Si8000m/Si9000e User Guide

Polar Instruments Ltd

Polar Instruments Ltd.
Garenne Park
St. Sampson
Guernsey
Channel Islands
GY2 4AF
ENGLAND

polarcare@polarinstruments.com
www.polarinstruments.com

MAN 207–2003
Si8000m/Si9000e User Guide

POLAR INSTRUMENTS LTD

COPYRIGHT

Copyright 2020 © by Polar Instruments Ltd. All rights reserved. This software and accompanying documentation is the property of Polar Instruments Ltd and is licensed to the end user by Polar Instruments Ltd or its authorized agents. The use, copying, and distribution of this software is restricted by the terms of the license agreement.

Due care was exercised in the preparation of this document and accompanying software. Polar Instruments Ltd. shall not be liable for errors contained herein or for incidental or consequential damages in connection with furnishing, performance, or use of this material.

Polar Instruments Ltd makes no warranties, either expressed or implied, with respect to the software described in this manual, its quality, performance, merchantability, or fitness for any particular purpose.

DISCLAIMER

1. Disclaimer of Warranties

POLAR MAKES NO OTHER WARRANTIES, EXPRESS, IMPLIED OR STATUTORY, REGARDING PRODUCTS. ALL OTHER WARRANTIES AS TO THE QUALITY, CONDITION, MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NON-INFRINGEMENT ARE EXPRESSLY DISCLAIMED.

2. Limitation of Liability.

POLAR SHALL NOT BE RESPONSIBLE FOR DIRECT DAMAGES IN EXCESS OF THE PURCHASE PRICE PAID BY THE END USER OR FOR ANY SPECIAL, CONSEQUENTIAL, INCIDENTAL, OR PUNITIVE DAMAGE, INCLUDING, BUT NOT LIMITED TO, LOSS OF PROFITS OR DAMAGES TO BUSINESS OR BUSINESS RELATIONS, WHETHER OR NOT ADVISED IN ADVANCE OF THE POSSIBILITY OF SUCH DAMAGES, THE FOREGOING LIMITATIONS SHALL APPLY, NOTWITHSTANDING THE FAILURE OF ANY EXCLUSIVE REMEDIES.

TRADEMARKS

Copyright Polar Instruments Ltd. ©2020

Microsoft®, Microsoft Windows®, Windows 7®, Windows 8®, Windows 10® and Microsoft Excel® are registered trademarks of Microsoft Corporation.

IBM® is the registered trademark of International Business Machines Corporation.

All other trademarks acknowledged.
## Personal Computer Requirements

<table>
<thead>
<tr>
<th>Item</th>
<th>Requirement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computer</td>
<td>IBM PC AT or compatible</td>
</tr>
<tr>
<td>Processor</td>
<td>Intel Pentium or compatible – 1GHz or better</td>
</tr>
<tr>
<td>Operating system</td>
<td>Microsoft™ Windows 7™ or later</td>
</tr>
<tr>
<td>System memory required</td>
<td>2GB recommended</td>
</tr>
<tr>
<td>Hard disk space required</td>
<td>100MB (min.)</td>
</tr>
<tr>
<td>Video standard</td>
<td>FHD (HD 1080) (1920 x 1080) minimum</td>
</tr>
<tr>
<td>Mouse</td>
<td>Microsoft compatible</td>
</tr>
<tr>
<td>Software key port</td>
<td>Parallel port/USB port</td>
</tr>
<tr>
<td>Spreadsheet</td>
<td>Microsoft™ Excel™ 2003 or later (32-bit only)</td>
</tr>
</tbody>
</table>
# Contents

**Introduction to the Si8000m and Si9000e**  
1

Si8000m/Si9000e Field Solvers  
1  
Lossless calculations  
1  
Multiline crosstalk  
1  
Monte Carlo Analysis  
2  
Frequency-dependent calculations  
2  
Surface roughness compensation  
2  
Extended substrate data  
2  
   Exporting/importing extended substrate tables  
3  
Sensitivity Analysis  
3  
Via checks  
3  
   Via pad/anti-pad calculation  
3  
Multiple dielectric builds  
4  
Surface coating modelling  
4  
Quick Solver goal seek  
4  
Integration with the Speedstack Stackup Design System  
4  
Structure spreadsheet functions  
5  
Evaluating PCB structure behaviour  
5  
Importing/exporting data  
5  
Si Projects – grouping of related structures  
5  
Loss Tangent Goal Seek  
5  
Importing/exporting data in Touchstone format  
6  
Importing CITS data log files  
6

**Introduction to Controlled Impedance PCBs**  
7

Controlled impedance  
7  
Impedance matching  
8  
Calculation methods  
8

**Transmission Line Structures**  
9

Microstrip and Stripline Transmission Lines  
9  
Single-ended Transmission Lines  
9  
   Surface Microstrip  
9  
   Embedded Microstrip  
10  
   Coated Microstrip  
11  
   Offset Stripline  
11  
Differential Transmission Lines  
12  
   Edge-coupled Surface Microstrip  
12  
   Edge-coupled Coated Microstrip  
13  
   Edge-coupled Embedded Microstrip  
13  
   Edge-coupled Offset Stripline  
14  
   Broadside-coupled Stripline  
15  
Coplanar Lines  
16  
   Surface Coplanar Strips  
16  
   Surface Coplanar Strips with Ground  
16  
   Differential Surface Coplanar Strips  
17
Installing the Si8000m/Si9000e 18

Activating the Field Solver and license options 18
Choosing purchased license options 18
Uninstalling the software 18

Using the Quick Solver 19

The Quick Solver interface 19
  Startup Mode 19
    Specifying the Startup tab 20
Quick Solver screen areas 20
  Lossless calculations 21
  Frequency-dependent calculations (Si9000e only) 22
  Sensitivity Analysis 23
  Via Checks 24
    Via modelling 24
    Via pad/antipad coaxial calculation 24
Menu/Toolbar 25
  Display structures 25
  Single ended/differential structures 25
  Coplanar single ended structures 25
  Coplanar differential structures 25
  Toggle hatch plane 26
  Process window 26
  Copy/paste structure parameters 26
  Copy/paste structures to/from Speedstack 26
  Copy structure to CGen 26
  Paste from Speedstack into Si Project 26
  Import measurement data 26
  Excel interface 27
  Track resistance calculator 27
Structure Bar 27
Structure graphics 28
Calculation Interface 28
Quick Solver operating configuration 29
  Calculation Options 29
  Setting parameter limits 30
    Goal Seek parameters 30
    Etch factor settings 30
Hatch configuration 31
  Quick Solver hatch plane/mesh module 31
  Modelling impedance on traces with hatch plane grounds 31
  XFE – Crosshatch Flex Enhancement 31
Startup Mode 32
Graph Style 32
Solver accuracy 32
TRC configuration 32

Lossless calculations 33

Lossless modelling 33
  Calculating single ended impedance 33
  Calculating propagation delay, inductance and capacitance 33
Field solving for board parameters (goal seeking) 34
   Specifying Goal Seeking convergence 34
   Using the Extended Interface 35
      Using multiple tolerances 36
   Calculating differential impedance 36
   Calculating propagation delay, odd, even and common mode impedance 37
Saving and recalling results 37
Copying and pasting parameters between structures 38
Si Crosstalk – modelling multiline crosstalk 39
   Forward and reverse crosstalk 40
   Conductor configurations 40
   Modelling single ended microstrip traces 41
   Modelling differential pairs 43
Monte Carlo impedance analysis 44
   Using Monte Carlo analysis 44
Exchanging stackup structure information with Speedstack 46
Creating a custom list of structures 48
Printing results 50
Using Si Projects 51
   Creating new projects 51
   Working with Si Projects in Speedstack 52
      Transferring structures from Speedstack to the field solver 52
      Adding/deleting and modifying structures 54
      Calculating impedance and insertion loss. 55
Importing CITS log file data 56
   Step 1 Reading the log file 56
   Step 2 Selecting the Data Log record 57
   Graph settings 58
   Impedance result filtering 59
   Applying statistical analysis 59
Sensitivity analysis 60
   Graphing impedance against multiple parameters 60
      Varying a single parameter 60
      Varying multiple parameters 61
      Constant impedance v changing parameters 63
      Process Window: Minimum / Maximum 64
   Using Sensitivity Analysis to graph crosstalk 65
      Setting the lossless parameters 65
      Using impedance v changing parameters 65
      Viewing tabular results 66
   Copying Field Solver data to external programs (Si8000m) 68
   Using Sensitivity Analysis to graph multiple impedances 69
      Displaying all impedances 72
   Using sensitivity analysis to model the effects of an adjacent copper layer 73
      Modelling the proximity of adjacent copper 73
Via Checks 75
   Stub checks 75
   Via pad/antipad coaxial calculation 76
Calculating the Conductor and Dielectric Loss 116
Calculating Loss Tangent 116

Speedstack Si to Si9000e data transfer (frequency dependent parameters) 118
Sharing structure properties 118
Transferring structures 121
   Transferring a single structure 121
   Solving for impedance 121
   Running frequency dependent calculations 122
   Transferring multiple structures via Si Projects 123
Modifying structures 125

Analysing S-parameters with the LA9000 126
The LA9000 S-parameter analyser 126
Exporting a single structure 126
Exporting results of a project 127

Importing/exporting data 128
   Importing/exporting data in Touchstone™ format 128
       Importing Touchstone files 128
       Exporting Touchstone files 130
   Importing insertion loss data from Polar Atlas 130
       Importing Atlas data via a text file 130
   Measured Attenuation and Measured Effective Er 132
       Measurement Data options 133
   Modelling Delta-L insertion loss with the Si9000e 134
       Delta-L measurement technique 135
   Notes 135

Using the Si Excel Interface 137

   Controlled impedance structure categories 138
   Moving through the structure sheets 139
   Calculating trace characteristics 140
   Choosing the calculation type 141
   Charting against varying board parameters 141
   Choosing other parameters 143
       Changing the parameters 143
       Modifying the chart 144
   Using the controlled impedance functions in other workbooks 147
       Charting results 150

   Plotting multiple data series 154
       Plotting $Z_0$ for surface and coated microstrip 154
       Plotting $Z_{even}$ and $Z_{odd}$ v trace separation 156

   Using more complex models 158
       Calculating the effect of etch back 158
       Calculating the effect of variations in Height 158
       Charting trace width error 159
       Charting etch back error 160

Terms used in this manual 162

References 163
Introduction to the Si8000m and Si9000e

Si8000m/Si9000e Field Solvers

The Polar Instruments Si8000m Multiple Dielectric Controlled Impedance Field Solver uses advanced field solving methods to calculate PCB trace impedance for most single-ended and differential circuit designs. Based on Boundary Element analysis, the Field Solver is able to provide rapid modelling for a wide range of microstrip, stripline and coplanar structures.

The Si9000e Transmission Line Field Solver incorporates fast and accurate lossless and frequency-dependent PCB transmission line modelling and extracts full transmission line parameters for a wide range of PCB transmission lines.

Lossless calculations

The Field Solvers provide for rapid calculation of single PCB trace impedance values against significant PCB parameters (e.g. trace height and thickness, dielectric constant, etc.) Given a target impedance the goal seeking functions of the Si8000m and Si9000e allow the user to calculate circuit parameter values to achieve the desired impedance.

For situations with structure dimensional constraints, the field solvers allow the designer and board fabricator easily to accommodate variations in supplier material dimensions.

Support is provided for single or multiple dielectric builds in a comprehensive range of trace and dielectric configurations. The field solvers provide models for structures with dielectric layers above and below traces, soldermask modelling and includes compensation for resin rich areas between traces.

Multiline crosstalk

Si Crosstalk multi line and differential pair (lossless) crosstalk add on option for the Si8000m and Si9000e allows you to model coupling between aggressor and victim traces.

The coupling is modeled against frequency and line length and allows a designer to plan for enough trace separation between individual signal lines or between differential pairs for crosstalk to be within safe limits. Both near and far end crosstalk are modeled for stripline and microstrip cases. Crosstalk is presented graphically and the lossless data may be exported in Touchstone™ format for further analysis.
Monte Carlo Analysis

Si8000m and Si9000e include Monte Carlo simulation of printed circuit board impedance to provide a graphical mechanism for predicting and presenting the variation of PCB trace impedance for a production run of PCBs.

The Si8000m/Si9000e Monte Carlo simulation can range from varying a single parameter (for example, the thickness of a layer of prepreg material) over a range of possible values to randomising all input parameters for a structure. The number of iterations can be specified to reflect the number of boards in a typical production run.

Frequency-dependent calculations

Employing Boundary Element Method field solving, Si9000e extracts RLGC matrices and 2-Port (single-ended) or 4-Port (differential) S-Parameters and rapidly plots a structure’s transmission line information. Frequency dependent modelling extends down to 1KHz. The Si9000e supports user defined S-parameter source and termination impedance.

Graphing against frequency is provided for impedance magnitude, loss (conductor loss, dielectric loss and insertion loss), inductance, capacitance, resistance, conductance, skin depth and effective Er.

When differential structures are selected, Near-End Crosstalk (NEXT) and Coupling Percentage results are calculated and displayed.

Surface roughness compensation

The Si9000e supports roughness modelling for both drum and treated side copper. Modeling is provided for smooth copper plus a choice of methods for predicting the additional attenuation owing to surface roughness. Four surface roughness compensation methods are available: Smooth, Hammerstad, Groisse and Cannonball-Huray.

The Hammerstad and Groisse methods use simple RMS roughness values (usually obtainable in consultation with the board manufacturer) as input parameters. The Huray method provides for higher data rates and allows for more complex input parameters to be specified. The All Losses plot will reflect the Surface Roughness Compensation method selected.

Extended substrate data

The Si9000e frequency-dependent calculations can be refined using extended substrate data. The Extended Substrate Data editor presents the option of assigning substrate values by frequency band to accommodate material from manufacturers who specify parameters (e.g. Er and loss tangent) that vary by frequency.
The Si9000e will accept constant (i.e. frequency independent) values for Er and TanD. However, using frequency independent permittivity is a source of non-causal time domain responses so causal interpolation of dielectric constant is implemented in the Si9000e via the Causally Extrapolate Er / TanD option from the Extended Substrate Data option group; this applies Svensson-Djordjevic modelling to each dielectric layer in the current controlled impedance structure.

Using the Multiple Er / TanD option the Si9000e can accept tables of multiple values of Er and TanD or use a single value to enable Svensson-Djordjevic frequency dependent permittivity modelling. When a single value table is used it employs the same modelling technique as implemented with the Causally Extrapolate Er / TanD option.

**Exporting/importing extended substrate tables**

Tables can be imported or exported in native (.ESL) format or pipe-delimited CSV format. Tables can, for example, be exported for editing in Microsoft Excel and the modified data subsequently reimported.

**Sensitivity Analysis**

The Si8000m and Si9000e allow designers and board fabricators to calculate and plot impedance changes against a range of values for a specified structure parameter (charting, for example, Z₀ for variations in H₁, Er, etc.)

