

# **Best Practices: Working with Allegro PCB SI Field Solvers**

**Product Version 23.1  
September 2023**

© 2023 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Allegro PCB Editor contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

All other trademarks are the property of their respective holders.

**Restricted Permission:** This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

---

**Restricted Rights:** Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.



---

# Contents

---

<u>Best Practices: Working with Allegro® PCB SI Field Solvers</u> .....	3
<u>Choosing a Field Solver</u> .....	4
<u>What We Recommend</u> .....	5
<u>Settings and Preferences</u> .....	6
<u>BEM2D Settings</u> .....	6
<u>EMS2D Settings</u> .....	7
<u>EMS2D Preferences</u> .....	7
<u>Stackup Settings</u> .....	9
<u>What We Recommend</u> .....	11
<u>Performance Tuning EMS2D</u> .....	12
<u>autoSolve Switch in SigXplorer</u> .....	12
<u>Pre-solved Interconnect Library</u> .....	12
<u>ABIML Tables</u> .....	13

## Working with Allegro PCB SI Field Solvers

---

---

# Best Practices: Working with Allegro® PCB SI Field Solvers

---

This *Best Practice* describes how electromagnetic field solvers provide the characteristic parameters necessary for modeling and analyzing Printed Circuit Board (PCB) interconnect.

These parameters include the impedance, propagation delay, and frequency-dependent electrical properties (RLGC, or S-Parameter) that are used in Signal Integrity (SI) and Electromagnetic Interference (EMI) analysis. In addition to SI/EMI analysis, these parameters are also used for certain electrical DRCs in the Allegro PCB suite of tools.

Starting with the SPB 16.2 release, Allegro PCB SI now contains the following fully-integrated, complementary 2D field solvers to handle typical PCB interconnect structures:

- BEM2D is a quasi-static solver based on the boundary element method, and has been part of Allegro PCB SI for many releases. Quasi-static solvers are used for frequencies up to 1 GHz. As with any field solver, it is important to understand all assumptions employed in the calculations. Skin effect can be modeled, but since static solvers do not directly solve Maxwell's equations, displacement currents are not included. Isotropic permittivity and permeability, as well as other material properties, are also assumed.
- EMS2D is a full-wave solver based on the finite element method, and was introduced with the Allegro PCB SI 16.2 release. Full-wave solvers can handle frequencies well into the Tera Hz (THz) range, but can be unstable at lower frequencies. To handle lower-to-mid frequencies, a quasi-static transverse electromagnetic (quasi-TEM) solver is part of EMS2D. Quasi-TEM combines with the full wave portion to provide an accurate solution from DC to THz. Maxwell's equations, and the coupling between magnetic and electrical fields, are handled at every frequency point. All electromagnetic effects including wave transmission, radiation, dispersion, and material losses are modeled.

