

m/mafl - The EM Technology File Editor for RFIC

Table of contents:

m/mafl - The EM Technology File Editor for RFIC	1
1. Introduction	2
2. What file formats and design flows are supported?	2
Read & write *.mafl	2
Read existing *.son simulation models	2
Export *.son and *.lay for Sonnet GDSII workflow	3
Export *.son and *.lyr for Sonnet DXF workflow	3
Export *.gym for Empire XCell 3D EM workflow	3
Export *.subst for ADS 2011 workflow	4
Import *.slm from Agilent ADS 2009 Momentum	5
Import *.subst from Agilent ADS 2011	5
Import from Assura Procfiles	5
3. Editor window and preview window	7
4. Preview settings	8
Fixed height view	8
Semi-proportional height	8
Metal Loss Property	8
Zoom	8
5. Dielectric Layers	9
Edit dielectric	9
Add new dielectric	9
Split dielectric	10
Merge dielectrics	10
6. Metals	11
Edit metal	11
Add new metal	12
Add new metal here	12
Clone metal	13
Metal display in the preview window	13
7. Vias	14
Edit via	14
Via calculator	15
Add new via	16
Add new via here	16

Document revised: 27. March 2013

Document revision: 3.0

1. Introduction

With m/matl, you can create, modify and visualize the material definition and cross section of EM stack-ups for use with the Sonnet EM simulator. For easy re-use of existing technology definitions, you can import and export from/to various other file formats.

If you need to support one semiconductor technology with multiple design flows, such as Cadence Virtuoso based and GDSII file based, you can use m/matl as the single source for all required technology, template and layer mapping files.

2. What file formats and design flows are supported?

Read & write *.matl

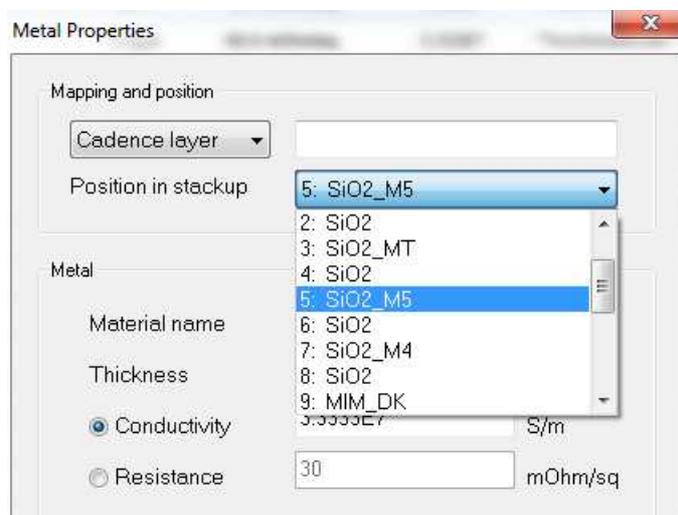
The primary file format for m/matl is the Sonnet *.matl file format, as used in Sonnet's interfaces to Cadence Virtuoso and Agilent ADS. The *.matl file includes all required information for using Sonnet in a Virtuoso or ADS workflow: metal, via and dielectric definitions and their position in the stack-up, along with the respective Cadence/ADS layer names.

To enable m/matl as a single data source for multiple design flows, we have extended Sonnet's *.matl file format, to also store optional layer mapping information for other file formats. These optional GDSII and DXF mappings in the *.matl file have no side effect when the file is used with Cadence and ADS workflows, they are simply ignored.

Read existing *.son simulation models

You can import *.son files into the m/matl editor. Contrary to the *.matl file, the metal and via definitions in an empty *.son project do not specify the actual position in the stack-up.

If the imported project includes actual geometries along with the material definitions, m/matl will evaluate their stack-up position and the used metal and via definitions, to identify which metal/via definitions are used in which stack-up position. If the imported Sonnet project is empty, with no geometries, there is no way how the software can identify the corresponding levels for metals and vias. In this case, you must assign the stack-up positions manually.

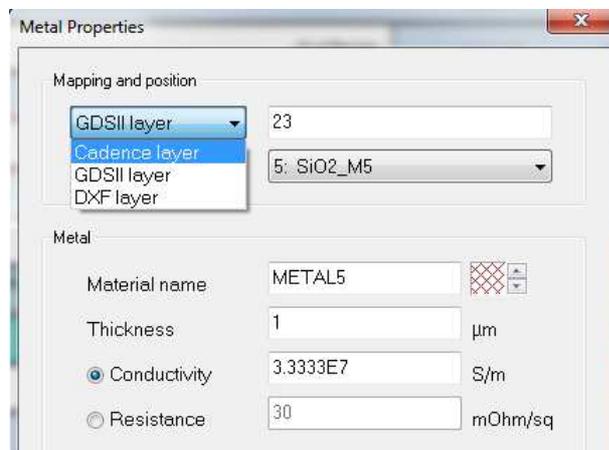


Export *.son and *.lay for Sonnet GDSII workflow

When using Sonnet with GDSII data, the Sonnet project editor xgeom can import the GDSII geometries with a template (empty Sonnet project) that holds the dielectric stack-up and metal and via definitions. Contrary to the *.matl file, only the conductor properties are defined in the template, but not their position in the stack-up. This data is supplied during the GDSII import process with an additional file, the layer mapping file (*.lyr).

To summarize: in a GDSII workflow, we need a *.son template file with the material definitions and a *.lay layer mapping file. With m/matl, you can create these files.

The required GDSII layer numbers must be defined from the m/matl metal and via dialogs, because they are not part of the usual *.matl information. However, once it is defined in m/matl, the information can be stored along with all other data in our extended *.matl file format.



Export *.son and *.lyr for Sonnet DXF workflow

The DXF workflow is similar to GDSII, with a *.son template and a layer mapping file. For DXF, the layer mapping file is *.lyr and uses a different syntax than the GDSII layer mapping file.

Export *.gym for Empire XCcell 3D EM workflow

With this export option, you can create an empty project (template) for the Empire XCcell 3D EM simulator. The template includes the dielectric and material definitions. Dielectrics are included, with the x and y dimensions as defined in m/matl under File > Preferences > Cell Size and Box size. Dielectric size is Cell Size x Number of Cells.