When calculating differential structures multiple impedances may be plotted on same graph

Result data calculated may be exported to clipboard in an Excel compatible format using the Edit - Copy Current Result Tab to Clipboard option

Graphs produced may be maximised, printed or exported to jpeg format

**Via checks**

The Via Stub check provides a simple color coded go/no go check on the potential for signal distortion of a via stub. The effects of the stub will increase as the stub length and Er increase and the signal rise time reduces.

**Via pad/anti-pad calculation**

The Via Checks tab includes via pad/anti-pad calculation. It provides for modelling plated through hole (PTH) vias with respect to impedance and signal integrity in order to allow the designer to ensure a constant impedance is presented to a signal as it propagates between devices.
Multiple dielectric builds

Advanced modelling allows the designer to predict the finished impedance of multiple dielectric PCB builds and also take into account the local variations in dielectric constant on close spaced differential structures (e.g. areas of high resin concentration between differential pairs).

Surface coating modelling

The resist thickness adjacent to, above and between surface traces is included in applicable models. This offers an elegant solution to modelling surface coating which can be tailored to the particular resist application method in use. Both field solvers also extract even, odd and common impedance. It is becoming increasingly necessary to control these characteristics on high speed systems such as USB 2.0 and LVDS.

The field solvers incorporate the Quick Solver for single impedance and parameter calculations along with a comprehensive set of advanced field solving methods incorporated as user-defined functions in the popular Microsoft Excel spreadsheet format.

Quick Solver goal seek

The Quick Solver’s goal seek provides for rapid calculation of single PCB trace impedance values against significant PCB parameters (e.g. trace height and thickness, dielectric constant, etc.) Given a target impedance the Quick Solver allows the user to calculate circuit parameter values to achieve the desired impedance. For situations with structure dimensional constraints, the Field Solver Tolerance fields allow the designer and board fabricator easily to calculate the effects of variations in supplier material dimensions.

Integration with the Speedstack Stackup Design System

The Quick Solver is integrated with the Speedstack Stackup Design System to allow the board designer or fabricator to add controlled impedance structures to layers in the stackup. The designer is able to utilise the goal seeking facility of the Quick Solver in conjunction with the Speedstack Stackup Editor to arrive at appropriate controlled impedance structures and parameters quickly and efficiently.

Speedstack Si (a software package comprising Speedstack plus Si9000e) caters for frequency dependent calculations and adds comprehensive insertion loss capability into Speedstack along with bidirectional copy and paste between Speedstack and the Si9000e.

Frequency dependent parameters include length of line, trace conductivity, dielectric constant and loss tangent,
frequencies of interest, causal extrapolation points for each substrate and roughness and roughness modeling methods.

**Structure spreadsheet functions**

The structure functions included in Excel format enable advanced functions, e.g. sensitivity analysis, graphing the effects of a range of parameter value changes. Single or multiple dielectric builds are supported in a comprehensive range of trace and dielectric configurations. Models are included for structures with dielectric layers above and below traces, soldermask configurations and compensation for resin rich areas between traces.

**Evaluating PCB structure behaviour**

The Si8000m and Si9000e offer as a purchasable option the familiar Microsoft Excel for Windows interface for easy graphing and data sharing. Using Excel’s powerful Autofill and Chart Wizard features, the field solving calculation engine can rapidly chart $Z_0$ against varying parameters, providing easy comparison and evaluation of the behaviour of most popular controlled impedance structures.

**Importing/exporting data**

Integration with the Polar Atlas PCB Insertion Loss test system allows direct importation of measurement data from Atlas SPP, SET2DIL or Delta-L into the Si9000e. The field solving engine parses and displays the imported and modelling data on the same graph – a single click then allows a user to goal seek loss tangent allowing exploration of the relationship between predicted and measured attenuation. The Si9000e provides for simple transfer of table data to external programs such as spreadsheets or databases for subsequent analysis.

**Si Projects – grouping of related structures**

Si Projects allows rapid copying and pasting of an entire stackup impedance structure set from Speedstack into the field solver for detailed analysis – or simply for storing groups of related structures. Once the set of Speedstack structures has been imported into the Si Project, use the frequency dependent calculation options to predict the conductor loss, dielectric loss and total attenuation for each structure. This is valuable for when designers need to control both impedance and insertion loss.

**Loss Tangent Goal Seek**

Measuring insertion loss yields the total losses of a transmission line, but sometimes it is useful to further process that information and deduce the contribution of copper losses and dielectric losses to the overall loss figure.
The Si9000e Loss Tangent Goal Seek provides a useful estimate of the dielectric loss tangent for the substrate material.

**Importing/exporting data in Touchstone format**

The Si9000e includes the capability to import Touchstone™ data so that measured and modelled S-parameter data may be compared.

Designers can import a Touchstone file containing S-parameter data, with options to display just the Touchstone data or combine this data with the current selected structure’s S-parameter data.

**Importing CITS data log files**

The Si8000m/Si9000e field solvers include the capability to import measurement data directly from the industry-standard Controlled Impedance Test System (CITS).

A CITS data log file (.CLF) contains comprehensive impedance measurement data and, along with existing modelled structure information, offers graphing capabilities and statistical analysis where the modelled and measured data can be presented together.
Introduction to Controlled Impedance PCBs

Controlled impedance

The increase in processor clock speed and component switching speed on modern PCBs means that the interconnecting paths between components (i.e. PCB tracks) can no longer be regarded as simple conductors.

At fast switching speeds or high frequencies (i.e. for digital edge speeds faster than 1ns or analog frequencies greater than 300MHz) PCB tracks must be treated as transmission lines.

That means that for stable and predictable high speed operation the electrical characteristics of PCB traces and the dielectric of the PCB must be controlled.

One critical parameter is the characteristic impedance of the PCB track (the ratio of voltage to current of a wave moving down the signal transmission line); this will be a function of the physical dimensions of the track (e.g. track width and thickness) and the dielectric constant of the PCB substrate material and dielectric thickness.

The impedance of a PCB track will be determined by its inductive and capacitive reactance, resistance and conductance. PCB impedances will typically range from $25\,\Omega$ to $120\,\Omega$.

In practice a PCB transmission line typically consists of a line conductor trace, one or more reference planes and a dielectric material. The transmission line, i.e. the trace and planes, form the controlled impedance.

The PCB will frequently be multi-layer in fabrication and the controlled impedance can be constructed in several ways. However, whichever method is used the value of the impedance will be determined by its physical construction and electrical characteristics of the dielectric material:

- The width and thickness of the signal trace
- The height of the core or pre-preg material either side of the trace
- The configuration of trace and planes
- The dielectric constant of the core and pre-preg material
Impedance matching

Components themselves exhibit characteristic impedance so the impedance of the PCB tracks must be chosen to match the characteristic impedance of the logic family in use.

If the impedance of the PCB tracks does not match the device characteristic impedance multiple reflections will occur on the line before the device can settle. This can result in increased switching times or random errors in high-speed digital systems. The value and tolerance of impedance will be specified by the circuit design engineer and the PCB designer, however, it will be left to the PCB manufacturer to conform to the designer’s specification and verify the finished boards meet the specification.

Calculation methods

The Si8000m/Si9000e incorporate field solving for single-ended and differential impedance structures. The discrete numerical analysis in the field solvers uses the Boundary Element Method to evaluate the residual field. A piecewise linear approximation is used with a weighted sub-division of the perimeter of the trace cross-section to predict the surface charge distribution on the trace. Knowing the boundary voltage conditions and the charge distribution allows the Boundary Element Method to predict the capacitance of the structure. This in turn allows the impedance of the structure to be calculated.
Transmission Line Structures

Microstrip and Stripline Transmission Lines

Controlled impedance PCBs are usually produced using microstrip and/or stripline transmission lines in single-ended (unbalanced) or differential (balanced) configurations.

A microstrip line consists of controlled width conductive traces on a low-loss dielectric (in practice the dielectric may be constructed from a single dielectric or multiple dielectric layers) mounted on a conducting ground plane. The dielectric is usually made of glass-reinforced epoxy such as FR-4. For very high frequencies PTFE may be used. Other reinforcement/resin systems are also available.

For close spaced differences on woven glass reinforced dielectrics, refer to application note AP139 on the Polar Instruments web site, www.polarinstruments.com

There are several configurations of PCB transmission line:

- Exposed, or surface, microstrip
- Coated microstrip (coating usually solder mask)
- Buried, or embedded, microstrip
- Centred stripline
- Dual (offset) stripline
- Coplanar strips and waveguides

The structures above can be constructed with single or multiple dielectrics.

Single-ended Transmission Lines

Single-ended transmission lines are the commonest way to connect two devices (i.e. a single conductor connects the source of a device to the load of another device). The reference (ground) plane provides the return path.

*Note that in the diagrams the trace is trapezoidal in profile and width, \( W \), refers to the trace width nearest the upper surface, \( W_t \); refers to the trace width nearest the lower surface.*

**Surface Microstrip**

In the diagram below (surface, or exposed, microstrip) the signal line is exposed (to air) and referenced to a power or ground plane. Structures are categorised according to the arrangement of the dielectric with respect to the trace (below or above the trace). The diagram below shows the surface microstrip structure using a single dielectric layer below the signal trace (designated 1B)
Surface microstrip with single dielectric below the trace

The diagram below shows the surface microstrip structure using two dielectric layers below the trace (designated 2B).

Surface microstrip with two dielectric layers below the trace

*Embedded Microstrip*

Embedded, or buried, microstrip is similar to the surface version, however the signal line is embedded between two dielectrics and located a known distance from the reference plane.

Embedded microstrip with two dielectric layers, one below and one above the trace

In this structure the two dielectrics are arranged one below and one above the trace (designated 1B1A). Embedding the
signal line can lower the impedance by as much as 20% compared to an equivalent surface microstrip construction.

**Coated Microstrip**

Coated microstrip with single dielectric below the trace

Coated microstrip is similar to the surface version, however the signal line is covered by a solder mask. This coating can lower the impedance by up to a few ohms depending on the type and thickness of the solder mask.

**Offset Stripline**

Offset Stripline 1B1A

In this configuration the signal trace is sandwiched between two planes and may or may not be equally spaced between
the two planes. This construction is often referred to as Dual Stripline.

A second mirror trace will be positioned \( H_1 \) from the top ground plane. These two signal layers will be routed orthogonally (crossing at right angles so as to minimise the crossing area).

**Differential Transmission Lines**

The differential configuration (often referred to as a balanced line) is used when better noise immunity and improved timing are required. In differential mode the signal and its logical complement are applied to the load.

The balanced line thus has two signal conductors and an associated reference plane or planes as in the equivalent single-ended (unbalanced) case. Fields generated in the balanced line will tend to cancel each other, so EMI and RFI will be lower than with the unbalanced line. External noise will be "common-mode out" as it will be equally sensed by both signal lines.

Note that in the following diagrams (except the Broadside-coupled Stripline) the traces are trapezoidal in profile and width, \( W \), refers to the trace width nearest the upper surface, \( W_1 \), refers to the trace width nearest the lower surface.

**Edge-coupled Surface Microstrip**

![Edge-Coupled Surface Microstrip 1B](www.polarinstruments.com)

Edge-coupled surface microstrip with single dielectric below the trace

In this construction the gap between the traces, \( S_1 \), defines the coupling factor and hence the differential impedance. The etch factor, plating density and undercut will make this construction simple to manufacture, but with a wider tolerance due to the extra processing required on external layers.
Edge-coupled Coated Microstrip

As in the case of the Surface Microstrip this construction is simple to fabricate, but the extra process of adding solder mask coating can cause impedance variations. The designer is able to specify the thickness of the coating outside, above and between the traces to allow for variations in the board fabricating process.

This construction is particularly sensitive to solder mask flooding with LPI (Liquid Photo Imagable) solder mask. This causes the dielectric constant in the edge coupling region to vary, depending on flood depth.

Edge-coupled Embedded Microstrip

The reduced processing of internal layers makes the Edge-coupled Embedded Microstrip construction easy to fabricate with more consistent results than the equivalent surface trace structure. During the manufacturing process resin will be forced in between the traces resulting in a resin-rich region (shown as Rer in the 1B1A1R model below) between the two traces. This region will result in a dielectric with Er different from the rest of the structure.
Edge-coupled embedded microstrip with resin-rich region between traces

*Edge-coupled Offset Stripline*

As in the case of the single-ended Offset Stripline construction this structure can be made up as a dual construction with a mirrored edge-coupled differential pair set a distance from the upper reference plane. The lower pair is routed orthogonal to the upper to minimise layer to layer coupling and cross-talk.

The model below shows a structure with two layers below the traces and one above and includes the resin rich region between the traces

Edge-coupled offset stripline structure modelling the resin-rich region between the traces
Broadside-coupled Stripline

Broadside-coupled offset stripline with two substrate dielectrics, H1, H2

This apparently simple construction is actually one of the most difficult to fabricate to produce consistent impedance results.

Despite having internal layers with minimal processing, the most common structure is that with both traces overlaid for maximum coupling.

Inner-layer mis-registration and slight offsets and differences in etching combine to make this more difficult to achieve consistent results, particularly if the traces are fine-line.

Broadside-coupled offset stripline with three substrate dielectrics, H1, H2 and H3

The broadside-coupled model assumes symmetry of dielectric in the two H2 and H3 layers — the two layers will normally be fabricated from the same material, i.e. with the same dielectric constant.

*Note that in the Broadside-coupled Stripline case the traces are trapezoidal in profile and width, W2 refers to the trace width nearest the surfaces, W1 refers to the trace width nearest the center.*
Coplanar Lines

Most microstrip and stripline transmission line structures can be manufactured in a coplanar version.

Coplanar structures have the advantage of single-sided construction with the signal line and ground on the same plane. Components can be grounded on the same plane as the signal line; this means the coplanar configuration is ideally suited for surface mounted devices.

In addition, the coplanar configuration shows only minor dispersion effects compared to microstrip lines.

Coplanar lines incorporate ground conductors adjacent to the controlled impedance trace(s) in the same plane as the trace(s).

Surface Coplanar Strips

Surface Coplanar Strips with Ground

Coplanar lines may be constructed with or without a ground plane underneath the controlled impedance trace(s).

This structure is an example of a controlled impedance trace on a single sided board that will typically be used in consumer applications.
Differential Surface Coplanar Strips

The diagram above shows a differential surface coplanar structure with strips and a lower ground plane fabricated using two dielectric layers.
Installing the Si8000m/Si9000e

Activating the Field Solver and license options

Note: It will be necessary to activate the product license prior to performing calculations with the field solver.

Versions 15.10 and later versions of Polar software are based on FlexNet Publisher v11.13.1.2.

Note: FlexNet Publisher v11.13.1.2 requires Windows 7 or later (see Polar Application Note AP605 System requirements for Polar software products)

If you are upgrading from an earlier version of Polar software it may be necessary to request an updated license file if the addresses referenced by a license file are no longer seen by the license manager. If you have either node-locked or 5/1 licenses you may therefore need to resubmit your HOSTID information for Polarcare to generate a new license in order for your 15:10 license to reactivate.

