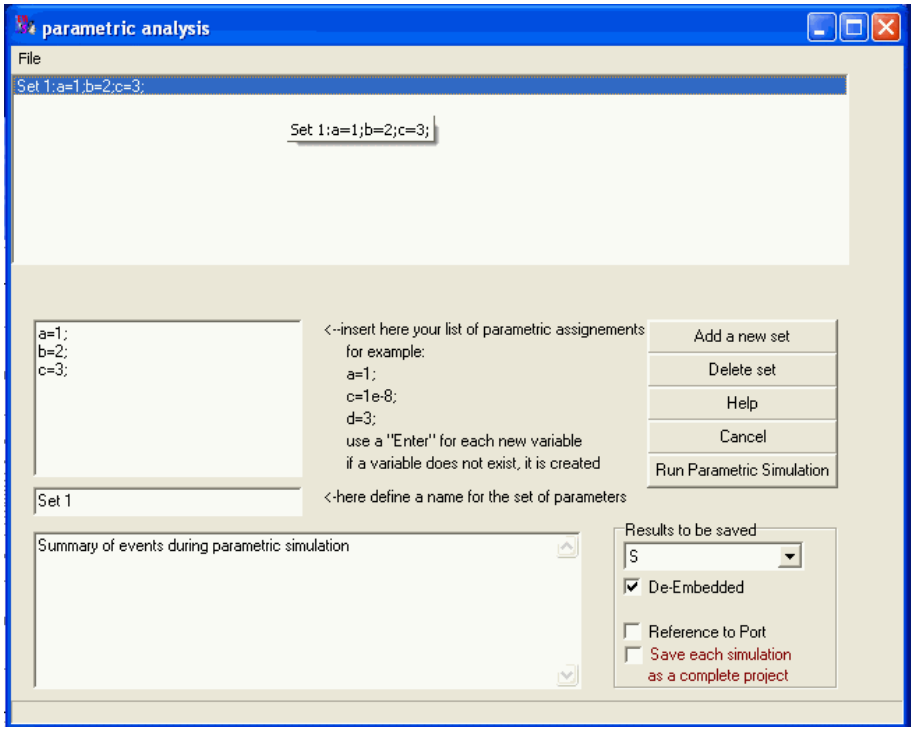


Fig.77: Parametric simulation panel

Insert in the edit box ("**insert here…")** the name of the variable and its value, e.g. "a=1;". Remember not to insert spaces, and end the definition by the symbol ";". Press Return (or Enter) to add additional variable definitions.
If a variable does not (yet) exist, it is created with the assigned value. Then select a unique name for such a set of variables, and enter it in the proper editbox ("**<- define here…**"). Then click on "**Add a new set**" button: at this point in the upper list it will appear a new item, showing the name of the set, and its content. To delete a defined set or a group of them, just click and press "**Delete set**" button.
When you click on "**Run Parametric Simulation**" a number of simulations will be performed, setting automatically the variables as required by your defined set. Resulting port parameters will be saved and re-imported in the project, so that you will have them available for plotting in any chart, and to make comparisons among different set of parameters. The kind of port to be saved is selected in the bottom of the form: choose which port parameters (S,Y,Z) and if either to work on De-Embedded or raw data. You can also request to create a separate, complete project, for each set of variables, by selecting "**Save each simulation as a complete project**". As a result, everything will be saved (currents, raw and calibrated data etc) for each set of data.

XXXII Post Processing: displaying network parameters and using the Data Browser

EM3DS includes complete and powerful post-processing abilities. Network plots are accessed either by clicking over **View/Graph menu**, or the **network parameters** button, or by the **Data browser** window. Data Browser has been added in ver. 5.0 in order to simplify post-processing. The Data Browser is basically a tree that can be expanded by clicking over the single leaves.
Some parts of the tree, such as the name of the graphs, may be edited, in order to assign them a new name: to this aim just click over the caption you want to modify.

By right clicking a popup menu allows adding more network plots, Smith Charts, Polar Antenna Plots, current plots, field plot (near and far), script macros written in Delphi Pascal, multimedia document and external data. External data are touchstone ascii files (see *to Probe Further*) and can be edited by the user.
Link to external data may be deleted by selecting the folder of the data and by accessing to the popup menu.

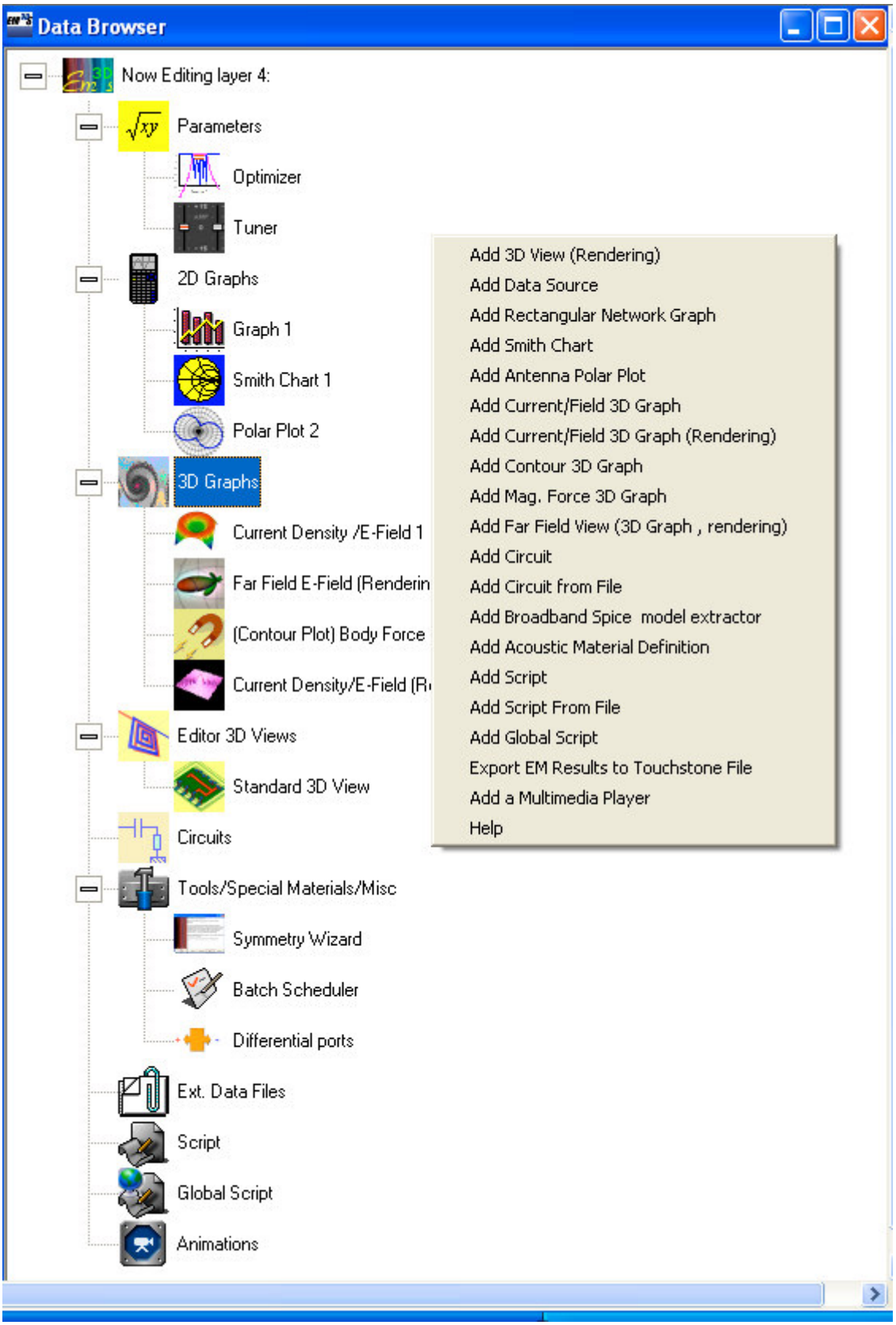


Fig.78:Data Browser

A plot is displayed by double clicking over the leave referring to it (e.g. **Graph1** in the picture above). Data files are reported in additional text windows whenever the user double-clicks over its folder. Information about settings, names and number of graphs is stored in a ".GOS" file, (Ascii file).

By any 2D chart the user can plot, compare and export network parameters. Network parameters

can also be exported via the menu **Tools/Export Touchstone**, and in this case settings about which kind of matrix exporting (if calibrated or not, whether S, Z or Y parameters) are taken from the first rectangular chart in the list. On the other end, if you are exporting from a rectangular network chart, the kind of exported matrix depends on the measurement being displayed (e.g., if the active trace is S, it exports S etc; if **de-embedded** is selected, calibrated results are exported). When moving mouse over the 2D chart, the lower bar displays the value of the point where the mouse is. Data referring to a structure being simulated (**Current EM Simulation** as Data Source) may be shown at the end of the calculation process, or during processing if **Refresh data during processing** in the pop-up menu is selected.

If de-embedding is available, the **de-embedded** check box allows showing and exporting de-embedded results. *Hence the user has always access to both raw and calibrated results. Moreover calibration plane may be shifted at the port plane by selecting "Reference To Port" check box.*

S-parameters are always exported normalized to 50 Ohm; however the user can *change the normalization* of the displayed parameters by **double clicking over the ports**. If the active trace in the 2D charts is the one of the em simulation, plot is automatically updated, otherwise just select the em trace, and force updating by e.g. switching temporary from dB to natural and vice-versa. The selected impedance also affects the visualization of the currents, whenever visualization is made "over load" and not "short circuit" (as default).

If a waveguide calibration set is enabled (see previous paragraph), S-parameters are displayed as normalized to TE10 characteristic impedance (default) or 50 Ohms.

Note: Whenever normalized to TE10, S parameters exported in Touchstone format are indicated as to be normalized to 50 Ohm. While this is not correct in principle (if you derive z or y-parameters from such a file you get wrong results), this expedient was needed for compatibility with some commercial software packages available.

You can plot several **traces** in the same plot: just right-click over the graph and select the additional menu item **Add Trace**. A new trace is made available and appears in the trace list. Any change you make -e.g. the kind of network parameters the user has selected- just affects the current trace, unless **Synchronize traces** is selected. In order to modify properties of a trace, just make this trace active from the list-box **traces**, and then perform your modifications.

By accessing to the popup menu, you can select **Zoom by mouse** or **Pan by mouse**: in this case, by dragging the mouse you can either zoom or pan your plot.

By default, parameters refer to the EM Simulation; however you can add several **Data Sources**, namely S, Z or Y Touchstone files. This way you can e.g. compare measured and calculated data or simply compare several simulations. Data sources added within a plot window are also added to the Data Browser and are available from any plot window. De-embed option is (possibly) available *only* for the current EM-Simulation. The user can delete traces (*Data Sources are only deleted by the Data Browser*).

The **Delete Trace** and **Edit Trace** menu only affect the selected trace.

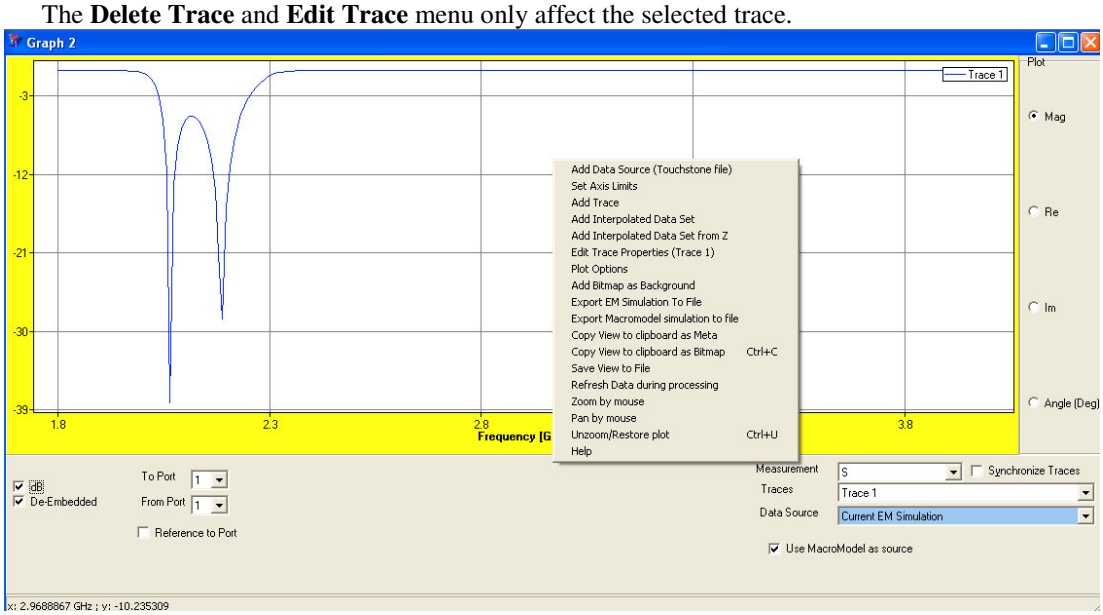


Fig.79: Displaying network parameters

If a macromodel is available, when the data source is the current EM simulation, you can select "**Use Macromodel as source**", and the macromodel response is displayed. The macromodel response is updated in real time when you modify the macromodelled parameter. The response of the macromodel for the current parameter can be exported by the popup menu **Export Macromodel simulation to file**, creating a standard touchstone file.

Note that if using the macromodel, the "De-embedded" checkbox has no effect.

Link to Data Sources (and not data sources) are saved within the EMS file: if the editor is unable to find a given path, it looks for the requested file in the same directory of the current EMS file.

From any chart you can create an Interpolated Data Set (**Add Interpolated data Set**), namely a rational interpolation of the input data; the new data set is added to the list of Data Sources, and saved in the HD with unique name. See the paragraph about SmartFIT for more information.

By clicking over the menu item **Set Axis Limits** it is possible setting the vertical (y) and horizontal (x) axis limit of the plot.

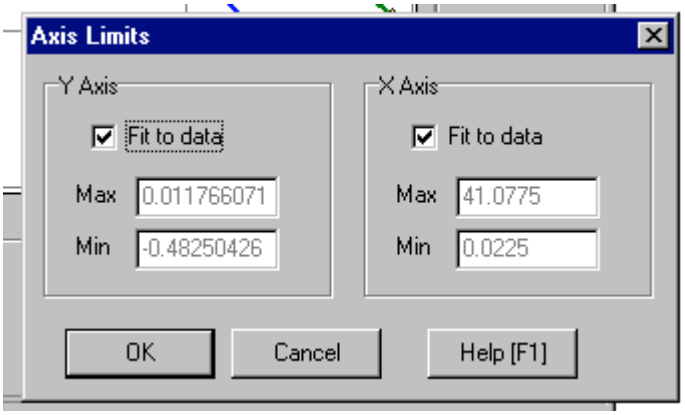


Fig.80:Axis Limits

By the **Fit to Data** check box limits are automatically sized.

Set the trace properties for the currently selected trace by clicking over **Trace Properties**. Double click over the **trace color** in order to set the trace color.

Select the desired **marker**, its **size** (in points), its border color (**Marker color**) and its **filling color.**

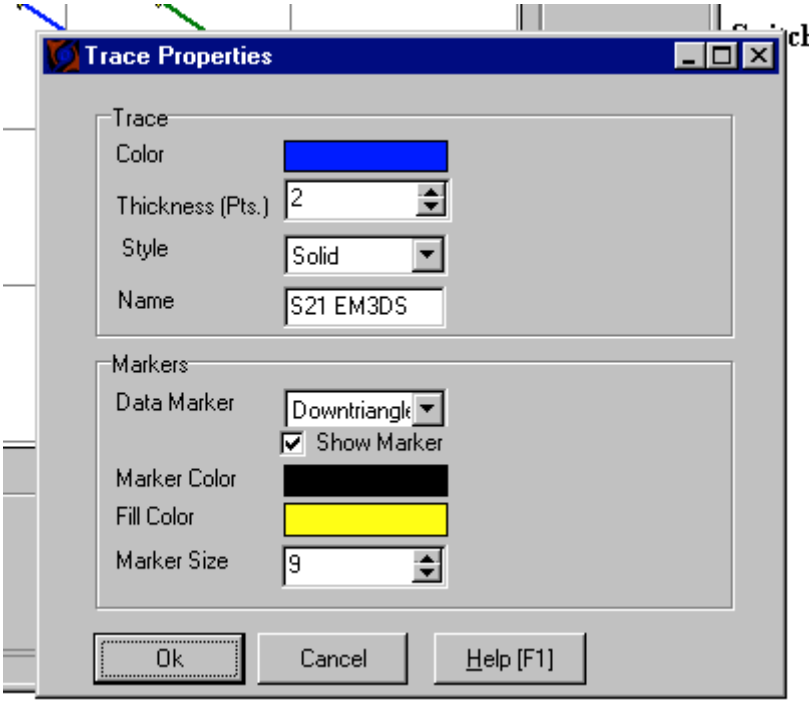


Fig.81:Trace Properties

Set the plot area properties by clicking over **Plot options** or by double clicking in the plot area:
x steps and **y steps** are the number of horizontal and vertical grids, respectively; grids are only displayed if the **Show x grid** and/or **Show y grid** boxes are selected.

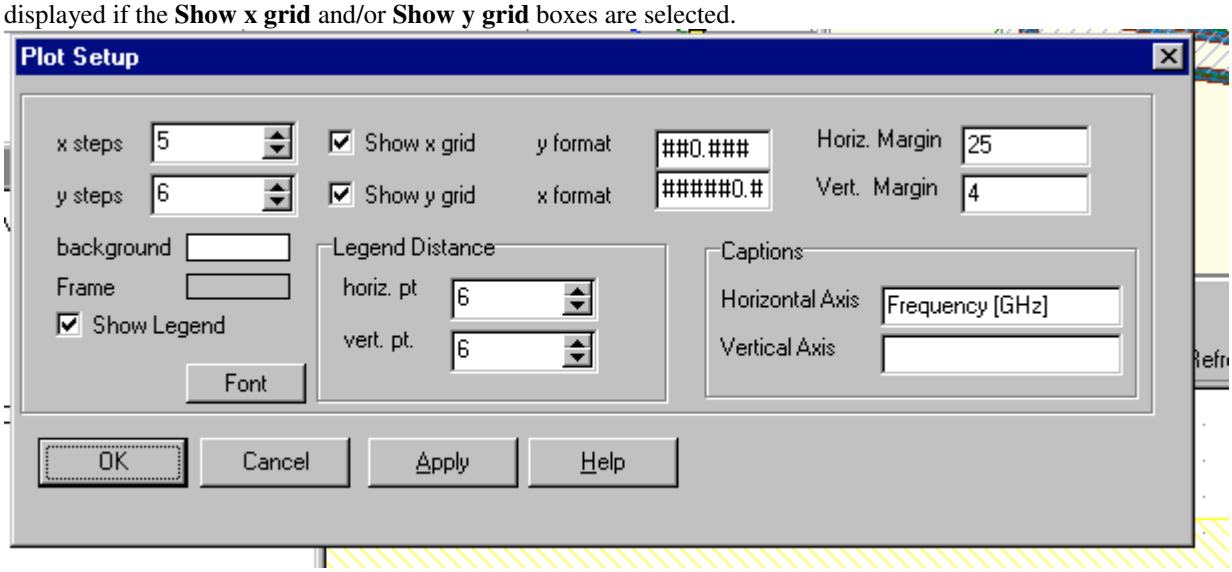


Fig.82:Plot Options

Background and Frame let you specify respectively the area and frame color.
Legend Distance is the distance from the upper right corner, where the legend (if any) is displayed.
Horiz. Margin and Vert. Margin is distance from the graph in pixels used to define the frame: they can be changed in order to keep space for the **captions**
X Format and **y Format** allow to specify the format number for abscissa and value. The value of **x/y Format** is a string that encodes the formatting of numeric data using the specifiers in the following table:

Specifier	Represents
0	Digit placeholder. If the value being formatted has a digit in the position where the "0" appears in the format string, then that digit is copied to the output string. Otherwise, a "0" is stored in that position in the output string.
#	Digit placeholder. If the value being formatted has a digit in the position where the "#" appears in the format string, then that digit is copied to the output string. Otherwise, nothing is stored in that position in the output string.
.	Decimal point. The first "." character in the format string determines the location of the decimal separator in the formatted value; any additional "." characters are ignored.
,	Thousand separator. If the format string contains one or more "," characters, the output will have thousand separators inserted between each group of three digits to the left of the decimal point. The placement and number of "," characters in the format string does not affect the output, except to indicate that thousand separators are wanted.
E+	Scientific notation. If any of the strings "E+", "E-", "e+", or "e-" are contained in the format string, the number is formatted using scientific notation. A group of up to four "0" characters can immediately follow the "E+", "E-", "e+", or "e-" to determine the minimum number of digits in the exponent. The "E+" and "e+" formats cause a plus sign to be output for positive exponents and a minus sign to be output for negative exponents. The "E-" and "e-" formats output a sign character only for negative exponents.
'xx'/"xx"	Characters enclosed in single or double quotes are output as-is, and do not affect formatting.
;	Separates sections for positive, negative, and zero numbers in the format string.

The locations of the leftmost "0" before the decimal point in the format string and the rightmost "0" after the decimal point in the format string determine the range of digits that are always present in the output string.
The number being formatted is always rounded to as many decimal places as there are digit placeholders ("0" or "#") to the right of the decimal point. If the format string contains no decimal point, the value being formatted is rounded to the nearest whole number.

If the number being formatted has more digits to the left of the decimal separator than there are digit placeholders to the left of the "." character in the format string, the extra digits are output before the first digit placeholder.

The following table shows the effect of various format strings:

DisplayFormat	Value	Result	Comment
#.##	12.2	12.2	Note extra digit to right of decimal still appears
#.00	2.5	2.50	Note extra zero: field will always show two decimal places
00.##	.006	00.01	Note extra 0s tor right of decimal point and rounding to two decimal places.

To allow different formats for positive, negative, and zero values, the format string can contain between one and three sections separated by semicolons.

- One section: The format string applies to all values.
- Two sections: The first section applies to positive values and zeros, and the second section applies to negative values.
- Three sections: The first section applies to positive values, the second applies to negative values, and the third applies to zeros.

If the section for negative values or the section for zero values is empty, that is, if there is nothing between the semicolons that delimit the section, the section for positive values is used instead.

If the section for positive values is empty, or if the entire format string is empty, the value is formatted using general floating-point formatting with 15 significant digits. General floating-point formatting is also used if the value has more than 18 digits to the left of the decimal point and the format string does not specify scientific notation.

The graph on the screen may be copied into the clipboard as Metafile, or as Bitmap (in the first case it may be edited by tools as Microsoft Word, and when changing its size no loss of details occurs; in the second case the graph is stored bit by bit). The graph may also be saved as graphic file (BMP or EMF) in file. The em simulation may be exported to file as touchstone, depending by the current setting.