Note that simulation settings default to lossless metal and lossless dielectrics, so you have to set these (and other) simulation settings manually.

Status of the Empire XCcell export option is: experimental
Carefully check the models that you have created with this option.

Export *.subst for ADS 2011 workflow

The new March 2013 release now introduces support for ADS 2011 *.subst files, which can be imported and exported.

IMPORTANT:

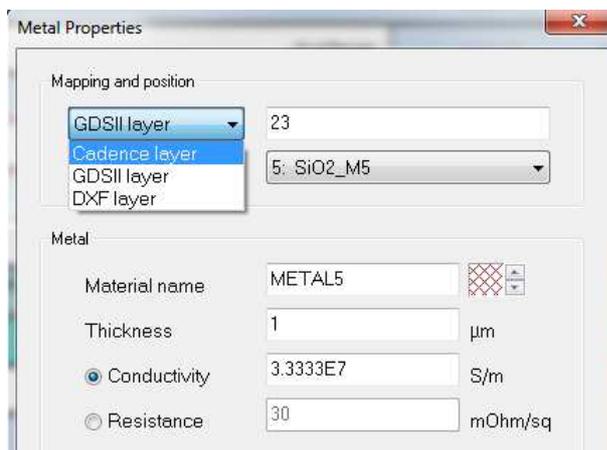
Unlike the old *.slm files, you cannot easily move/copy the ADS 2011 substrate files!

The *.subst file does not include the full EM stackup information, and references the shared material definitions file. That shared file materials.matdb is always located in the same directory as the *.subst file, and has all material definitions for all substrates in this library. Thus, if you just copy it around, this will have side effects on the other substrate files!

The location for all substrate files is the library folder. When m/matl exports an ADS 2011 substrate file, the corresponding material definitions are appended to materials.matdb. If materials.matdb does not yet exist, it is created. If you have multiple substrate files in one library, make sure that you export to that shared file location, because that will append the material definitions to an existing materials.matdb.

Another detail to be aware of with ADS 2011 substrate files: The layer names for metals and vias must be specified by the layer **number**, not the layer name. This is caused by Agilent's definition of the *.subst file.

To set the Agilent layer number for metals and vias in m/matl, use the GDSII layer input field. We decided to share that setting because in RFIC/MMIC technologies, we often have ADS layer name = GDSII layer name. By referencing the GDSII layer number setting, we can easily export existing m/matl substrate file with GDSII layer number to ADS 2011 *.subst files.



The name of the *.subst file is appended as a suffix to the material name, to avoid side effects on other material definitions in the same file.

Known limitations: The m/matl tool was originally designed for Sonnet stackup files, and supports metals that have the same thickness as the dielectric they are embedded into, or even penetrate into the dielectric above. This is valid in Sonnet, but not valid in ADS 2011. In the present version, m/matl exports the metal thickness as you have defined it, and does not fix possible metal thickness violations. Invalid metal thickness will be reported as an

error by the ADS substrate editor. You, the user, should make sure that you define metal thickness that is valid in ADS. One feature that might help you is the merging of dielectric into a new thick dielectric with the equivalent permittivity, as described in chapter 5.

Import *.slm from Agilent ADS 2009 Momentum

*.slm is the file format used by Agilent ADS version 2009 and earlier. It is a single file which has all the EM stackup information for the ADS Momentum EM simulator.

The *.slm already includes the stack-up position and layer mapping information. Some differences exist in the definition of thick metal between Momentum and Sonnet. These should be handled automatically in many cases, but you should always check the result after import.

Import *.subst from Agilent ADS 2011

*.subst is the file format used by Agilent ADS version 2011 and later. The file does not include the full EM stackup information, and references the shared material definitions file. That shared material definitions file is always called materials.matdb and is located in the same directory as the *.subst file.

Note that *.subst option thick metal expansion down is not supported at the present time. Also, finite dielectrics (dielectric vias) for the FEM solver are not supported.

Import from Assura Procfiles

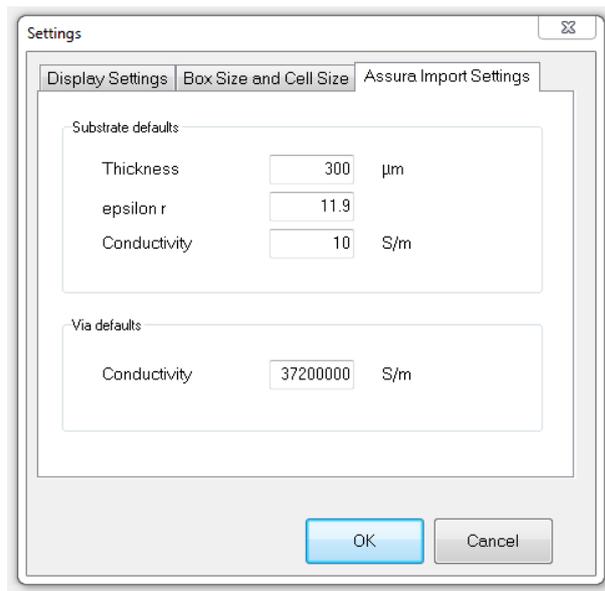
You can import substrate information from existing Cadence Assura procfiles.

To use this option, use File > Import and select an Assura procfile. The software will evaluate the procfile, and also try to find and evaluate two additional files in the same directory: p2lvsfile and lvsfile. If these additional files are not available, the via information is missing and you need to set the via layout (from... to ...) manually.

Unfortunately, the layer names used in the Assura files do not always agree with the layer names used in the layout editor. During import, m/matl tries to figure out the layer names, as best as it can. Sometimes, this is not possible and you need to set the correct Cadence layer names manually.

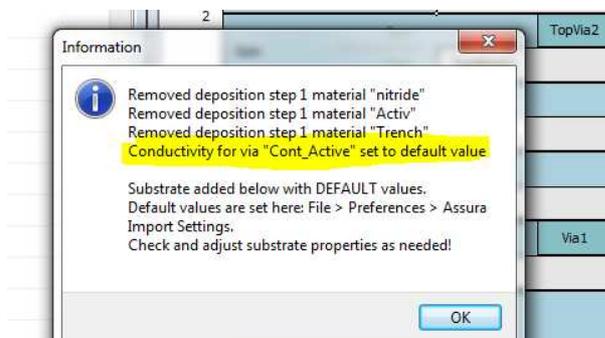
Important limitations for using Assura procfile import:

It is important to understand that the substrate itself is not included in any of these files. The substrate thickness, permittivity and conductivity are added based on m/matl default values. These default values can be set with File > Preference > Assura Import Settings.