The Polar licensing system supports both floating licenses and licenses node-locked to a machine’s ethernet address or to FLEXnet ID dongles.

Floating (counted) licenses will require the server-side installer, available from the Polar web site support page.

If a hardware key (dongle) license has been purchased it will be necessary to download and install the key drivers (available from the Polar web site support page.)

Contact Polarcare at polarcare@polarinstruments.com or your local office for licensing information.

Choosing purchased license options

Select the Configuration menu and choose the License Options command to display the License Options dialog; click your purchased License options and click Apply.

Uninstalling the software

To uninstall the software click the Windows Start button and choose Control Panel. Double-click Programs and Features and choose Si8000m or Si9000e from the list. Right click and choose Uninstall.
Using the Quick Solver

The Quick Solver interface

<table>
<thead>
<tr>
<th>Lossless Calculation</th>
<th>Frequency Dependent Calculation</th>
<th>Sensitivity Analysis</th>
<th>Via Checks</th>
</tr>
</thead>
</table>

**Si9000e Quick Solver**

**Startup Mode**

Calculation modes (Lossless Calculation, Frequency Dependent Calculation, Sensitivity Analysis and Via Checks) are selected via the associated tabs.

**Lossless Calculation**

The Lossless Calculation Interface displays structure parameters and tolerances and calculation/goal-seeking results.

**Frequency Dependent Calculation**

The Si9000e includes the Frequency Dependent Calculation tab. The Frequency-dependent Calculation Interface displays frequency dependent and structure parameters.

It will often be necessary, however, to begin frequency dependent calculations by selecting the Lossless Calculation tab to enter structure parameters. In this case it
may be found more convenient to change the Startup tab to the Lossless Calculation tab – see Specifying the Startup tab below.

Sensitivity Analysis tab

Select the sensitivity analysis tab to display the effects of varying parameters such as charting the variation in impedance as substrate height varies.

Via Checks tab

Select the Via Checks tab to run simple colour coded go/no go checks on the potential for signal distortion of a via stub.

Specifying the Startup tab

From the Configuration menu choose Startup Mode to specify the tab displayed when the program is started.

Choose the option and click Apply.

Quick Solver screen areas

The Quick Solver is divided into the following areas:

The Menu system – comprising the File menu, containing the commands to save and open databases and projects, print results and import and export files in third party formats; the Edit menu, for copying parameters and results to the clipboard; the Configuration menu, to set up the operating parameters, licensing options and paths to optional components and the Help menu, to view license information and controlled impedance application notes on the Polar Instruments web site

The Toolbar – containing all the commands and structure range select buttons

Structure Bar – displaying available structures within the selected range

Structure Graphic – displaying the selected structure and associated parameters
Lossless calculations

The Lossless Calculation Interface displays structure parameters and tolerances and calculation/goal-seeking results.

Calculation Options allow the user to select parameter units, standard or extended interface style and goal seeking convergence (see Field solving for board parameters).

Calculations include delay, inductance, capacitance, effective dielectric constant and velocity of propagation.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impedance (Z₀)</td>
<td>75.178</td>
</tr>
<tr>
<td>Delay (ps/in)</td>
<td>144.114</td>
</tr>
<tr>
<td>Inductance (nH/in)</td>
<td>10.834</td>
</tr>
<tr>
<td>Capacitance (pf/in)</td>
<td>1.917</td>
</tr>
<tr>
<td>Effective Dielectric Constant</td>
<td>2.893</td>
</tr>
<tr>
<td>Velocity of Propagation (CTS)</td>
<td>0.588</td>
</tr>
</tbody>
</table>
Frequency-dependent calculations (Si9000e only)
The Frequency-dependent Calculation Interface displays frequency-dependent and structure parameters.

Frequency-dependent Result Graph and Tables display frequency-dependent results in graphical and tabular form.

Options and parameter settings for the presentation of frequency dependent data include: linear or logarithmic scales, units (inches/meters) for line length, Er/TanD options, user-editable surface compensation methods, measurement data options, s-parameter configuration.
**Sensitivity Analysis**

Sensitivity analysis provides fast and interactive built-in graphing of impedance variation against a range of physical structure parameters.

Select the sensitivity analysis tab to display the effects of varying parameters (for example, chart the variation in impedance as substrate height varies.) The Si9000e graphs impedance for single-ended and differential structures: odd mode / even / differential / common / all.

Results are displayed both graphically and in table form for export via the clipboard for use in Excel, etc.

Alternatively, the graph may be exported to JPEG for easy and convenient inclusion in your documentation.
Via Checks

Via modelling

Via modelling provides for simple modelling of plated through hole (PTH) vias with respect to impedance and signal integrity, recognizing the need to present to a signal a constant impedance as it propagates between devices.

The Quick Solver incorporates a Via Check tab that provides a simple color coded go/no go check on the potential for signal distortion of a via stub. Interactive controls let you run some basic checks to calculate whether via stubs are likely to be visible to signals at your chosen operating speed.

Unconnected via stubs have the potential for a far larger effect on the signal than the geometry of the via itself.

Simple interactive controls allow rapid analysis of the potential effects of a via's stub for different values of stub length, signal risetime and dielectric constant.

Via pad/anti-pad coaxial calculation

The Via Checks tab also includes via pad/anti-pad calculation. The anti-pad is the void area (shown as the blue annular ring in the diagram above) between the pad and the copper of the plane. It is generally designed so that it maintains the impedance of a transmission line as it passes through the plane.

Detailed analysis of vias and pads and antipads can prove complex, requiring the analytical functions of a 3-D solver, however, in many cases this straightforward check will ensure that any major mismatches are removed before you resort to more exhaustive analysis.
**Menu/Toolbar**

The menu system and toolbar contain all the field solver commands, the structure range select buttons, the Speedstack/CGen Copy and Paste buttons, the Atlas and CITS data log import and Track Resistance Calculator buttons.

Display structures

- Display all structures
- Display user structures
- Si Project

Single ended/differential structures

- Display single ended structures
- Display differential structures
- Display groundless differential structures

Coplanar single ended structures

- Display surface coplanar structures
- Display coated coplanar structures
- Display embedded coplanar structures
- Display offset coplanar structures

Coplanar differential structures

- Display surface coplanar differential structures
- Display coated coplanar differential structures
- Display embedded coplanar differential structures
Display offset coplanar differential structures

**Toggle hatch plane**

- Toggle lower hatch plane
- Toggle upper hatch plane

**Process window**

- Multiline Crosstalk
- Launch process window
- Monte Carlo Analysis

**Copy/paste structure parameters**

- Copy structure parameters
- Paste structure parameters

**Copy/paste structures to/from Speedstack**

- Copy structure to Speedstack
- Paste structure from Speedstack

**Copy structure to CGen**

- Copy structure to CGen

**Paste from Speedstack into Si Project**

- Paste structures from Speedstack into Si Project

**Import measurement data**

- Import measurement data from CITS
- Import measurement data from Atlas
Using the Quick Solver

**Import measurement data from Touchstone**

**Excel interface**

**Launch Si Excel interface**

**Track resistance calculator**

**Launch Track Resistance Calculator**

Use the File menu commands to save, recall and print results and the Edit menu to copy frequency-dependent tabular data via the Windows clipboard to a spreadsheet or database for analysis.

Use the Configuration menu to set structure parameter minimum and maximum values and goal seeking convergence settings used by the calculation engine.

The Help menu contains the product license status and links to controlled impedance-related pages on the Polar Instruments web site.

Clicking each Toolbar structure buttons selects the associated range of controlled impedance structures (single-ended, differential, coplanar, etc.) for display in the Structure Bar.

Use the Copy and Paste buttons to exchange controlled impedance information with the Speedstack PCB Stackup Builder.

**Structure Bar**

Use the Structure Bar to select a controlled impedance structure from the list of structures displayed. The range of
structures displayed is controlled by the associated button on the Toolbar.

Structure graphics

The Structure Graphic reflects the chosen controlled impedance. For lossless modelling click the parameter "hotspot" (the parameter label, H1, Er1, etc.) to select a parameter field for editing.

Calculation Interface

Use the Calculation Interface to enter and modify the structure parameters and tolerances, calculate impedance values and goal seek for parameter values for a target impedance.
Extended Interface

Use the Extended Interface to apply tolerances to a calculation. The colored text fields indicate which parameter affects the minimum and maximum impedance values; e.g. consider the green colored fields – variations in the minimum value of Er1 affect the maximum value of impedance.

Quick Solver operating configuration

Calculation Options

Use the Calculation Options to specify the calculation units, interface style and goal seeking convergence settings.
Setting parameter limits

The field solver is designed to work with “real world” values. If the parameter values used in calculation are beyond its operating limits, the calculating engine returns a value of zero. The user is able to control the range of values used by the field solving engine during calculation.

Click the Configure menu and choose Parameters; the Configuration dialog is displayed.

Enter the values for minimum and maximum for each parameter.

**Goal Seek parameters**

Specify the Goal Seek parameters, the number of calculation iterations (Goal Seek Tries) and convergence settings.

**Etch factor settings**

Specify the etch factor settings (the default value is 1.000) to be applied to the upper trace width to model the effect of the etching process. Quick Solver will accept both positive and negative values.
Hatch configuration

Quick Solver hatch plane/mesh module

The Quick Solver provides a practical method of predicting the impedance of stripline and microstrip PCB traces when crosshatching (or meshed) return paths are deployed rather than the solid copper return paths of traditional rigid PCBs.

Modelling impedance on traces with hatch plane grounds

Careful use of crosshatched planes on flex and flex-rigid PCBs has proved a practical method of keeping controlled impedance traces at wider, more manufacturable dimensions while also achieving the desired flexibility of the assembly. Crosshatching is also deployed to keep impedance controlled line widths at reasonably manufacturable geometries – for example, on interposer boards.

The XFE field solver enhancement can be used to model more closely the impedance as fabricated on a flex-rigid PCB using crosshatch return planes.

XFE – Crosshatch Flex Enhancement

The Quick Solver’s proprietary technique, XFE (Crosshatch Flex Enhancement) employs Polar’s 2-D field solvers but uses a unique algorithm to correct for the effects of flex over a wide range of typical controlled impedance structures.

![Hatch Configuration](image)

The XFE option, applicable to the lossless mode of the Quick Solver allows for configuration of hatch pitch (HP) and width (HW) as shown in the above graphic.

Hatch pitch and width may be specified either directly or by association with a choice of copper area settings. Set the pitch by simply dragging the Hatch Pitch slider and visually monitoring the Copper Area for the required percentage.

If the desired Copper Area is known, select the %age from the preset value buttons — the hatch width will automatically be calculated for a given hatch pitch.
**Startup Mode**

From the Configuration menu choose Startup Mode to specify the tab displayed when the program is started.

Choose the option and click Apply.

**Graph Style**

Choose between the Default and Enhanced graph styles to display the graphs of loss, impedance, etc. with standard or heavy line weights.

**Solver accuracy**

From the Configuration menu switch the solver accuracy between Default and Enhanced modes.

The Enhanced Mode is especially useful when calculating fine trace thickness geometries.

Note: Enhanced Mode will increase calculation time.

**TRC configuration**

If necessary, specify the location of the Track Resistance Calculator executable.
Lossless calculations

Lossless modelling

The Si8000m/Si9000e Field Solver allows the operator to perform rapid single calculations of PCB trace values against significant PCB parameters. The Field Solver solves for impedance, propagation delay and inductance and capacitance per unit trace length along with effective dielectric constant and velocity of propagation.

Click the Field Solver icon on the desktop to start the program.

Click the **Lossless Calculation** tab

| Lossless Calculation | Frequency Dependent Calculation |

Calculating single ended impedance

Click on the structure type from the Structures Bar.

Select the dimension units (mils, inches, microns or millimetres) from the Units option group.

Enter the values for:

- **H1 (Height)** — dielectric thickness
- **W1 and W2 (Width)** — signal trace width (allowing for finished etch factor)
- **T1 (Thickness)** — signal trace thickness
- **Er1** — dielectric constant

into the associated text boxes and press the Impedance **Calculate** button. The calculated impedance will appear in the Impedance (Zo) box.

Add explanatory notes on your particular construction, if necessary, in the Notes text box.

Calculating propagation delay, inductance and capacitance

Click on the configuration from the **Structures** menu or from the Structures Bar.

Enter the parameter values as described above into the text boxes and press the **More...** button. For the Standard Interface single ended results are shown below.
The More Information dialog displays the results of impedance, propagation delay, inductance and capacitance along with effective Er and velocity of propagation (useful for Test Editor | Vp entry within the Polar CITS test editor) in the selected units. Press Close to exit.

Field solving for board parameters (goal seeking)

The Field Solver can solve (goal seek) for board parameters given a nominal (target) impedance value.

Enter the given board dimensions in their associated fields and the nominal impedance value in the Impedance field and click the Calculate button against the unknown dimension, e.g. Substrate 1 Height, Trace Width, etc.

Specifying Goal Seeking convergence

The convergence values used during goal seeking are specified in the Configuration screen (see Setting parameter limits). Choose between fine convergence to derive parameters whose values may be infinitely variable (e.g. trace width) and coarse convergence for parameters whose values may be fixed by supplier (e.g. height).

Typically the user will enter all the known parameters and goal seek for the desired impedance on the dielectric height. Using the coarse convergence option can speed up goal
seeking on complex structures. The trace width can then be derived using fine convergence.

**Using the Extended Interface**

Selecting the Extended Interface Style displays additional fields, Tolerance, Minimum and Maximum allowing the user to specify a range of values for each parameter and observe the effect of manufacturing process variations.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Tolerance</th>
<th>Minimum</th>
<th>Maximum</th>
<th>Calculate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Substrate 1 Height</td>
<td>H1</td>
<td>8.5000</td>
<td>±</td>
<td>0.0000</td>
</tr>
<tr>
<td>Substrate 1 Dielectric</td>
<td>E1</td>
<td>4.2000</td>
<td>±</td>
<td>0.0000</td>
</tr>
<tr>
<td>Lower Trace Width</td>
<td>W1</td>
<td>7.0405</td>
<td>±</td>
<td>0.0000</td>
</tr>
<tr>
<td>Upper Trace Width</td>
<td>W2</td>
<td>6.0405</td>
<td>±</td>
<td>0.0000</td>
</tr>
<tr>
<td>Trace Thickness</td>
<td>T1</td>
<td>1.2000</td>
<td>±</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

Fields which control the maximum impedance value are shown in green, fields which control the minimum impedance value are shown in orange.

In this example we specify a nominal impedance value of 80 ohms and observe the effects on the nominal impedance of a manufacturing variation of ±1 mil in the substrate height.