In ver 6.1.3 it has been added the ability to import BMP and JPG files to be used as background of your graph (**Add Bitmap as Background**). This is especially useful to compare calculated results with data in the literature, by means of some scanning of available graphs, and setting appropriately the options and limits of the graphics in EM3DS.

In rectangular plots displayed Parameters are
S: Scattering Matrix
Z: Impedance Matrix
Y: Admittance Matrix
Quality Factor Q defined according to Im(Yii)/Re(Yii), where i is the selected port
Group Delay: it is defined as

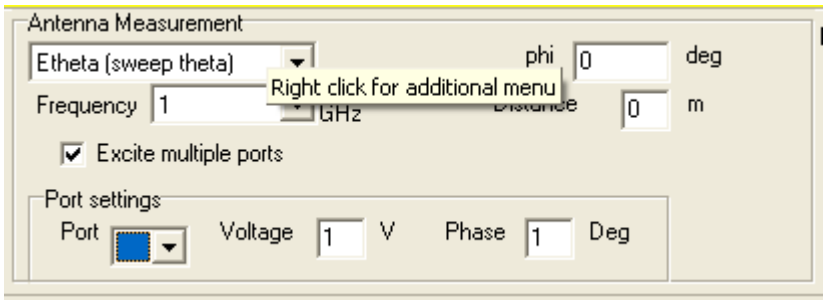
$$-\frac{\partial \phi_{ij}}{\partial \omega}$$

where ϕ_{ij} is the phase of S_{ij} parameter. The quantity is expressed in time units (nsec). Note that the derivative is actually an incremental ratio, and data have to be available at least at two frequency points.

Phase Difference: useful function to evaluate the phase balance of hybrids: phase difference is in DEG and represents the phase difference between S parameters, according to the selected ports.

Losses: for a selected port p in an N-port it evaluates $\sum_{k=1}^{N}\left|S_{p,k}\right|^2$. If a device is lossless, such a value is 1, namely 0 dB. For a lossy device, its value in dB represents the fraction of power dissipated by the device due to several loss factors (dielectric losses, conductor losses and possibly radiation losses), when fed from port p.

Antenna Measurement: This measurement allows to see the calculated far-field, if a sense layer was previously added. Basically, a new panel is made available



The user can plot either Etheta (Eθ) or Ephi (Eϕ), as function of theta (and a fixed phi) or of phi (and a fixed theta). Theta and phi are angular coordinates in spherical coordinates (see details and underlying hypotheses in the antenna section). Usually the antenna is excited at one port, selected by the user in the "**To Port**" list, while the remaining ports are shorted to ground. However the user can

check the **box "Excite multiple ports**", and in the new panel to select for each port voltage amplitude and phase, in order to plot far field of antenna arrays.

In this chart field is not normalized, so that the user can specify the distance (note: no checks against meaningless distances –e.g. not satisfying the "far" field requirements- are performed in the present version). If "0" is inserted, the distance factor e^{-jkr}/r is simply replaced by 1.

Custom Measurements: you can create your own measurements; see next section.

TDR Time Domain: it enables the panel for the TDR Time Domain Reflectometry. When selecting this checkbox the new panel is made available (see the image below).

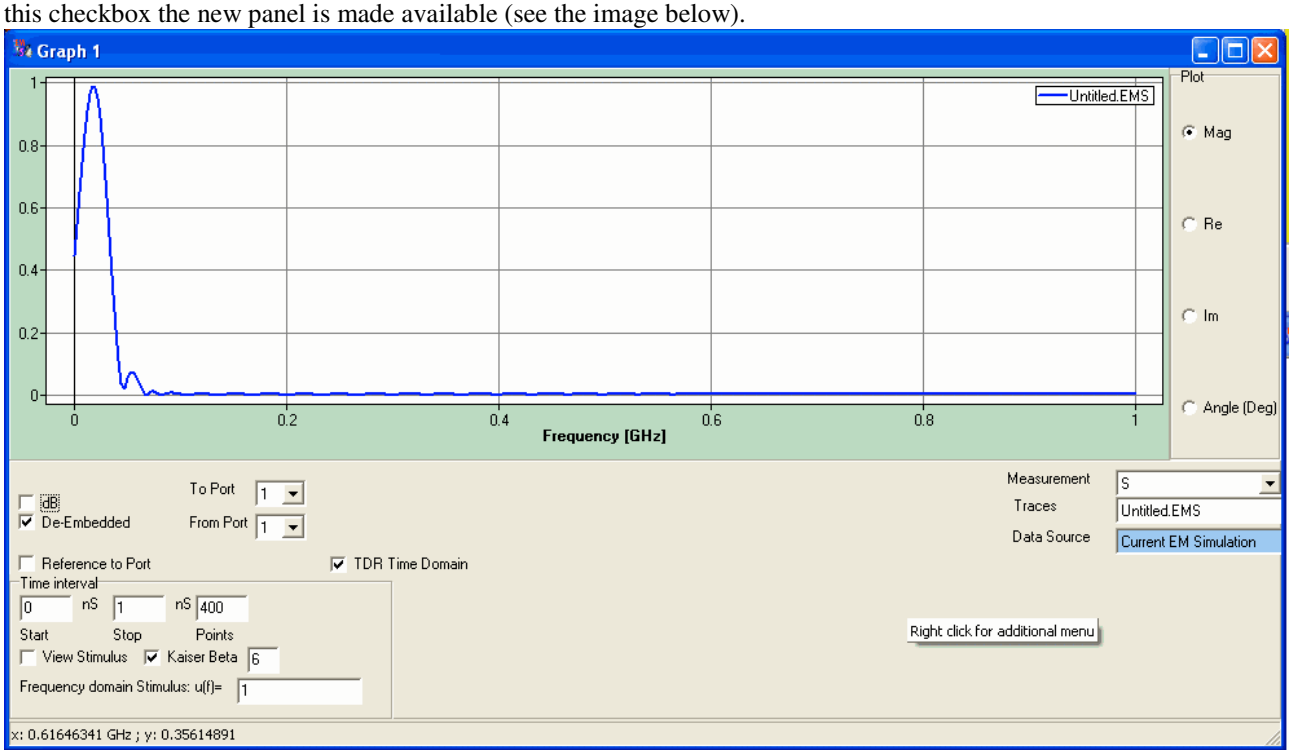


Fig.83:Additional Time Domain Panel

Here one enters the time interval, the number of plotted points in time: there are no restrictions. By default the stimulus is assumed to be 1 over the whole frequency band: if frequency points are equispaced, this equivalent to a finite pulse, whose width is determined by the upper frequency point (1/fmax) and which repeats after a time correspondent to the lower frequency point (1/fmax). However it is possible to enter any frequency dependent expression to create your own stimuli. The actual stimulus is displayed when checking "view stimulus". A Kaiser Beta windowing is applied by default in order to reduce the ripple induced by the finite bandwidth used. See the document the section Time Domain in EM3DS for more information

A subset of the above commands are available for Smith Charts.

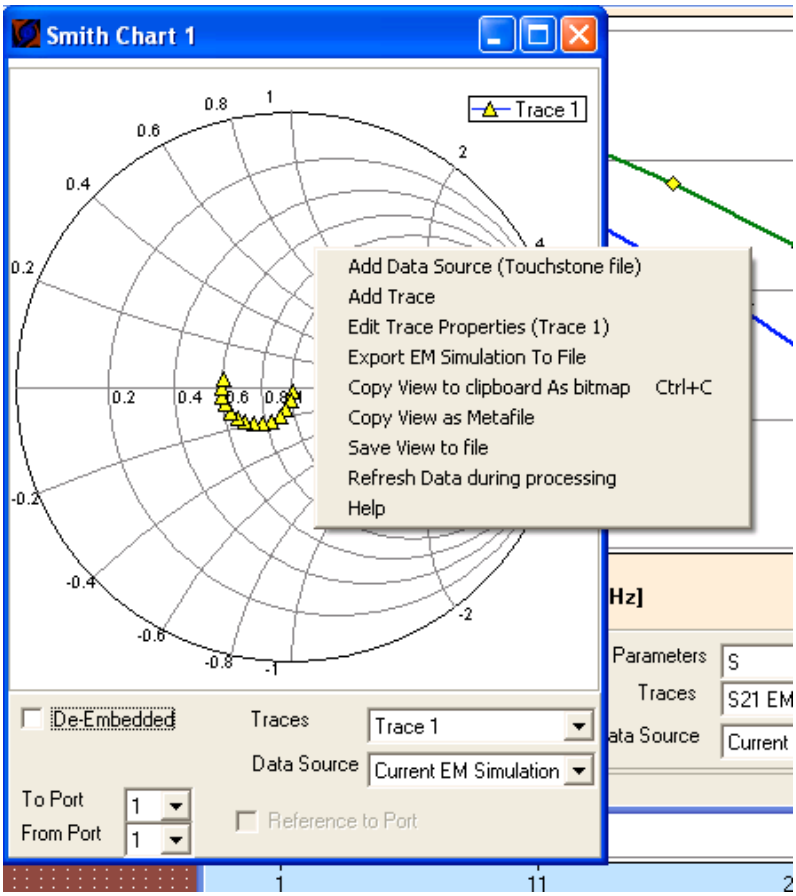


Fig.84:Smith Chart

An additional chart for antenna measurement is the Polar Plot.

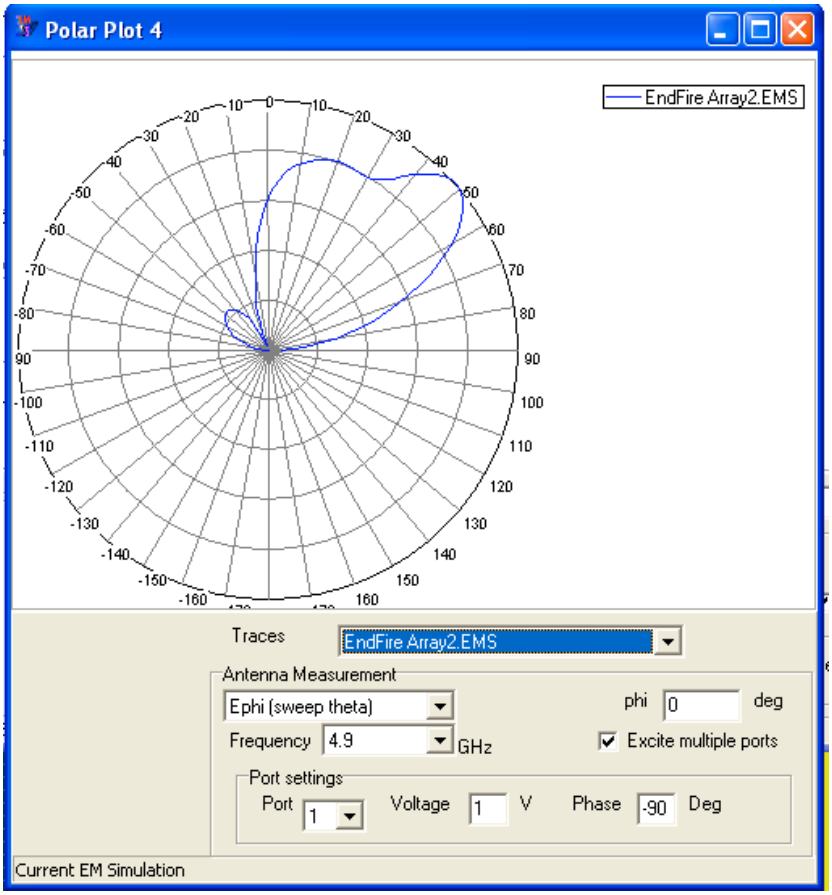


Fig.85:Antenna Polar Plot

Even in this case, the antenna plot is available only if a sense layer was added to the project. In particular, in the present version, it is plotted the far field (one polarization) *normalized* to its maximum value in the displayed chart.

Several of the above features are also available when displaying **De-embedding Line Parameters** (menu **View/ De-embedding Line Parameters**). In fact when de-embedding information is available, you can display the characteristic parameters of the feeding lines, namely characteristic impedance, propagation and attenuation constant etc.
Data refers to left, right, top and bottom ports, whenever some reference plane has been defined.
Results about the displayed parameter can be exported as text data file by clicking over the menu **Export to file as text**, appearing when right clicking over the graph.

A few restrictions apply:

- the de-embedding plane distance should not exceed λ/4; if this is the case, the displayed parameters are unreliable. Some minor tests are performed in order to provide a warning message, but the user should check after the simulation. *Note that even if this condition is not respected, the de-embedding algorithm does not fail*, as de-embed does not assume any pre-defined topology: only the "line parameters" will probably be useless.

Additionally the user can edit the **trace properties** and the **plot** area **option**. In this graphs only one trace is currently allowed.

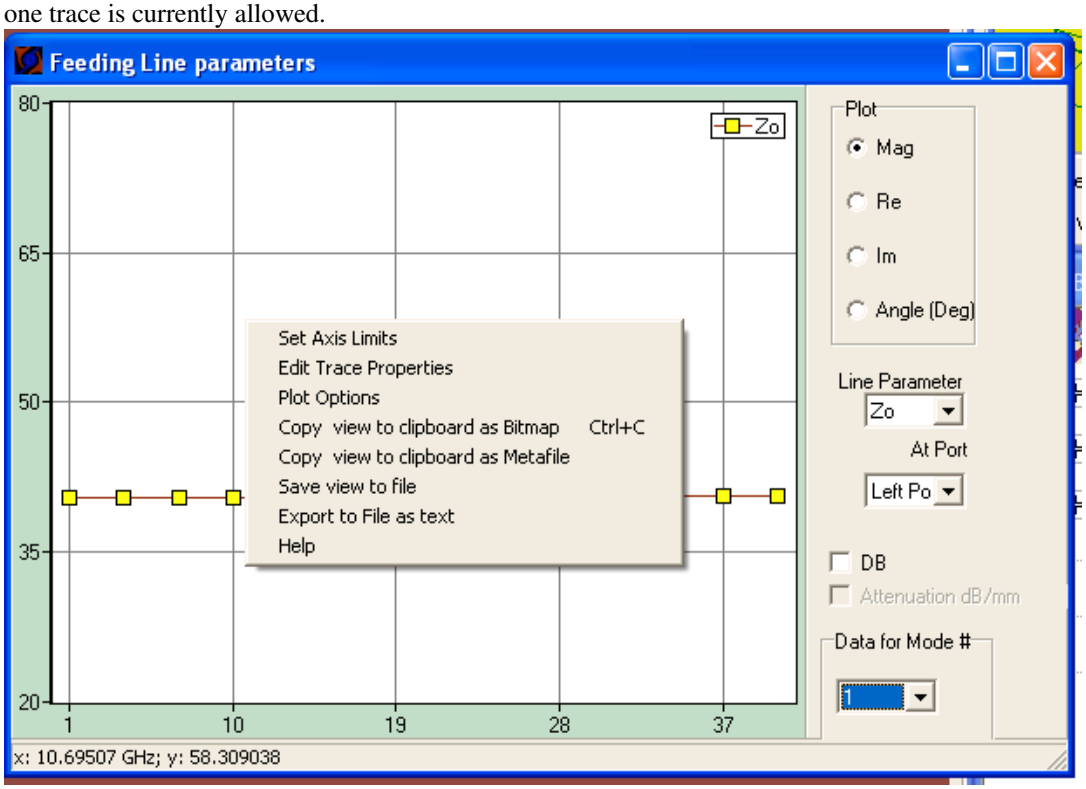


Fig.86:Feeding line parameters

When more than one port is define at the same side, characteristic impedance for any of the quasi-TEM modes is calculated, and can be displayed by selecting "Data for Mode #". However in these conditions, the propagation constant and the effective permittivity are only an average value over the different modes.

***To Probe Further**: The Touchstone Data File Format*

EM3DS imports and exports data file in Touchstone format. Touchstone files are ascii files, having a particular extension: for example ".s2p" means a 2-port network parameters representing Scattering matrix. Em3ds handles S,Z and Y network parameters with an arbitrary number of ports. When exporting a simulation form a network graph, the current visualization settings are used in order select which parameter to export. Hence if z parameters of the de-embedded structure are displayed, file of z-type is exported including calibrated results.

Comments are inserted in files by an exclamation (!) point. Comments persist until the end of the line.

Before data, header has to be specified. The header is indicated by a "#" character. Hence the accepted

headers are

HZ/KHZ/MHZ/GHZ/THZ S/Y/Z MA/DB/RI [R x]
Where the "/" character is used to separate different choices, and the "[ ]" brackets indicate an optional entry. Coming to the meaning of the entries:

#
Beginning of the header
HZ / KHZ/| MHZ/ GHZ / THZ
Specifies the frequency units of the data file (choose one)
S/ Y / Z
Specifies the parameter type of the data file (choose one)
MA / DB / RI
Specifies how the complex data are presented (choose one)
[R x]
x is a real number that specifies the reference impedance (optional)

Example Headers:

MHZ S MA R 50
GHZ S DB
HZ Y RI
Notes
1) The reference impedance is only needed when the parameter type is S (scattering coefficients).
2) MA and DB means the complex data are in polar form (mag, angle), the angle of which is always in units of degrees; DB further specifies that the magnitude has been transformed via 20*Log(mag). RI means the data are in rectangular form (real, imag).
4) The network parameter matrices are in row major order, except for two port matrices, which are in column major order.
5) Each network parameter is a complex number that is read as two sequential real numbers.
6) Each line may contain a maximum of four network parameters (8 real numbers). If the matrix contains more than four network parameters per row, i.e. it is larger than a four port, then the remaining network parameters are continued on the following line.
7) Each row of the network parameter matrices begins on a new line.
8) The first row of a network parameter matrix is proceeded by the frequency at which the data was generated.
9) The filename extensions, by convention, are s1p, s2p, ..., s9p, s10, s11, ..., s99.
The network data syntax is:
<freq point 1> <row 1>
[<row 1 cont.>]
<row 2>
[<row 2 cont.>]
.....
<row n>
[<row n cont.>]
...
<freq point m> <row 1>
[<row 1 cont.>]
<row 2>
[<row 2 cont.>]
.....
<row n>
[<row n cont.>]

where
m Number of frequency points
n Matrix size

Example file for a two-port:

Example Data File Using Format

GHZ S RI R 50
!Freq S11 S21 S12 S22
f1 ReS11 ImS11 ReS21 ImS21 ReS12 ImS12 ReS22 ImS22
f2 ReS11 ImS11 ReS21 ImS21 ReS12 ImS12 ReS22 ImS22
f3 ReS11 ImS11 ReS21 ImS21 ReS12 ImS12 ReS22 ImS22

XXXIII Post Processing: Creating Custom Measurements

This form is accessible either by the main menu>Tools>Measurement Editor, or the button 

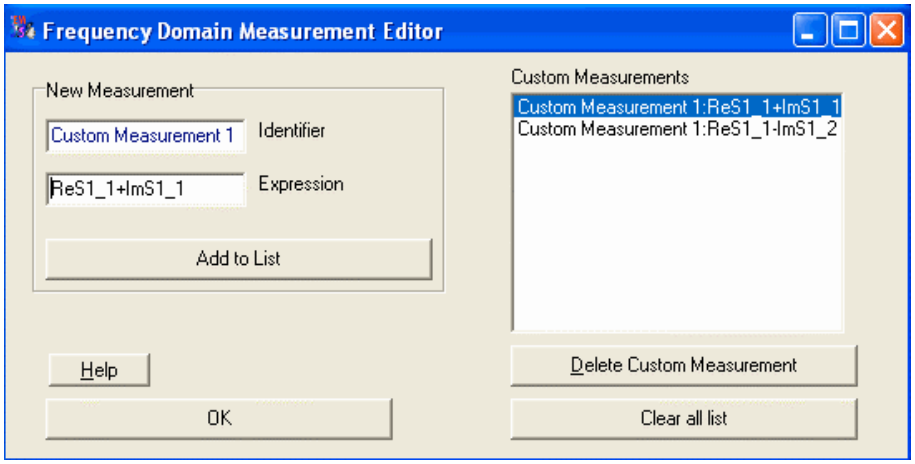


Fig.87:Custom Measurement Editor

The Measurement Editor, allows to create your custom measurements; any new measurement will be available in all Charts

Create a unique **Identifier** and enter the **Expression** defining the new measurement. The "Expression" has to be created using the reserved symbols ReS*fromport_toport* and ImS*fromport_toport:* for example ReS1_1 indicated the real part of S11. The expression can make use of any of the built in functions. When clicking OK the list of measurements is updated and any chart will have available the new measurement. The list of custom measurements is stored in a text file named CustomMeas.prf, automatically loaded by EM3DS at the startup. In order to remove a measurement, select it by mouse, and click Delete. Multiple selection is possible by keeping pressed the SHIFT on the keyboard.

Note

The expression correctness is only checked when it is used, namely when evaluating the measurement. If for example you make reference to two ports and you apply the measurement to a data source involving only one port, an exception is raised by the expression parser.

XXXIV Post Processing: Time Domain in EM3DS

Time Domain Reflectometry is a powerful technique, allowing to determine the origin and the position of discontinuities in transmission lines, as well as to study transients in a circuit. Usually it relies on the anti-Fourier Transform of the frequency data, in order to obtain the time-domain evolution of a signal, and programs implement this procedure practically by using FFT algorithms. This poses some restriction: for example frequency points have to be equispaced. When processing data which are not equispaced this imposes some kind of interpolation, with all the precision issues related to this choice.
In EM3DS it is implemented a different point of view. When looking at the frequency response of a system at a single frequency fo, let's say S(fo) we are basically implying a term exp(j ω t), so that the actual time domain response is s(t)=Re(S(fo) exp(j ωo t)): this is the response to the stimulus cos(ω t). If we have two frequencies f1 and f2, considering the linearity of the systems involved in our analysis, we can evaluate by superposition of effects s(t)=Re(S(f1) exp(j ω1 t)+ S(f2) exp(j ω2 t)), which will be the response to the stimulus cos(ω 1t)+ cos(ω 2t). There is no restriction on the position and the number of frequencies. Hence continuing we get the general case

$$s(t) = \mathrm{Re} \sum_{f_i} \left(S(f_i) e^{j 2 \pi f_i t} \right)$$

as response to the stimulus

$$s(t) = \sum_{f_i} \mathrm{Re}\left(e^{j2\pi f_i t}\right)$$

Of course, if frequencies are equispaced, include the DC term, and the minimum frequency is equal to the interval between the different frequency points, we are basically doing a Fourier transform. The key is that the EM3DS choice let you explore more possibilities, taking under control what is going on.

In the case of a Fourier Transform, you are seeing in time the response to a pulse with finite bandwidth, basically a convolution between a pulse and a sinx/x function. The larger the upper frequency, the narrower is the pulse. The lower frequency define the "range", namely the maximum time after which you get a replica of your pulse (1/fmin).