In this dialog, there is also a settings for via conductivity. The via conductivity might be defined in the Assura files, or it might be missing. If the information is missing, the default will be used and this default assignment is listed in the import summary message.

Another note in the import summary message is related to the deposition steps. At the surface of the substrate, there are multiple deposition steps 1 defined, for different areas of the chip. The m/matl import uses only the first line for deposition step 1, which is usually for field oxide, and skips the other deposition step 1 entries. These skipped lines are listed in the import summary message.



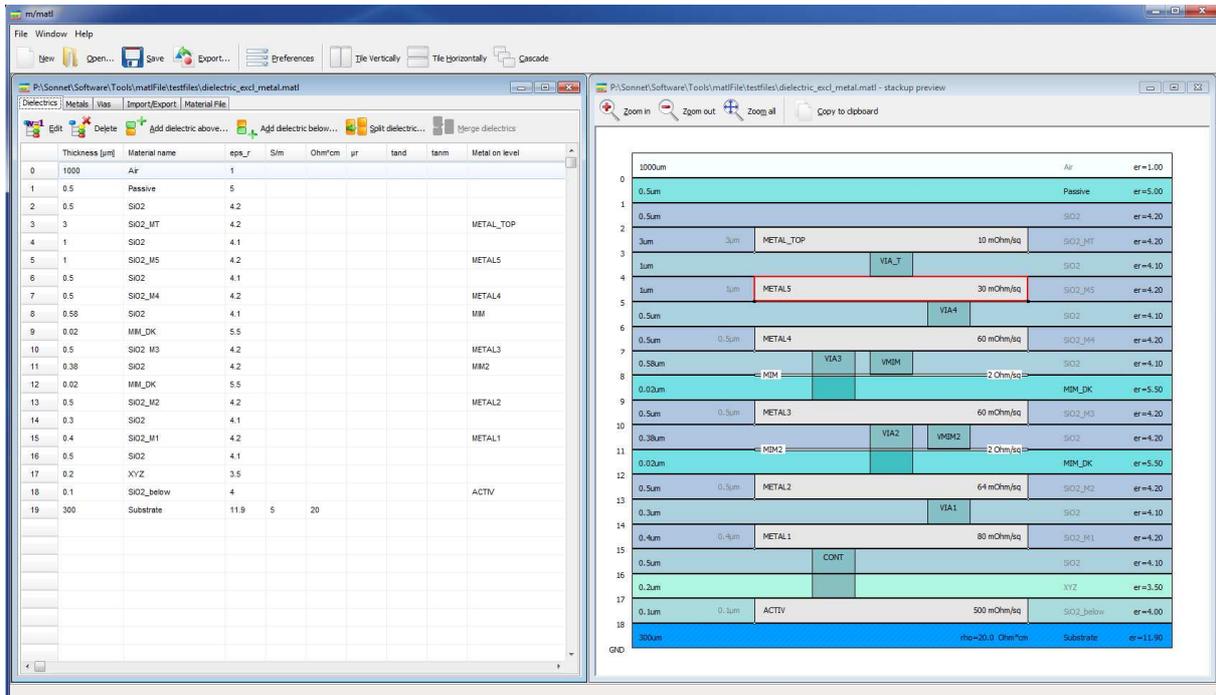
For metal sheet resistance specified in the p2lvsfile as (Ri,wi) pairs for width dependent sheet resistance, m/matl uses the last entry, which is supposed to be valid for widest metal traces.¹

Carefully check your results! The Assura import has been tested and optimized with many different design kits. However, there is some “educated guess” required in some cases, especially for layer names. If you experience problems, please let us know!

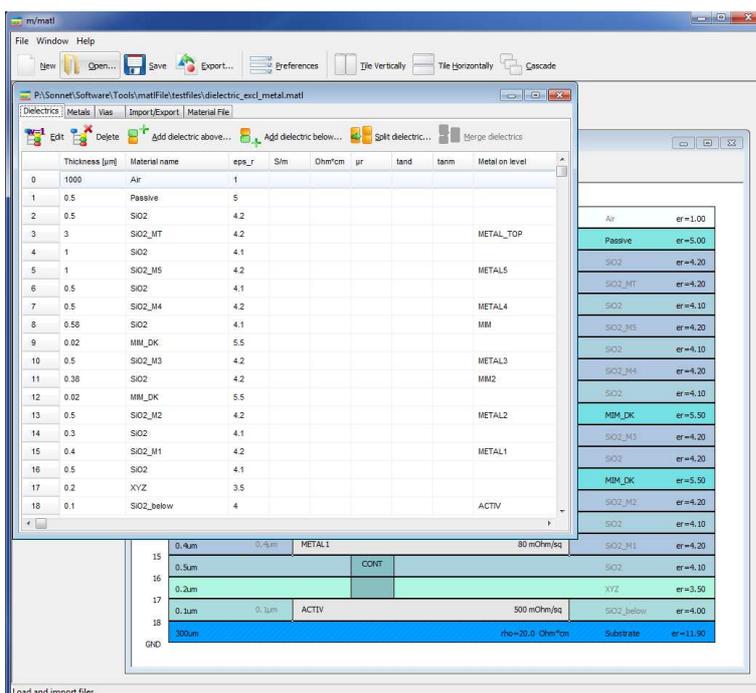
¹ From the Assura documentation: “Additionally, the (ri,wi) pairs should be specified in order of increasing width.”

3. Editor window and preview window

When you load in a file, two windows are shown: the editor window and the preview window. The editor window is the place where you can apply modifications, and has several "tabs" for dielectrics, metals, vias and file view. The preview window shows a graphical representation of the stack-up.



You can re-arrange the windows by using the Window menu or the Tile/Cascade icons. When you try to close a preview window, it is only minimized. To actually close a file, you have to close the editor window.