Select the Extended Interface, enter a value of 80 ohms in the Impedance field and click the Substrate 1 Height Calculate button

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Tolerance</th>
<th>Minimum</th>
<th>Maximum</th>
<th>Calculate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Substrate 1 Height</td>
<td>H1</td>
<td>9.8301</td>
<td>±</td>
<td>0.0000</td>
</tr>
<tr>
<td>Substrate 1 Dielectric</td>
<td>E1</td>
<td>4.2000</td>
<td>±</td>
<td>0.0000</td>
</tr>
<tr>
<td>Lower Trace Width</td>
<td>W1</td>
<td>7.0405</td>
<td>±</td>
<td>0.0000</td>
</tr>
<tr>
<td>Upper Trace Width</td>
<td>W2</td>
<td>6.0405</td>
<td>±</td>
<td>0.0000</td>
</tr>
<tr>
<td>Trace Thickness</td>
<td>T1</td>
<td>1.2000</td>
<td>±</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

The nominal Substrate 1 Height is calculated at 9.83 mil.

To calculate the effect on impedance of a ±1.0 mil tolerance in the substrate height, enter a value of 1 mil in the Substrate 1 Height Tolerance field – the minimum and maximum fields are automatically completed.
Click the Impedance Calculate button to calculate the range of impedance for a 1mil variation in H1 as 76.30–83.37 ohms.

**Using multiple tolerances**

Other parameter tolerances can included as necessary. Enter a value of 0.5 in the Substrate 1 Dielectric Tolerance field and click the Impedance Calculate button. The impedance range should now show 72.74–87.88 ohms.

**Calculating differential impedance**

Calculating differential impedance is similar in technique to that for the single-ended models, but with the addition of trace separation or offset.

For some models the dielectric constant of the separation region can be specified separately from the substrate dielectric constant bulk value.

Enter the parameter values and tolerances if required into their respective fields and Click Calculate to calculate the resulting impedance.

Use the other Calculate buttons to goal seek for the parameter values required to return a target impedance.
Calculating propagation delay, odd, even and common mode impedance

For the Standard Interface clicking More... displays results for differential impedance include odd, even and common mode impedance, effective Er and near end crosstalk.

Clicking More... on the Extended Interface displays the range of results for the selected tolerances.

Saving and recalling results

Impedance calculation results for a board type or vendor, for example, may be saved to disk and recalled for future reference.

From the File menu choose the Save As... command. Choose a name and destination and press Save.

The program will only save calculated results.

To recall a set of results choose Open... from the File menu and choose the desired results file and press Open.
Copying and pasting parameters between structures

The parameters, both lossless and frequency dependent (Si9000e only,) of a controlled impedance structure may be copied to the clipboard and then pasted to another structure.

Example: model impedance with and without coating.

Select the single ended Coated Microstrip 1B structure and goal seek using trace width for 50 ohms.

Copy the structure parameters to the clipboard

Select the single ended Surface Microstrip 1B structure.

Paste the coated microstrip structure parameters from the clipboard to the new structure and click Calculate.

Si9000e calculates the impedance of the same structure prior to coating, illustrating the approximately 2 ohms difference.
Si Crosstalk – modelling multiline crosstalk

*Note:* Prior to performing multiline crosstalk modelling ensure that the Multiline Crosstalk License (XTALK) option is selected.

Select the Configuration menu | License Options command to display the License Options dialog; click the Multiline Crosstalk License (XTALK) check box and click Apply.

The Si Crosstalk multiline and differential pair (lossless) crosstalk add on option for the Si8000m and Si9000e allows you to model coupling between aggressor and victim traces.

**Crosstalk** (the unwanted coupling of energy between two or more adjacent lines on a PCB) can alter the required signal. The Si8000m / Si9000e presents crosstalk graphically for easy inspection and the lossless data may also be exported in Touchstone™ format for further analysis.

The coupling is modeled against frequency and line length and allows a designer to plan for enough trace separation between individual signal lines or between differential pairs for crosstalk to be within safe limits. Both *near* and *far-end* crosstalk are modeled for stripline and microstrip cases.
**Forward and reverse crosstalk**

*Forward,* or *far-end,* crosstalk is energy that is coupled from the active signal line, the aggressor, onto a quiet passive victim line so that the transferred energy "travels forward" to the end of the victim line. *Forward,* or *far end,* crosstalk can be a problem if it is necessary, for example, to use long traces on outer layers.

*Reverse,* or *near-end,* crosstalk is energy that is coupled from the actual signal line, the aggressor, onto a quiet passive victim line so that the transferred energy "travels back" to the start of the victim line.

*Reverse,* or *near end,* crosstalk can be an issue when using high speed circuit components with adjacent input and output signal lines.

With the Si Crosstalk option it is easy to illustrate, for example, how surface traces are much more prone to far end crosstalk than stripline traces.

See Polar Application Note AP8164 *Introduction to forward and reverse crosstalk.*

**Conductor configurations**

Modelling is provided for both *Aggressor – Victim* and *Aggressor – Victim — Aggressor* conductor configurations.

<table>
<thead>
<tr>
<th>Conductor Configuration</th>
<th>1</th>
<th>3</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conductor / Port Number</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Voltage</td>
<td>+1</td>
<td>0</td>
<td>+1</td>
</tr>
</tbody>
</table>

The aggressor and victim lines and associated ports for the selected conductor configuration are reflected in the accompanying graphics as shown in the dialogs below.
**Aggressor – Victim – Aggressor conductor configuration**

**Modelling single ended microstrip traces**

This example will model the crosstalk between a pair of adjacent surface microstrip traces.

In the Lossless Calculation tab choose the single ended Surface Microstrip structure and enter the parameters for the target impedance.

Click the Multiline Crosstalk icon to display the Multiline Crosstalk dialog – the Multiline Crosstalk option will run a pair of surface microstrips alongside each other and model the resulting crosstalk; i.e. it uses the selected structure and models a pair of those lines side by side.

For this example, an aggressor trace and a victim trace, choose the Aggressor – Victim conductor configuration.
The Crosstalk graphic displays the two lines and four ports associated with the model.

Choose the aggressor and victim lines and voltages. Supply the parameters, line length, separation, etc. for the model:

<table>
<thead>
<tr>
<th>Conductor / Port Number</th>
<th>1</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Voltage</td>
<td>+1</td>
<td>&lt; S &gt;</td>
</tr>
<tr>
<td>Length of Line</td>
<td>LL</td>
<td>1.0000</td>
</tr>
<tr>
<td>Separation</td>
<td>S</td>
<td>0.0000</td>
</tr>
<tr>
<td>Frequency Minimum (MHz)</td>
<td>FMin</td>
<td>500.000</td>
</tr>
<tr>
<td>Frequency Maximum (GHz)</td>
<td>FMax</td>
<td>10.000</td>
</tr>
<tr>
<td>Frequency Steps</td>
<td>FSteps</td>
<td>300</td>
</tr>
</tbody>
</table>

Check the ports to be displayed and click Calculate.

Change the line length, separation, frequency, etc. to examine the effect of changing parameters on near and far end crosstalk.
Modelling differential pairs

Select the Edge-Coupled Surface Microstrip structure and supply the parameters (goal seeking if necessary) for the target impedance.

Click the Multiline Crosstalk icon to open the Multiline Crosstalk dialog. Choose the aggressor and victim lines and applied voltages. The Multiline crosstalk graphic reflects the differential pairs in the model.

Check the ports to be displayed and choose between Odd/Even/Single Ended – click a data series to display the Picked Data Point Information frequency and magnitude.

Lossless calculations • 43
Monte Carlo impedance analysis

Si8000m and Si9000e provide analysis tools for production control in manufacturing when building high volumes of printed circuit boards.

Si8000m and Si9000e include Monte Carlo simulation of printed circuit board impedance to provide a graphical mechanism for predicting and presenting the variation of PCB trace impedance for a production run of PCBs.

The Si8000m/Si9000e simulation can range from varying a single parameter (for example, the thickness of a layer of prepreg material) over a range of possible values to randomising all input parameters for a structure. The number of iterations can be specified to reflect the number of boards in a typical production run.

Using Monte Carlo analysis

Choose the structure to be analysed.

Use the Lossless Calculation interface to model the structure and arrive at the parameter values for the target impedance.

Click Monte Carlo Analysis to open the Monte Carlo Analysis interface.

Supply the Tolerance values for each parameter to vary – either as absolute values or as percentages. (For the dimensional values in the graphic above the field solver will simulate a 5% variation in all parameters.)

Choose the number of iterations.
Choose Uniform Distribution or Normal Distribution

Click Calculate – the field solver will build a bar graph – in this case a normal distribution curve – of the variations in trace impedance for the range of parameter values.
Exchanging stackup structure information with Speedstack

Polar field solvers exchange controlled impedance structures with the Speedstack Stackup Design System. Use the buttons shown below to import stackup layer data from Speedstack for use with the field solver goal seeking facility, and to export calculated values to Speedstack.

Paste structure from Speedstack

Copy structure to Speedstack

Select the structure within Speedstack.

With the stackup parameters displayed in the Speedstack Controlled Impedance window, click the Speedstack To Field Solver button to transfer the current Speedstack parameters to the field solver.

Switch to the field solver and click the Paste Structure From Speedstack button to load the parameters into the associated fields.
Field solver parameter fields with Speedstack data loaded

In the sample diagram above, the target impedance for the structure in the stack in Speedstack will be 100 Ohms. The designer has chosen to use just the values for Trace Width to goal seek for the target of 100 Ohms. Click the Upper Trace Width Calculate button to goal seek on trace width to obtain the target impedance. The goal seek returns values for trace width to produce 100 Ohms final impedance.

Solved values for impedance

Click the Copy structure to Speedstack button

Switch to Speedstack and click the From Field Solver button – choose the properties to import and click Apply.
The structure is returned to Speedstack.

Rebuild and recalculate to refresh the stack. It may also be necessary to round some dimensions (e.g. dielectric heights) to the nearest practical values and recalculate the impedance.

**Creating a custom list of structures**

Use the My Structures function to create and edit a custom list of structures (the My Structures group).

Choose the Configuration|Structures command to display the complete list of structures – select the structure to be edited and click Edit.
Add the descriptive text to provide a label for the structure in the Alias field:

Click Yes to add to the My Structures list then click Apply.

The structure is added to the list of custom structures and is displayed with the edited title.
The structure alias is applied to both structure image titles and graph titles in the frequency dependent tab.

![Graph 1](image1)

and the sensitivity analysis tab.

![Graph 2](image2)

**Printing results**

Choose Print from the File menu to print a hard copy of the Quick Solver screen.
Using Si Projects

Si Projects allows the designer to store groups of related structures or rapidly to copy and paste an entire stackup’s impedance structures from Speedstack into the field solver for detailed analysis. This will allow the designer to group together a set of related structures for a particular design.

The Projects function will also be found useful for creating multiple instances of the same structure type with different parameter values. Integration with Speedstack allows the easy import of a complete set of Speedstack structures in a single step.

Creating new projects

Click the Projects button to create a new project.

With the structure displayed, click the Add Structure to Project button

Supply a descriptive name for the project structure:

```
Project Structure Name - Add
```

Structure Name (for example, L1 50 ohms)

```
M-Board L1 SE 50 Ohms
```

The structure is renamed and added to the project group.
A selected structure belonging to a Project is denoted by the grey background.

To add another structure, select the structure, modify its parameters to achieve the target impedance then click the Projects button.

**Working with Si Projects in Speedstack**

The Si Projects feature incorporated in Speedstack and Si8000m/Si9000e allows for easy transfer of controlled impedance structures from the Speedstack stackup design tool into the field solver.

Si Projects allows groups of structures to be saved and recalled in Si8000m/Si9000e and entire stackups of structures to be pasted from Speedstack into the field solver with just a few clicks of the mouse – the toolbar option copies a group of structures from Speedstack and places them onto the clipboard, these structures can then be pasted directly into the Si Project group.

**Transferring structures from Speedstack to the field solver**

The example stackup below in Speedstack's Stackup Editor contains controlled impedance structures in the layers indicated by the Ohms symbol.
Click Speedstack's Controlled Impedance tab to display the structures.

Use the toolbar buttons in the Speedstack and the field solver interfaces to transfer the structures to the field solver. Within Speedstack click the To Si Project toolbar button:

Switch to the field solver and click the Paste from Speedstack into Si Project toolbar button:

The set of structures appears in the Project window.
$L_n$ indicates the layer number in Speedstack.

**Adding/deleting and modifying structures**

Selecting each structure displays its associated graphic in a grey background.

Right click on a structure in the structure list to view the structure options. Structures can be renamed, moved up or down, duplicated or deleted. Select Clear Project to remove all structures.
Click the + and – buttons in the structure graphic to add additional structures from the Si structure library or remove selected structures from the Project folder. Click the Rename Structure (the pencil icon) to assign the structure a descriptive name.

**Calculating impedance and insertion loss.**

With a structure selected the structure parameters can be modified as required and the impedance recalculated.

Once the Speedstack structures have been imported into the Si Project, use the frequency dependent calculation options to predict the conductor loss, dielectric loss and total attenuation for each structure.
Importing CITS log file data

The Si8000m/Si9000e can import and read CITS data log files containing measured impedance data for analysis, comparing modelled and measured data for controlled impedance structures in Si Project files. This allows for display of the logged data against the predicted values.

Click File|Open Project… and choose the project (.SIP) file. The Si Project file is opened with the project structures displayed in the structure bar.

To import the CITS data log file for analysis click the Import measurement data from CITS button. The Import CITS File dialog is displayed.

**Step 1 Reading the log file**

Click the File Import button and navigate to the log file and click Open – then click Read to load the data and associated dialog fields.

Summary data includes the CITS model and serial number along with the total number of data records, number of coupons per board and number of boards in the log file.

(In the example above the Data Log Record Count, 160, reflects the 40 coupons with 4 tests per coupon.)
**Step 2 Selecting the Data Log record**

When the log file is read the measured data for each structure in the project may be selected for display and compared with the associated modelled impedance.

Click the Data Log Records drop-down to select the data log for each structure.

Click the Project structure drop-down to select the project structure for the associated data log record.

The selected structure graphic is displayed along with the resulting chart displaying the logged data for the structure.

The chart above displays the logged measured data for each board along with the nominal impedance value.
The structure graphic for the project structure is shown alongside the log summary.

Graph settings

The Graph Settings dialog allows modelled and measured impedance to be displayed and compared; options for display include modelled nominal, minimum and maximum values and measured nominal values and tolerances.

The chart below adds modelled and measured nominal values and tolerances to the displayed data.
Click any data point to show detailed test results in the Picked Data Point Information box.

**Impedance result filtering**

Outlying or invalid data values may be excluded from the chart; open or short circuit readings that occurred during testing will typically not be regarded as valid for logging so will generally be excluded.

Use the Include Impedance Results options to filter out errant data log values that could cause the plots to become difficult to read due to axis scaling issues. (Note that all the data log file is read by the software – the Impedance Result filtering is applied only during the graphing phase.)

Ticking or clearing the Pass or Fail checkboxes will display or exclude PASS or FAIL data (i.e. results exceeding the tolerance limits) respectively. Displaying failed readings only would allow detailed analysis of the failed tests.