In order to reduce the ripple, it is possible to weight the spectral content by some windowing function, making smoother the transition in frequency at the edge of the band. EM3DS implements internally the Kaiser-Beta function, so that, when Kaiser Beta is enabled, the response is

$$s(t) = \mathrm{Re}\sum_{f_i} \left(K(f_i)S(f_i)e^{j2\pi f_i t}\right)$$

The ripple is reduced, but the stimulus results larger in time. The more the Beta factor, the smoother is the response.

Actually EM3DS let you define also your own weighting function u(f), so that you can evaluate the response

$$s(t) = \mathrm{Re}\sum_{f_i} \left(u(f_i)S(f_i)e^{j2\pi f_i t}\right)$$

to the stimulus

$$s(t) = \mathrm{Re}\sum_{f_i} \left(u(f_i)e^{j2\pi f_i t}\right)$$

You can always see the stimulus and the response, which are evaluated in real-time.

When looking the TDR response in a chart, remind that by default S parameters are normalized at 50 Ohms: your transmission line has 50 Ohm both as source and as load impedance, unless you double click over the ports of your project and you specify different normalization impedances.

Note that the TDR option can be applied also when displaying current/field distributions (any possible plot, with the exception of contour plots), and that time animation showing a transient can be generated (even though this procedure is quite slow!). However when looking at the TDR in current distribution remind some assumptions:

1. the source impedance is always 0 (Y parameters are used in this calculation), so that if a wave reaches the source, it is reflected back completely.
2. The load impedance is by default 0, but it can be changed to the port impedance value, much like is done for the aforementioned charts

Finally, when creating time-domain animations of the current distributions, remind to select in the proper panel "**set fixed range**": when this is not done, the plot is automatically, adjusted to show variations in currents; any frame in the time animation would have a different scaling, resulting in a useless animation.

XXXV Post Processing: displaying volume current distributions in surface plots

At the end of processing, by clicking over the **View/3D graph** menu **Show 3D Plot of the currents** button or double-clicking over data browser **Current density**, a window with the current distribution within the conductor/dielectric brick volume is displayed. Currents are displayed layer by layer (hence if layer 1 is selected only currents flowing across objects in layer 1 are displayed).

The parameters appearing at the bottom of the window are

Number of Samples: Sampling points over the box used to plot the surfaces: if the number of points is insufficient for a given structure, details may be lost or visual aliasing may appear.

Animate: (Animation) Animate current view vs. frequency (magnitude) or vs. phase (using phasor technique) at the specified frequency or in time. The time animation allows to get images about transients, and makes use of the assumptions explained in the "Time Domain in EM3DS" section and it is quite time-consuming. When selecting Time in the animation, time-domain

current is shown and an additional panel is made available

Now Showing reports the time instant of the current display. Here one enters the time interval, the number of plotted points in time: there are no restrictions. By default the stimulus is assumed to be 1 over the whole frequency band: if frequency points are equispaced, this equivalent to a finite pulse, whose width is determined by the upper frequency point (1/fmax) and which repeats after a time correspondent to the lower frequency point (1/fmax). However it is possible to enter any frequency dependent expression to create your own stimuli. The actual stimulus is displayed when checking "view stimulus". A Kaiser Beta windowing is applied by default in order to reduce the ripple induced by the finite bandwidth used. The stimulus impedance is always 0 Ohm, while the load impedance can be modified by double clicking over the ports of your project and by specifying different normalization impedances. See the previous section about Time Domain in EM3DS for more information

In **Frequency Animation** the first 2D plot window of network parameters is brought to front, and a circle over the plot indicates the frequency being displayed.

Steps: (Graph Options) lets you select the number of frames that are used in the animation vs. phase, starting with the **starting phase**.

When more than one expanding function is used along the y (conductor thickness) direction in the current layer, the user can select a **Slice in layer** (Layer), namely one of the "slices" used to subsection the conductor over the thickness. The current distribution refers to such subsection, so the user can also investigate e.g. details of the "skin effect". Slices are numbered starting from the floor of the conductor. (see also pp.5-6)

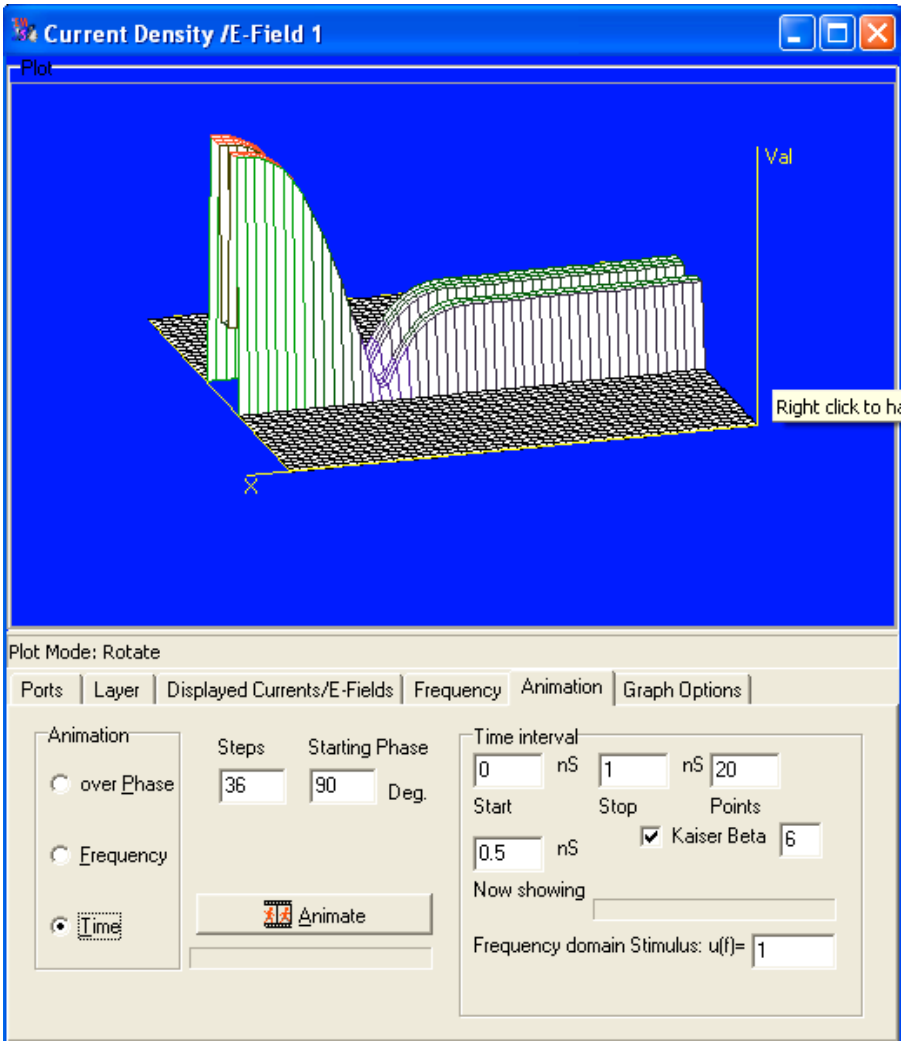


Fig.88: Surface graph of the volume current distribution

Layer: display currents for the selected layer

Current Component: (Displayed Currents) select if displaying Jx, Jy, Jz or the total current

density.

Termination: (Ports) it is activated for 2 or more ports; if **Short** is selected, short circuit currents are displayed, otherwise currents are calculated for the selected port impedance. In order to change this impedance, just double click over the port.

Values: (Graph Options) shows the max and min values for the displayed plot; these values are automatically updated, unless the **Set fixed Range** box is selected.

Keep Aspect Ratio: (Graph Options) if selected, the x-z dimensions are displayed respecting their real ratio (default)

Right click over the plot to handle it, namely to rotate it, move it or zoom. Operations are performed by dragging mouse over the graph. You can hide the lower control panel by deselecting from the popup menu **Display Setup panel**.

Operations are performed by dragging the mouse over the image.

The current view may be exported to file as Windows BMP or EMF (Enhanced Metafile [vectorial]). By the same token, view may be copied into clipboard.

In the **Animation Window**, also accessible by menu View, you can select start and stop frames, and save animation as AVI or GIF files.

The size of the surface plot, is changed by changing the window size. The animation produced has the same size of the surface plot.

From the **Data Browser** or from the menu View, you can add several standard surface plots. You can also add advanced 3D plots, involving lights and rendering, very effective for presentations.

Note that this kind of graph strongly interacts with your graphic card, and may not work properly in any PC.

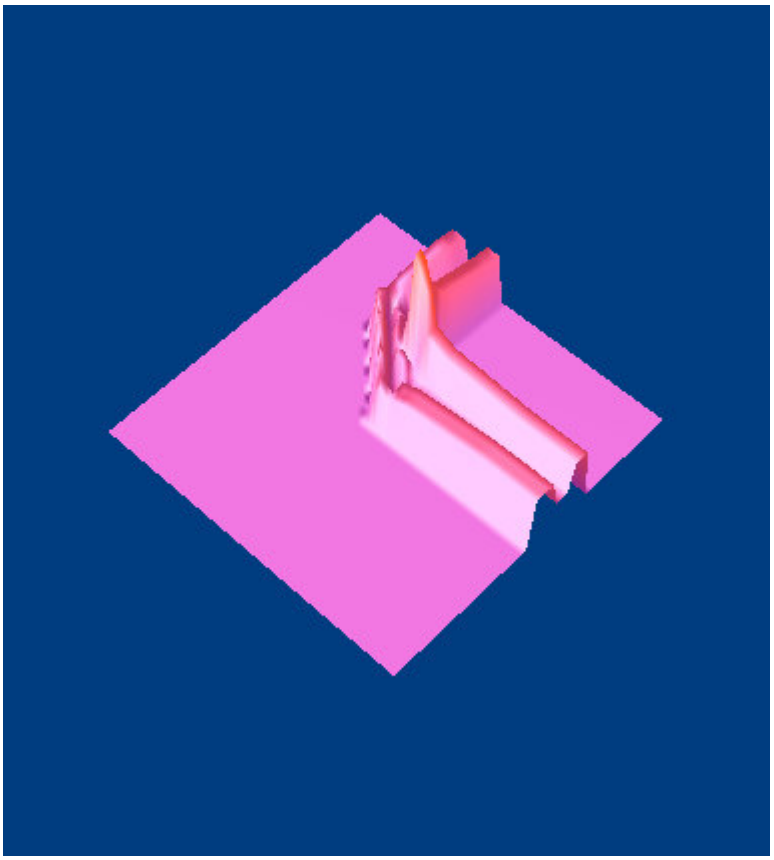


Fig.89: Surface graph of the volume current distribution with rendering

XXXVI Post Processing: displaying volume current distributions in contour plots

EM3DS adds one more representation of the current density, namely a contour plot. To this aim just select from the data browser **Add contour Graph 3D** or click over the icon if your have already added such a graph. A window with the current distribution within the conductor/dielectric brick volume is displayed.

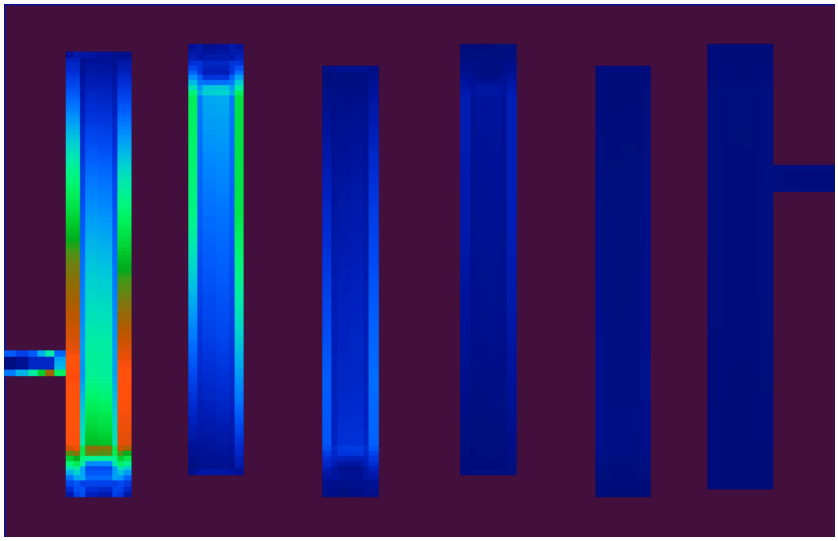


Fig.90: Contour Plot of the volume current distribution

Currents are displayed layer by layer (hence if layer 1 is selected only currents flowing across objects in layer 1 are displayed), but if **All layers** is selected, every layer, starting from the lower one, is displayed in the same picture. Note that in the latter case, if more slices are defined for a layer, only the lower one is displayed. In order to control the graphic appearance, select from the popup menu **Display Setup Panel**.

Several parameters are common to the standard 3D surface plot (see previous section). By moving mouse across the area, the current density value is displayed in the status bar (Note: this will not work if All Layers is selected). Also a legend may be displayed by selecting **Show Legend**. You can drag and drop by mouse the legend across the image. In copy and paste operations, legend will not appear. Instead you can copy and past directly legend.

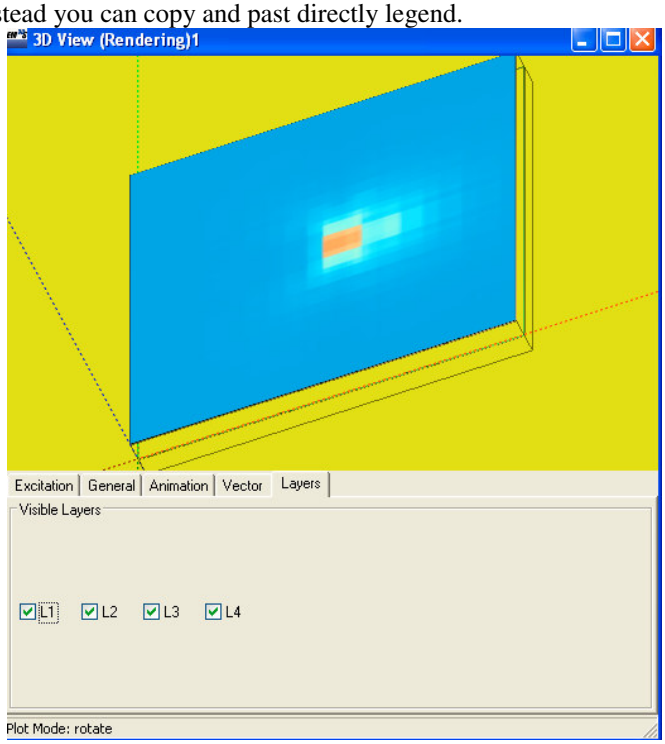


Fig.91: Contour Plot of the volume current distribution over the 3D structure with rendering, including a top sense later.

The same kind of graph is available directly over the ***3D views with rendering*** of the structure. In order to show currents just select "**Show Currents**" from the popup menu of the 3D view. If a **sense layer** was added, the quantity displayed in that layer is the E-Field Distribution. Also there will be a legend for E-field (if a sense layer is applied) or for currents. Objects in each layer can be selectively hidden: this is specially useful when e.g. the sense layer hides the other objects. The sense layer is always layer1.

As the drawing is a quite lengthy process, by default only currents over the upper surfaces are shown. However by selecting **Display Setup Panel**, a lower panel is shown where possibly deselect "**Show only on up- Surf.**". Additionally the currents being displayed, its frequency and other parameters may be modified. Finally current distribution may be animated.

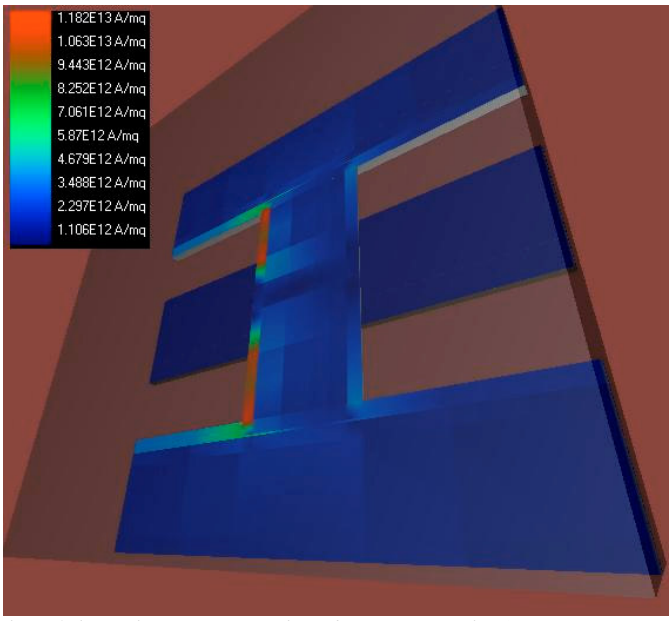


Fig.92: Contour Plot of the volume current distribution over the 3D structure with rendering

Additionally, vector currents may also be displayed in the same plot. Items in the **setup panel** are common to those shown in other plots of the currents. Additionally, number and position of the arrows is selected according to the "**num. sampling cells**". A value of 20 for x, for example, indicates that a cell is a/20, being a the box size: the viewer fits any object by cells with this size, and in the middle of each cell places a vector showing direction and magnitude of the current in that point.

You can also see and create time-domain animations of the current/field distribution. Menu is identical to what described in other current plots. See also the section about Time Domain in EM3DS for more information, or the tutorial about time domain.

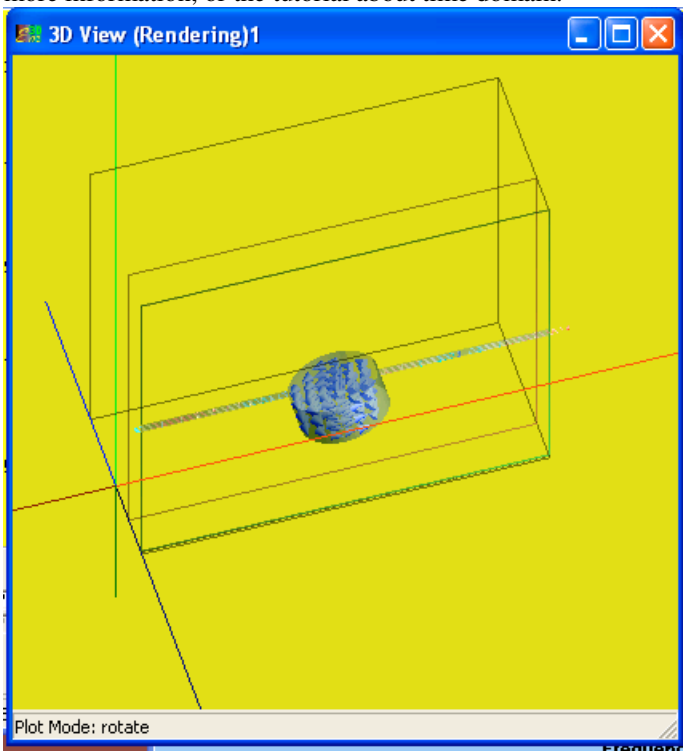
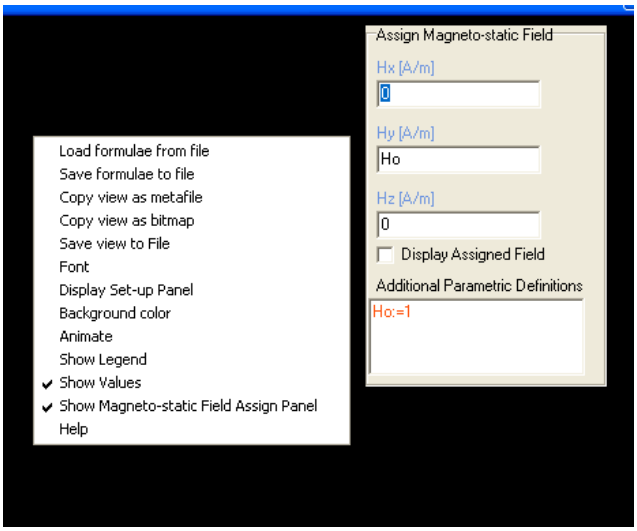


Fig.93: Vector plot

From the **Data Browser** you can select **Add Mag. Force**. A window similar to the one described in the previous section is created.

In this window a contour plot of the approximate body force distribution due to an assigned magnetostatic field is displayed. The underlying assumption is that the assigned magnetic field is not distorted by the existing structure, or that the expression the user inserts already takes into account for that. Expressions for magnets having simple shapes are known. The user inserts an expression for static H **Assign Magnetostatic Field** panel:



Expressions may be parametric, and the user can define parameters in the **Additional Parametric definition** box. Expressions can make use of a complete list of functions (see Appendix for built in functions). They can make reference to x, y and z as spatial co-ordinates in meters. X=0 Z=0 is the left-hand upper part of the screen (and of the box). Y=0 is the floor of the box, where usually ground is located. Note that all quantities in this window are metric and not imperial units!

The entered formulae may be saved to text files by the popup menu **Save Formulae to file**. Being text files, they may also be edited externally. Formulae may be reloaded by clicking over the item **Load Formulae from file.**

Any time the user leaves an edit box, the graph is updated: the tool just calculates $\mathbf{B}=\mu\mathbf{H}$, by using the magnetic permeabilities of the dielectric stack, and then calculates $\mathbf{F}=\mathbf{J}\times\mathbf{B}$ [N/m^3] where **J** is the RF volume current distribution calculate from the em analysis. This operation is done on the floor of the selected current slice.

The assigned magnetostatic field may be displayed by selecting **Display Assigned Field** in the **Assign Magnetostatic Field** panel. Field at a given co-ordinate in the xy or xz or yz plane is shown, according to the selection in the Set-up (or control) panel.

The control panel is only displayed if **Display Setup panel** in the popup menu is checked.

From the popup menu you can:

load formulae from file: load text containing desired expression

save formulae to file: save text containing expression

copy view as metafile: copy into the clipboard as EMF (note the legend is not copied).

copy view as bitmap: copy into the clipboard as BMP (note the legend is not copied).

copy Legend to the Clipoboard: copy legend into the clipboard as BMP (visible only if legend is visible).

save view to file: save view to file (note the legend is not saved).

save Legend to file: save legend.

Display Set-up panel: show the control panel.

Background color: change the background color (default is black).

Animate: Animate view, either in phase or frequency according to the selection in the control panel.

Show Legend: Display the legend of values; legend can be dragged and dropped over the graph.

Show values: Display current values in the status bar when moving mouse. Values are not

calculated if more than one layer is displayed

Show Magnetostatic Field Assign Panel: Display the panel used to assign magnetostatic field. Panel may be dragged and dropped by mouse over the graph

In the control Panel:

When more than 1 expanding function is used along the y (conductor thickness) direction in the current layer, you can select a **Slice in layer**, namely one of the "slices" used in order to subsection the conductor over its thickness. The current distribution refers to such subsection, so you can also investigate details of the "skin effect". Slices are numbered starting from the floor of the conductor.