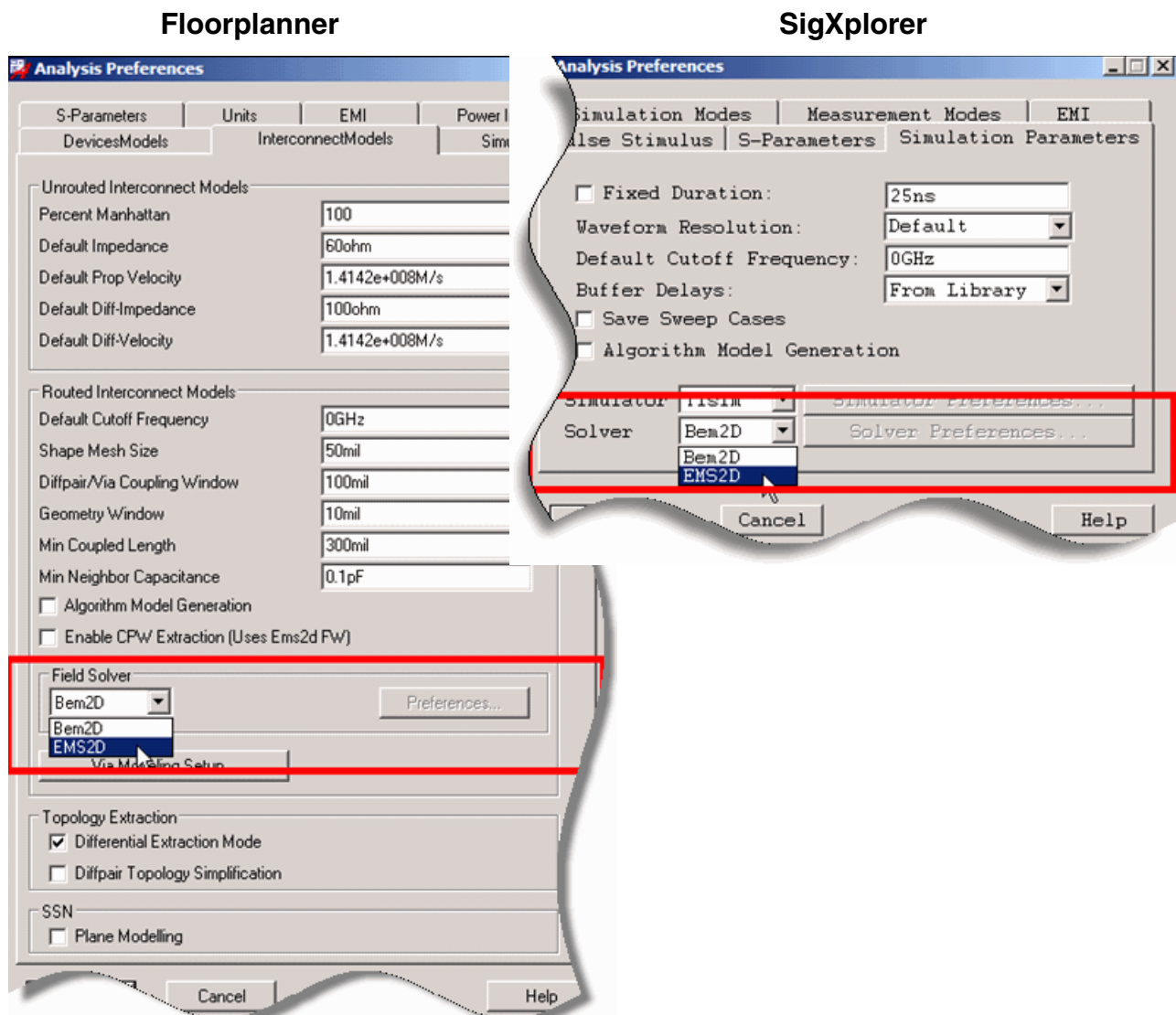
Both solvers are available in the Allegro PCB solution for SI/EMI, as well as electrical DRC analysis, and share the same use model. Static solvers run quickly, but may not provide the desired advanced capabilities. A full-wave solver is more capable, but requires more computation resources.

This paper answers questions related to selecting the right field solver for a specific application, and provides recommendations for configuration and usage.

## Choosing a Field Solver

When you invoke either the floorplanner or SigXplorer, the default selection for a solver is BEM2D. You can choose a specific field solver through the drop-down menus, as shown below.

Figure 1-1 Analysis Preferences Dialog Box





## What We Recommend

BEM2D and EMS2D correlate over a wide range of conditions. When choosing a field solver, you need to consider many factors, including simulation frequency, the interconnect structures used by the nets of interest, and materials.

Use BEM2D . . .

- for general use and for electrical DRC analysis.
- for frequencies below 1 GHZ.

Use EMS2D . . .

- for frequencies at or above 2.5 GHz (5 Gbps).
- for designs with complex materials, or when better accuracy is needed to account for dielectric loss, skin effect, proximity effect, dispersion, or wave behavior.
- to work with SPB 16.2 features, such as:

- ☐ Frequency-dependent Materials

The dielectric constant and loss tangent properties of PCB materials are factors that describe electrical behavior, and are often input as scalar constants at frequencies below a few gigahertz. At higher frequencies these properties can be defined as a complex number, which varies with frequency.