**Applying statistical analysis**

Click the Analysis (1) tab to display the log statistically – i.e. chart the number of boards v impedance; Analysis(1) displays a line graph histogram.

Click the Analysis (2) tab to display the log statistics in bar chart form.

Click the Measurement Data tab to display the log of raw data in tabular form.
Sensitivity analysis

Graphing impedance against multiple parameters

The Sensitivity Analysis tab provides access to fast and interactive built-in graphing of impedance variation against a range of physical structure parameters. Sensitivity analysis allows for:

- Graphing impedance against any varying structure parameters
- Setting a target impedance line on the graphs
- Exporting the graph data to clipboard for use in Excel
- Graphing impedance for single-ended structures
- Graphing differential structures: Odd mode / Even / Differential / Common / All
- Exporting graphs to JPEG for easy and convenient inclusion in your documentation

Select the sensitivity analysis tab to display the effects of varying parameters (for example, chart the variation in impedance as substrate height varies.)

Varying a single parameter

*Charting $Z_0$ as $H_1$ varies*

In this example, given the values below in the Lossless Calculation tab, switch to the Sensitivity Analysis tab (all values in mil.)

<table>
<thead>
<tr>
<th>Substrate 1 Height</th>
<th>$H_1$</th>
<th>Tolerance</th>
<th>Minimum</th>
<th>Maximum</th>
<th>Calculate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Substrate 1 Dielectric</td>
<td>$E1$</td>
<td>4.2000</td>
<td>± 0.0000</td>
<td>4.2000</td>
<td>4.2000</td>
</tr>
<tr>
<td>Lower Trace Width</td>
<td>$W1$</td>
<td>7.0000</td>
<td>± 0.0000</td>
<td>7.0000</td>
<td>7.0000</td>
</tr>
<tr>
<td>Upper Trace Width</td>
<td>$W2$</td>
<td>6.0000</td>
<td>± 0.0000</td>
<td>6.0000</td>
<td>6.0000</td>
</tr>
<tr>
<td>Trace Thickness</td>
<td>$T1$</td>
<td>1.2000</td>
<td>± 0.0000</td>
<td>1.2000</td>
<td>1.2000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Impedance</th>
<th>$Z_0$</th>
<th>Minimum</th>
<th>Maximum</th>
<th>Calculate</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Z_0$</td>
<td>75.18</td>
<td>75.18</td>
<td>75.18</td>
<td>Calculate</td>
</tr>
</tbody>
</table>

The parameters shown, including a nominal value of $H_1$ of 8.5, result in an impedance of 75.18 Ohm. To chart the effect on impedance of varying $H_1$, specify the range of $H_1$ values, from a minimum of 4 to a maximum of 10; use an increment of 1.
Click Calculate – the range of impedance values against substrate height is shown below.

Varying multiple parameters

**Charting Z₀ as H₁ and H₂ vary**

For this example, choose an edge coupled stripline and chart the variation in Z₀ as the two substrate heights are varied.

Choose the structure Edge-Coupled Offset Stripline 1B1A
Enter the parameters below and goal seek on trace width for a differential impedance of 75 Ohms.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>H1</th>
<th>Tolerance</th>
<th>Minimum</th>
<th>Maximum</th>
<th>Calculate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Substrate 1 Height</td>
<td>8.000</td>
<td>0.000</td>
<td>8.000</td>
<td>8.000</td>
<td></td>
</tr>
<tr>
<td>Substrate 1 Dielectric</td>
<td>4.200</td>
<td>0.000</td>
<td>8.000</td>
<td>8.000</td>
<td></td>
</tr>
<tr>
<td>Substrate 2 Height</td>
<td>8.000</td>
<td>0.000</td>
<td>8.000</td>
<td>8.000</td>
<td></td>
</tr>
<tr>
<td>Substrate 2 Dielectric</td>
<td>4.200</td>
<td>0.000</td>
<td>8.000</td>
<td>8.000</td>
<td></td>
</tr>
<tr>
<td>Lower Trace Width</td>
<td>7.4980</td>
<td>0.000</td>
<td>7.4980</td>
<td>7.4980</td>
<td></td>
</tr>
<tr>
<td>Upper Trace Width</td>
<td>6.4980</td>
<td>0.000</td>
<td>6.4980</td>
<td>6.4980</td>
<td></td>
</tr>
<tr>
<td>Trace Separation</td>
<td>5.000</td>
<td>0.000</td>
<td>5.000</td>
<td>5.000</td>
<td></td>
</tr>
<tr>
<td>Trace Thickness</td>
<td>1.2000</td>
<td>0.000</td>
<td>1.2000</td>
<td>1.2000</td>
<td></td>
</tr>
</tbody>
</table>

Note the nominal value for both H1 and H2 of 8 mil.

Switch to the Sensitivity Analysis tab and specify the parameters below – in this example vary both H1 and H2: increment H1 between 3 and 10 with an interval between increments of +0.1 mil, set the value of H2 to decrement by the same interval (–0.1 mil)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>H1</th>
<th>H2</th>
<th>Calculate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Range Start Value</td>
<td>3.000</td>
<td>11.000</td>
<td></td>
</tr>
<tr>
<td>Range Finish Value</td>
<td>10.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Increment</td>
<td>0.1000</td>
<td>-0.1000</td>
<td></td>
</tr>
</tbody>
</table>

Click Calculate – the result is shown below
Constant impedance v changing parameters

In this example, the differential impedance is held constant at 75 Ohms and the trace width, $W_1$, varied as $H_1$ and $H_2$ vary. Choose the varying parameter as $W_1$, set the target impedance as 75 Ohms and click Calculate below.

![Constant Impedance vs Changing Parameters](image)

The result is shown below.

![Edge-Coupled Offset Stripline 1B1A - 75 Ohms](image)

Choose the 3D option button to display the result in three dimensions, i.e. as all three parameters vary.

![Edge-Coupled Offset Stripline 1B1A - 75 Ohms](image)
Process Window: Minimum / Maximum

Click the Process Window: Minimum / Maximum check box to chart the effects of varying parameters within defined limits; in this example hold the differential impedance constant at 75 Ohms as above and vary the trace width, W₁ as H₁ and H₂ vary. Choose the varying parameter as W₁, set the target impedance as 75 Ohms and the upper and lower limits as shown below and click Calculate.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>75.0000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Target Impedance</td>
<td>75.0000</td>
</tr>
<tr>
<td>Process Window Minimum / Maximum</td>
<td>67.5000</td>
</tr>
</tbody>
</table>

The results are shown below.
Using Sensitivity Analysis to graph crosstalk

This example describes how to use the sensitivity analysis facility to graph the effect on crosstalk (NEXT and FEXT) of changing both the separation, S1, and trace width, W1 and W2, of a differential pair while maintaining constant impedance.

Setting the lossless parameters

Begin in the Lossless Calculation tab.

Select the Edge-Coupled Coated Microstrip 1B structure; use the default structure parameters but change the substrate height, H1, to 5.5 mils and calculate the impedance; differential impedance, Z\text{diff}, should equal close to 100 ohms.

Using impedance v changing parameters

Switch to the Sensitivity Analysis tab.

Under the Impedance vs Changing Parameter section select the first Parameter, to trace separation, S1, set the Range Start Value to 7 mils and the Range Finish Value to 50 mils. In the Constant Impedance vs Changing Parameters set the Parameter to trace width, W1 and the Target Impedance to 100 ohms.

Click Calculate in the Constant Impedance vs Changing Parameters section.
The Constant Impedance plot charts trace width vs trace separation over the selected range of values of S1 while maintaining a constant value of 100 ohms differential impedance.

Viewing tabular results

A subset of the results as trace width and separation vary is shown below. The results tab shows W1 / W2 / S1 changing; the impedance, ZDiff, calculates to 100 ohms but the NEXT / FEXT values for each W1 / W2 / S1 combination of parameter values are displayed.

This data can be exported to other tools (for example, Microsoft Excel®) for further analysis.
Sensitivity analysis includes graphing for differential, common, odd and even mode impedances along with near and far-end crosstalk.

Change the Display Series from Constant Impedance to NEXT / FEXT. The plot below shows NEXT / FEXT as $S1$ increases and $W1$ changes to maintain the target impedance of 100 ohms.
Copying Field Solver data to external programs (Si8000m)

To export the results of a sensitivity analysis calculation, ensure the Sensitivity Analysis tab is displayed and its Results tab is selected and displaying the table of calculated values.

<table>
<thead>
<tr>
<th>Graph</th>
<th>Results</th>
</tr>
</thead>
<tbody>
<tr>
<td>H1</td>
<td>E1</td>
</tr>
<tr>
<td>4.0000</td>
<td>4.2000</td>
</tr>
<tr>
<td>5.0000</td>
<td>4.2000</td>
</tr>
<tr>
<td>7.0000</td>
<td>4.2000</td>
</tr>
<tr>
<td>8.0000</td>
<td>4.2000</td>
</tr>
<tr>
<td>10.0000</td>
<td>4.2000</td>
</tr>
</tbody>
</table>

From the Edit menu choose Copy Current Results Tab to Clipboard

Si8000m takes the current results from the field solver and locates them on the Windows clipboard.

The result tables may then be pasted to a suitable location in a spreadsheet or database (e.g. a Si8000 or Si8000Expert workbook.) For a spreadsheet the values are inserted beginning at the active cell location. The number of cells required depends upon the structure chosen. Ensure no important data are overwritten in the process.

For example, the operator can use Excel’s Paste Function command to insert the associated function using the pasted Quick Solver parameter values as arguments.
Using Sensitivity Analysis to graph multiple impedances

The Polar field solver Sensitivity Analysis function allows the designer to graph multiple impedances.

For example, USB 2.0 guidelines specify routing the DP/DM signals with 90 ohms differential impedance, and 22.5~30 ohms common impedance. This example describes how to use sensitivity analysis to achieve both the differential (Zdiff) and common mode (Zcommon) specifications.

Using Constant Impedance vs Changing Parameters mode, setting the Target Impedance to 90 ohms and looking at the Zdiff, Zcommon or All Impedance display series allows the user to select a W1 / W2 / S1 combination that meets both differential and common impedance requirements.

Begin by clicking the Lossless Calculation tab.

Select the Edge-Coupled Coated Microstrip 1B structure; use the default structure parameters but change the substrate height, H1, to 5.5 mils and calculate the impedance; set a target differential impedance, Zdiff, of 90 ohms and goal seek on trace width to achieve 90 ohms. Parameters are shown below.

Switch to the Sensitivity Analysis tab.

Under the Impedance vs Changing Parameter section set the Parameter to trace separation, S1, set the Range Start Value to 3 mils and the Range Finish Value to 20 mils – choose an increment of 0.5 mils.
In the Constant Impedance vs Changing Parameters set the Parameter to trace width, W1 and the Target Impedance to 90 ohms.

Click Calculate in the Constant Impedance vs Changing Parameters section.

The Constant Impedance plot charts trace width v trace separation over the selected range of values of S1 while maintaining a constant value of 90 ohms differential impedance.

A subset of the sensitivity analysis results as trace width and separation vary is shown below.
The Results tab shows W1 / W2 / S1 changing; the differential impedance, $Z_{\text{diff}}$, calculates to 90 ohms but the table indicates how the common impedance value, $Z_{\text{common}}$, changes for each W1 / W2 / S1 combination of parameter values. This data can be exported to other tools (for example, Microsoft Excel®) for further analysis.

The associated graph (showing Zdiff at 90 ohms and Zcommon varying between 44 ohms and 24 ohms) is shown below.
Displaying all impedances

The sensitivity analysis function includes graphing for differential, common, odd and even mode impedances along with near and far-end crosstalk. Change the Display Series from Zdiff, Zcommon to All impedances. The plot below shows differential, odd mode, even mode and common impedances as S1 increases and W1 changes while maintaining the target differential impedance of 90 ohms.
Using sensitivity analysis to model the effects of an adjacent copper layer

In this example, sensitivity analysis is used to predict the effects on impedance of a copper layer adjacent to a controlled impedance structure – in this example, interstitial copper leaves in bookbinder flex.

It is sometimes useful to model the effects on a structure's impedance from an interstitial copper layer – either from a folded flex or from an interstitial leaf in a bookbinder flex.

Modelling the proximity of adjacent copper

To model the effects of an interstitial copper layer, use one of the multi-dielectric controlled impedance structures in the field solver, setting the Er of one of the dielectrics in the multi-dielectric substrate to the approximate value for air, i.e. set Er = 1.0000.

From the structure list choose the Offset Stripline 1B2A structure. In this example H3 serves as the separation from the interstitial copper leaf.

Set the structure parameters to their appropriate values, but set the value of Er3 to 1.0000. If necessary, goal seek on trace width to achieve the target impedance (In this example, 50 ohms.) Set the height of H3 to 20 mils.

Modelling the Er3 section of the structure above as air will model the proximity effect of the copper on the next structure above.

Switch to the Sensitivity Analysis tab.
From the Impedance v Changing Parameters dialog (above), choose the H3 parameter and set its start value to 3 mils and the end value to 20 mils with a 0.5 mils increment. Click Calculate.

The chart displays the impedance varying between 45 and 50 ohms as H3, the distance to the interstitial copper leaf, varies between 3 mils and 20 mils.

So, for this example, at anything greater than 20 mils distance the impedance is largely unaffected, at 10 mils the impedance drops by 1 ohm and at 3 mils distance the impedance has dropped by 10%.
The Si8000m/Si9000e incorporate a Via Check that provides a simple colour coded go/no go check on the potential for signal distortion of a via stub. The designer can run some basic checks to calculate whether via stubs are likely to be visible to signals at the chosen operating speed. The effects of the stub will increase as the stub length and Er increase and the signal rise time reduces.

Stub checks

Click the Stub option to calculate the effect of a stub

The Via Stub Check supports three modes:

- Stub Length, Effective Er and Bit Rate
- Stub Length, Effective Er and Single Frequency
- Stub Length, Effective Er and Rise Time
Click Bit Rate, Frequency or Rise time as appropriate and use the sliders to specify the stub length, Er value and your chosen parameter.

The Si9000e will change from green through amber and red to indicate the effects of the stub.

**Via pad/antipad coaxial calculation**

Via pad/antipad coaxial calculation provides for modelling plated through hole (PTH) vias with respect to impedance and signal integrity in order to allow the designer to ensure a constant impedance is presented to a signal as it propagates between devices.

The Via Checks tab includes via pad/anti-pad calculation. The anti-pad is the void area (shown as the blue annular ring in the diagram below) between the pad and the copper of the plane. It should be designed so that it maintains the impedance of a transmission line as it passes through the plane.

For example, assume a transmission line characteristic impedance of 50 Ohms; choose a via pad size (VP) of 12 mils (0.3mm) and calculate the anti-pad (AP) size that is required to present a nominal 50 Ohm impedance at this point.

For this calculation it is also necessary to specify the dielectric constant (Er1 illustrated above) in the region of the via. FR-4, a composite of resin (Er 3.2) and glass fibres (Er 6.1), will have a bulk Er of around 4.1 with significant local variations.
It is reasonable to assume that the $E_r$ value in the immediate vicinity of the via will be lower than the bulk $E_r$ of the dielectric material as more resin will tend to flow into this type of region. In this example specify $E_r1$ with a value of 3.5.