4. Preview settings

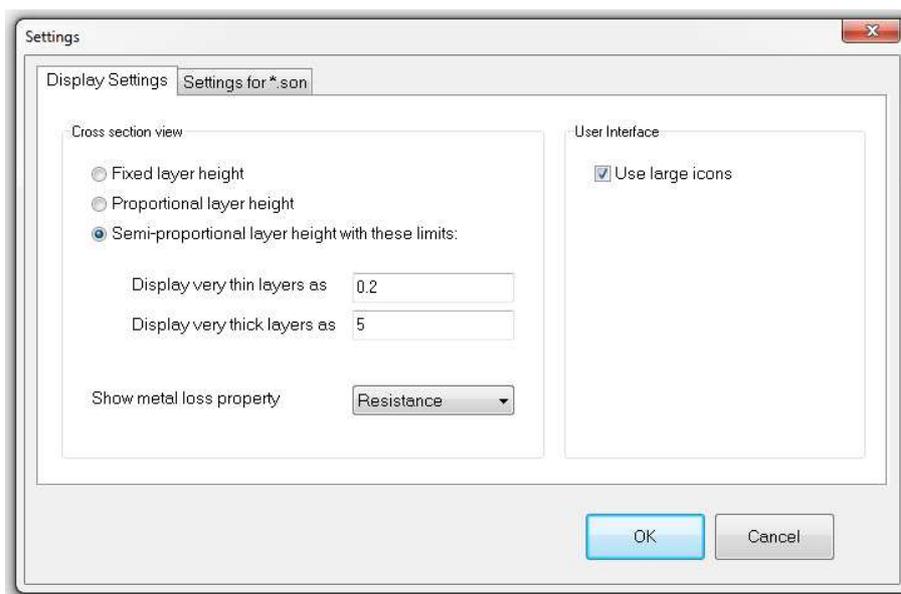
For preview, you can choose between "fixed height" view and "semi-proportional height" view. This setting is found in File > Preferences > Display Settings.

Fixed height view

With "fixed height" view, all dielectrics are shown with the same height, no matter what their physical thickness is. Metals that have the same thickness as the dielectric are shown over the full height of the dielectric. Metals that are thinner than the dielectric are shown with half the dielectric height. Metals that are thicker than the dielectric penetrate into the dielectric above.

Semi-proportional height

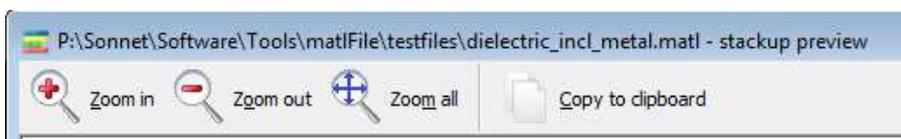
With "semi-proportional height" view, you can visualize the differences in dielectric thickness and metal thickness. However, limits apply for very thin dielectrics (such as MIM layers) and very thick dielectrics (such as the substrate or air above), to obtain a nice and clean view without. You can set the effective display thickness limit that is applied to very thick and very thin layers.



Metal Loss Property

In the Display settings dialog, you can choose if metal loss is shown in the preview window as conductivity [S/m] or sheet resistance [mOhm/square].

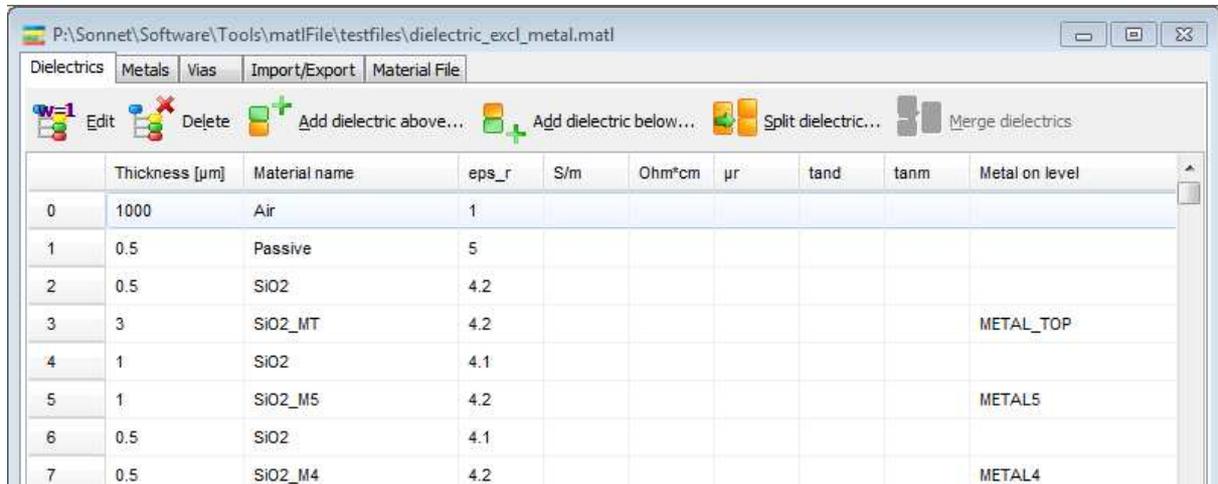
Zoom



Use the Zoom in, Zoom out and Zoom all icons to select the desired level of detail for the stack-up preview. Some dimensions will be hidden if the zoom level is too small.

5. Dielectric Layers

The dielectrics tab of the editor window shows a list of all dielectrics and their properties. The order of the list corresponds to the physical position in the stack-up.