- ☐ Anisotropic Materials

These are materials with physical properties that vary with direction, such as dielectric values that are different, depending on the direction of the electric fields. They are defined using a multi-dimensional matrix known as a Tensor.

- where arbitrary cross-sections are involved, such as:
  - ☐ co-Planar Waveguides (CPW), whose trace interconnect and DC shapes are on the same layer.
  - ☐ a trapezoidal representation of the manufacturing etching process
  - ☐ designs with a conformal coating layer

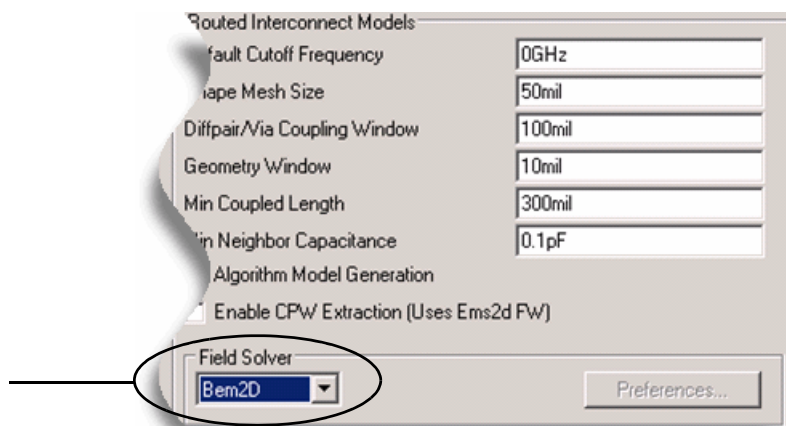
**Note:** For frequencies between 1 GHZ and 2.5 GHZ, base your field solver selection on the structures and specialty materials used in the design.

## Settings and Preferences

This section addresses setting up the environment and the design for SI/EMI simulation and Electrical DRC Analysis, including preferences and cross section detail.

Figure 1-1 on page 4 shows the *Analysis Preferences* dialog boxes for both the floorplanner and SigXplorer. Figure 1-2 shows the floorplanner's *InterconnectModels* tab of the *Analysis Preferences* dialog box in detail.

**Figure 1-2 Analysis Preferences: InterConnectModels tab**



## BEM2D Settings

In both *Analysis Preferences* dialog boxes, when the Field Solver is set to BEM2D, the *Preferences* button is disabled. It is only enabled when you select EMS2D.

## Cutoff Frequency

Use the *Cutoff Frequency* parameter to establish a bandwidth for generating interconnect parasitics. When you set *Cutoff Frequency* to 0, and you choose BEM2D as the field solver, it generates DC values for the RLGC matrix only, and does not account for internal inductance.

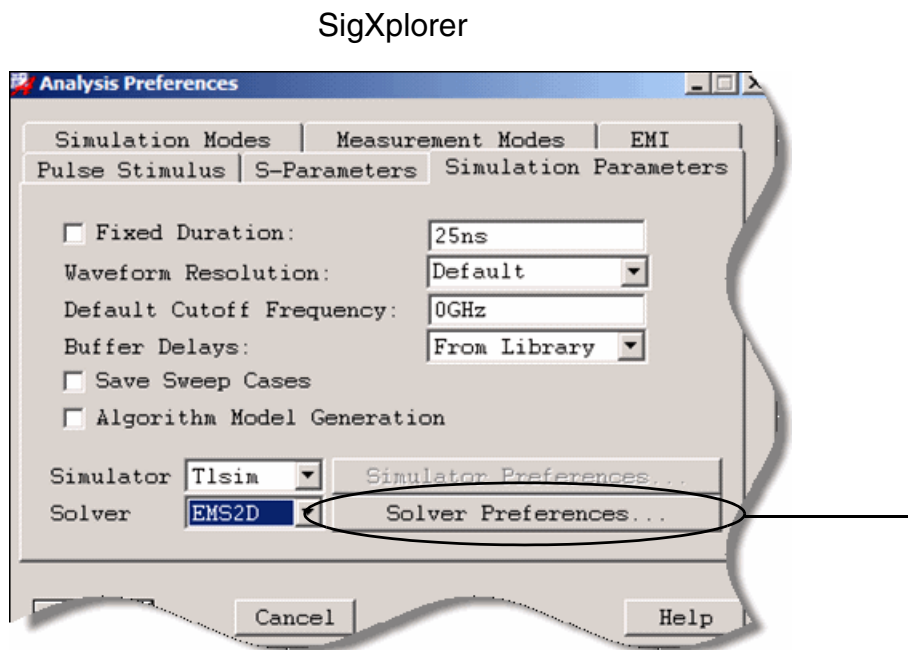
The section that follows explains the EMS2D settings used to generate frequency data. One option is to mimic how BEM2D uses the cutoff frequency setting; when you set this value to 0, EMS2D provides internal inductance.