Enter the values of the via pad diameter, $V_P$, of 12mil (0.3mm) and the $E_r1$ of 3.5 into their respective fields. Move the slider bar for the anti-pad diameter, $A_P$, until the Impedance ($Z_0$) field displays 50 Ohms (alternatively, type the value into the Anti-Pad Diameter text box.)

Note: for this calculation the drilled size is required, not the finished size.
The Si8000m/Si9000e include the optional Track Resistance Calculator (TRC.) Calculating trace resistance will be found useful, for example, when working with fine geometry tracks where series loss must be considered.
Calculating track resistance

The TRC will accept values for track shape and length, along with material type and provide the DC resistance of the track in Ohms for the specified trace.

Specifying track dimensions

When the TRC is started, trace resistivity is automatically passed to the TRC alongside other parameters (Upper and Lower trace widths, W1, W2, Trace thickness, T1, and the length of the line, LL) specified in the lossless calculation tab.

The values for track dimensions can also be typed in directly or changed via the associated slider controls.

The TRC can work in all field solver units, Thou (mils), inches, microns (um) or millimetres. Click on each unit option to convert between units.

Choosing material resistivity

Choose from the dropdown material list to specify the material of the board or coupon trace.

The resistivity and conductivity of the selected material is displayed on screen; both trace resistivity (Ohm Metres) and conductivity (Siemens / m) are supported.
Editing material resistivity values

To add new materials or edit existing material values choose Tools|Edit Materials. The TRC displays both resistivity and conductivity values under the Materials and Resistivity Values editing dialog.

When adding or editing materials, either value can be specified; the reciprocal value is automatically calculated and added. Step through the available material images and assign a material image as required.
Frequency-dependent calculations (Si9000e only)

The Si9000e incorporates fast and accurate frequency-dependent PCB transmission line modelling, and extracts full transmission line parameters across its range of controlled impedance structures.

The Si9000e uses Boundary Element Method field solving to extract SPICE RLGC matrices and 2-Port S-Parameters for single-ended models or 4-Port S-Parameters for differential structures and provides high speed plotting of transmission line information for the structure under design.

The designer can choose graphing against frequency for impedance magnitude, loss (conductor loss, dielectric loss and insertion loss), inductance, capacitance, resistance, conductance and skin depth.

Click the Frequency Dependent Calculation tab; the Frequency-dependent interface is displayed.

Click the Graph tab and select the data series from the Display Series dropdown. The Si9000e displays results over the specified frequency range.

The graph below (All Losses) charts conduction loss, dielectric loss and insertion loss from 100MHz to 10GHz for a surface microstrip structure with the specified parameters.
To change the structure parameters, switch to lossless mode and modify values as required.

Select other data series and change parameters as required; the graph below shows the variation in impedance magnitude between 100Mhz and 1GHz.

The graph below show the variation in skin depth between 100MHz and 10 GHz.
Frequency-dependent Result Graph and Tables

Use the Result Graph and Table interface to view the frequency-dependent calculation results in both graphical and tabular form. The graph below charts all losses, and includes conductor loss and attenuation with roughness compensation.

**Viewing data in table form**

Switch to the table tabs to view the raw data in table form.

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>Impedance Real Dc (Ohm)</th>
<th>Impedance Imaginary Dc (Ohm)</th>
<th>Impedance Magnitude Dc (Ohm)</th>
<th>Inductance Birne (nH/m)</th>
<th>Resistance Dc Birne (Ohm)</th>
<th>Capacitance Birne (pF/m)</th>
<th>Conductance Birne (mS/m)</th>
<th>Skin Depth Birne (mm)</th>
<th>Conductor Loss Birne (W/m)</th>
<th>Dielectric Loss Birne (W/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.000E+03</td>
<td>5.0E03 ± 0.1</td>
<td>-2.22E-01</td>
<td>5.0E03 ± 0.1</td>
<td>7.687E-09</td>
<td>6.99E-01</td>
<td>2.99E-12</td>
<td>1.0E06 ± 0.1</td>
<td>1.14E-04</td>
<td>-5.47E-02</td>
<td>-3.09E-02</td>
</tr>
<tr>
<td>1.000E+04</td>
<td>5.0E04 ± 0.1</td>
<td>-3.41E-02</td>
<td>5.0E04 ± 0.1</td>
<td>7.6E-09</td>
<td>6.99E-01</td>
<td>2.99E-12</td>
<td>1.0E06 ± 0.1</td>
<td>1.14E-04</td>
<td>-7.31E-02</td>
<td>-7.11E-02</td>
</tr>
<tr>
<td>1.500E+04</td>
<td>5.0E04 ± 0.1</td>
<td>-4.32E-02</td>
<td>5.0E04 ± 0.1</td>
<td>7.6E-09</td>
<td>1.0E06 ± 0.1</td>
<td>2.99E-12</td>
<td>4.91E-04</td>
<td>6.77E-05</td>
<td>-9.43E-02</td>
<td>-8.65E-02</td>
</tr>
<tr>
<td>2.000E+04</td>
<td>5.0E05 ± 0.1</td>
<td>-5.33E-03</td>
<td>5.0E05 ± 0.1</td>
<td>7.6E-09</td>
<td>1.2E06 ± 0.00</td>
<td>2.99E-12</td>
<td>8.73E-04</td>
<td>7.16E-05</td>
<td>-1.03E-01</td>
<td>-1.43E-01</td>
</tr>
<tr>
<td>2.500E+04</td>
<td>5.0E05 ± 0.1</td>
<td>-6.35E-04</td>
<td>5.0E05 ± 0.1</td>
<td>7.6E-09</td>
<td>1.4E06 ± 0.00</td>
<td>2.99E-12</td>
<td>1.15E-03</td>
<td>8.71E-05</td>
<td>-1.28E-01</td>
<td>-1.72E-01</td>
</tr>
</tbody>
</table>

Single ended structures include graphs and data for 2 port s-parameters.
Differential structures include impedance values for odd and even modes, along with values for crosstalk and effective Er.

<table>
<thead>
<tr>
<th>Frequency</th>
<th>Impedance Real Value</th>
<th>Impedance Imaginary Value</th>
<th>Inductance Value</th>
<th>Resistance Value</th>
<th>Capacitance Value</th>
<th>Conducitance Value</th>
<th>Skin Depth</th>
<th>Conductor Loss (dB/line)</th>
<th>Conductive Loss (dB/line)</th>
<th>Attenuation (dB/line)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.000E+06</td>
<td>5.00E+02</td>
<td>0.00E+03</td>
<td>0.00E+01</td>
<td>0.00E+01</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>1.000E+07</td>
<td>5.05E+03</td>
<td>0.00E+03</td>
<td>0.00E+02</td>
<td>0.00E+02</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
<tr>
<td>2.000E+07</td>
<td>5.04E+04</td>
<td>0.00E+03</td>
<td>0.00E+03</td>
<td>0.00E+03</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>

Near and far end crosstalk values

Differential structures include graphs and data for 4 port and mixed mode s-parameters.

Choose from the Graph Settings Display Series drop-down list to choose results, including those for loss, impedance, inductance, resistance, capacitance, conductance, skin depth and attenuation and effective Er.
**Viewing detailed data point information**

Click a data point on any of the data series to expand into detailed picked data information.

**Frequency-dependent calculation interface**

Use the Frequency-dependent calculation interface to enter or modify parameter values used in frequency-dependent calculations.

- **Frequency Distribution** options allow choosing between logarithmic or linear graphing.
- **Result Presentation** options specify the vertical chart axis; choose between dB/line length, dB/inch or dB/m.
- **Extended Substrate Data** options are used to specify parameters by frequency range.

**Frequency independent modelling**

Choose **Constant Er / TanD** to employ fixed Er and TanD values. *Note that modelling complex dielectric permittivity and loss tangent as fixed (i.e. frequency-independent) values leads to non-causal results.*
Frequency dependent modelling

Using frequency independent permittivity is a source of non-causal time domain responses so causal interpolation of dielectric constant is implemented in the Si9000e via the Causally Extrapolate Er / TanD option from the Extended Substrate Data option group; this applies Svensson-Djordjevic modelling to each dielectric layer in the current controlled impedance structure.

Causally extrapolating substrate data

The Svensson-Djordjevic model is a physically correct model of dielectric loss in the frequency domain that is well-behaved after transformation to the time domain. It works best when a single frequency is nominated for Er and the Svensson-Djordjevic interpolation calculates the appropriate Er vs frequency.

Choose the Causally Extrapolate Er / TanD option from the Extended Substrate Data option group to apply Svensson-Djordjevic modelling to each dielectric layer in the current controlled impedance structure.

Click Edit… the Causally Extrapolate Substrate Data dialog is displayed.

Each substrate in the controlled impedance structure may be assigned causal extrapolation reference points.

Set the causal extrapolation reference points, frequency, Er and TanD for each substrate then set the frequency range and number of steps and click Calculate.
Select Dielectric Constant or Loss Tangent for display.

Click a data point on the graph to display the value at the frequency of interest in the Picked Data Point box. Results may be displayed in graphical or tabular format.

<table>
<thead>
<tr>
<th>Frequency</th>
<th>Dielectric Constant</th>
<th>Loss Tangent</th>
<th>Dielectric Constant</th>
<th>Loss Tangent</th>
<th>Dielectric Constant</th>
<th>Loss Tangent</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hz</td>
<td>Er: H1</td>
<td>TanD: H1</td>
<td>Er: H2</td>
<td>TanD: H2</td>
<td>Er: CEr</td>
<td>TanD: CEr</td>
</tr>
<tr>
<td>5.00E+09</td>
<td>4.437061</td>
<td>0.019324</td>
<td>4.226143</td>
<td>0.019324</td>
<td>4.034419</td>
<td>0.019324</td>
</tr>
<tr>
<td>1.00E+09</td>
<td>4.400000</td>
<td>0.019500</td>
<td>4.200000</td>
<td>0.019500</td>
<td>4.000000</td>
<td>0.019500</td>
</tr>
<tr>
<td>1.50E+09</td>
<td>4.378513</td>
<td>0.019598</td>
<td>4.178513</td>
<td>0.019598</td>
<td>3.978506</td>
<td>0.019598</td>
</tr>
<tr>
<td>2.00E+09</td>
<td>4.362139</td>
<td>0.019689</td>
<td>4.163860</td>
<td>0.019689</td>
<td>3.963689</td>
<td>0.019689</td>
</tr>
<tr>
<td>2.50E+09</td>
<td>4.349550</td>
<td>0.019724</td>
<td>4.152225</td>
<td>0.019724</td>
<td>3.951550</td>
<td>0.019724</td>
</tr>
<tr>
<td>3.00E+09</td>
<td>4.339882</td>
<td>0.019770</td>
<td>4.142719</td>
<td>0.019770</td>
<td>3.945447</td>
<td>0.019770</td>
</tr>
<tr>
<td>3.50E+09</td>
<td>4.331572</td>
<td>0.019800</td>
<td>4.134632</td>
<td>0.019800</td>
<td>3.937252</td>
<td>0.019800</td>
</tr>
<tr>
<td>4.00E+09</td>
<td>4.324279</td>
<td>0.019841</td>
<td>4.127730</td>
<td>0.019841</td>
<td>3.931162</td>
<td>0.019841</td>
</tr>
<tr>
<td>4.50E+09</td>
<td>4.317844</td>
<td>0.019871</td>
<td>4.121575</td>
<td>0.019871</td>
<td>3.925313</td>
<td>0.019871</td>
</tr>
<tr>
<td>5.00E+09</td>
<td>4.312089</td>
<td>0.019898</td>
<td>4.116085</td>
<td>0.019898</td>
<td>3.920381</td>
<td>0.019898</td>
</tr>
<tr>
<td>5.50E+09</td>
<td>4.306883</td>
<td>0.019922</td>
<td>4.111115</td>
<td>0.019922</td>
<td>3.915348</td>
<td>0.019922</td>
</tr>
</tbody>
</table>

Using frequency independent capacitance modelling

To illustrate frequency independent modelling, select Constant Er / TanD from the Extended Substrate Data option group. Choose a simple Surface Microstrip structure (i.e. a single substrate region.) On the Frequency Dependent Calculation tab specify 100 frequency steps.

From the drop-down choose (for this example) Capacitance and click Calculate – the graph of capacitance is shown below.
Using frequency independent permittivity, however, is a source of non-causal time domain responses.

Causal interpolation of dielectric constant is implemented in the Si9000e lossy line field solver by employing the Extended Substrate Data options.

_Causally extrapolating substrate data_

For this example, choose the Causally Extrapolate Er / TanD option from the Extended Substrate Data option group to apply Svensson-Djordjevic modelling to each dielectric layer in the current controlled impedance structure.

The example below illustrates a controlled impedance structure with two dielectrics.

The Causally Extrapolate Substrate Data dialog is displayed with an entry for each dielectric. Use the dialog to set substrate causal extrapolation reference points: values for frequency, Er and TanD.

<table>
<thead>
<tr>
<th>Substrate 1 Height</th>
<th>H1</th>
<th>1.00*10⁹</th>
<th>4.2000</th>
<th>0.0195</th>
</tr>
</thead>
<tbody>
<tr>
<td>Substrate 2 Height</td>
<td>H2</td>
<td>1.00*10⁹</td>
<td>4.2000</td>
<td>0.0195</td>
</tr>
</tbody>
</table>

Set the frequency range and number of steps (or frequency increments.)

<table>
<thead>
<tr>
<th>Frequency Minimum (MHz)</th>
<th>FMin</th>
<th>500000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency Maximum (GHz)</td>
<td>FMax</td>
<td>10000</td>
</tr>
<tr>
<td>Frequency Steps</td>
<td>FSteps</td>
<td>20</td>
</tr>
</tbody>
</table>

Click Calculate: the Si9000e charts Dielectric Constant (Er) v frequency for each dielectric.
From the Graph Settings dropdown select Loss Tangent (TanD) to display the change in TanD over the selected frequency range.

Click a data point on the graph data series to display the value v frequency of the selected point.
Using extended substrate tables

Si9000e frequency-dependent calculations can be refined using extended substrate data.

The Si9000e contains an Extended Substrate Data Library that allows the user to enter tables of Freq vs Dielectric Constant (Er) and Loss Tangent (TanD). These tables can then be associated with each substrate region for a given structure and are used during the frequency dependent insertion loss calculations.

Si9000e provides the option to import / export individual material tables or the complete library.

Users can assign substrate values by frequency band to accommodate material from manufacturers who specify parameters that vary by frequency. Manufacturers may specify, for example, differing values of Er across a range of frequencies, Er = 4.2 for frequencies up to 100MHz, Er = 4.15 from 100MHz up to 1GHz, Er = 4.1 from 1GHz to 10Ghz, etc.