Layer: display currents for the selected layer

All Layers: All layers are shown at the same time; plot is drawn starting from the lower layer; upper layers overwrite lower layers.

Current Component: select if displaying Jx, Jy, Jz or the total current density.

Termination: it is activated for 2 or more ports; if **Short** is selected, short circuit currents are displayed, otherwise currents are calculated for the selected port impedance. In order to change this impedance, just double click over the port.

Stimulus Voltage: specify the voltage (V) applied to the selected port

Show Legend: display legend with pseudo-colors. The legend may be dragged by using mouse; it can be exported or copied into the clipboard by using the popup menu

Show Values: display a status bar, where current density value for the position of the cursor is reported

When **Display Assigned Field** in the **Assign Magnetostatic Field** panel is selected, the user can specify at which plane assigned H has to be calculated and displayed.

The current view may be exported to file as Windows BMP or EMF (Enhanced Metafile), and copied into the clipboard (use the popup menu).

More views may be added and handled by he Data Browser or the main menu **View**.

XXXVIII Post Processing: Far Field 3D View

From the **Data Browser** you can select **Add Far Fiel View**. A window similar to the one described in the previous section is created.

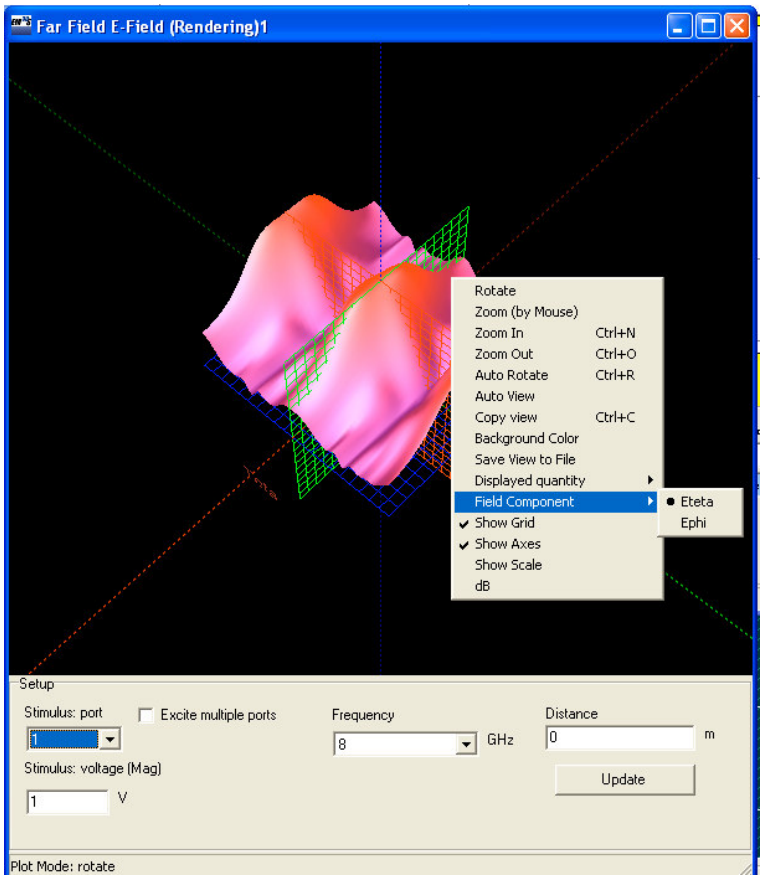
The far field is calculated and displayed by using a powerful rendering and 3D-like perspective: you should read the section about Antenna modeling to know about assumptions and limitations.

Far Field is only calculated if a Sense Layer was added to the project (menu **Action/Add Sense Layer** or button **Add Sense Layer**). Radiated field is plotted in spherical-coordinates.

In the setup panel the user can select the excitation port (**Stimulus port**), the amplitude of the excitation (**Stimulus Voltage**), Frequency and distance (meters). By default the distance is settled to zero, meaning that the r-dependent factor in the far field equation is replaced by 1.

One can also select multiple excitations, specifying voltage amplitude and phase at each port, by checking "**Excite Multiple Ports**": this is needed to see diagrams for arrays of antennas.

Right click over the plot to handle it: a popup menu allows selecting the plot mode

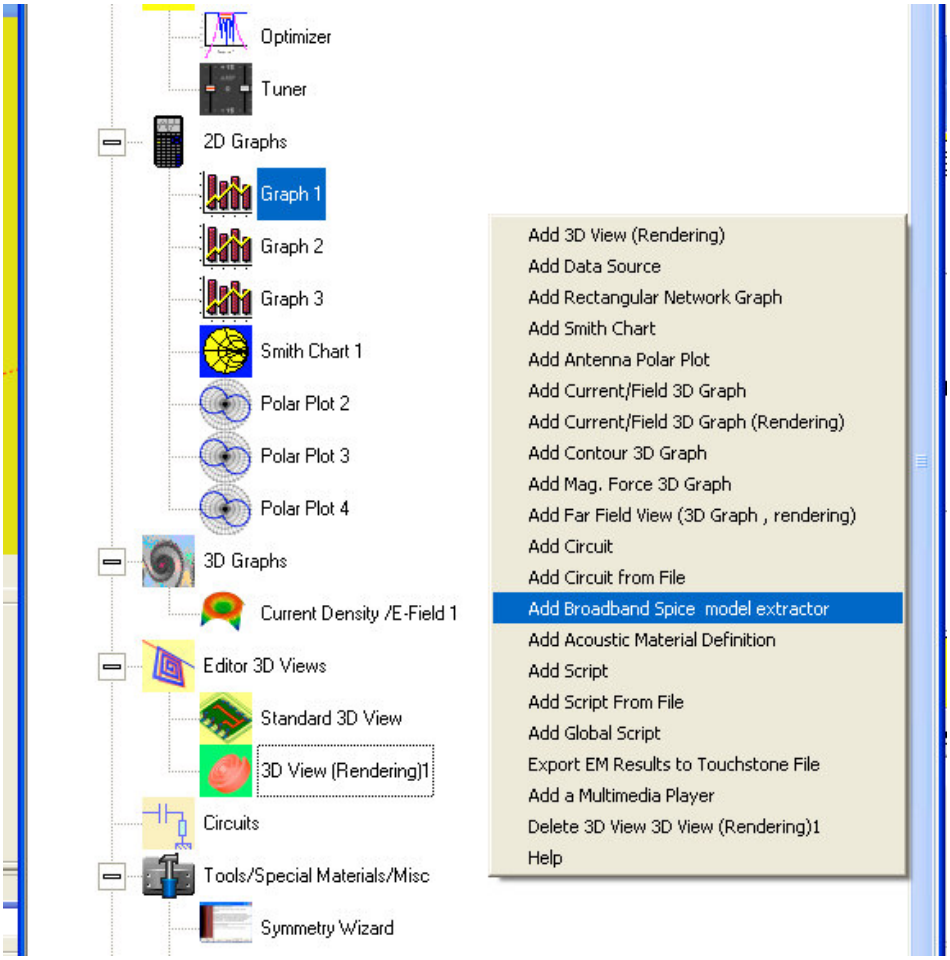


copy view: copy into the clipboard as BMP
save view to file: save view to file.
Background color: change the background color (default is black; modification is stored as default).
Displayed Quantity: Choose either magnitude, Real part or Imaginary part.
Field Component: Select the polarization of the displayed field.
Show Axes: Display reference axes.
Show Scale: Display a scale of values.
Help: or F1: show the help window

The view may be exported to file as Windows BMP or EMF (Enhanced Metafile), and copied into the clipboard. More views may be added and handled by he Data Browser or the main menu **View**.

XXXIX Post Processing: Broadband Spice Model Extractor

EM3DS includes a simple Spice Model Extractor, that can be accessed by **Tools/Spice Model Extractor** main menu, or by a button in the tool-Bar.



Several extractors may be allocated by the user: in order to allocate a Spice model Extractor, you can either select **Tools/Add Spice Model Extractor**, or select **Add Spice Model Extractor** from the Data Browser: every Model extractor may be hence accessed by clicking over its symbol in the Data Browser, in the **Circuits** Set.

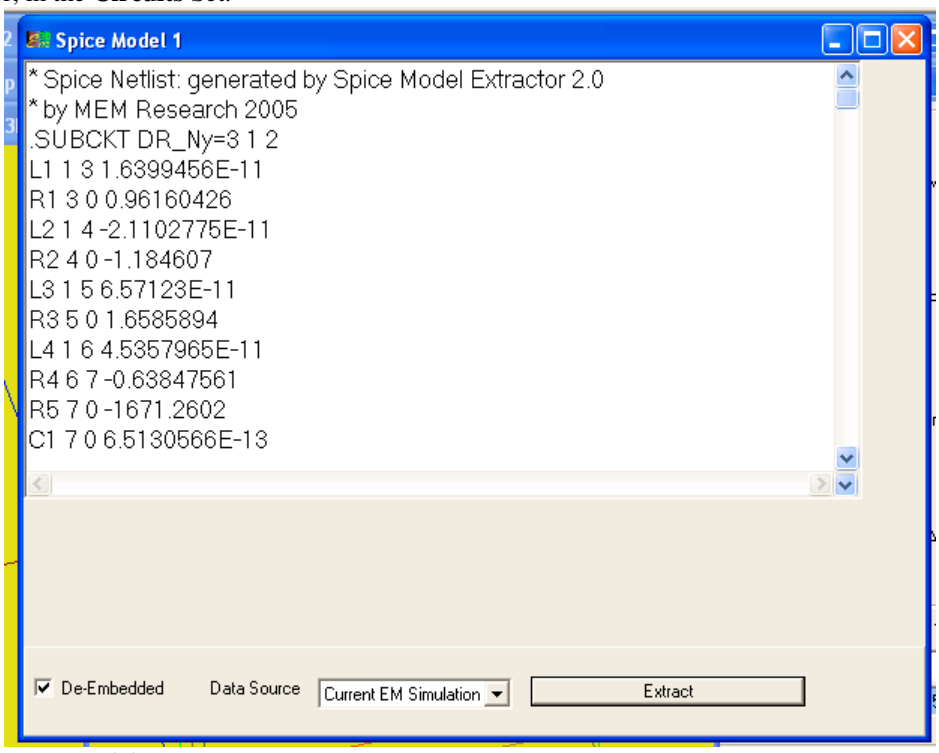


Fig.94:Spice Model Extractor

The model is described as a set of nodes and a set of lumped elements between the nodes that can reproduce the electrical behaviour of electromagnetic structure.

The new spice model extractor allows extracting a lumped model for arbitrary structures.

The model is described as a set of nodes and a set of lumped elements between the nodes that can reproduce the electrical behavior of electromagnetic structure. In order to extract the model, all frequencies are used: the order of the circuit is related to the number of frequency points, so it is not advisable to provide too many frequency points. One can test the quality of the circuits either by re-inserting the extracted circuit in the circuit simulator or, more simply, by creating an interpolated data set from any available chart or graph: the quality of the interpolation will usually mimics the quality of the extracted circuit. If the data source is the current em simulation, evaluated by using SmartFIT, usually there is no problem as the number of frequency points is reduced as possible. Otherwise, if data are from external imported sources, or the current EM was not obtained by SmartFIT, one should perform such a test. There are no limits on the electrical length (even lines may be modeled within a frequency band). The extracted circuit may also be able to extrapolate, to some extent, the circuit response out of the initial band

The model is displayed into a text window, where it can be modified by the user, and saved. The model is in SPICE format.

Spice model may be extracted even from external network parameters, the only restriction being that the structure is reciprocal.

No validity checks are performed in the present version.

The model is displayed into a text window, where it can be modified by the user, and saved. The model is in SPICE format.

For more information about SPICE model implemented please read the paper by M. Farina, A. Morini, T. Rozzi, "Broadband frequency-domain electromagnetic analysis and automatic extraction of Spice-compatible equivalent circuits", *IEE Electronic Lett.*, vol. 41, No. 22, October 2005

XL Post Processing: Linear Circuit Simulator

EM3DS provides also a simple linear circuit simulator.

From a given circuit topology, including lumped (e.g. resistors, inductors etc.), distributed (transmission lines, external data, em simulations) elements, it calculates network parameters.

In order to create a **Circuit Solver** Window, right-click on the Data Browser window and select "**add Circuit**", or do the same operation from the main menu **Tools**.

The circuit solver works on a net-list, namely a list of nodes, the components are connected to, including the component values (or definition).

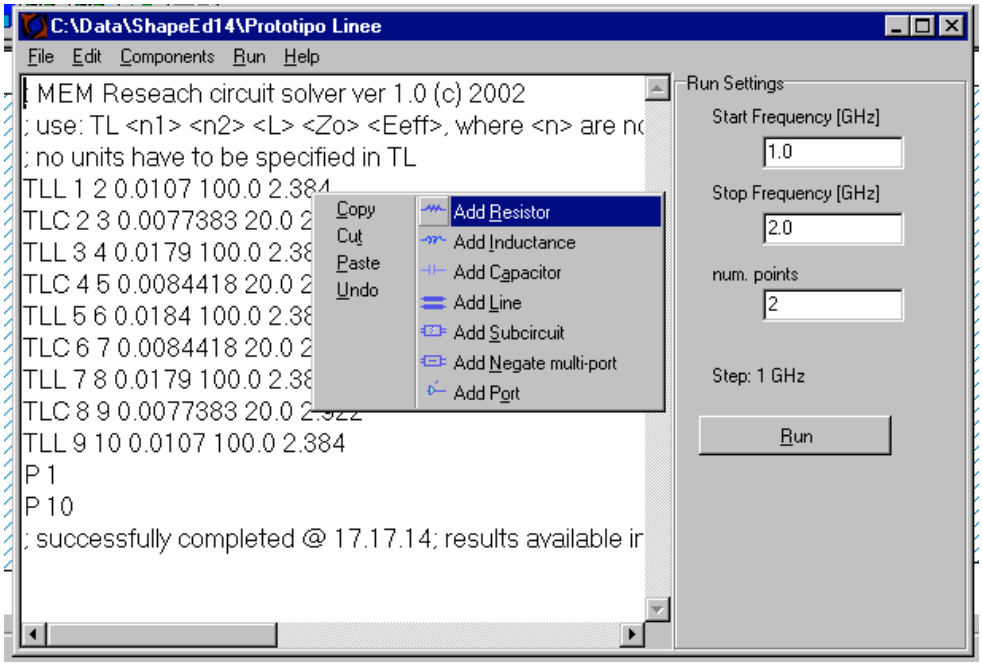


Fig.95: Linear Circuit Solver window

There are few syntax rules to be respected:
- comments begin with symbols ";" or "*" or "\"
- ground is indicated as node "0" or "gnd"
- ports are defined with respect to ground
- the first character is reserved to understand what kind of component you are dealing with:
    R: resistors
    G: conductance
    L: inductor
    C: capacitor
    X: subcircuit
    N: negate subcircuit
    P: port

For example Ra1 is considered a resistor

General 2-port components have the following syntax
<Component> <Node1> <Node2> <Value> [<Units>]
where <Units> is optional.
For example

R1 1 0 100 k

Indicates a resistor having 100kOhm between node 1 and ground
Only the first character of the unit parameter is considered, hence "k" is for Kilo, "m" for milli, "M" for mega etc
Only units are case-sensitive.
You do not need to write directly the above expressions: you can select from the menu the desired component: a string is added correspondingly. Hence you can edit nodes and values according to your need.

Transmission Lines have the following syntax
TL <n1> <n2> <L> <Zo> <Eeff>, where <n> are nodes, <L> length in meters, <Zo> in Ohm, and <Eeff> effective dielectric permittivity
Note that no unit has to be specified for this kind of component
For example

TL1 1 2 1e-2 50.0 4.0

Is an ideal transmission line between nodes 1 and 2, 1 cm long, 50 Ohm and 4 as effective dielectric permittivity

You can add references to external data or em simulation as subcircuits. The syntax is
X <n1> <n2>...<nn> subckt <name>. The name is one of the names appearing in Data Browser window, in the **Ext. Data Files** folder.
For example

X1 1 2 3 subckt circuit.y3p

Is the circuit circuit.y3p, connected to nodes 1 2 and 3. You can simply add a subcircuit by menu, by selecting **add subcircuit**, and hence by clicking over the desired subcircuit in the **Ext. Data Files** folder of the Data Browser

An additional particular kind of subcircuit, the **Negate multi-port**, is included, in order to allow manual calibration procedures. Such a component allows to "remove" the effect of the given subcircuit. (see figure below). The syntax is the same of subcircuits, but the are labelled with string beginning with "N".
For example

N1 1 2 3 subckt circuit.y3p

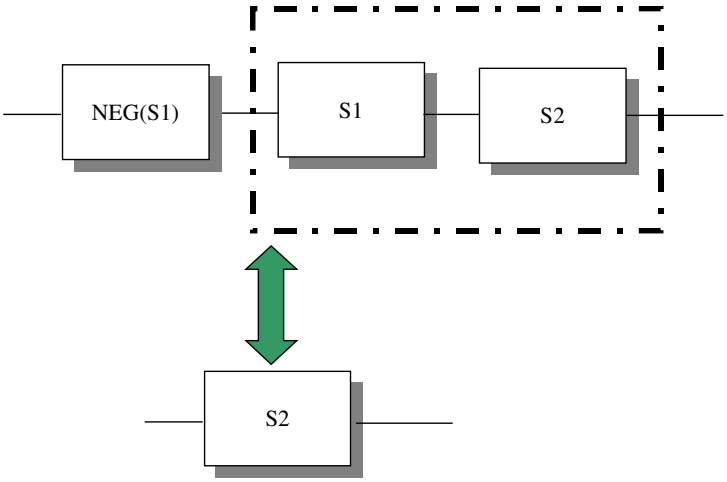
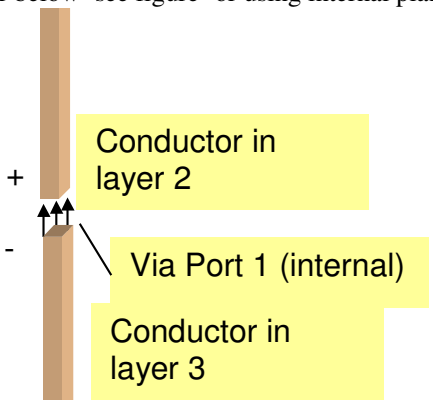


Fig.96: Effect of connecting a NEG component

If a given circuit may be considered as the interconnection of S1 and S2, and we have an evaluation of S1, by connecting Neg(S1) to the circuit we can recover S2. This works for arbitrary multiports. Calibration of via ports (that are not automatically calibrated) may be performed by using this strategy: S1 in this case would be the em simulation of the only via-port.

NOTE about using NEG: whenever you have *n* ports, in order to negate circuit S1, port *i* of S1 has to be connected to port *n-i+1* of NEG(S1); in the simple case of a 2-ports, either you have to connect port 2 of S1 to port 1 of NEG(S1) or port 1 in S1 to port 2 in NEG(S1). If S1 is a 4-ports, port 3 has to be connected with port 2 of NEG(S1), while port 4 to port 1 of NEG(S1).

NOTE INTERNAL PORTS: EM3DS can also create results where one of the ports is off-ground (either using a via port, attached at the bottom of a conductor which in turn is connected to an other conductor below -see figure- or using internal planar ports -see section about ports-)



Let us suppose that you want to connect a resistor at circuit level between points + and – (local ground). The correct connection is to attach one lead of the resistor to the port, and the other to the common ground, as shown below

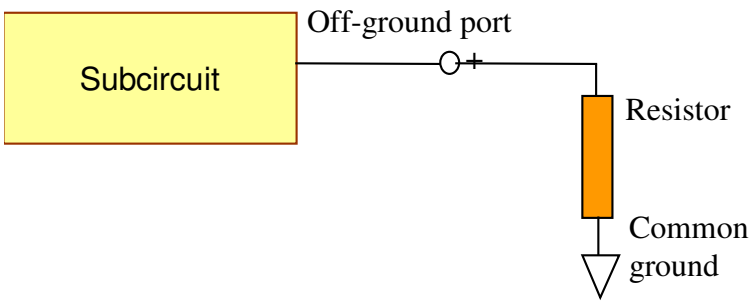


Fig.97: How to connect to an off ground port

Ports have to be connected to the circuit under test: the syntax is

P <n>
Where n is the node; numbering of PORTS is automatically assigned according to the order of appearance in the net list.

Results are automatically saved as Touchstone file according to the name of the circuit. The name of the circuit may be modified by a simple click in the corresponding element in the tree of the Data Browser.
Moreover Results are added in the "**External Data Files**" folder in the Data Browser window; they are available as data "*sources*" in any graph, either rectangular or Smith Chart, (in the data source list) where they can be plotted

If an EM3DS project contains a circuit, it is automatically reopened as far as the path is preserved, or the circuit is in the same folder of the EM3DS project.

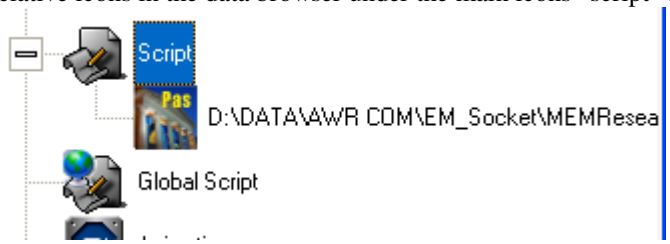
XLI Post Processing: Object Oriented Pascal Script

EM3DS implements an object pascal compiler with a reduced set of commands to write scripts, namely macros to be executed to customize EM3DS or to iterate some process.

Scripts can be **GLOBAL** or **LOCAL**: if local, they are only loaded when opening a given project, while if global, EM3DS will always look for them.

A script can be **AUTORUN** (select this option by selecting the icon in the Data Browser and right clicking); in that case the script is immediately compiled and executed when the project is open (or em3ds starts, if global). In any case a confirmation dialog is displayed in order to avoid execution of malicious software; the warning can be disabled in the **Preferences**.

A local script can be made global and vice-versa by simply dragging and dropping by mouse the relative icons in the data browser under the main icons “script” and “global script”.