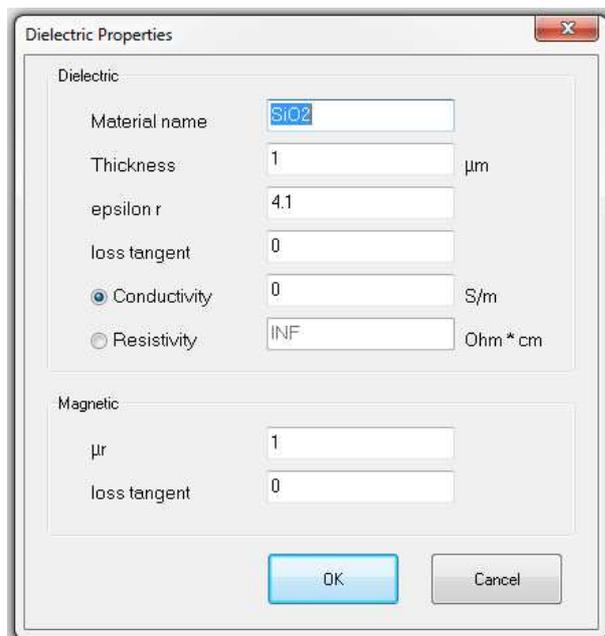


	Thickness [µm]	Material name	eps_r	S/m	Ohm*cm	µr	tand	tanm	Metal on level
0	1000	Air	1						
1	0.5	Passive	5						
2	0.5	SiO2	4.2						
3	3	SiO2_MT	4.2						METAL_TOP
4	1	SiO2	4.1						
5	1	SiO2_M5	4.2						METAL5
6	0.5	SiO2	4.1						
7	0.5	SiO2_M4	4.2						METAL4

Edit dielectric

To edit a dielectric, double click on an entry in the dielectric table, or double click on a dielectric in the preview window. This will bring up the "Edit" dialog.

For conducting dielectrics, e.g. the substrate, you can choose if you want to specify the conductivity as [S/m] or specify the resistivity as [Ohm*cm]. Your input will be converted to the format that Sonnet uses internally.



Dielectric Properties

Dielectric

Material name: SiO2

Thickness: 1 µm

epsilon r: 4.1

loss tangent: 0

Conductivity: 0 S/m

Resistivity: INF Ohm*cm

Magnetic

µr: 1

loss tangent: 0

OK Cancel

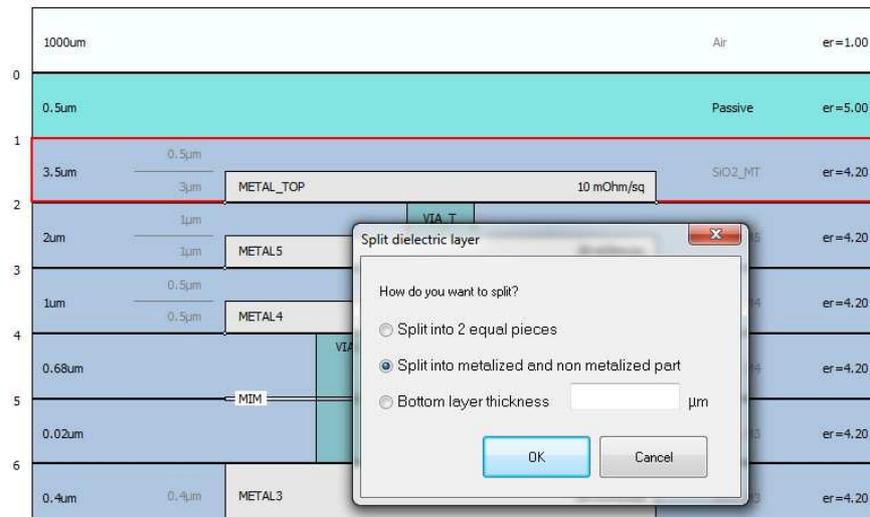
Add new dielectric

To add a new dielectric, select an existing dielectric in the editor window, or in the preview, and then use "Add dielectric above" or "Add dielectric below".

Split dielectric

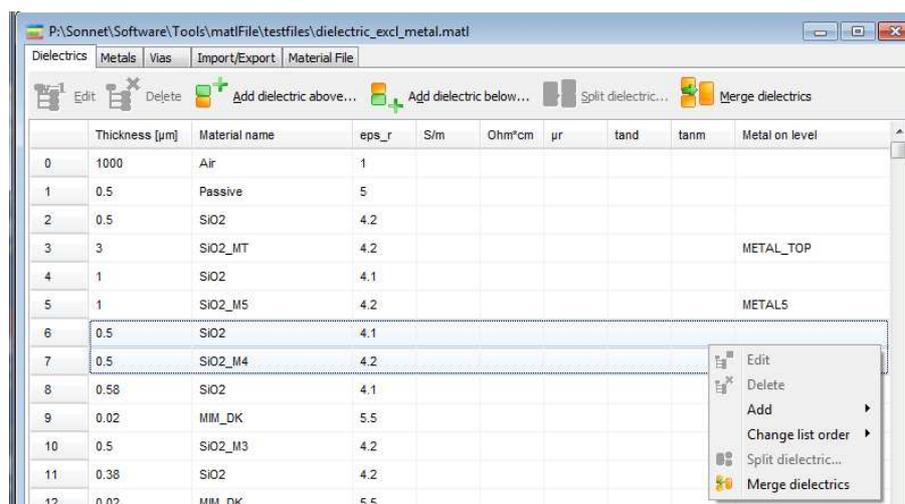
You can split an existing dielectric layer into two pieces. This can be done, for example, if you prefer a stack-up where thick metals have their "own" dielectric, instead of penetrating partially into the dielectric above.

To split a dielectric, select it in the editor window or in the preview window, and then click on the "Split dielectric" icon, or use the right mouse button menu. You will be prompted how you want to split. If there is a thick metal defined on that dielectric, you can choose to split into a metalized and a non-metalized part.



Merge dielectrics

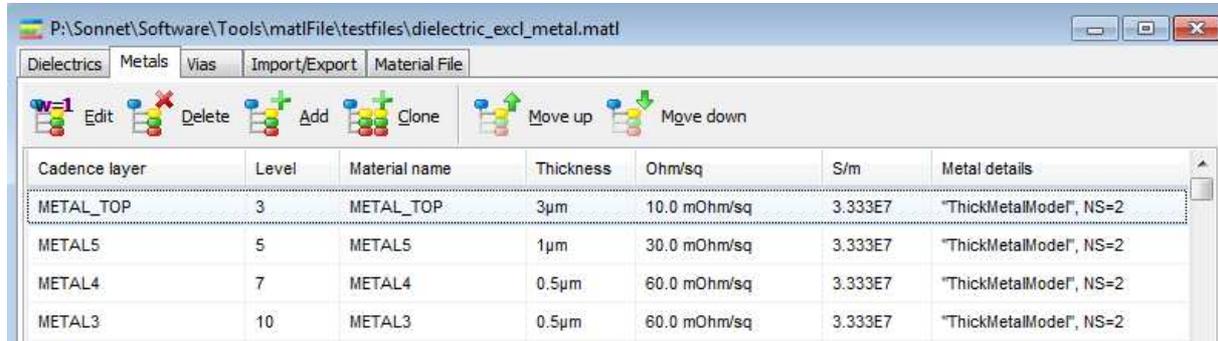
This is the opposite of splitting: you can also combine two dielectric layers into one. To do so, use the editor window to select the dielectrics that you want to merge (multi-selection in the dielectrics table), and then click on the "Merge dielectrics" icon.