Multiple Er / TanD option

Using the Multiple Er / TanD option the Si9000 can accept tables of multiple values of dielectric constant and loss tangent or use a single value to enable Svensson-Djordjevic frequency dependent permittivity modelling.

When a single value table is used it employs the same modelling technique as implemented with the Causally Extrapolate Er / TanD option.

Creating a single entry table

To use the Svensson-Djordjevic method and enforce causal modelling choose the Multiple Er / TanD option and click Edit to display the Extended Substrate Data dialog and add a single entry table. Click the Add New Table button, supply a descriptive name and add a single entry for Frequency, Er and TanD as shown in the example below. An entry at 1GHz is recommended.
Choosing the table

With the table defined, specify the table in the Substrate 1 Height drop-down.

Close the dialog and Calculate – the Si9000 implements causal modelling using the Svensson-Djordjevic dielectric loss model.

For this example, choose Capacitance – note the variation in capacitance with frequency. Compare with the frequency independent modelling above.

Note that the Si9000e applies frequency dependent permittivity modelling even though Er and TanD are specified with single values (i.e. as constants.)
Choosing a dielectric layer frequency profile

To choose a dielectric layer frequency profile, click the Edit button in the Extended Substrate Data screen area; the Extended Substrate Data dialog is displayed. A frequency profile table may be specified for each dielectric layer.

For each substrate region select the appropriate extended substrate data table. The number of substrate regions displayed is dependent upon the structure selected.

Click the dropdown listbox arrow to display the list of available tables. For each dielectric layer choose a layer profile. Click Close.

To use the layer profile in frequency-dependent calculations ensure the correct Extended Substrate Data option is ticked.
Adding and modifying extended substrate data tables

The Si9000e allows users to add or modify tables describing the frequency-dependent behaviour of substrate material. In the table below Er decreases with frequency.

Tables may be added and edited as described below or imported and exported in pipe-delimited .ESL format or in comma-separated .CSV format, suitable for editing, for example, in Microsoft Excel®.

Adding a table

Click the Add Table button and choose a descriptive table name and click Add Table; the new table is added to the Extended Substrate Data Library.

Adding data to the table

Click the Add Entry button to add dielectric constant and loss tangent values for the lowest band of frequencies and click the Add Entry button. Repeat for each frequency band.

Each band is added to the table in ascending order of frequency. In this example the dielectric constant, Er decreases with frequency, but Loss Tangent, TanD remains constant.

---

Frequency-dependent calculations (Si9000e only) • 93
Editing and deleting table data

To delete an entry in the table click into the data row and click the Delete Entry button.

To change the data values in a table entry click into the table row and click Edit Entry; modify the values as required and click Edit Entry.

To use the new table, select the table from the dropdown list in the Set Extended Substrate Data Tables section of the dialog.
Importing and exporting material tables

Extended substrate tables can be imported and exported individually or as a complete library as illustrated below.

Use the Import and Export Table controls to read in or export existing tables. Tables can be exported into .EST (pipe delimited) or .CSV format suitable for editing in Microsoft Excel®.

Libraries of tables can be imported or exported as .ESL files or as .CSV files.
Importing individual tables

Click the Import Table control to select a table to be appended to the substrate data library. Navigate to the file location and choose the file type, .EST or .CSV.

The table will be appended to the Library and may be selected via the Extended Substrate Table Name drop-down.

Exporting individual tables

To export a single table, select the table from the Extended Substrate Table Name drop down and click the Export Table control, choose the file format, .EST or .CSV, and SAVE.

If the table is exported to .CSV it may be opened for inspection or editing in a text editor or a spreadsheet such as Microsoft® Excel®.

If necessary, edit the table to reflect parameter changes; the edited table may then be reimported to the Si9000e as described above.

Note: The layout and format of the extended substrate table must be preserved when editing – alterations may prevent a successful subsequent import.
Importing/exporting libraries

The extended substrate tables may be exported as a group, i.e. as a library, for example, to share with other members of a design group or imported from other users.

Exporting the library

To export the whole library of tables, click Export Library. Navigate to a suitable folder; select the file type, .ESL for the Si9000e native library format or .CSV for comma separated text file format, name the file and click Save.

Open the file in a text editor or spreadsheet; a typical library export is illustrated below – each table is shown as a row in the spreadsheet.

Note: If the library table is edited, the layout and format of the table must be preserved when editing – alterations may prevent a successful subsequent import.

Importing a library

Groups of tables may be imported as a library. Click Import Library, choose the file and file type and click Open.

The Si9000e will request confirmation to replace the current library of tables with the new library.

Click Yes to replace the library with the new import.
Viewing the Si9000e data tables

The Si9000e makes a comprehensive range of data for the selected structure available in a convenient tabular form.

Once calculation is complete, in single-ended mode click on the associated tab to view the single-ended data, SPICE RLGC, 2-Port S-Parameter data and measured attenuation and effective Er.

<table>
<thead>
<tr>
<th>Frequency Hz</th>
<th>R Matrix (Ohms/m)</th>
<th>L Matrix (H/m)</th>
<th>G Matrix (S/m)</th>
<th>C Matrix (F/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5000E+06</td>
<td>1.398E+00</td>
<td>8.952E-03</td>
<td>2.127E-04</td>
<td>3.473E+12</td>
</tr>
<tr>
<td>1000E+06</td>
<td>1.955E+00</td>
<td>8.858E-03</td>
<td>4.255E-04</td>
<td>3.473E+12</td>
</tr>
<tr>
<td>1500E+06</td>
<td>2.458E+00</td>
<td>8.858E-03</td>
<td>6.382E-04</td>
<td>3.473E+12</td>
</tr>
<tr>
<td>2000E+06</td>
<td>2.846E+00</td>
<td>8.858E-03</td>
<td>8.518E-04</td>
<td>3.473E+12</td>
</tr>
<tr>
<td>2500E+06</td>
<td>3.185E+00</td>
<td>8.818E-03</td>
<td>1.056E-03</td>
<td>3.473E+12</td>
</tr>
<tr>
<td>3000E+06</td>
<td>3.492E+00</td>
<td>8.808E-03</td>
<td>1.276E-03</td>
<td>3.473E+12</td>
</tr>
</tbody>
</table>

Single-ended mode data

For differential models the Si9000e provides data for odd and even mode, SPICE RLGC, 4 Port and mixed mode S-Parameters along with crosstalk and effective Er.

<table>
<thead>
<tr>
<th>Frequency Hz</th>
<th>Impedance Real (Oms)</th>
<th>Impedance Imaginary (Oms)</th>
<th>Impedance Magnitude (Oms)</th>
<th>Conductance (S/m)</th>
<th>Skin Depth (m)</th>
<th>Attenuation (dB/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5000E+06</td>
<td>5.000E+00</td>
<td>5.000E+00</td>
<td>5.000E+00</td>
<td>2.571E-12</td>
<td>1.290E-04</td>
<td>5.276E-02</td>
</tr>
<tr>
<td>1000E+06</td>
<td>5.063E+00</td>
<td>5.063E+00</td>
<td>5.063E+00</td>
<td>2.571E-12</td>
<td>1.290E-04</td>
<td>5.276E-02</td>
</tr>
<tr>
<td>1500E+06</td>
<td>5.118E+00</td>
<td>5.118E+00</td>
<td>5.118E+00</td>
<td>2.571E-12</td>
<td>1.290E-04</td>
<td>5.276E-02</td>
</tr>
<tr>
<td>2000E+06</td>
<td>5.174E+00</td>
<td>5.174E+00</td>
<td>5.174E+00</td>
<td>2.571E-12</td>
<td>1.290E-04</td>
<td>5.276E-02</td>
</tr>
<tr>
<td>2500E+06</td>
<td>5.230E+00</td>
<td>5.230E+00</td>
<td>5.230E+00</td>
<td>2.571E-12</td>
<td>1.290E-04</td>
<td>5.276E-02</td>
</tr>
<tr>
<td>3000E+06</td>
<td>5.287E+00</td>
<td>5.287E+00</td>
<td>5.287E+00</td>
<td>2.571E-12</td>
<td>1.290E-04</td>
<td>5.276E-02</td>
</tr>
</tbody>
</table>

Differential mode data

98 • Si8000m Controlled Impedance/Si9000e Transmission Line Field Solver
Graphing impedance variation with frequency

Transmission line impedance

Transmission line impedance is broadly constant over a wide range of frequencies – however, impedance on a uniform transmission line is derived from:

\[ Z_0 = \sqrt{\frac{L}{C}} \]

(where \( L \) and \( C \) are inductance and capacitance per unit length of line respectively). Dielectric constant tends to fall slightly with increasing frequency; this example graphs \( Z_0 \) through the frequency range.

Choose the Coated Coplanar Strips with Ground 1B structure.

Frequency-dependent calculations

Using the Extended Substrate Data table Edit function, add a table describing the frequency-dependent behaviour of the substrate material.

For this example, supply the values in the table below. \( \varepsilon_r \) and Loss Tangent are defined for the frequencies of interest; for this material \( \varepsilon_r \) decreases with frequency.
Graphing $Z_0$ against frequency in the Si9000e, it can be seen that as the $\varepsilon_r$ decreases the impedance, $Z_0$, also decreases.

![Coated Coplanar Strips With Lower Gnd 1B](www.polarinstruments.com)

Displaying the table of underlying values (the Single Ended tab) shows that the solver is also solving the inductance.

<table>
<thead>
<tr>
<th>Frequency Hz</th>
<th>Impedance Magnitude Ohms</th>
<th>Inductance $\mu$/line</th>
<th>Resistance $\Omega$/line</th>
<th>Capacitance $\mu$/line</th>
<th>Conductance $\mu$/line</th>
<th>Skin Depth $\mu m$</th>
<th>Conductor Loss $\mu$dB/ln</th>
<th>Dielectric Loss $\mu$dB/ln</th>
<th>Attenuation $\mu$dB/ln</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.00E+06</td>
<td>5.274E+01</td>
<td>7.382E-09</td>
<td>7.503E-02</td>
<td>2.955E-12</td>
<td>1.684E-06</td>
<td>1.164E-03</td>
<td>-6.244E-03</td>
<td>-3.775E-04</td>
<td>-6.622E-03</td>
</tr>
<tr>
<td>5.60E+06</td>
<td>5.116E+01</td>
<td>7.660E-09</td>
<td>7.110E-01</td>
<td>2.927E-12</td>
<td>1.839E-04</td>
<td>1.099E-04</td>
<td>-4.338E-02</td>
<td>-4.086E-02</td>
<td>-8.424E-02</td>
</tr>
<tr>
<td>1.11E+07</td>
<td>5.104E+01</td>
<td>7.618E-09</td>
<td>7.191E-01</td>
<td>2.924E-12</td>
<td>3.671E-04</td>
<td>7.792E-05</td>
<td>6.119E-02</td>
<td>8.133E-02</td>
<td>-1.426E-01</td>
</tr>
<tr>
<td>1.67E+07</td>
<td>5.100E+01</td>
<td>7.593E-09</td>
<td>8.793E-02</td>
<td>2.921E-12</td>
<td>5.535E-04</td>
<td>6.367E-05</td>
<td>-7.495E-02</td>
<td>-1.233E-01</td>
<td>-1.988E-01</td>
</tr>
<tr>
<td>2.22E+07</td>
<td>5.098E+01</td>
<td>7.588E-09</td>
<td>1.04E+00</td>
<td>2.919E-12</td>
<td>7.582E-04</td>
<td>5.516E-05</td>
<td>8.635E-02</td>
<td>-1.673E-01</td>
<td>-2.542E-01</td>
</tr>
<tr>
<td>3.33E+07</td>
<td>5.095E+01</td>
<td>7.54E-09</td>
<td>1.240E+00</td>
<td>2.918E-12</td>
<td>1.148E-03</td>
<td>4.954E-05</td>
<td>-1.057E-01</td>
<td>-2.542E-01</td>
<td>-3.598E-01</td>
</tr>
<tr>
<td>3.89E+07</td>
<td>5.094E+01</td>
<td>7.52E-09</td>
<td>1.338E+00</td>
<td>2.917E-12</td>
<td>1.342E-03</td>
<td>4.172E-05</td>
<td>-1.141E-01</td>
<td>-2.969E-01</td>
<td>-4.109E-01</td>
</tr>
<tr>
<td>4.44E+07</td>
<td>5.093E+01</td>
<td>7.50E-09</td>
<td>1.430E+00</td>
<td>2.917E-12</td>
<td>1.535E-03</td>
<td>3.902E-05</td>
<td>-1.220E-01</td>
<td>-3.396E-01</td>
<td>-4.616E-01</td>
</tr>
<tr>
<td>5.00E+07</td>
<td>5.092E+01</td>
<td>7.46E-09</td>
<td>1.517E+00</td>
<td>2.917E-12</td>
<td>1.730E-03</td>
<td>3.673E-05</td>
<td>-1.239E-01</td>
<td>-3.925E-01</td>
<td>-5.113E-01</td>
</tr>
</tbody>
</table>

The Inductance column shows that at the lower frequency end the inductance is changing as indicated above – and as $\varepsilon_r$ only has a square root effect on the capacitance the inductance has the dominant effect on $Z_0$ at the lower frequencies.

The inductance, L, depends on the skin depth. L decreases with skin depth, which in turn decreases with frequency. Once the skin effect is fully developed the inductance changes are minimal and at the higher frequencies $\varepsilon_r$ will have more predominant effect. However, as the frequency rises the change in $\varepsilon_r$ also slows down. So the $Z_0$ reduction that is observed as the frequency rises to fully develop the skin effect is expected behaviour.
Si9000e S-parameters and Smith charts

The Si9000e allows graphical representation of S-parameters $S_{11}$ and $S_{21}$ via a Smith Chart, a widely used tool for graphical solution of transmission-line networks. The Smith Chart displays reflection coefficient in terms of constant normalised resistance and reactance circles.

S-parameters

A linear network can be characterised by a set of simultaneous equations describing the waves, $b_1$ and $b_2$, exiting from each port in terms of incident waves, $a_1$ and $a_2$,

where S-parameters:

$S_{11} = b_1 / a_1$

$S_{12} = b_1 / a_2$

$S_{21} = b_2 / a_1$

$S_{22} = b_2 / a_2$.

S-parameters are the reflection and transmission coefficients between the incident and reflected waves (i.e. the voltage ratios of the waves) fully describing the behaviour of a device (in this example, a transmission line) under linear conditions at radio frequencies.

S-parameters are complex (i.e. comprising both magnitude and angle) because both the magnitude and phase of the signal are changed by the network.

S-parameters can be graphed in several ways; one option is to use two graphs (magnitude v frequency and phase v frequency) to represent one s-parameter.

Another popular method, described briefly here, is via the use of Smith Charts.
Smith Charts

A Smith Chart is a polar plot with several different scales/axis overlaid onto the graph. This example briefly considers an important scale, implied but not drawn on the Smith Chart, that of reflection coefficient, \( \rho \).

Consider the following graphs of reflection coefficient, \( \rho \). Smith Charts are constructed within the circle described when \( \rho \) is unity.