## EMS2D Settings

With EMS2D selected as the Field Solver, the *Solver Preferences* button is enabled (see Figure 1-3).

**Note:** The floorplanner provides a similar button.

**Figure 1-3 Analysis Preferences: Field Solver Preferences**



Both the *Simulator Preferences* and the *Solver Preferences* buttons invoke their respective preferences dialog box. Figure 1-4 on page 8 shows the *EMS2D Preferences* dialog box.

## EMS2D Preferences

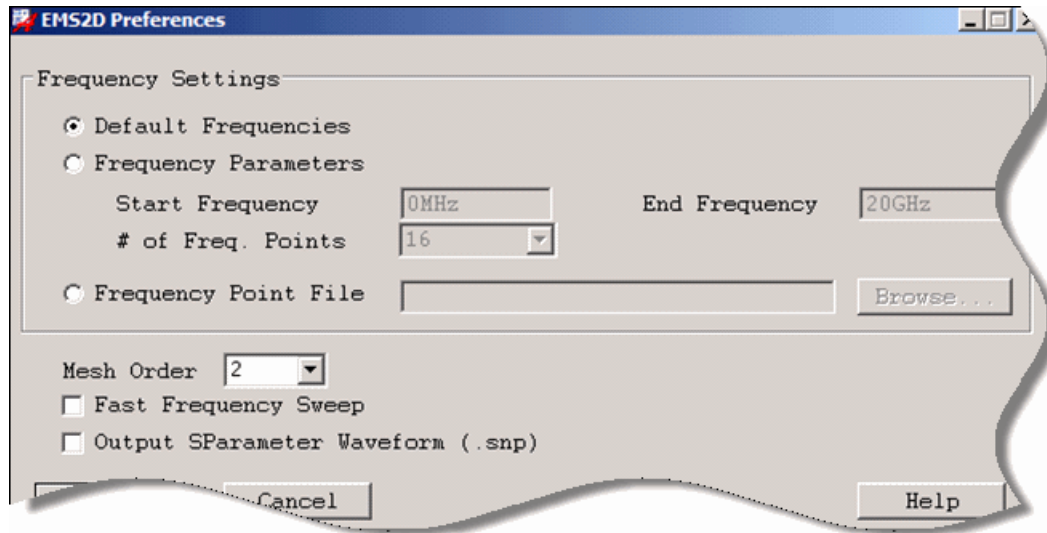
EMS2D frequency and mesh order preferences include:

### Frequency Preferences

#### ■ Default Frequencies

Used to create frequency data values. If the cutoff frequency is set to 0 GHz (the default) in the *Preferences* dialog, EMS2D generates only DC values. If a cutoff frequency is set, EMS2D generates a range of frequencies similar to those of BEM2D.

**Figure 1-4 EMS2D Preferences**



- **Frequency Parameters**

Used to specify a frequency range and number of points.

- **Frequency Point File**

Used for specify frequency points in a text file. The format is a single column of values in GHz.