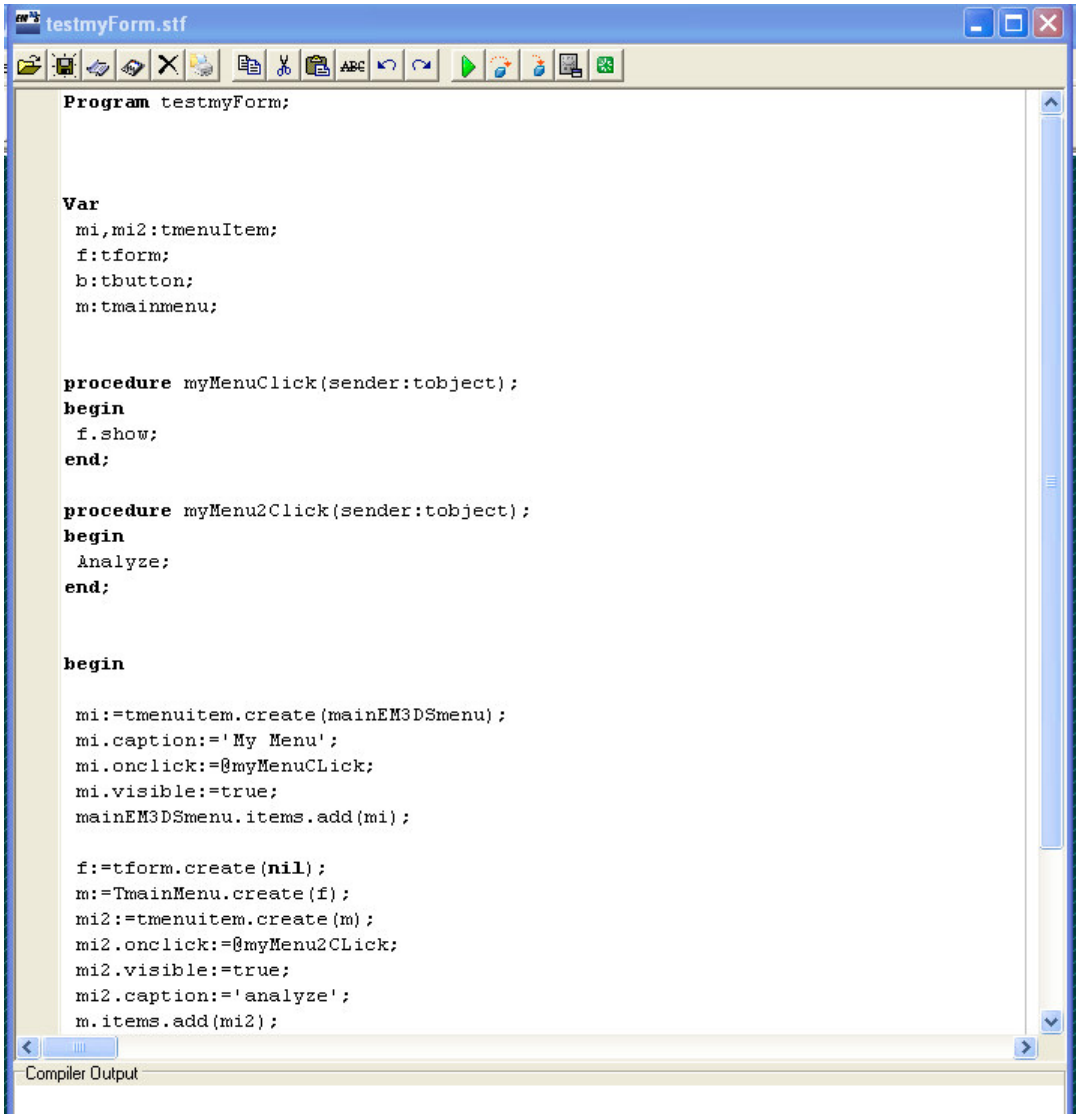


Fig.98: Script Compiler

Scripting will always be in the source format, and stored separately in a file linked to the project (local) or to EM3DS itself (global). It is compiled and then run. For debugging purposes, one can add break points by the popup menu (right click) or execute step-by-step (see menu) or compile etc. A complete menu and new toolbar is made available.

So far there are some restrictions: "**uses**" clause is not yet implemented, pointers are not allowed, and is not possible to create new classes. But a number of classes are already implemented.

They include (see *Borland Dephi* or the online help for reference about their use)
Classes
TSPEEDBUTTON,
TCUSTOMGROUPBOX
TGROUPBOX
TCUSTOMLABEL
TLABEL
TCUSTOMEDIT
TEDIT
TCUSTOMMEMO
TMEMO
TCUSTOMCOMBOBOX
TCOMBOBOX
TBUTTONCONTROL

TBUTTON
TCUSTOMCHECKBOX
TCHECKBOX
TRADIOBUTTON
TCUSTOMLISTBOX
TLISTBOX
TSCROLLBAR
TRADIOGROUP
TSHAPE
TIMAGE
TPAINTBOX
TBEVEL
TTIMER
TCUSTOMPANEL
TPANEL
TFORM
TAPPLICATION
TPOPUPMENU
TMAINMENU
TMENU
TMENUITEM
TSTRINGS
TSTRINGLIST
TBITS
TSTREAM
THANDLESTREAM
TMEMORYSTREAM
TFILESTREAM
TDATETIME
TMSGDLGTYPE
TMSGDLGBTN
TMSGDLGBUTTONS

And the functions
function MessageDlg(const Msg: string; DlgType: TMsgDlgType; Buttons: TMsgDlgButtons; HelpCtx: Longint): Word

It includes all standard Pascal types, and general routines.
Additionally some strings/date types and utilities

function EncodeDate(Year, Month, Day: Word): TDateTime;
function EncodeTime(Hour, Min, Sec, MSec: Word): TDateTime;
function TryEncodeDate(Year, Month, Day: Word; var Date: TDateTime): Boolean;
function TryEncodeTime(Hour, Min, Sec, MSec: Word; var Time: TDateTime): Boolean;
procedure DecodeDate(const DateTime: TDateTime; var Year, Month, Day: Word);
procedure DecodeTime(const DateTime: TDateTime; var Hour, Min, Sec, MSec: Word);
function DayOfWeek(const DateTime: TDateTime): Word;
function Date: TDateTime;
function Time: TDateTime;
function Now: TDateTime;
function DateTimeToUnix(D: TDateTime): Int64
function DateToStr(D: TDateTime): string;
function StrToDate(const s: string): TDateTime;
function FormatDateTime(const fmt: string; D: TDateTime): string;

Complex numbers are handled via the “COMPLESSO” type, and a number of functions are implemented:

Type COMPLESSO=Record
Re : double;
Im : double;
end;

```
Procedure CONIUGATO( C1 : Complesso; var C2 : Complesso); //in C2 the complex conjugate of C1
Function MODULO( var C1 : Complesso) : double; //Modulus of C1
Procedure CSUM( a, b : complesso; var c : complesso); //Complex sum: c=a+b
Procedure CSUB( a, b : complesso; var c : complesso); //Complex subtraction c=a-b
Procedure CMULT( a, b : complesso; var C : complesso); //c=a*b
Procedure CRMULT( a : complesso; b : extended; var C : complesso); //same, but b is a real
Procedure CIMULT( a : complesso; b : extended; var C : complesso); //same but b is purely imaginary
Procedure CDIV( a, b : complesso; var c : complesso);// c=a/b
Procedure CRDIV( a : complesso; b : extended; var C : complesso); //same but b real
Procedure CSIN( a : complesso; var c : complesso); // c=sin(a)
Procedure CCOS( a : complesso; var c : complesso); //c=cos(a)
Procedure CEXP( a : complesso; var c : complesso); //c=exp(a)
Procedure CTAN( x : complesso; var tangent : complesso); //tangent=tan(x)
Procedure CTANH( x : complesso; var tangent : complesso);// tangent=tanh(x)
Procedure CATANH( x : complesso; var Atangent : complesso); //atangent=arc tanh(x)
Procedure CSINH( x : complesso; var C : complesso); //c=sinh(x)
Procedure CCOSH( x : complesso; var C : complesso); //c=cosh(x)
Procedure INVrp( x : complesso; var modulo, fase : double);// modulo=magnitude(x), fase=phase(x)
Procedure INVpr( modulo, fase : double; var x : complesso);// for a given mag and phase, give x
Procedure CSQRT( x : complesso; var y : complesso); //y=complex square root(x)
Procedure CLN( x : complesso; var y : complesso); //y=ln(x)
Procedure CACOS( x : complesso; var y : complesso); //y=arc cos(x)
```

A static matrix of complex numbers is also used to exchange results with the solver

```
TSCOMPLEXARRAY=array[0..maxstaticsize] of complesso
TSCOMPLEXMATRIX=array[0..maxstaticsize] of TSCOMPLEXARRAY
```

The constant **maxstaticsize** is available to the user. Also, a constant indicates the script library version, namely **LIBRARYVER**, which is a string. Library will be often enriched with new functionalities.

A number of functions allow to interact with the solver:

Function FrequencyIndexExists(f:double; var index:integer):boolean;

After a calculation, if one passes the value of the frequency (GHz) as *f*, the function returns in *index* the position in the solution array; the function returns FALSE if the solution wasn't calculated @*f*

Procedure LoadNetMatrixFromFile(FileName:string; var errorString:string)

Load in EM3DS Touchstone file named *Filename*. *ErrorString* is empty if successful; otherwise it returns an error message. NOTE: in order to make accessible to the user the loaded file, Procedure LoadNetMatrixFromFile should be ALWAYS followed by the instrucion **AddSourceOnList**.

Procedure AddSourceOnList(namesource:string)

Adds a previously loaded Touchstone file named *nameSource* to the list of available external file in the Data Browser and in any chart. ALWAYS needed after loading a Net matrix from file.

Procedure SaveTouchStoneFile(FileName:string; AlreadyOpen:boolean; KindMatrix:char;
Y:TSComplexMatrix; NumPorts:integer; frequency:double;autoExtension:boolean);

Creates a new Touchstone file or append results to an existing Touchstone file, named *Filename*. If *AlreadyOpen* true then the file is supposed to have been created and the procedure appends the Y matrix to it, otherwise a new file is create with all needed heading. *Y* is a network (either S, Y, Z) matrix describing a *NumPorts* circuit. If *autoExtension* is true, the kind of matrix to be exported is defined by *KindMatrix* (either s,z,y), and string *FileName* should not include extension. Otherwise, *autoExtension* false implies that *FileName* also includes the extension (e.g.: filename is myTouchstone.s2p for a 2 port S-parameter)

Function GetVariableList:tstringList

Creates a stringList indicating all existing variables in a given project.
For example:

```
Program TestStringList;
var
 s:TstringList;
 i:integer;

Begin
  s:=GetVariableList;
  For i:=0 to pred(s.count) do
    MessageDlg('Variable '+intToStr(i)+'= '+s.strings[i],mtInformation,[mbok],0);
End.
```

This example recovers the list of variables in s, then looks through s and issues a message (type information) with a button OK and no help.

Function ParseExpression(s:string):extended;

This function evaluates the numeric value of an expression *s*. The expression may involve variables defined in the EM3DS project. For example, if the variable list is defined a variable 'x' you can pass as *s* the argument 'sin(x)', and get its value.

Procedure SetVariableToValue(variable:string; value:double)

Set the variable *variable* to the numeric value *value.* If *variable* is not in the variable list, a new variable is created.

Variable MainEM3DSMenu:TMainMenu;

Such a global variable is related to the main EM3DS menu, so that you can add items, or change the existing ones.
For example

```
Program testmMenu;

//creates a new menu in the EM3DS

Var
 mi:tmenuItem;
 m:tmainmenu;



procedure myMenuClick(sender:tobject); //event when clicking in the new meny
begin
 Analyze;
end;


begin //Main program

 mi:=tmenuitem.create(mainEM3DSmenu); //create a new item
 mi.caption:='My Analyze';
 mi.onclick:=@myMenuCLick; //link to the event
 mi.visible:=true;       //shows
 mainEM3DSmenu.items.add(mi); //adds to the main list
end.
```

In this case a new item in the menu of EM3DS (the main one, appearing with the editor) has a new item, "My Analyze". When clicked, an analysis is started.
NOTE: when there are events, consider that if you discard the script, the additional menu remains, but the event is no longer linked to a located procedure. Hence there will be an Access Violation error. To avoid this problem, modifications of the main EM3DS should be done always in Global Scripts.

By the same token, it is easy to create your own form: for example

```
Program TestFormAndButton;

Procedure MyOnClick(Sender: TObject);
Begin
 MessageDlg('Hello',mtinformation,[mbok],0);
End;


var
m:tButton;
f:TForm;

begin
f:=TForm.create(nil);
f.caption:='This is my form';
m:=tbutton.create(f);
m.caption:='test';
m.parent:=f;
f.show;
m.show;
m.onclick:=@Myonclick;

end.
```

Variable Application:Tapplication

It is the main EM3DS process.

Variable nodialogs:Boolean

If *true*, EM3DS does not issues message dialogs during calculations, but only events recorded in the even window. Useful for batch simulations. Its default value is *false*, and should be always reset to false after calculation.

Function deembAvailable: Boolean

Returns true if also calibrated results are available, false otherwise.

Procedure OpenFile (const FileName:string)

Open a file with name *FileName*. **NOTE**: this command should only be used from a GLOBAL SCRIPT, as, when opening a new project, project related scripts are de-allocated. Execution from a local script of this command will cause an Access Violation Error.

Procedure BrowseFile

Opens the Browse file dialog in EM3DS

Procedure SaveFile

Save the current project

Procedure SaveAsBrowse

Browse to select a name for a file to be saved

Procedure Analyze;

Analyze the current project

Procedure Optimize

Run the optimizer

Function GetCurrentYRawMatrix(indexOfFreq:integer; var success:boolean):TSComplexMatrix

Provides the uncalibrated Admittance matrix for the calculated structure, at the frequency in position *indexOfFreq* in the frequency list. Use function *FrequencyIndexExists* to recover the position in the list of a frequency f

Function GetCurrentYCAlibratedMatrix(indexOfFreq:integer; CalibrateToPort: boolean; var success:boolean):TSComplexMatrix

Provides the calibrated Admittance matrix for the calculated structure, at the frequency in position *indexOfFreq* in the frequency list. Use function *FrequencyIndexExists* to recover the position in the list of a frequency f. If *CalibrateToPort* is false, results are at the specified calibration plane, while if true, results are calibrated at the port plane.

XLII Printing

Since version 12, a set of utilities to print out in a standard printer any chart, graphic or image in EM3DS is made available. For example, from the menu **File->Print**, or from the **Print** popup menu in any chart or graphic, you access a **Print Preview** form.

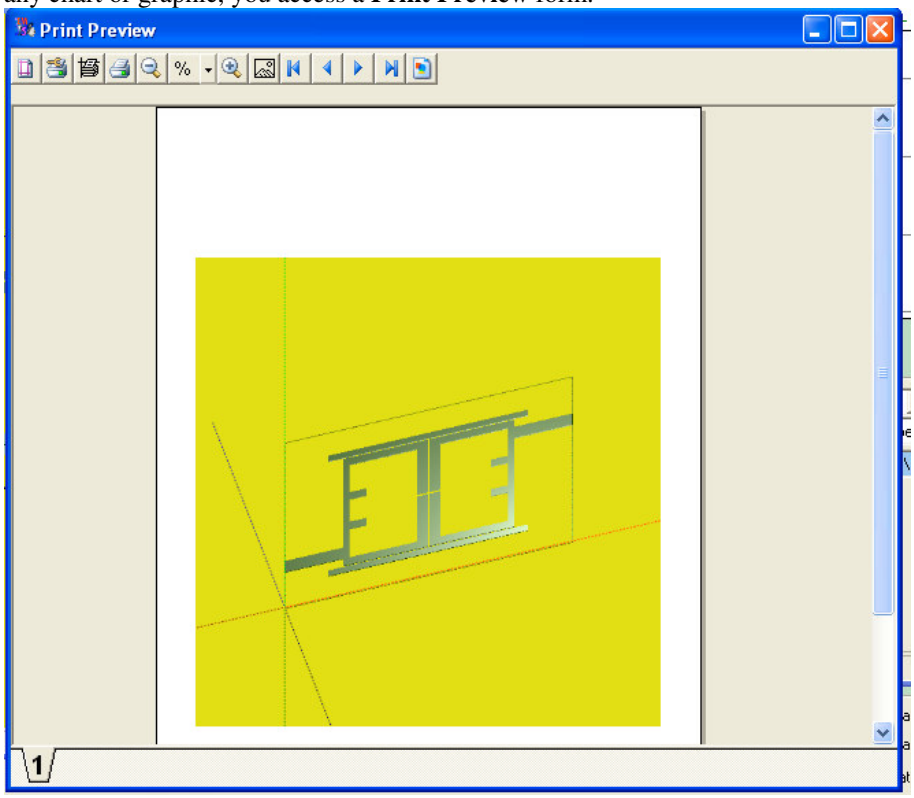


Fig.99: Print Preview Form

From this form, clicking on the buttons shown in the toolbar, you can: change the page setup, setup the printer, show/hide margins, print, zoom/reduce the image, show in Black/white or Color, browse between pages and save as TIFF. By moving the mouse across the buttons it is possible to see a hint about the function of each individual button.

XLIII 3D View

The structure being drawn is displayed in its 3D view. The view can be handled by right clicking over the figure, by selecting **zoom, move** or some other action and dragging the mouse over the image. Boundary walls are only shown if **Show WR Walls** is selected in the 3D view. **Keep Ratio** allows seeing the figure in its real aspect ratio, while otherwise size is automatically scaled to fit the window.

Real Time Refresh allows to update in real time the image when rotating the structure. This can be a slow operation in some old computer; in such a case disabling this option may be useful.

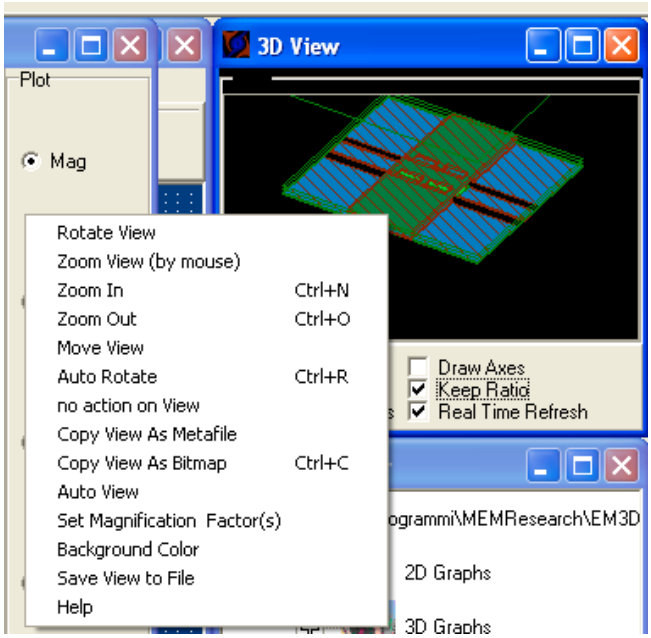


Fig.100: 3D View

By the editor **View** or by the popup menu in the **Data Browser** you may also create very special 3D views: the actual look of the structure being edited. These additional windows include visual processing and rendering, very effective for presentations, but hardware-dependent (they may not work well on all graphic cards).

Current densities may only be displayed over 3D views with rendering.

The 3D view can be exported to file (Save View to File) either as bitmap or Windows Metafile. It can be copied (Copy View) into the clipboard as metafile.

Sometimes a real structure may have dimensions very different along the 3 dimensions; in order to see details it may be useful to exaggerate magnification along one or more directions selectively. This may be done by clicking over the **Set Magnification Factor(s)** item in the popup menu and selecting a selective zoom along one direction.

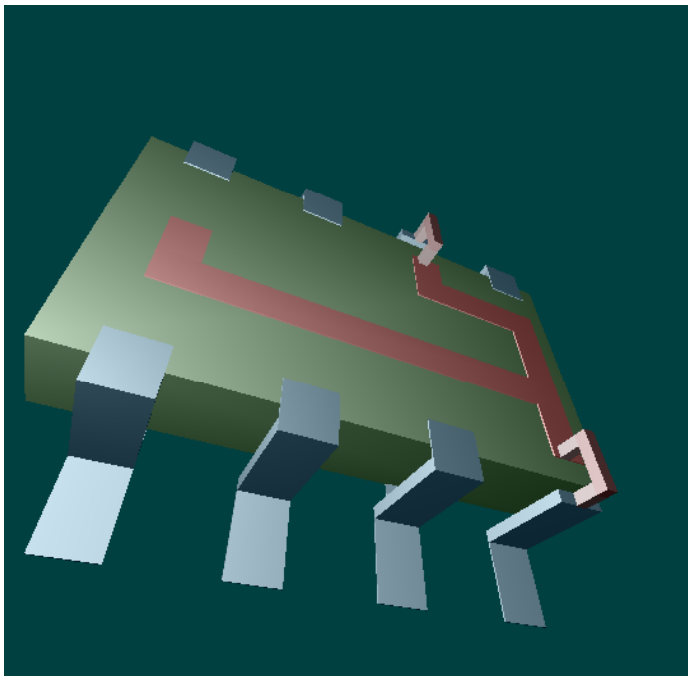


Fig.101: 3D View with rendering

XLIV Project Description

EM3DS provides a window where notes about the project may be written and displayed: an RTF file (indicated as .DES) is created automatically and reopened along with the project. This file may be modified and saved independently.

XLV Translators: importing and exporting DXF and GDSII files

EM3DS has also some ability in translating its native files (EMS) into other popular formats. It is possible to import Autocad DXF files, Calma GDSII files and Windows Bitmap (BMP) files.
If the user selects to import DXF or GDSII file, the **layer list** on left-hand side reports the list of layer found in the file and, for each layer, in which layer of EM3DS the objects will be imported. On the right hand side a list of commands ignored by the filter is shown.

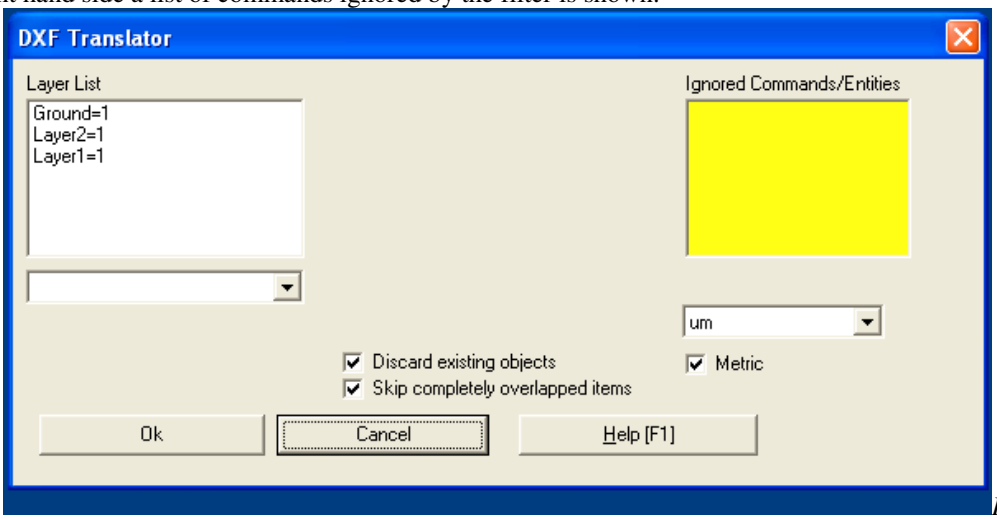


Fig.102: DXF Import window

In the lower dialog list you can change the layer of EM3DS where the highlighted item of the **Layer list** will be imported. **Skip completely overlapping items**: if selected requires EM3DS to test if an object is completely contained into an other object in the same layer (what would produce a complete failure of the analysis), and possibly to discard it.