The resulting dielectric will be called "merged" and the new permittivity ϵ_r will be calculated based on the effective vertical permittivity. This ensures that a plate capacitor would still see the same vertical permittivity ϵ_r after merging.

6. Metals

The metals tab of the editor window shows a list of all metals and their properties.



Cadence layer	Level	Material name	Thickness	Ohm/sq	S/m	Metal details
METAL_TOP	3	METAL_TOP	3µm	10.0 mOhm/sq	3.333E7	"ThickMetalModel", NS=2
METAL5	5	METAL5	1µm	30.0 mOhm/sq	3.333E7	"ThickMetalModel", NS=2
METAL4	7	METAL4	0.5µm	60.0 mOhm/sq	3.333E7	"ThickMetalModel", NS=2
METAL3	10	METAL3	0.5µm	60.0 mOhm/sq	3.333E7	"ThickMetalModel", NS=2

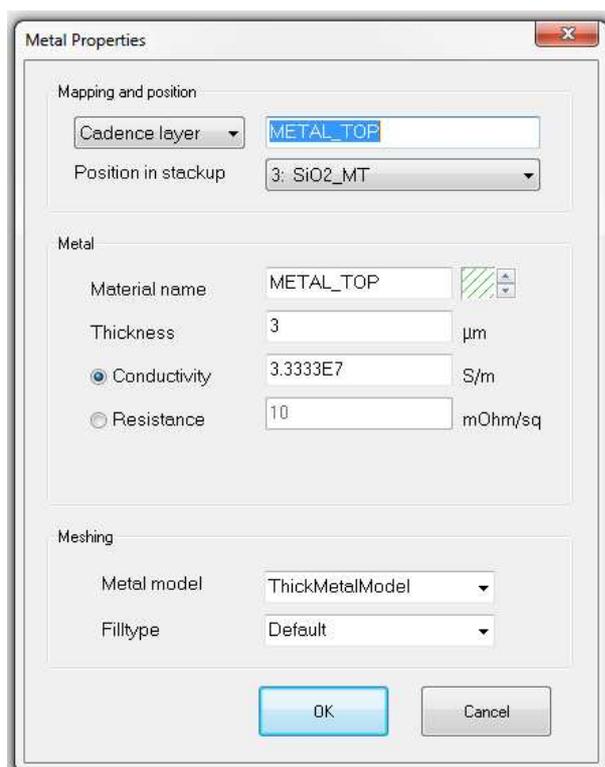
For metals, the list order does not have any effect on results. If you prefer to change the list order, you can use the "Move up" or "Move down" icons.

Edit metal

To edit a metal, double click on an entry in the metal table, or double click on a metal in the preview window. This will bring up the "Edit" dialog.

In the "Mapping and position" section, you can set the Cadence/ADS layer name for this metal, and the position in the stack-up. If these two settings are not defined, the definition is stored in the *.matl file as a "MATERIAL" entry instead of "METAL" entry, and can be mapped later in the Cadence/ADS interfaces.

The material name is only a name used internally in Sonnet. By default, when you assign the Cadence layer name, this will also set the material name to the same value.



Metal Properties

Mapping and position

Cadence layer: METAL_TOP

Position in stackup: 3: SiO2_MT

Metal

Material name: METAL_TOP

Thickness: 3 µm

Conductivity: 3.333E7 S/m

Resistance: 10 mOhm/sq

Meshing

Metal model: ThickMetalModel

Filltype: Default

OK Cancel

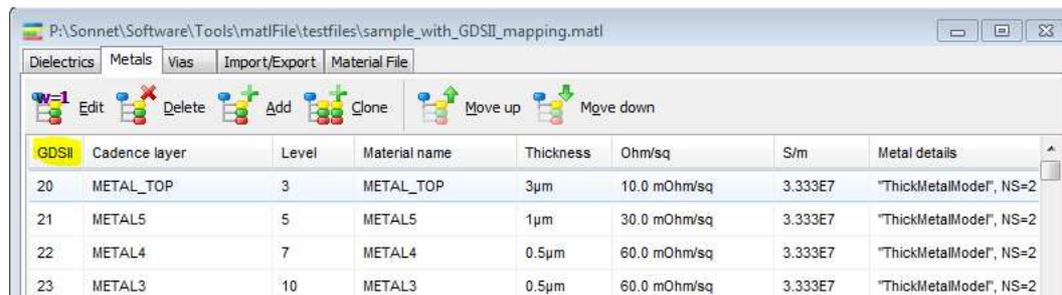
For the metal losses, you can specify conductivity σ in [S/m] or DC sheet resistance in [ohm/square]. The sheet resistance is calculated as $1/(\sigma * \text{thickness})$.

If you want to define a lossless metal, you can type "INF" for the conductivity, or set sheet resistance to zero. For lossless metal, the thickness may be defined as 0 μm . For lossy metal, you must assign a valid thickness value.

The Sonnet metal model - normal (thin) metal or thick metal model - can be set with the drop down list at the bottom of the dialog. If the *.matl file was imported, it might also contain some other, unusual metal definitions, which are then shown in the drop down list.

The fill type is set in the bottom drop down list. Typically, this is set to default, so that the actual mesh type (staircase or conformal) for that layer is determined by the setting in the Cadence/ADS interface. If the fill type is fixed to "Staircase" or "Conformal", instead of "Default", then this value will always be used for this metal/layer, regardless of the setting in the Cadence/ADS interface.

For use with GDSII and DXF, you can also set GDSII and/or DXF layer names. Just change the layer name drop down list to "GDSII layer" or "DXF layer" and type in the value. If at least one DXF or GDSII layer is defined, the corresponding column will also be shown in the metal table.