### **Miscellaneous Preferences**

- **Mesh Order drop-down**

Used to control the polynomial order for solving each grid cell in the mesh generated by the field solver. The default setting is 2. A setting of 1 speeds-up performance without significantly sacrificing accuracy. A setting of 3 provides the highest accuracy, but requires more computing resources.

- **Fast Frequency Sweep**

Used only when solving many frequency points. Rather than generating a full solution for each point, this option allows the field solver to find acceptable ranges where values can be interpolated. This does *not* mean a degradation in accuracy, as the solver compares fully solved values to points where interpolation is used, and does so with a very tight tolerance.

## Working with Allegro PCB SI Field Solvers

### Best Practices: Working with Allegro® PCB SI Field Solvers

#### ■ Output S-Parameter Waveform (.snp)

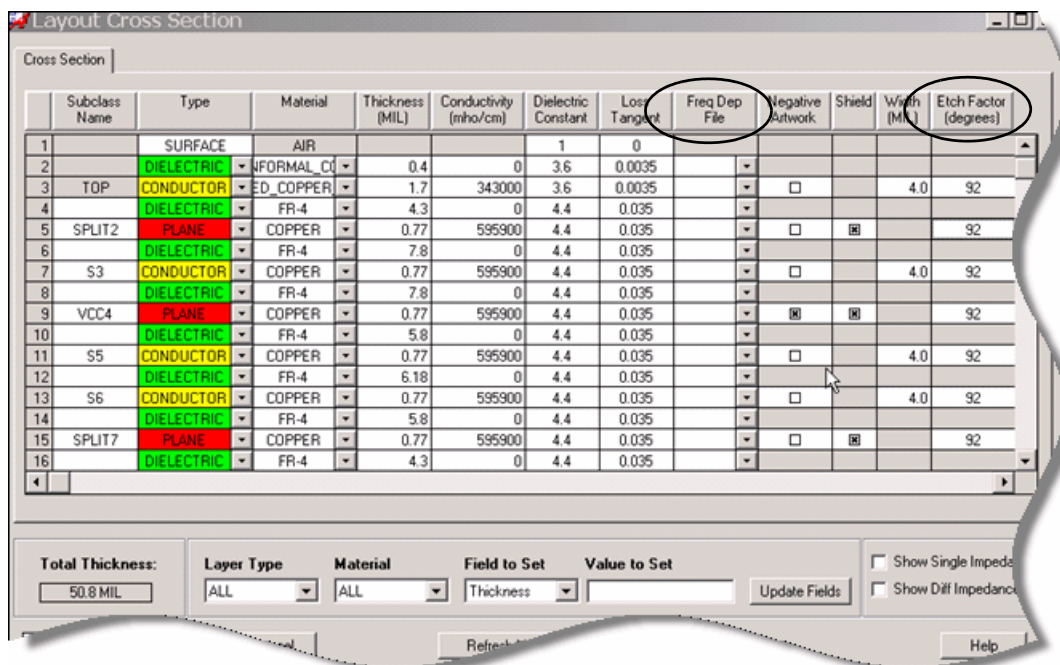
Used to generate an S-Parameter waveform in addition to a trace model. S-Parameter is used for comparison to other 3<sup>rd</sup> party field solvers that generate S-Parameters. This option is enabled only when the Frequency Settings are set to specific Frequency Parameters or when using a Frequency Point File.

## Stackup Settings

Stackup settings provide data for the field solver, including the thickness and type of specific layers, and the material properties of the dielectric and conductive layers. This data is crucial for the field solver to generate accurate trace models for simulations, and for providing interconnect models for topologies extracted to SigXplorer.

In Allegro PCB SI, you enter stackup data in the *Layout Cross Section* dialog box (choose *Setup – Cross-section*).

**Figure 1-5 Layer Cross Section dialog box**



Introduced in the SPB 16.2 release, the *Layout Cross Section* dialog box now includes an *Etch Factor* column to account for manufacturing process variations, such as when a copper trace is routed within in a trapezoid rather than within a rectangle (see [Figure 1-6](#) on page 10).