Usually disabling the automatic mesh while importing external sources is recommended, as often some additional editing is needed in order to simplify the structure.

During import actually only "POLYLINES" of DXF and "Boundary" of GDSII are handled so as to produce objects (namely conductors) in EM3DS; rectangular polygons are automatically converted in rectangles allowing to add ports. Other entities are only plotted in the editing window (**DXF Image**), so to e.g. help to redraw some structure. The **DXF Image** is available until a new file is selected or the menu item **Edit/Clear DXF Image** is selected.
Importing BMP file has the effect of producing a BMP image to be e.g. used as guideline in drawing. You can clear the **BMP Image** by selecting the menu **Edit/Clear BMP Image**

Link to BMP and DXF images are not saved in the EMS file.

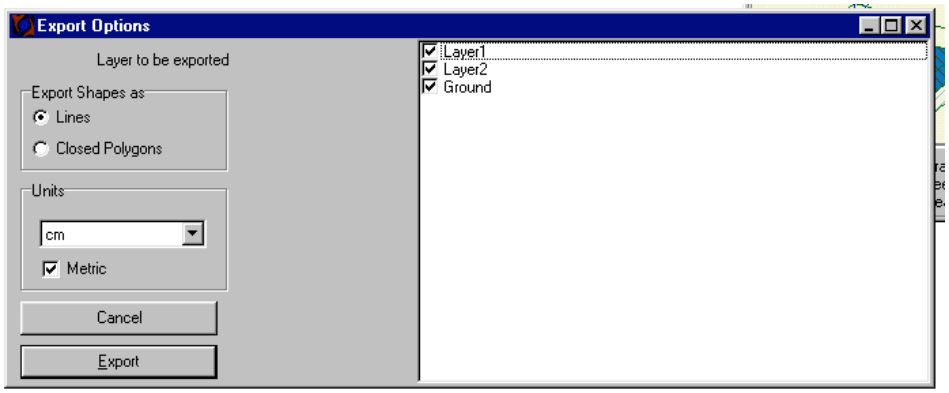


Fig.103:

DXF Export window

Files can exported as Autocad DXF or GDSII by selecting **File/Export**.

A checklist on the left-hand side allows selecting which layers have to be exported. Layer "Ground" is just a layer describing the box. When exporting in DXF format, shapes are either translated into lines or polygons (POLYLINES), according to the user selection

NOTE: DXF standard has been modified several times since its definition, hence exporting or importing DXF files may not work properly over any external program (or different versions of programs). GDSII is surely a very robust standard, and its use for exchanging data is recommended.

GDSII import filter is shown below. If **Discard Existing Objects** is selected (default), importing a file results in creating a completely new project. Otherwise it is possible to import several structures in the same file.

Auto map Layers allows that EM3DS automatically creates a number of layers adequate to include all layers being imported. Each GDSII layer is imported into an EM3DS layer.

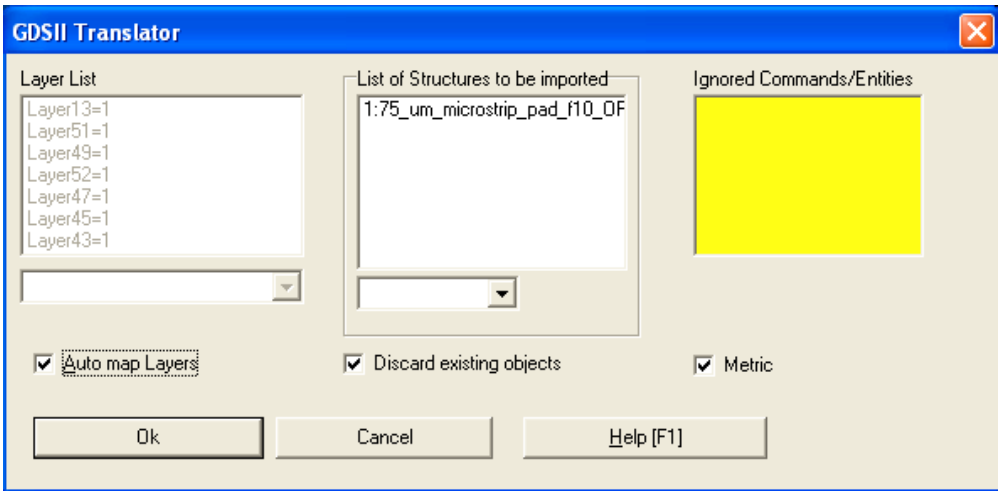


Fig.104: GDS Export window

GDS translator is very robust. The only assumption in the present version is that, whenever more structures are defined, no cross-reference between structures is used. I all entities are grouped into just one structure, no problem arises.

XLVI Using EM3DS with command line: files that can be associated to EM3DS

When clicking over a file that has been associated by Windows to an application, the action that Windows performs is to pass the name of the file being clicked to the associated application. EM3DS is able to handle this way the following files:

.EMS: EM3DS project; the action is opening the project
.GDSII: GDSII file; the action is importing file in EM3DS
.S?P, Z?P, Y?P: Touchstone files: EM3DS imports them as External Data Files
.NET: circuit: EM3DS creates a "circuit" window and imports the file

XLVII Antenna modeling

As highlighted at the beginning of this manual, the structures analyzed are enclosed in a waveguide, that may be either infinite in the vertical (y) direction (possibly filled by any lossy or lossless material [hint: using lossy material allows rigorous modeling of lossy ground planes in microstrip, backed CPW etc.]), closed with perfect conductors so as to form a closed box, or terminated over **absorbing boundaries** (approx. 377 Ohm, the characteristic impedance of plane waves; see p.6).

This particular assumption poses some limitation on the class of antennas that can be modeled. Nonetheless, antennas may be simulated by using some care.

A "large box" may be used to model the open space. The matter is what we mean by the term "large".

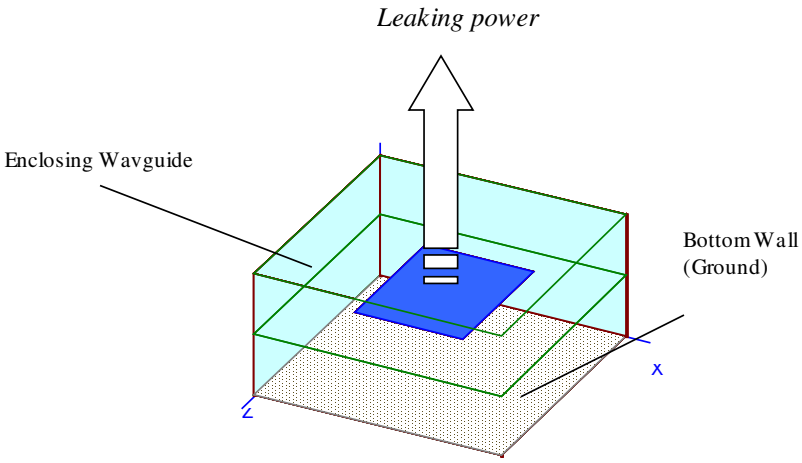


Fig.105: Modeling radiation

Let us refer to a radiating patch, schematically shown in fig. *105*. In order to model radiating energy we need power leaking in the vertical direction: this way if we draw a 2-port, some energy is lost in spite of having selected a lossless structure. Such a leakage is only possible if *at least* one mode of the waveguide is over cut-off over the whole frequency range. Multi-mode leakage and large box allows simulating several kind of planar antennas more than satisfactory. The upper boundary may be either infinite waveguide or approximately open. In this last case care has to be used so to ensure that the absorbing wall be placed far enough so to not directly interact with the near field of the antenna. The **approx. Open** boundary is helpful in that sometimes it allows to reduce the size of the box, relaxing the need of over-cutoff modes: we know from the previous sections how reducing the size of the box may substantially save a large amount of computation time. The absorbing boundary are hence placed according to the trade-off between reducing interaction with near field, and providing power leakage even with relaxation of constraints about over cut-off modes. Usually half a wavelength is OK for most structures.

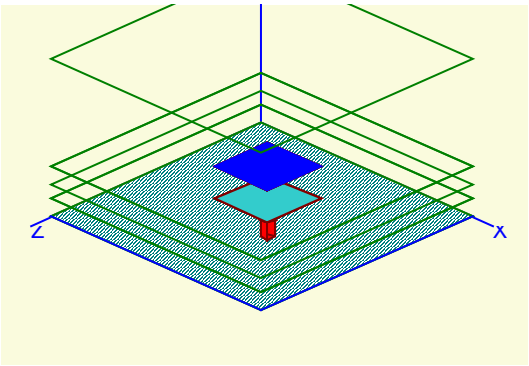


Fig.106: L-band patch antenna, fed by coaxial probe.

Excitation of patch antennas by coaxial feed is usually well modeled by adding a via port, as shown in figure. If de-embed is enabled, and edge ports are used with non-zero de-embedding planes, the use should be careful about the problems discussed in section XXII.

Radiation pattern can also be calculated and visualized either as polar cuts (selecting one of the angular coordinates in the spherical set -see section about polar plots) or as surface plots, as indicated in the post-processing sections.

To this aim one has to add a Sense Layer (menu **Action/Add Sense Layer**)
When selecting "**Add Sense Layer**" EM3DS

1 sets the top enclosure to "**Approx. Open**", namely the top cover of the metal box enclosing the circuit is replaced by an absorbing boundary, which mimics the 377 Ohms seen by a plane wave in free space. This approximation is a good approximation for open space if the lateral metal wall are kept far enough from the structure, and the absorbing boundary is at a distance so as not intercepting the radiated near-field
2 adds two layers (1 and 2) on top of your structure: the first one touches the absorbing boundary ad is labeled as "sense layer", while the second one is a spacer between the absorbing boundary and the structure. Layer 0 is settled to 0 thickness and labeled as "null layer", basically is the same as

if we were removed that layer.

3 Inserts in layer 1 a thin lossy conductor: the conductor is so thin and so lossy that it nearly does not affect the calculation of the network parameters (basically "transparent"). However, owing to Ohm's law, induced currents are proportional to electrical fields, and this way EM3DS recover the field distribution in the sense layer, without the explicit need of the Green's function. Such a field is available in any current plot. The thin conductor cannot be edited by the user, however the user can modify its mesh properties. This is important, because the additional conductor in the sense layer affects the computational load, and in spite of this, its mesh only affects the resolution in viewing the field and the quality of the radiation diagrams. Hence, the user can and should check and eventually modify its mesh properties, according to the needs. **NOTE**: if you add any object in layer 2 (layer 1 is forbidden if it is a sense layer), when removing the sense layer, layer 2 is removed at the same time, along with all object hosted. Anyway a warning is issued.

Once the E-field is known in the sense layer, equivalence theorem is used.
Basically it is assumed that the cavity radiates over a perfect conducting plane.

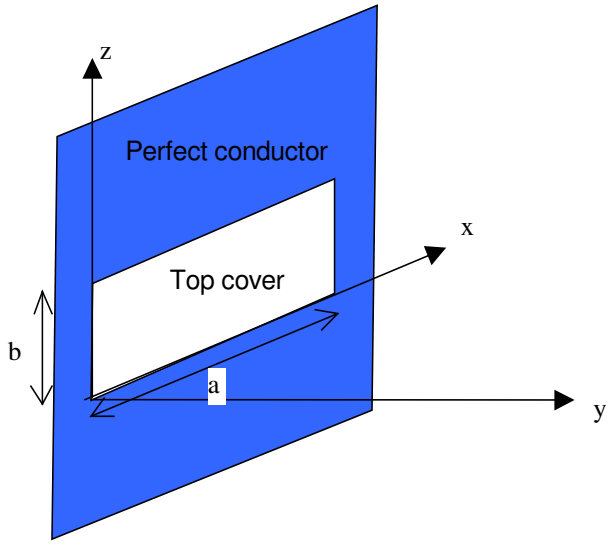


Fig.107: Top of the box surrounded by a perfect conductor.

By means of the equivalence theorem, the tangential E-field across the absorbing boundary is replaced with Magnetic Sources

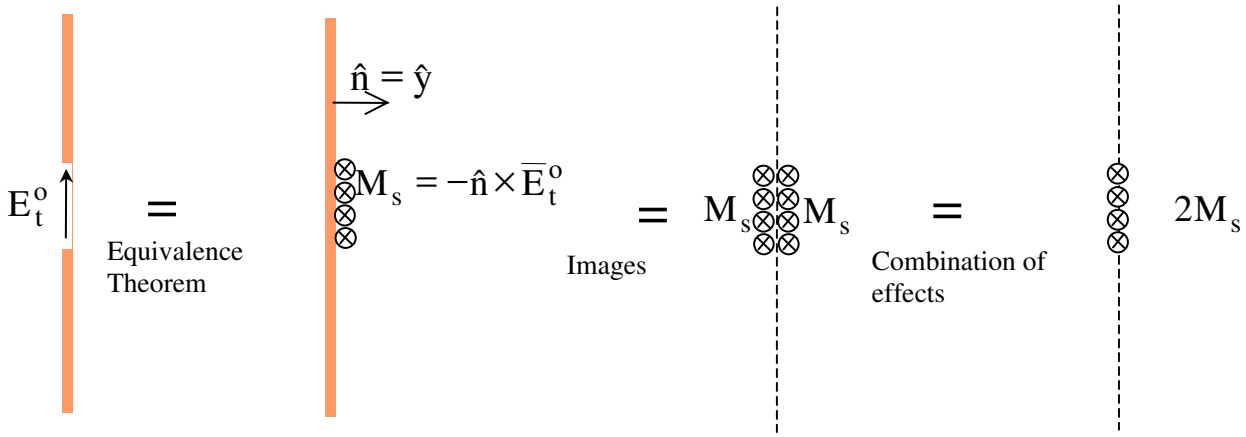


Fig.108:Equivalence Theorem and Image Theorem are combined across the top boundary to evaluate the radiated field..

The image principle gives the equivalent problem, where a magnetic source is radiating in free space. This is done by using the magnetic potential

$$\mathbf{F} = \frac{\varepsilon}{4\pi}\frac{e^{-jkr}}{r}\int_{-\infty}^{\infty}\int_{-\infty}^{\infty}\mathbf{M}_{eq}(x',z')e^{jk\sin\theta\cos\phi\, x'}e^{jk\sin\theta\sin\phi\, z'}dx'dz'$$

and the far field expressions relating the E-field with the magnetic sources

$$\overline{E} = -\frac{1}{\varepsilon}\nabla\times\overline{F} \rightarrow -\frac{1}{\varepsilon}jk\mathbf{u}_r\times\overline{F} = -j\omega\eta\mathbf{u}_r\times\overline{F}$$

This approach is quite good, and works well, at least for calculating broadside antennas. Results in terms of radiation patterns, however, have to be interpreted knowing the above assumptions: one recovers the E-field radiated in presence of the perfect conductor seen above.

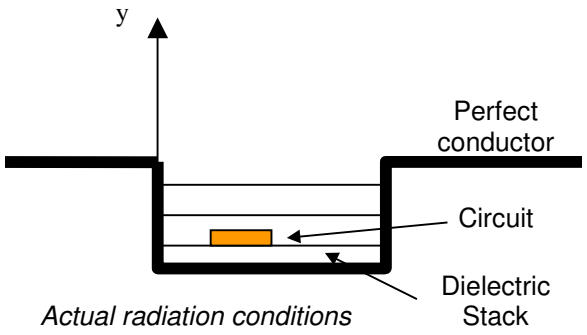
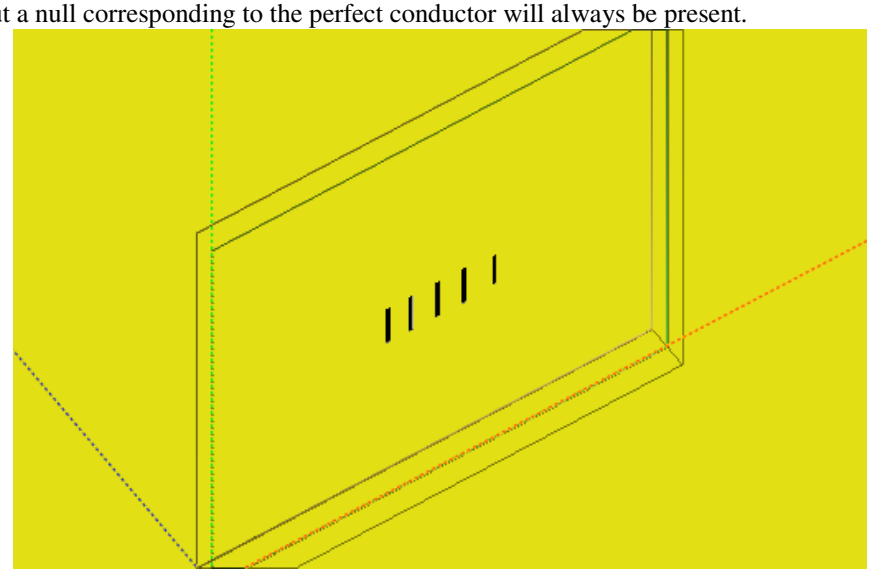


Fig.109:One should always keep in mind that the radiation pattern is obtained considering that the radiating structure is radiating in a large cavity embedded in a metal plane.

The figure above shows a cross section (side view) of a microstrip circuit and the actual conditions used in calculating radiation patterns. It is evident that the absorbing boundary position is critical: if too close to the circuit, there will be interaction with the near field while, if too far, the radiation pattern will become the one of an aperture in a standard rectangular waveguide. Hence a trade-off is usually needed; as indicated above, for a microstrip antenna and several others, λ/2 resulted to be a good choice.

While modeling an end-fire antenna, e.g. a Yagi-Uda, the end-fire characteristic will be apparent, but a null corresponding to the perfect conductor will always be present.



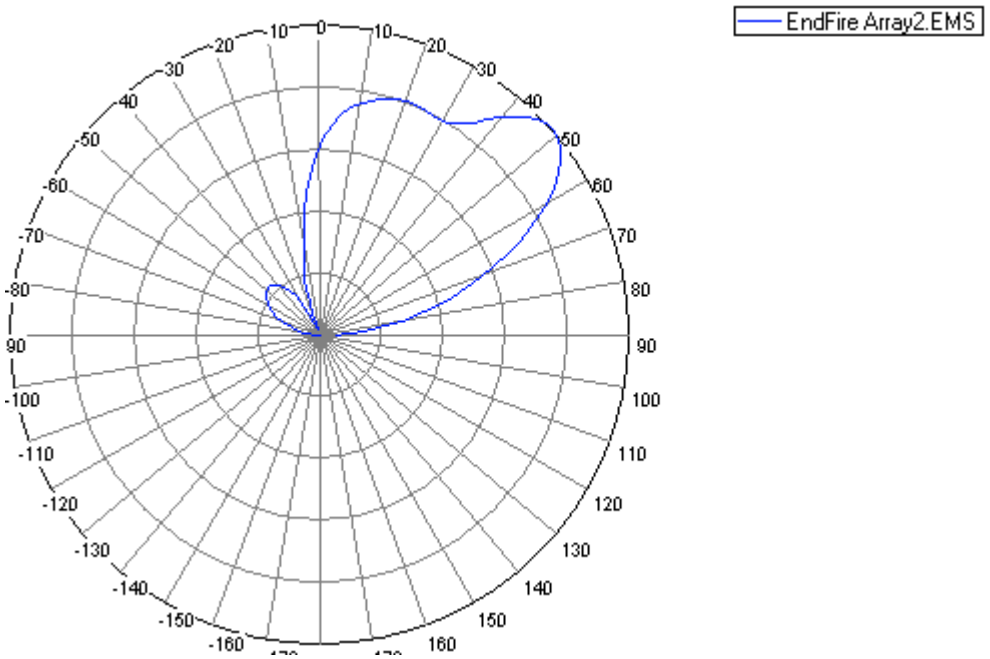
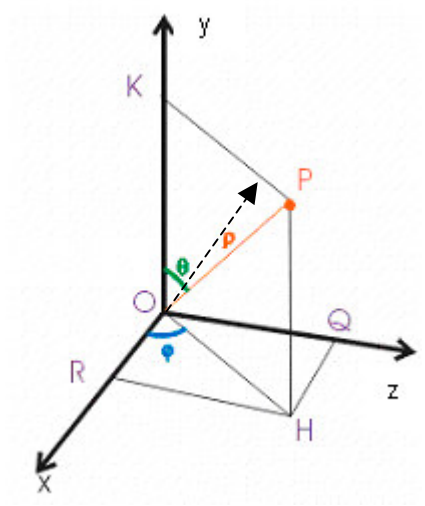


Fig.110:Three-elements (plus one reflector and one director, phase difference between elements 90°) planar Yagi-Uda: Eϕ with ϕ=0, sweeping θ. End-fire behavior is seen, but maximum is not at 90°, where it should be and where there is a null due to the ground plane.

The figure above represents a 3-element planar Yagi Uda (plane xz) *Eϕ* with ϕ=0 (plus one reflector and one director; phase difference between elements 90°), sweeping θ. End-fire behavior is seen, but maximum is not at 90°, where it should be and where there is a null due to the ground plane.

Radiated field is plotted in spherical-coordinates.



Note that actually the above system -maintained for backward compatibility issues- would not be right-handed. Hence in calculation of the far field x axis (and hence ϕ) is actually inverted (see the dotted axis).

XLVIII Modeling Bulk Acoustic Wave devices

EM3DS allows the coupled acoustic/em analysis, especially developed so far for pure mode bulk acoustic wave resonators.

In order to perform this kind of modelling, you have to create an **Acoustic Material/Analysis form**. This may be done either from the popup menu in the **Data Browser** or from the main menu **Tools**.