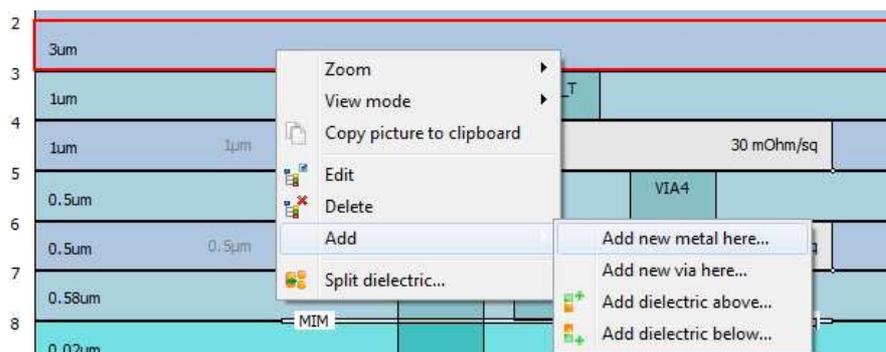
GDSII	Cadence layer	Level	Material name	Thickness	Ohm/sq	S/m	Metal details
20	METAL_TOP	3	METAL_TOP	3 μm	10.0 mOhm/sq	3.333E7	"ThickMetalModel", NS=2
21	METAL5	5	METAL5	1 μm	30.0 mOhm/sq	3.333E7	"ThickMetalModel", NS=2
22	METAL4	7	METAL4	0.5 μm	60.0 mOhm/sq	3.333E7	"ThickMetalModel", NS=2
23	METAL3	10	METAL3	0.5 μm	60.0 mOhm/sq	3.333E7	"ThickMetalModel", NS=2

Add new metal

In the metal tab of the editor window, you can use "Add" to add a new metal. By default, it is appended at the end of the metal list. You can use the "Move up" or "Move down" icons to move the metal in the list. This has no effect on the physical position in the stack-up. To change the physical position in the stack-up, edit the metal and assign a different position.

Add new metal here

If you have selected a dielectric in the preview window, you can use the right mouse button menu to bring up the "Add new metal here" command. This adds a new metal, and already sets the metal position to that dielectric layer.



Clone metal

The clone metal command creates a copy of the selected metal, with the same thickness and conductivity and mesh properties. You only need to set Cadence/ADS layer and the position in the stack-up.

This command can be useful if you have imported a *.son file, where one metal definition is shared between multiple positions in the stack-up. In the *.matl file format, each metal can only have one unique position in the stack-up. Thus, you might need to clone a metal definition before you can use it in multiple positions.

Metal display in the preview window

In the preview window, metal is shown with the corresponding Cadence/ADS layer name, and the loss specification. The loss specification can be [S/m] or [mOhm/square], depending on your settings in File > Preferences > Display settings.

0.4µm	METAL3	80 mOhm/sq
0.6µm	VIA2	
0.4µm	METAL2	80 mOhm/sq

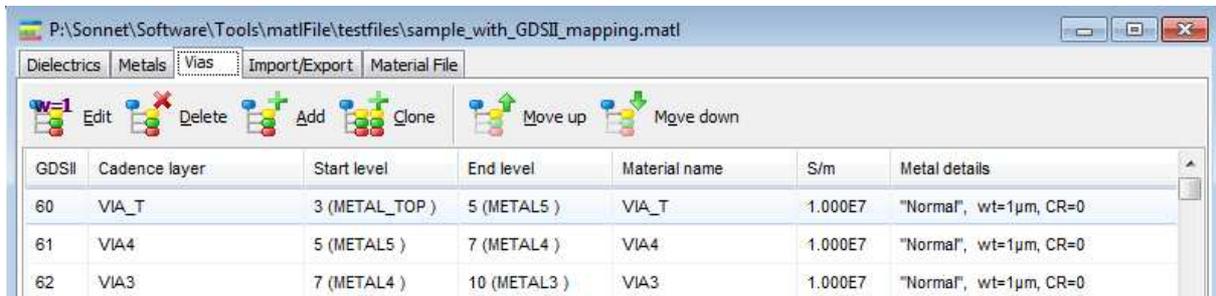
If a thick metal partially penetrates into the dielectric, the thickness of the metal and the remaining dielectric thickness above are shown, if the zoom level permits.

If the metal name is shown with yellow highlighting, this means that the Cadence/ADS layer name was not assigned. In this case, the material name will be shown instead of the layer name, with yellow background color. To create a valid *.matl file, you need to assign the Cadence/ADS layer name.

3µm	METAL_TOP	10 mOhm/sq	SiO2_MT
	VIA_T		SiO2
1µm	METAL5	30 mOhm/sq	SiO2_M5
	VIA4		SiO2

7. Vias

The vias tab of the editor window shows a list of all vias and their properties. If at least one DXF or GDSII layer is defined, for use with these file formats, then the corresponding column will also be shown in the via table.



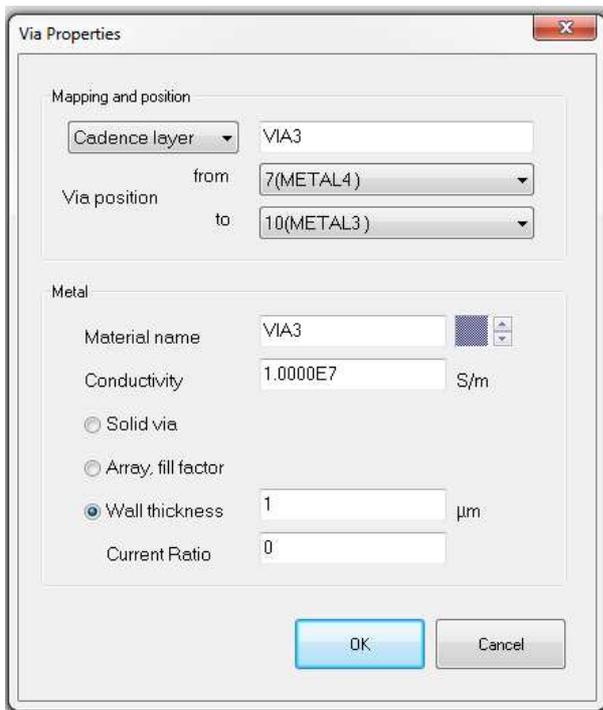
GDSII	Cadence layer	Start level	End level	Material name	S/m	Metal details
60	VIA_T	3 (METAL_TOP)	5 (METAL5)	VIA_T	1.000E7	"Normal", wt=1µm, CR=0
61	VIA4	5 (METAL5)	7 (METAL4)	VIA4	1.000E7	"Normal", wt=1µm, CR=0
62	VIA3	7 (METAL4)	10 (METAL3)	VIA3	1.000E7	"Normal", wt=1µm, CR=0

For vias, the list order does not have any effect on the output files. If you prefer to change the list order, you can use the "Move up" or "Move down" icons.