These trapezoids can be upright or inverted, depending on the layer and the process used.

**Note:** Etch Factor replaces the environment variable `trapezoidal_angle_in_degree`, which you can now define by layer.

Figure 1-6 shows the relationship between the angle used to represent the deviation of the trace-to-trace geometry and the range of values used to represent various orientations.

**Figure 1-6 Trace Geometry Deviations**

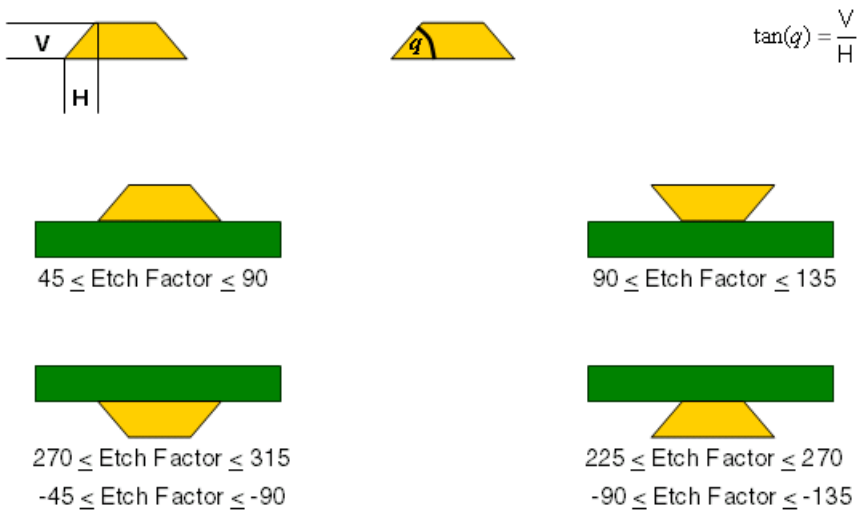


Figure 1-5 on page 9 shows the existing columns for *Dielectric Constant* and *Loss Tangent*, which are constants. To the right is a new column, *Freq Dep File*, with put-downs that search for files with a `.material` extension in the directories specified by the `materialpath` environment variable.

These frequency-dependent material files support:

- Isotropic

A one dimensional array, where each line has a frequency value and a complex number for the dielectric constant. Each line has the following format

```
freq  $\epsilon'$   $\epsilon''$ 
```

where `freq` is the frequency of interest, expressed in GHz, and  $\epsilon'$  and  $\epsilon''$  are the real and imaginary parts of the dielectric constant, respectively. The real component is what we usually refer to as the dielectric constant. Because we ignore the DC current in dielectric material, the imaginary portion simplifies to the loss tangent \* dielectric constant.

■ **Anisotropic**

A multi-dimensional array, or tensor, of the format

$\text{freq } \epsilon_{xx} \ \epsilon_{xy} \ \epsilon_{xz} \ \epsilon_{yx} \ \epsilon_{yy} \ \epsilon_{yz} \ \epsilon_{zx} \ \epsilon_{zy} \ \epsilon_{zz}$

where *freq* is the frequency of interest, expressed in GHz, and each  $\epsilon_{nn}$  is the directional complex dielectric constant.

Material file matrices are used to define surface roughness, among other parameters, where the behavior is reflected by the new complex dielectric constants for a specified range of frequencies.

## What We Recommend

No matter which solver you choose, we recommend these best practices:

- Ensure that your design is properly set up for high speed design and analysis. If you are unaware of the specific steps, consult the Setup Advisor (choose *Tools – Setup Advisor*) in PCB SI.
- Understand the Field Solver's default settings, and adjust accordingly. These settings are typically most appropriate for electrical DRC analysis or for multiple simulations where interconnect losses are not important.