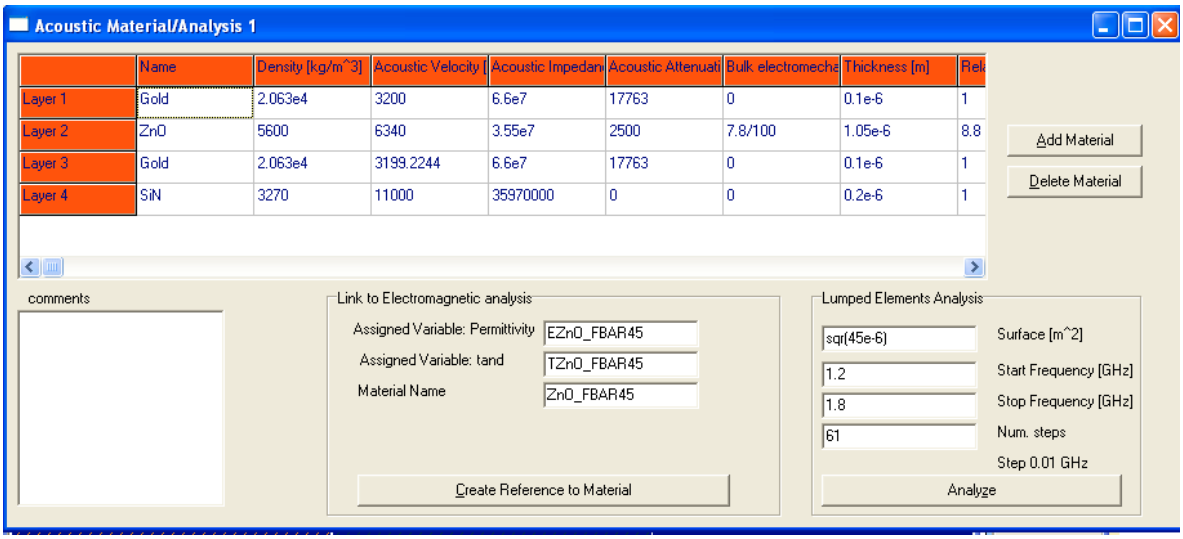


Fig.111: Form used to define acoustic properties of a BAW .

You can add several forms of this kind, and access them by double clicking over icons in the **Data Browser**, in the folder **Tools/Special Materials/Misc**.

In this form you specify the acoustic properties of the material composing the bulk acoustic wave resonator. There are a few assumptions beneath the performed analysis:

- the analysis only accounts for pure mode (either longitudinal or shear) and the propagation properties have to refer to the considered modes. Mode coupling due to imperfect crystal orientation is not accounted for

- only one piezoelectric film may be specified for a resonator; the piezoelectric film is the only one having non zero bulk electromechanical coupling (k_t^2). The number of the additional materials is on the other hand completely arbitrary and the mechanical loading effect is rigorously accounted for.

- While generally piezoelectric materials are generally anisotropic, so far material is handled as isotropic (and considering permittivity on the direction orthogonal to the piezoelectric film); this is generally acceptable in pure mode thin film structures, FBAR.

You can add a material (always added on the bottom of the list) or delete any selected material. Not all parameters are independent: e.g. by modifying the density or the acoustic velocity, the acoustic impedance is automatically updated and vice-versa. Cells of the table may be resized by dragging the mouse over the separation lines.

The material properties may be saved as text file (indicated with extension **.amt**, acoustic material model). Any allocated table will be reopened automatically when reopening the project, provided that the corresponding amt file is in the same directory of the project. You can add also some line of comments that will be saved into the amt file.

EM3DS will exploit the acoustic analysis to create a special material, in particular a special dielectric, having the dispersive, lossy and resonant behaviour due to the piezoelectric phenomena. In order to do that, select a name for this material (**Material name**) and then click over the **Create Reference to Material** button: two variables, one for the permittivity and one for the tand, will be created as displayed in a dialog box. In the electromagnetic simulator you can use these variable to define permittivity and tand of the piezoelectric film, that you can place e.g. by defining a dielectric brick (same variables are also available for definition of the layers in the substrate).

When you run the em simulation, at each frequency em3ds performs a preliminary acoustic analysis, and assigns the necessary electrical parameters to the em model.

It is also possible to perform an additional stand-alone circuit analysis of the BAW/FBAR: this calls for a set of additional assumptions

-the circuit is electrically small (electrical lumped analysis)

-parallel plate capacitance formula is used to evaluate the clamped inter-electrode capacitance

The electrode surface has only to be specified for this kind of analysis (it is not used in the em analysis). In order to perform the lumped elements analysis only select the frequency range and then

press **Analyze**: the results of this circuit analysis are automatically exported to external touchstone file, added to the **Ext. Data File** folder in the **Data Browser** and available in any 2D Chart. They may be used directly into the circuit solver.

Figures below are a good example of the performances of this modelling. They show comparison between experimental and EM3DS results for a ZnO FBAR.

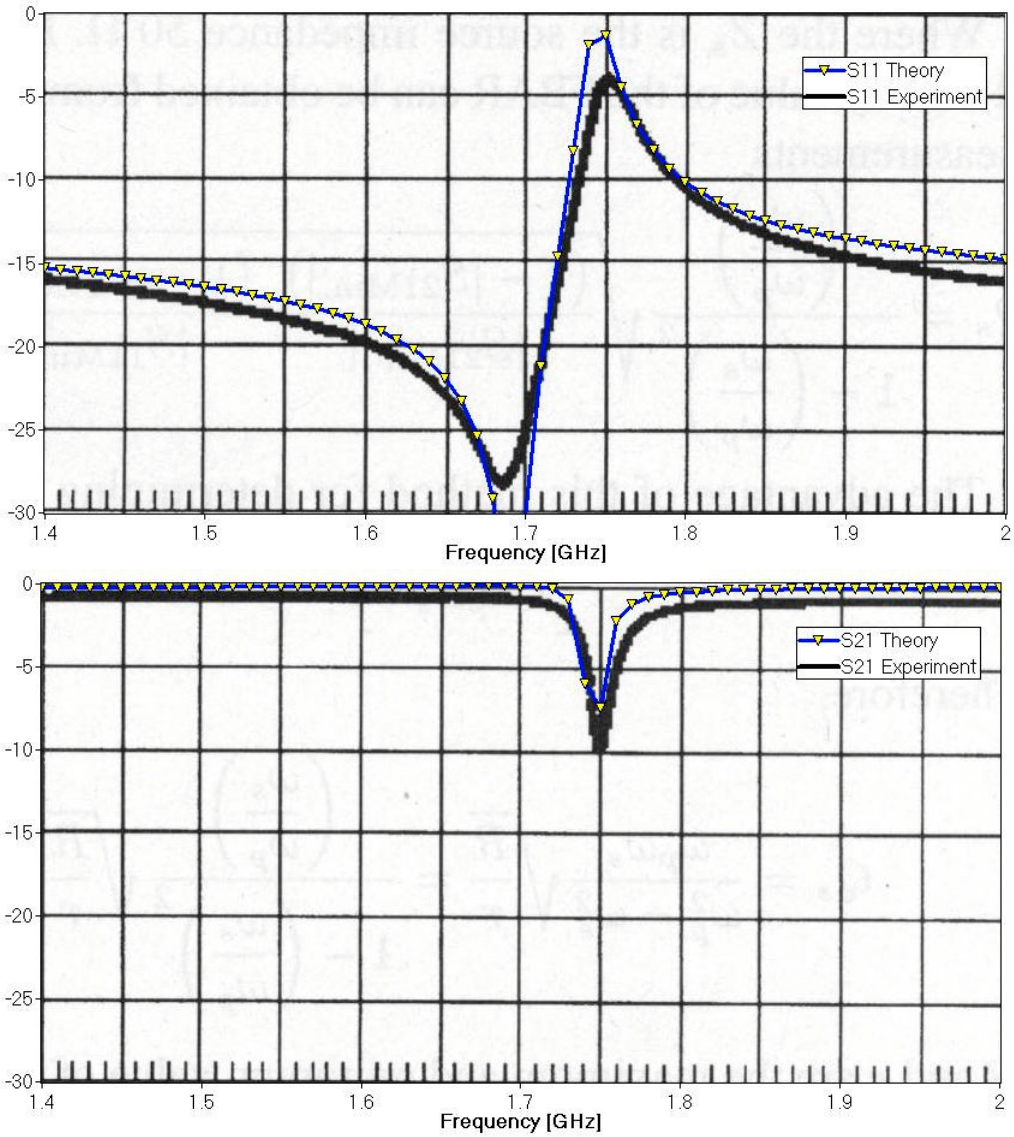


Fig.112:Experimental validation of the full-wave technique inEM3DS for a ZnO pure mode FBAR

.

XLIX Creating queues of simulations: the Batch Planner

Batch Planner is a form used to create a queue of multiple simulations/optimizations.

It can be accessed in a number of ways, such as the Tools main menu, or the icon in the Data Browser, or the button in the toolbar.

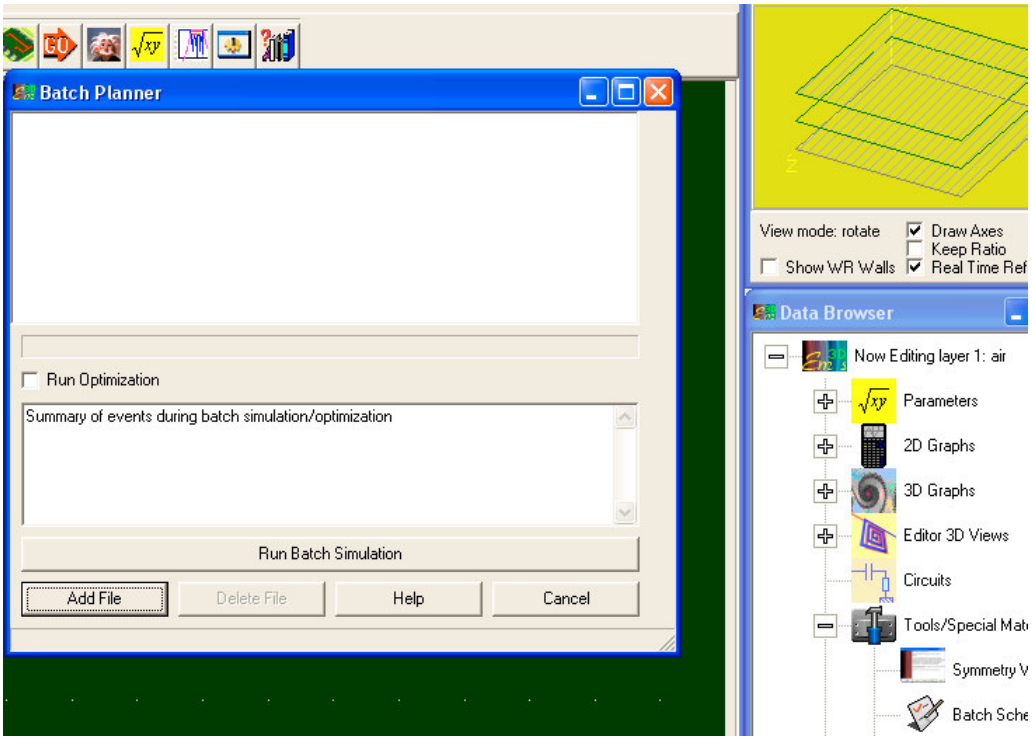


Fig.113:batch planner/scheduler .

When you click **Add File**, you are prompted to select an EMS file: the project will be opened, in order to check if it is a complete and valid file. Hence the name of the selected file appears in the list of the batch planner. Repeat the procedure, as many times you need, to create a queue, namely a complete list. You can also select a task from the list and remove it by pressing **DEL** key, or button **Delete File**. Check "**Run Optimization**" if actually your projects involve any parametric optimization. Otherwise leave this item unselected. Hence press **Run Batch Simulation**. Any file is opened, simulated and saved automatically, while a summary of events with timing is reported in the summary list below.

L Accessing and using EM3DS from AWR Microwave Office.

EM3DS works also as server for AWR Microwave Office 6 or later: This means that AWR's customers are able to see EM3DS as an electromagnetic engine, still using Microwave Office editor and post-processing capabilities. This capability is devoted to the seamlessly transfer of geometry and parameters between the AWR suite and EM3DS. To date this feature has been developed mostly in one direction: any structure designed in Microwave Office may be seen and simulated directly into EM3DS, but modifications made to the structure by EM3DS are not reported back to Microwave Office. However work is in progress to make fully bidirectional the link to the object.

In order to access this feature the user has to run the registration utility REGEM3DS.exe If the main EM3DS file, EM3DS 12.exe, is not in the same directory of the registration utility, locate it. Then click on the **Register** button. If AWR Microwave Office 6 (or following versions) is registered in your system, the process will successfully terminate.

At this point EM3DS should be available into your AWR Environment.

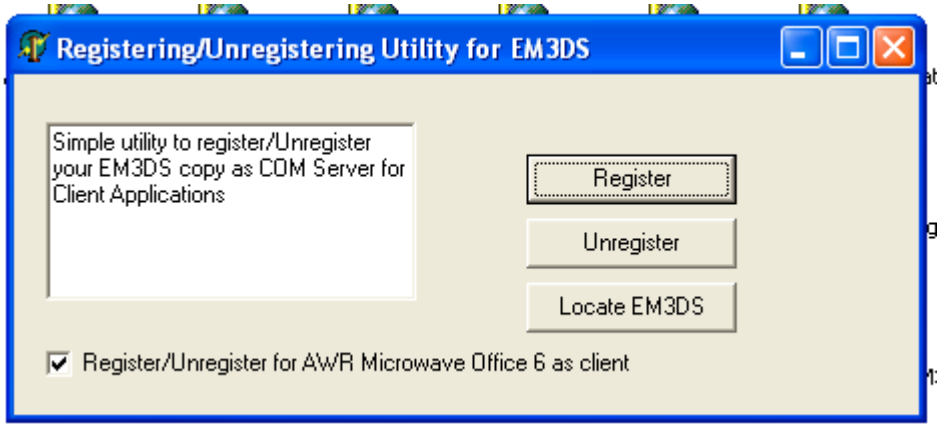


Fig.114:Registration Utility REGEM3DS

Just create a new EM structure.

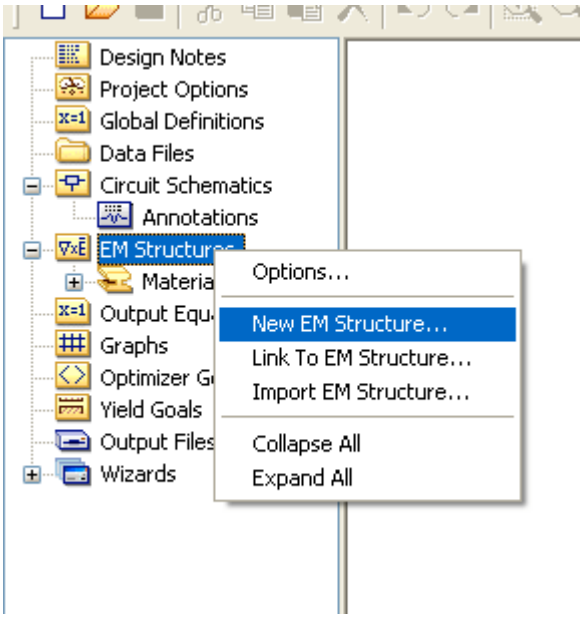


Fig.115:AWR Microwave Office 6: adding an EM structure

You will be requested of selecting if creating an EmSight or an EM3DS structure (you can also set the simulator later). Then you can proceed to draw your structure in Microwave Office. The structure can be further edited directly in EM3DS by selecting the popup menu **Open in Native Editor**.

A few notes should be considered:

- mesh options in EM3DS are still basically different from the ones of Microwave Office, mostly because in EM3DS there is not grid constraining the design. A good effort has been made to map some of the options of Microwave Office into sound mesh in EM3DS, and more work is in progress. While meshing structure only involving Manhattan polygon should be already robust, general non-Manhattan or curved structure should be checked for correct mesh into EM3DS.
- Microwave Office is 2.5D, so the structure is imported in 2.5D mode as default. If and when switching in 3D mode, check if there is any dummy layer to be deleted. Moreover if there are via connection, it may be necessary to set slicing, possibly non-uniform, for layers hosting quasi-planar parts
- Resistive conductors in Microwave Office (MWO) are so far handled by means of surface impedance. When reading the structure from MWO Em3DS assigns a fictive thickness to planar conductors and a resistivity ensuring the same DC resistance. If and when switching to 3D mode, be sure to assign the correct physical resistivity (Ωm) to the conductors in EM3DS.

Refers to the AWR's user's manual for further information about Microwave Office

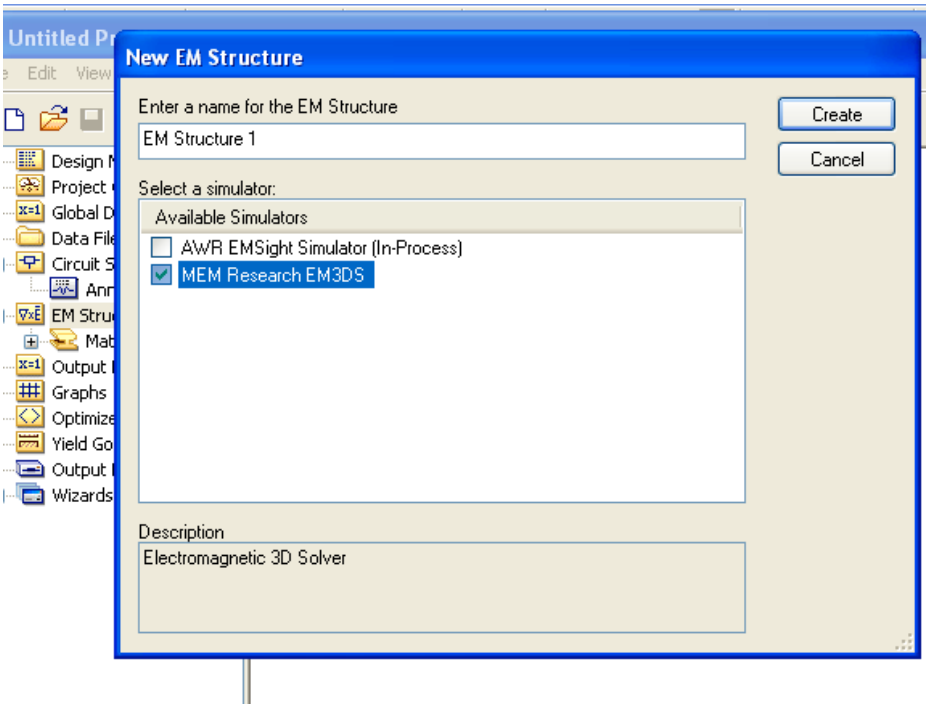


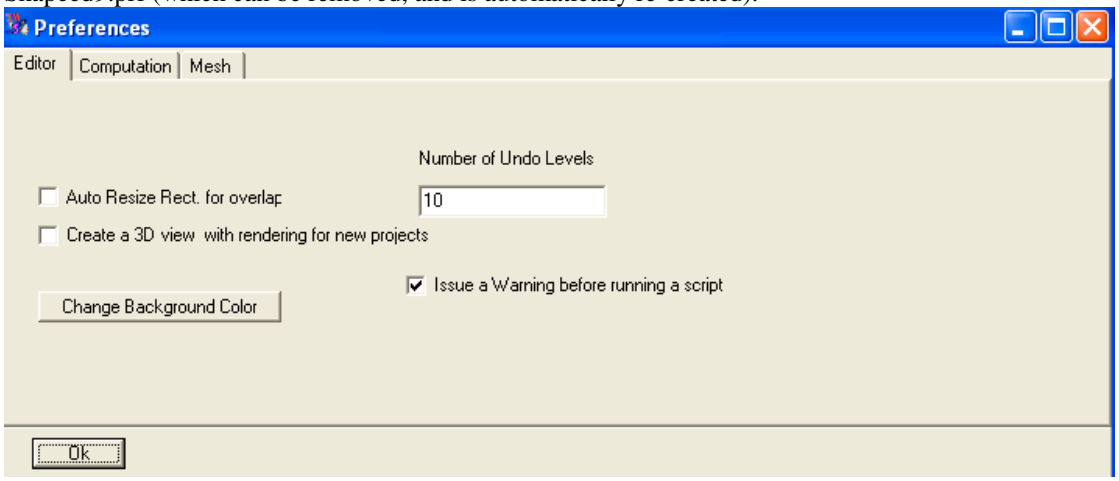
Fig.116:AWR Microwave Office 6: adding an EM3DS structure

LI The form "Preferences"

This form is accessible either by the main menu>Settings>Preferences

Aim

This is to set up some general preferences in EM3DS. The preferences are stored in a file Shapeed9.prf (which can be removed, and is automatically re-created).



Editor Preferences:

Auto resize rect. for overlap: Up to ver. 4.2, when drawing conductors having some common parts, the program deformed the design so as to guarantee the needed continuity conditions. If 2 conductors were in contact, one of them was resized in order to overlap the second one; the amount of overlapping was selected according to the "border ratio" of the larger conductor. When working with polygons, such an overlapping was not automatically performed, and the user had to do the work. In the current version this is no more necessary. Consider that in EM3DS positions are not constrained over any grid, so that establishing if two items are or not in contact is a primary issue. By right-clicking over an item, the command "Highlight Contact Shapes" in the pop up menu gives an answer to this problem, by verifying if several items visually close are or not in contact. In fact sometimes the round-off on the screen may give false impressions. Hence, in the present version no overlap is needed, even if overlapping is possible. Overlapping usually results in a denser mesh that may or may

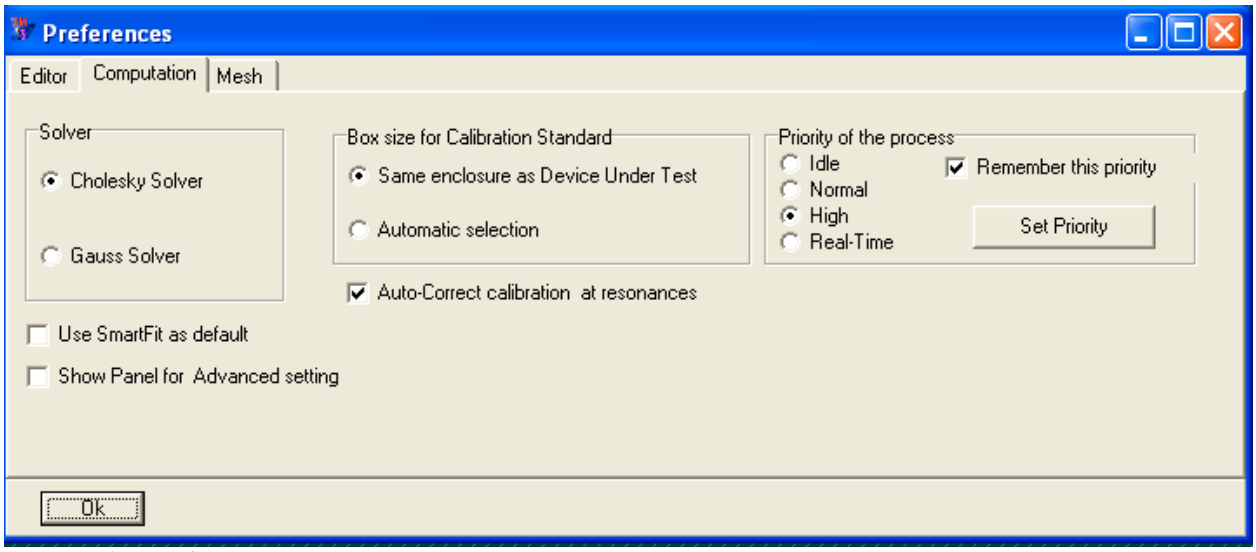
not be desired by the user. The older editing mode, namely the automatic "resize to overlap" may be enabled if desired by selecting this check box. (Note: the "resize to overlap" mode should be selected when *creating* your project. Avoid to create a project partly developed with this selection enabled and partly disabled: the editor may not be able to reopen calculated data)

Create a 3D View with rendering for new projects: Requires to create automatically at least one enhanced 3D View when defining a new project.