Edit via

To edit a via, double click on an entry in the via table, or double click on a via in the preview window. This will bring up the "Edit" dialog.

In the "Mapping and position" section, you can set the Cadence/ADS layer name for this metal, and the from-to position in the stack-up. If these two settings are not defined, the definition is stored in the *.matl file as a "MATERIAL" entry instead of "METAL" entry, and can be mapped later in the Cadence/ADS interfaces.



Via Properties

Mapping and position

Cadence layer: VIA3

Via position: from 7(METAL4) to 10(METAL3)

Metal

Material name: VIA3

Conductivity: 1.0000E7 S/m

Solid via

Array, fill factor

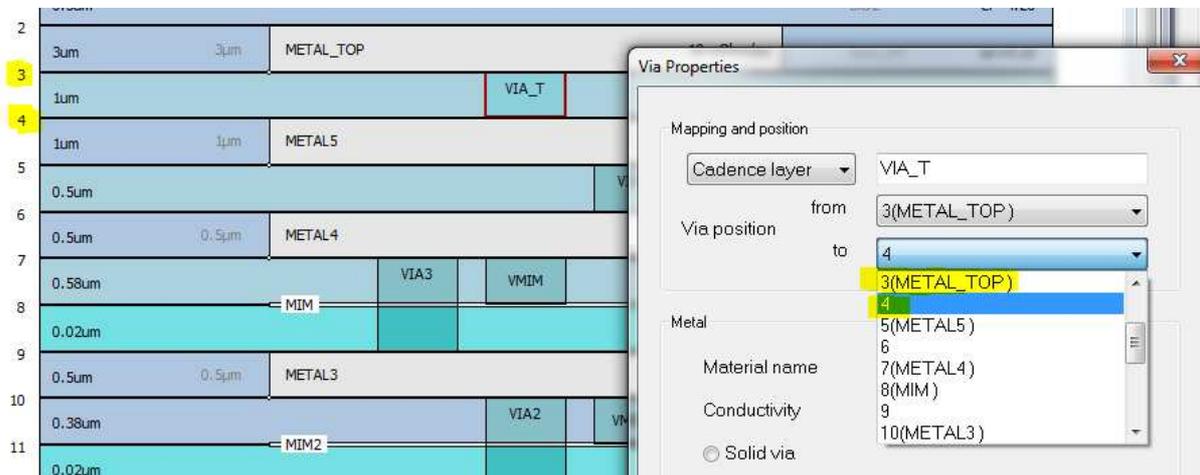
Wall thickness: 1 µm

Current Ratio: 0

OK Cancel

If you want to create output files for Sonnet version 12, your vias must be defined with wall thickness. Solid and array vias are only supported for Sonnet 13 or later.

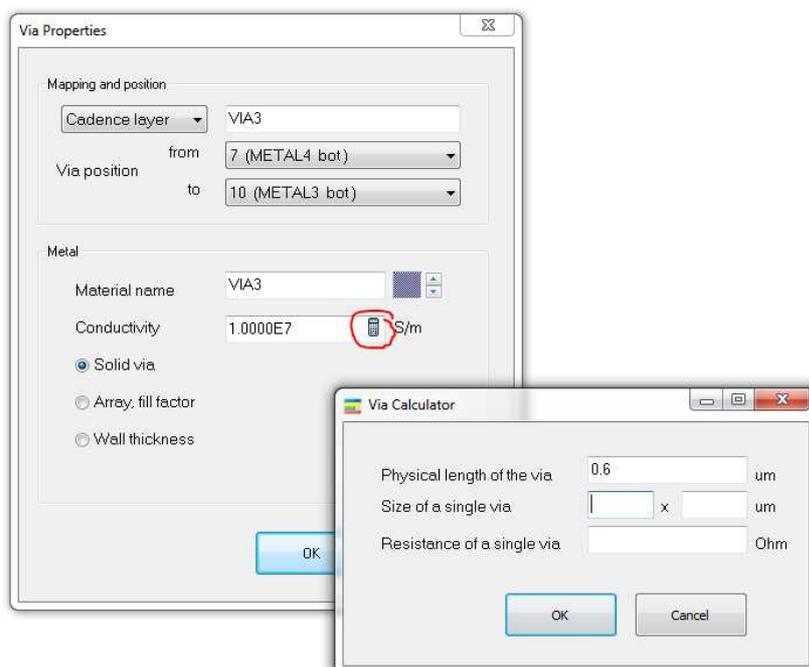
To simplify the from-to layer choice, the drop down list will display the dielectric levels along with the name of the metal that is placed on this level. However, for thick metal that fills a complete dielectric, the via can start at the top of the thick metal, which is actually one level above the base of the thick metal.



Note: If a via is defined to go down to the bottom of a thick metal, like levels 3-5 in the example above, the part of the via that is inside the thick metal will not be shown in the stack-up preview. This is to ensure a clean view.

Via calculator

If you have chosen a "Solid via", you can open the via calculator by clicking on the small calculator icon located at the right side of the Conductivity input field. This calculator will help you to convert via resistance and dimensions from the datasheet to the corresponding conductivity value. Of course, if you already know the conductivity, you can type it in without using the calculator.



Add new via

In the via tab of the editor window, you can use "Add" to add a new via. By default, it is appended at the end of the via list. You can use the "Move up" or "Move down" icons to move the via in the list. This has no effect on the physical position in the stack-up. To change the physical position in the stack-up, edit the via and assign a different position.

Add new via here

If you have selected a dielectric in the preview window, you can use the right mouse button menu to bring up the "Add new via here" command. This adds a new via, and already sets the via from-to position to that dielectric layer.