Default choices are BEM2D for the solver and 0 GHz for the Cutoff Frequency

- To enable lossy line simulations when using BEM2D, set a non-zero value for Cutoff Frequency. For example, a value of 10 GHz will span up to 50 GHz, and is sufficient for most Multi-GHz applications.
- The default frequency preference in EMS2D is set to *Default Frequencies*, which tracks how BEM2D handles frequency values. The primary use of this setting is for comparing results between EMS2D and BEM2D.
- With the solver set to EMS2D, a frequency range should be specified by using either the *Frequency Parameters* or *Frequency Point File* fields in the *Frequency Settings* section of the *EMS2D Preferences* dialog box (see [Figure 1-4](#) on page 8).

You do not have to specify too many frequency points, unless you are generating S-Parameters. The default, 16 points, is appropriate for a typical transient simulation in PCB SI.

- The default Mesh Order setting of 2 offers a reasonable trade-off between performance and accuracy.
- Consult with your manufacturing resources to obtain specific cross section parameters and material properties. Manufacturing values for impedance may be based on finished trace geometries; therefore, you may need to inquire about Etch Factor values.

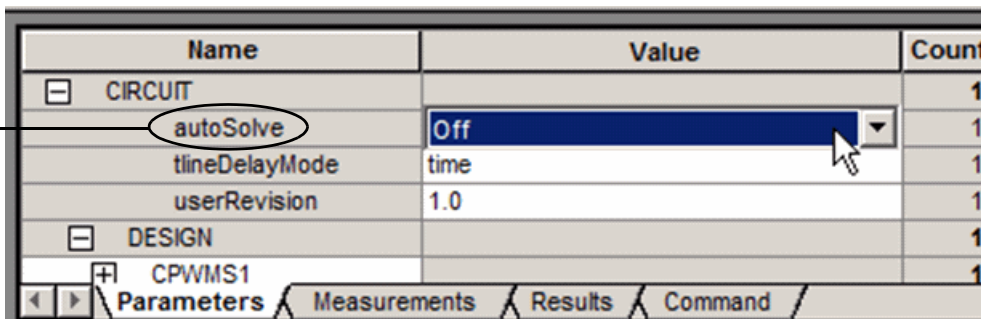
## Performance Tuning EMS2D

EMS2D, like any other full wave electromagnetic solver, is computationally intensive. The SPB 16.2 release includes the following new features to improve performance:

### autoSolve Switch in SigXplorer

To reduce the calls to the field solver, you can shut off the *autoSolve* mode, which will delay field solutions until they are needed for simulation.

**Figure 1-7 AutoSolve Switch Parameter**



The screenshot shows the 'Parameters' tab in the SigXplorer interface. A table lists various parameters under the 'CIRCUIT' and 'DESIGN' categories. The 'autoSolve' parameter is highlighted with a blue oval, and its value is set to 'Off'. A mouse cursor is pointing at the dropdown arrow next to the 'Off' value. The 'tlineDelayMode' parameter is set to 'time', and 'userRevision' is set to '1.0'. The 'DESIGN' category is expanded, showing 'CPWMS1' with a count of 1. The bottom of the panel has tabs for 'Parameters', 'Measurements', 'Results', and 'Command'.

Name	Value	Count
<b>CIRCUIT</b>		<b>1</b>
autoSolve	Off	1
tlineDelayMode	time	1
userRevision	1.0	1
<b>DESIGN</b>		<b>1</b>
CPWMS1		1

### Pre-solved Interconnect Library

We now supply a default interconnect library, `cds_interconn.iml`, which contains a pre-solved, EMS2D-formatted trace model for each of SigXplorer's default trace models. You can add parts represented by these models directly to SigXplorer's canvas.



## **ABIML Tables**

The *Algorithm Model Generation* checkbox in the *Analysis Preferences* dialog box (see [Figure 1-1](#) on page 4) enables the algorithmic generation of trace models directly from pre-solved tables. These EMS2D-generated tables are very accurate and contain a range of values for specific trace parameters, such as line width. As long as the trace model is within the limits of the table, their effect is instantaneous; if not, EMS2D is called.

**Note:** We provide a ABIML table for each of SigXplorer's default trace models.

# **Working with Allegro PCB SI Field Solvers**

## **Best Practices: Working with Allegro® PCB SI Field Solvers**

---