Change Background Color: modify the editor background color

Number of Undo Levels: Storing information for roll-back (undo) may require some time, so that in complex structures it may be necessary to reduce the number of undo levels in order to keep editing fast enough. The user can select the number of undo levels according to its exigency.

Issue a Warning before Running a Script: if a global script is defined in EM3DS (see Tools/Script Editor), issues a warning before running it



Computation Preferences:

Solvers:

Cholesky Solver: this is the method used to solve the final system. Is the faster and lighter one even if, in order to make faster the solver, the number of controls is reduced to a minimum. It works only for reciprocal structures: when modeling active devices EM3DS will load the Gauss solver regardless the user's selection

If this solver fails, the user can try with the alternative Gauss Solver.

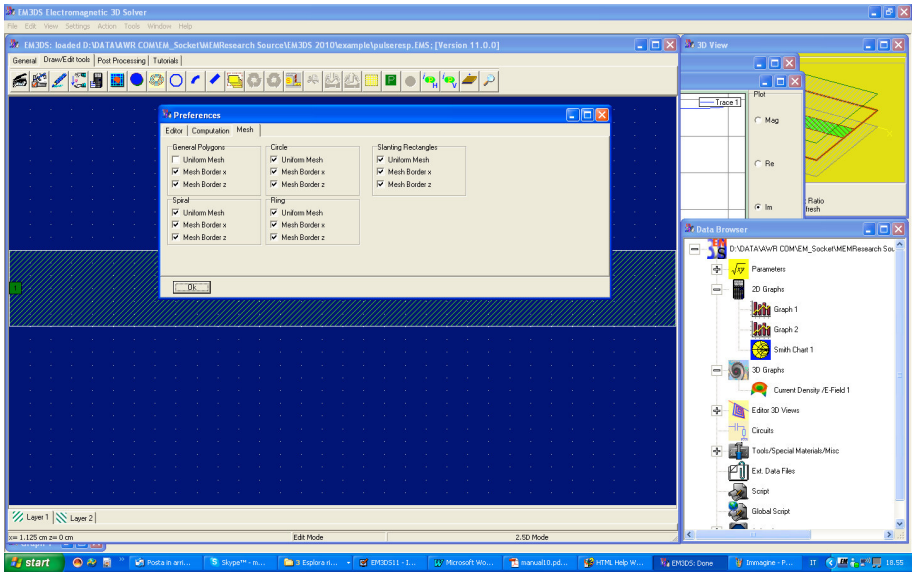
Gauss Solver: Its is a robust routine, but much slower and heavier than the previous one. For complex structures it may be several times slower than the Cholesky solver, and the peak memory occupied may be as high as double of the one needed by the Cholesky solver.

Use SmartFIT as default: if selected, any new project will automatically invoke SmartFit when running an EM simulation

Box Size for calibration standard: select the default settings for the calibration standard needed when de-embedding a simulation at the port plane. See Standard calibration for more information. Same considerations apply for Auto-Correct calibration at resonance.

Show Panel for Advanced Setting: if selected, a panel is shown; this has to do with the internal solver, and how are treated modes below cut-off; the default value is 3, and should not be changed as this is most accurate for general calculation; it can be raised to 6 or 9 for some kind of specific waveguide computations, or for the calculation of the far field; however increasing this value will deteriorate accuracy if long vias or very thick structures in 3D mode are used. For this reason, the entered value is not stored and after re-starting EM3DS it will reload the default value.

Priority of the process: It allows to select the priority in your OS of the EM3DS process. The priority is effective only when you click "set priority". The priority will be automatically settled only if you select the checkbox "Remember the priority"



Mesh Preferences:
Let you specify the default mesh options to be used when placing polygons, spirals etc.

Note: the preferences are stored in file Shapeed9.prf. You can remove this file if any issue is related to your settings: in this case EM3DS will create a new file with standard settings.

Appendix I: Built-in functions and syntax

Any parameter in EM3DS may be entered by means of expressions. The expressions may involve the following functions and operators (they are not case sensitive):

Trigonometric functions

Sin(x)
Cos(x)
Tan(x)
Cotan(x)
Arctan(x)
Arg(x)

Exponential/Logaritmic functions

Exp(x)
Ln(x)
Log10(x)
Log2(x)

General

Sqrt(x)
Sqr(x)
Abs(x)
Sign(x)
PI

Integer

Trunc(x)
Ceil(x)
Floor(x)

In the following is reported a subset of the possible messages issued by the EM3DS, with a reference to where arguments highlighting how to solve the problem (if any) may be found. Message Dialog indicates a dialog box where the message is displayed, while Event indicates that the message only appears in the Events/Messages window (silent messages). Fatal messages are the only one interrupting calculation, while any other message does not necessarily indicate errors in results.

Message Dialog: The highlighted conductor seems to contain too many mesh cells; mesh procedure may take a while. However it is possible that this high number is a consequence of an error when drawing the structure. Such a condition occurs if you have a very large structure with respect to the wavelength [e.g. you may have selected wrong units either when specifying the box size, or when setting the upper frequency of analysis] or simply you have manually selected very small mesh cells in the Mesh Options.

This is a rather self-explaining message: before performing automatic mesh EM3DS checks if the number of cells is suspect. In this case the above message is issued. By clicking over Abort button mesh is not performed and the user can check if there is actually an error. In order to avoid repeated display of this message, disable temporarily the automatic mesh (menu **Action/Mesh**)

Event : Warning: the reference plane at the XX Hand Side should be placed after at least one mesh cell. Hint: Try by raising the Border Mesh ratio XX and by checking Draw Border Ratio XX

You need to define at least one mesh cell between the reference plane and the ports to have reliable calibrated results. If this is not the case, the quality of de-embedding may be poor.

Event : Hint: as a rule of thumb [strictly valid for microstrips], in order to get reliable results the reference plane should be placed at a distance greater than the substrate thickness; check the XX Hand Side plane

If a too short reference plane, when analyzing the "standard" structure coupling via high order modes may happens between ports. This way the calibration quality may be poor.

Event : Remark: you have placed your shape in a highly conductive layer

A common error is assigning the conductivity of a shape to its embedding layer. This remark is only added with the purpose of bringing to your attention this possibility.

Message Dialog: Verify the distance of the reference plane: it is outside the conductor; it has been automatically displaced

Probably you have resized a conductor, or simply misplaced the reference plane, so that now it is defined out of the shape. Hence EM3DS set this plane so as to fit in the shape

Message Dialog: The active file belongs to a previous version of EM3DS; available results may be lost. Continue?

When developing new versions, MEM Research pays attention to backward compatibility. Hence you can always open an old project with a new version. However any saving operation will transform the older project to a new one. From one version to another the mesh algorithm may be changed (in order to improve it), and as results (.DAT) are current distributions, it is possible that the new mesh will be different from the previous one, and results not usable. Only dynamic results are lost, hence, of course, if you have exported S parameters they are still there.

Message Dialog: This file has been generated by a previous version of EM3DS: while it can be opened without problems, if you have to modify it or running a simulation, please check the mesh options; note that in the present version the "Border mesh ratio" refers to the box

This message appears when opening files generated by very old versions of EM3DS: the meaning of one parameter was changed in the newer versions, so that editing the structure should be made also considering that the Border Mesh Ratio parameter may be wrongly settled.

Event : WARNING: the suggested value for the number of modes is high.--- check the mesh options and the position of your conductors!

The number of suggested mode exceeds 500 either for x or z directions. This happens as two adjacent mesh lines are too close, and are highlighted by red color. Displacing, if possible, the red lines is

recommended

Event : Warning: the Asymptotic estimator may need more modes than presently selected
The asymptotic estimator allows to reduce the number of modes needed for the computation (with the only exception of the first frequency point, the first time a structure is analyzed). In the **Project Data** window (menu **Settings/Analysis and**....) set the **Involved Modes** to numbers *at least* equal to those recommended (otherwise accuracy at the higher frequency range is lost)

Message Dialog: Unable to start computation: no port specified
You need to specify at least one port in order to analyze a structure (fatal error)

Message Dialog: Hint: via ports are not de-embedded; no de-embed will be performed at via ports
In the present version via ports are not de-embedded.

Message Dialog: The suggested value for the number of modes to be used in the MoM matrix doesn't match the current setting. Do you want to verify the current settings?
EM3DS uses the modes of a rectangular waveguide (the one represented by the sides of your workspace) in order to build the em solution. The number of such modes needed for the solution is directly related to the minimum distance between mesh lines (forming the mesh cells). [Hence, pay attention to this parameter to reduce the computational load!]
The number of needed modes is usually equal to the ratio between the waveguide size and such a distance. If this message is being displayed, the currently selected values do not agree with this rule. By selecting "Yes", a window is displayed where you can select the button "use suggested values". Hence you will able to start the computation

Message Dialog: Warning: the dielectric brick should not assume the same permittivity of the embedding substrate!
You cannot ignore this message. For EM3DS a dielectric brick having the same permittivity as the embedding substrate is a non-sense, and calculation will be aborted

Message Dialog: Warning: the dielectric brick cannot assume zero resistivity!
You cannot ignore this message. Set correctly the dielectric brick resistivity.

Event : Warning: the "scale resistivity" option should not be used if the layer embedding the conductor has more than 1 vertical (thickness direction) subsections [Ny>1]
Checking "**Scale Resistivity**" option in **Resistivity/Material Specifications** of a shape is nearly equivalent to switching to a 2.5D point of view. This is not compatible with rising the number of **vertical subsections (Ny)**

Event : Warning: the imported structure seems to be rather complex; editing the structure before starting computation is recommended
The file contains more than 50 items. This is not a problem. However, if this is the result of an import (e.g. GDSII) you could want to consider editing of the structure before mesh or running the simulation

Event : Unable to find file XXX: Command "//CMDLOADDATA=" [load data source file] Ignored
Simply, your EMS file references to a link to an external data file (Touchstone) that EM3DS was unable to find (EM3DS looks for either in the specified path or in the same directory where the EMS file is located). This is not an error. Only plots containing that information are not displayed.

Event : Error while computing network parameters @X GHz: <solver message>; Unable to complete the system solution: unable to perform the Cholesky decomposition. Try by setting Gauss Solver as system solver
Some error occurred while solving the final system (as specified by the solver message). Normally this should not happen. If the problem is the hill conditioning of the final system you can try by switching to Gauss Solver, one of the 2 solvers EM3DS avails (menu **Settings/Preferences**). The implementation of the Gauss Solver is numerically more stable, but requires much more memory and it is much slower.

Event : Error while opening current distribution: <message>
When opening file including calculated data (.Dat), network parameters are calculated from the stored

current distributions. This error happen if EM3DS is not able to load DAT file. The structure is displayed, but results are lost. An other possibility is that a DAT file does exist, but is not calculated for the structure currently described by the EMS file (e.g. you have manually placed a wrong DAT file with the right name, or for some reason the structure in EMS was changed but results not updated). To this aim EM3DS just performs a size-check: is the number of unknowns stored in the DAT file does not match the number obtained by meshing the EMS file the message is "data not correctly linked to the EMS file". This kind of error may produce a number of additional errors in computing network parameters. Results may have no meaning.

Event : Error while computing network parameters @X <message>
There is some case where EM3DS is not able to compute the network parameters, mainly due to numerical properties of the calculated Y matrix

Event : Error while computing Network parameters for de-embed structure @X GHz: <Message>
Error in calculation of the network parameters for the calibration standards. Calibrated results may not be available (or miss some points), but raw results are calculated.

Message Dialog: The MoM Matrix may be incomplete: do you want to proceed in computing Network parameters with the available matrix?
You have aborted calculation when EM3DS was filling the system matrix (namely not all modes according to the number of selected modes were used to fill the matrix). If you believe that the number of used modes was enough (e.g. because computation was going to terminate), network parameters may be calculated from the available (incomplete) matrix

Event : Error while loading de-embedding files: <message>
Files DL*.*, DR*.* etc are where error networks are saved. These files are needed in order to re-calculate calibrated results. If there is some error in loading these files, de-embedded results may be not available.
Note that if these files cover just a sub-range of the available raw results, calibrated results will be available in that sub-range, in spite of this error message. As a general rule, de-embedded results are available where error network parameters are available.

Message Dialog: The user required waveguide calibration cannot be performed: calibration data are not calculated at the same frequency points of the current e.m. simulation
Waveguide calibration requires a calibration file calculated at the same frequency points as the available raw results.

Event : Warning: waveguide below cut-off at f X
Current theory for the waveguide calibration was developed under the hypothesis of just one mode over cut-off being accessible. Tests for general cases are in progress

Message Dialog: Unable to complete the network parameter calculation; numerical error
An unexpected numerical error has occurred (probably floating point overflow)

Message Dialog: The number of frequency points is not sufficient: at least 2 values are needed
2 frequency points (at least) are required for Spice Model Extraction

Event : Warning: de-embedding plane larger than half wavelength; check de-embedding line parameters @XX GHz'
When the error network parameters are known from the calibration procedure, they can be used so as to calculate the feeding line parameters, provided that some topology is hypothesized. The feeding line parameters are calculated correctly only if one period of the feeding line is in the calibration structure. Otherwise the displayed feeding line parameters are affected by a significant error, up to become completely meaningless. Note that calibrated results do not rely on any hypothesis and hence are reliable. Note that the feeding line parameters may lose accuracy well before this message is displayed (it is just sufficient that the whole line length in the calibration structure be near resonance: this is common to any system exploiting circuit theory!). If the feeding line parameters were of primary interest for you, try by reducing the distance of that planes from the ports.

Event : Remark: be careful; layer XX includes lossy conductors while being "non uniform y-subsections" checked; this option works only in an approximate mode for lossy conductors

Conductors may be vertically “sliced” and the user may require that the first and the last slices be smaller than the others. While such a condition allows to model correctly flowing of currents in ideal conductor, in actual lossy conductors currents may distribute itself within the whole section, and the non-uniform slicing constraints the solver to a unphysical assumption. Owing to an internal compensation algorithm, results are reliable also in such a case (only current visualization may be affected) and this remark only reminds that the internal compensation algorithm will be used.

Message Dialog: Missing nodes at line XX (Circuit solver)
Syntax error in using the Circuit solver: the parser was unable to find the nodes the component is connected to (see p.54)

Message Dialog: Missing Component Value at line XX (Circuit solver)
Syntax error in using the Circuit solver: the parser was unable to find the value for the given component

Message Dialog: Missing SUBCKT symbol at line XX (Circuit solver)
Syntax error in using the Circuit solver: any component labeled by string beginning with “X” or “N” is treated as subcircuit and has to respect subcircuit syntax (see p.54)

Message Dialog: No name specified for subcircuit at line XX (Circuit solver)
Syntax error in using the Circuit solver: any component labeled by string beginning with “X” or “N” is treated as subcircuit and has to respect subcircuit syntax (see p.54)

Message Dialog: Missing Length/Zo/Effective Dielectric value at line at line XX (Circuit solver)
Syntax error in using the Circuit solver: any component labeled by string beginning with “T” is treated as transmission line and has to respect transmission line syntax (see p.55)

Message Dialog: Node XX is floating (Circuit Solver)
Label XX indicates a floating node

Message Dialog: No Port Specified (Circuit Solver)
At least one port has to be connected to the circuit

Message Dialog: Merge Algorithm Failed
At the moment Merge does not work properly if vertices of a polygon lie directly over one of the side of the merging polygon: a small partial overlap allows to circumvent such a problem.

Message Dialog: Be careful: the control plane is defined outside the channel; its co-ordinates should fall in the channel
The control plane is the plane along which the control voltage is calculated. It is a plane parallel to the gate electrode. Usually the control region is the depletion region beneath the gate electrode.

Message Dialog: Be careful: the control layer should not be the same containing the channel
The control plane is the plane along which the control voltage is calculated. Usually the control region is the depletion region beneath the gate electrode; it cannot be the channel itself.

Message Dialog: A controlled source cannot be applied to an ideal conductor; please modify the resistivity value
Channel cannot have infinite conductivity!

Message Dialog: A controlled source cannot be inserted into a layer having "non-uniform" slicing
In this version you cannot define an active region in a layer having “non-uniform” slicing selected

Message Dialog: Warning: the inserted dielectric brick will share the magnetic permeability (>1) of its embedding layer
This is just a reminder; the relative magnetic permeability of any object defined in a layer is the same of the embedding layer.

Event : Remark: de-embedded results could lose accuracy over and near XX GHz: the enclosure of the structure is a waveguide having a different number of over cut-off modes with respect to the calibration structure for the Left ports; Hint: if this is the case you could try by displacing

the de-embedding plane or add to your structure top and bottom metal covers
Calibration requires that a set of different structures be automatically built and calculated, the "standards", so as to be able to extract any effect due to port discontinuity and feeding lines. Whenever the device under test (DUT) is placed in an open waveguide or an infinite waveguide (along y), it may happen that the different structures are enclosed in waveguide having different modes above cut-off. If this is the case, each structure has a different amount of energy leaking toward infinity or free space. The calibration algorithm assumes this energy be the same: if this condition is violated you may find even reflection coefficients>1 in a *calibrated* passive circuit (note that the raw results are ok). The remark highlights that there is such a possibility. A different position of the de-embedding plane produces different calibration structures, and this is the reason of the suggestion. Of course if the structure is completely boxed no problem of energy leakage may arise.

Event : WARNING: no active brick should be defined between the port and its de-embedding plane; this happens at XX port(s); the standard being defined for de-embedding may lead to wrong results!
Active regions cannot appear in calibration structures; hence you cannot place active regions between a reference plane and a port.

Message Dialog: Cannot start computation in 2.5D mode: either a vertical (via) port is connected to a shape in an odd (path) layer or a horizontal (edge) port is is connected to a shape in an even (via) layer. Please remove the wrong port or switch to 3D mode by means of the menu "Settings/Analysis and subs. Settings".
In 2.5D mode the odd indexed layers (1,3,5…) are considered to host only standard thin conductors, while even layers only via conductors. Standard thin conductors are modeled by surface currents (Jx and Jz), so that only Edge ports (which are calculated by evaluating currents Ix and Iz) can be attached to those conductors. On the other hand via conductors are modeled by just volume Jy, and consequently only via ports may be attached. If any of the two conditions is violated in 2.5D mode, the engine will stop the calculation with this error message.

Event : WARNING: de-embedding plane larger than half wavelength; check de-embedding line parameters @..
Possible failure of the extraction of parameters for feeding lines; however this failure does not affect the accuracy of the calibration.

Event : Caution! interpolation is made over a large set of input points; interpolation might be poor.
Interpolation, either during a SmartFIT run or in post-processing, is performed over a too large data-set. This may raise numerical issues. One should attempt to reduce the number of input points, below 40-50.

Event : Caution: either interpolation or EM solution are not reliable at…GHz.
Results obtained during interpolation or SmartFIT for a reciprocal circuit violate PR conditions, namely does not respect passivity conditions. This may be either to the interpolation, or to a poor calibration in the EM results (e.g. when simulating antennas or open structures)

Message Dialog: the selected polygon includes collinear (redundant) vertices. Deleting collinear vertices is usually advisable, and it may be particularly important when defining ports. Delete collinear vertices before attempting to define a port? .
This message is issued when trying to add an edge port to a polygon, on a side, which includes collinear points. Collinear points are not allowed when defining ports.

Message Dialog: In order to add an edge port you have to click over a conductor completely sharing at least one side with the enclosure; additionally no more than one port PER SIDE can be added to a SINGLE object; click near the enclosure at the desired side.
This message is issued when trying to add an edge port when:
1) you have clicked "too far" from an edge
2) the edge is not touching the box (check out also for round-off errors in coordinates)
3) you have a polygon which share more than one side with the box

Message Dialog: This is BASIC Mode license; simulation in 2.5D mode is not allowed
Calculation aborted as your license does not allows 2.5D simulation
Message Dialog: The number of ports no longer matches the requirements in the goal list; some

of the edge ports is probably no longer available; Aborting
You have probably inserted in the optimization list variables controlling the position of conductors with edge ports attached; as effect of the optimization process, some port is no longer defined (the conductor no longer touches the box) and consequently the optimization is aborted.

Message Dialog: no variable to optimize; please indicate by "Y" which variable in the above list should be used in the optimization; you can define variables in the variable sheet and use them for the parametric definition of geometries
Optimizer needs a parametric geometry to be defined

Message Dialog: Attempt to connect to remote license server failed: message.
You are using EM3DS license in floating mode, and EM3DS was not able to connect to the license manager in the host computer.

Message Dialog: Server cannot enable remote license; the max number of allowed license may have been exceeded
You are using EM3DS license in floating mode; EM3DS connected to the license manager but did not get permission to run; this may be either because the number of available licenses is zero, or because the license manager was settled to filer out requests from your PC, namely from you TCP/IP.

Message Dialog: License for a previous version of EM3DS xx. In order to run x.x version you need to remove the existing license and request an updated license code;Note that the new code will keep enabled the previous version of EM3DS. Do you want to remove the existing license now?
EM3DS 8 (and more) will require a new license; the new license will work for previous versions too, allowing different versions to coexist in the same system.

Message Dialog: Running Server is for an older version of EM3DS
Since ver EM3DS 8, each new version of EM3DS requires its updated license manager. Note that e.g. License Manager 8 will work for EM3DS 7 too.

Message Dialog: the number of ports of the two network files (even and odd) MUST be equal; your current selection is invalid.
This error is reported by the symmetry wizard: it indicates that files selected for recovering the full port response are inconsistent.

Message Dialog: Cannot start computation: an edge port is attached to a magnetic wall (PMC). Please remove the wrong port or modify the enclosure by "Settings/Analysis and subs. Settings".
A port is defined by means of an E-field orthogonal to the conductor section; hence it cannot be defined between a conductor and a PMC, as E-field orthogonal to PMC is zero.

Finally there are specific messages, pertaining to the importing/exporting filters.

Moreover set of messages refers to the GOS file, namely the file where graph settings and formatting information are saved. GOS files are ascii files, and hence EM3DS also indicates where the error occurred. This kind of error has no effect, and only graph formatting (legends, axis limits etc) may be lost. The most common error is the one issued whenever an external data file is not found, and some graphics refer to that file. Recall that EM3DS seeks for external data file either 1) in the complete path stored when saving EMS file or 2) in the same directory of the EMS file (hence you can quickly fix the problem).

NOTE

While the Authors believe that the information and the codes are correct, all parties must rely upon their own skill and judgement when making use of it. The Authors do not assume any liability to anyone for any loss or damage caused by any error or omission, whether such error or omission is the result of negligence or any other cause. Any and all such liability is disclaimed.

Summary