A point plotted at the origin shows no reflection, i.e. a transmission line perfectly terminated.

Commonly, a point plotted at the left-hand edge shows 100% –ve reflection, i.e. unity reflection with 180 degrees phase change, implying a transmission line terminated with a short circuit.
Similarly, a point plotted at right-hand edge shows 100% +ve reflection, i.e. unity reflection with no phase change, implying a transmission line terminated with an open circuit.

A point plotted anywhere on this circle, a reflection coefficient of unity, shows a perfect reflection at different phase angles.

**Plotting s-parameters on the Si9000e Smith chart**

Consider a 50 Ohm structure

The structure can be represented as a transmission line (terminated in ZL = 50 Ohms) and the s-parameters of the network/black box can be obtained.
Plotting reflection coefficient

\[ S_{11} = \frac{b_1}{a_1} \]

In a perfect system \( Z_0 = Z_L = 50 \text{ ohms} \), the network is exactly terminated, and there is no reflection – the magnitude of \( S_{11} = 0 \).

\( S_{11} \) of this surface microstrip would plot as a dot in the centre of the Smith chart — no reflection.

Plotting transmission coefficient

\[ S_{21} = \frac{b_2}{a_1} \]
In a perfect system there is no loss and the signal passes through the transmission line unattenuated: the magnitude of $b_2 = \text{magnitude of } a_1$ and the magnitude of $S_{21} = 1$.

A single frequency reading of $S_{21}$ of our surface microstrip would plot as a dot somewhere on the outer circle of the Smith chart.

Where this dot is plotted would depend on the phase shift through the transmission line. If the frequency were increased, and other $S_{21}$ frequency readings obtained, the magnitude of $S_{21}$ would still be 1 but the phase shift would change.
Adding more $S_{21}$ readings at increasing frequency

As more $S_{21}$ readings are plotted at increasing frequencies it can be seen that the plotted graph increases in a clockwise direction — typical of a transmission line.

S-parameters can thus be used to completely characterise a network; the values of the s-parameters change with frequency and can be plotted on 2 conventional graphs, magnitude v frequency & phase v frequency or on a Smith Chart.

The Smith Chart portrays reflection and phase shift: the centre of a Smith Chart represents no reflection / transmission, the unity circle of a Smith Chart represents perfect reflection/transmission.
The Si9000e Smith Chart

Plotting the real response $S_{11}$ and $S_{21}$ of the 50 ohm surface microstrip shown above produces the following chart on the Si9000e (in this example only 10 points are plotted). The $S_{21}$ graph starts at the right hand edge at 100Mhz and circles around with the last plot at 15GHz.

The previous graphs plotted an ideal network with no loss. This plot shows this transmission line with a small amount of loss, as the 15GHz point is no longer sitting on the unity outer circle. If the loss increases with frequency the reflection coefficient becomes smaller and the plotted line spirals inwards.
Surface roughness compensation

The Si9000e allows the user optionally to provide compensation for surface roughness in frequency dependent calculations; the Si9000e will chart dielectric losses along with conductor losses and attenuation values that include compensation for surface roughness. Modelling extends into both RLGC and S-parameter data.

Surface roughness effect on PCB trace attenuation / loss

The thermal stability (and hence the reliability) of a PCB structure will relate to the mechanical strength of the bond between dielectric and copper layers. In order to provide good adhesion between copper and dielectric materials in core layers PCB materials vendors control the roughness of the associated copper layers (typically by chemical treatment). Since the roughness is a random quantity it is commonly specified in terms of the rms (root mean square) height \( h \) of the surface unevenness.

The surface roughness of the copper layers will have no effect on current at low frequencies as, at low frequencies, the depth of current penetration will exceed the value of \( h \). At high frequencies, however (i.e. in the GHz region), the skin effect (see below) will be significant as, at high frequencies, most current flows in the outside of the conductor (in a very narrow skin on the conductor – hence the name.)

The skin effect

Skin effect refers to the phenomenon where electromagnetic fields (and hence the current) decay rapidly with depth inside a conductor.

![Diagram showing skin effect](attachment:skin_effect_diagram.png)
The diagram above graphs the amplitude of magnetic field against depth ($z$) into a conductor and shows the variation of the amplitude of magnetic field $H_y$ in the $z$-direction where $H_0$ is the amplitude at the conductor surface. As a consequence of Ampere's Law in a conductor, a conduction current is associated with $H_y$. This current will be perpendicular to $H_y$. Thus there is a conduction current of density $J_x$, (where $J_0$ is the current density at the surface) whose amplitude will vary in the same manner as that for $H_y$. The distance $\delta$ is the value of $z$ at which $|J_x| = J_0/e$. This is also the same value at which the rectangular area $\delta J_0$ in the diagram equals the area under the exponential curve. $\delta$ is known as the Skin Depth.

**Surface roughness**

At very high frequencies (where skin depth $\delta$ is less than $h$, i.e even smaller than the conductor surface roughness) current follows the contours of the surface of the copper, effectively increasing the distance over which current must flow and hence the resistance of the copper. Chemical treatments producing roughness heights of several microns are typical with FR-4 dielectrics resulting in signal attenuation at high frequencies.

Attenuation factor variations with frequency for different roughness values (in µm) are shown as shown in the graph below. From the chart it can be seen that as the surface roughness increases attenuation occurs at lower frequencies; at low values of roughness attenuation is insignificant below 1GHz, at higher values attenuation can begin at frequencies in the low hundreds of MHz.
Conductor losses in PCBs

Losses that need to be considered by the PCB designer/fabricator can be summarised as conductor and dielectric losses. Conductor losses include DC, skin effect and surface roughness losses and the designer will need to balance the trade-off associated with foil roughness and conductor loss with the requirement for robust packaging — the challenge is to optimize conductor loss while ensuring good dielectric/foil adhesion. Designers and fabricators will need to discuss with the PCB vendor the surface treatments and dielectric materials available.

Surface roughness compensation methods

The Si9000e provides several commonly used methods for surface roughness compensation. The frequency dependent tab allows you to choose between:

- Smooth copper, (no compensation for Cu loss at all)
- Hammerstad modelling
- Groisse modelling
- Huray / Cannonball-Huray modelling

The Smooth copper option provides for no compensation for copper loss at all.

Hammerstad modelling is a proven technique that has stood the test of time but has practical limitations when used over 4GHz as the model tends to saturate.

Groisse modeling can, with care, be used to extend the modelling up to 7 to 10 GHz before saturation in the model blunts its accuracy.

Huray modelling extends the roughness modeling validity up to 40 to 50GHz and possibly higher, but is more demanding in terms of input. However, as a rule of thumb if you do not have the detailed SEM measurement information needed for Huray, many OEMS find they can get a good empirical match by feeding the Huray settings with a sphere radius of 0.5um and a number between 45 spheres for the smoothest copper and 85 for the roughest with 60 spheres being the nominal (for surface conditions typical in 2017).

Hammerstad or Groisse modelling

Using the modified Hammerstad or Groisse conductor roughness models the Si9000e allows the user optionally to include the RMS value for surface roughness in frequency dependent calculations and chart dielectric losses along with conductor losses and attenuation values that include compensation for surface roughness.
The Si9000e graph above charts all losses, the dielectric loss and the significant increase in the overall loss due to surface roughness, allowing the materials supplier to isolate the contributions of the different loss mechanisms.

Tick the Use Roughness Compensation check box and click the Hammerstad or Groisse option and Click Edit… to specify the RMS values for trace and plane roughness.

Values for surface roughness (obtainable in consultation with the board manufacturer) are specified in the currently chosen units.

<table>
<thead>
<tr>
<th>Parameter Entry Units</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td>Mile</td>
</tr>
<tr>
<td>Inches</td>
</tr>
<tr>
<td>Micron</td>
</tr>
<tr>
<td>Millimetres</td>
</tr>
</tbody>
</table>

Roughness values may be set for each surface in the structure – the example below includes settings for all the surfaces of a broadside stripline structure.
Values for surface roughness (obtainable in consultation with the board manufacturer) are specified in the currently chosen units. Typical values for RMS roughness could be 0.8 µm (0.03mils) for stripline, 1.6µm (0.06mils) for surface microstrip. The Si9000e assumes losses on all sides of a copper trace.

**Huray modelling**

Click the Huray option to apply Huray modeling and click Edit… The model is based on a non-uniform distribution of stacked copper nodules shapes resembling “snowballs”.

Huray modelling extends the roughness modeling validity up to 40 to 50GHz (and possibly beyond).

Click the Huray option button:

Click the Huray Edit button and specify the parameters for the Huray spheres (snowballs.)
Supply the values in the associated fields and click Apply. If the Huray values are not available, click Enable Cannonball-Huray and supply the Rz values for matte and drum side roughness and click Calculate to populate the Huray fields, then click Apply. Click Calculate to refresh the graph.

Click the Application Note link to access the paper *Practical Modeling of High-speed Channels Based on Data Sheet Input* (Bert Simonovich, LamSim Enterprises Inc.) which includes a description of roughness modelling using the Cannonball-Huray Model.
Using the Si9000e Loss Tangent Goal Seek

Measuring insertion loss yields the total losses of a transmission line, but for some applications it may be found useful to further process that information and deduce the contribution of copper losses and dielectric losses to the overall loss figure.

The Si9000e simplifies the complexity of the process of estimating dielectric loss by allowing the designer to:

- enter the total measured attenuation
- calculate an estimate of copper losses from cross section data
- remove the copper loss from the total attenuation to leave the losses from the substrate alone.

This figure can then be processed to provide a useful estimate of the dielectric loss tangent for the substrate material. This procedure describes the sequence of steps to goal seek for loss tangent.

Surface microstrip example

Select Surface Microstrip 1B structure with the default parameters:

From the Frequency Dependent tab select the Loss Tangent Goal Seek option.
The Loss Tangent Goal Seek dialog is displayed.

**Entering the Total Attenuation**

Under Step 1 enter the Total Attenuation for the point of interest, the frequency and the loss per length of line – dB / LL. The length of line will be the value entered into the Length of Line (LL) parameter on the main frequency dependent tab, so may be 1000 mils for dB/inch or 10mm for dB/cm.
Note: The values for total attenuation can also be loaded from the currently picked data point on the displayed graph.

In the case of the Surface Microstrip 1B structure, default for LL is 1000 mils so dB/inch. In this example the Total Attenuation at 8GHz = –0.8 dB/inch

**Calculating the Conductor and Dielectric Loss**

Under Step 2, Calculate the Conductor and Dielectric Loss. This will take the parameters for the current selected structure (Surface Microstrip 1B) and calculate the conductor loss at 8GHz and then take the calculated conductor loss from the total attenuation entered in Step 1 to calculate the remaining dielectric loss.

In this case the conductor loss is –0.2981 dB/inch so the remaining dielectric loss is –0.5019 dB/inch

**Calculating Loss Tangent**

The Step 3 Calculate Loss Tangent option will allow the Si9000e to calculate the Loss Tangent (TanD) required to achieve a dielectric loss of -0.5019 dB/inch. Using the Goal Seek Parameters to limit the min / max range of TanD the Si9000e will now sweep the range of TanD values until a suitable TanD is calculated to achieve a dielectric loss of –0.5019 dB/inch.

The result in this example is that the Loss Tangent (TanD) of 0.0196 is required to achieve a dielectric loss ~ –0.5019 dB/inch – the convergence value is used to give the target dielectric loss a tolerance.

To verify, update the current structure with the calculated loss tangent result. This will copy the TanD value of 0.0196 into the TanD field on the frequency dependent tab. Close the dialog and click the frequency dependent Calculate button to update the Loss v Frequency graph.
Query the Conductor Loss with Roughness, Dielectric Loss and Attenuation with Roughness curve data points at 8GHz – notice the Attenuation with Roughness (total attenuation) is now ~ 0.8 dB/inch as specified in the Goal Seek dialog.

If the total attenuation value entered in Step 1 of the Loss Tangent Goal Seek option is unachievable the Si9000e displays the message alert below:

The minimum and maximum Loss Tangent values as specified in the Setup Goal Seek Parameters are displayed along with the calculated Total Attenuation.
Speedstack Si to Si9000e data transfer (frequency dependent parameters)

Speedstack and Si9000e incorporate the facility to realise bidirectional transfer of all structure parameters (i.e. both lossless and frequency dependent – including surface roughness parameters) for a single structure or all structures via the clipboard.

Parameter transfer is accomplished via the data transfer icons:

**Single structures**

Use Speedstack’s To Field Solver icon to transfer the parameters of a single structure via the clipboard from Speedstack to the Si9000e.

Use Speedstack’s From Field Solver icon to transfer the parameters of a single structure via the clipboard from Si9000e to Speedstack.

Use the Si9000e’s Paste Structure from Speedstack to paste the whole structure with all its parameters into the Si9000e – the currently displayed structure will be replaced.

With all calculations complete click the Copy Structure to Speedstack to return the structure to the stackup in Speedstack.

**Multiple structures**

Use Speedstack’s To Si Project icon to transfer all structures as a project from Speedstack to the Si9000e.

Use the Si9000e’s Paste from Speedstack into Si Project to paste the set of structures into the Si9000e as a project.

**Sharing structure properties**

Each structure in Speedstack can store a complete set of frequency dependent parameters, so each structure can have its own Length of Line, range of frequencies (FMin, FMax, FSteps and Frequency of interest) substrate data, surface roughness compensation and loss budget.

Using the data transfer icons within Speedstack allows a selected set of structure properties to be shared between other structures on the same electrical layer on the stackup.

To share parameters between structures, select the source structure (structure 1, Edge Coupled Coated Microstrip 1B.)
Select the Frequency Dependent Properties button to display the frequency dependent properties.

All the structure’s properties, including all the frequency dependent parameters, will be available for sharing with the target structure.

Close the dialog and click the To Field Solver button to copy the parameters to the clipboard.

Select the target structure (in this example, structure 2, single ended Coated Microstrip 1B as shown below) and click the From Field Solver button.
Speedstack displays the Paste Structure Properties dialog

Select the properties to be pasted – in this case, the impedance parameters are unchecked as the source structure’s 100 ohm differential impedance does not apply.

The frequency dependent parameters, along with the causal extrapolation reference points (frequency, Er and TanD) and surface roughness compensation method are applied to the target structure.
Transferring structures

Si9000e transmission line field solver is fully integrated with Speedstack Si. Users can transfer structures with all parameters from Speedstack Si to the field solver for processing then transfer the solved properties back to Speedstack Si.

Transferring a single structure

Within Speedstack, select the structure to be copied to the Si9000e.

Click the To Field Solver button to transfer the structure and all parameters to the Si9000e.

Switch to the Si9000e.

Click the Si9000e’s Paste Structure from Speedstack button to paste the structure complete with all impedance and frequency dependent parameters into the Si9000e.

The Si9000e displays the Paste Structure Properties dialog.

Choose which groups of properties are to be pasted into the field solver and click Apply. The impedance, lossless and frequency dependent properties are pasted into the field solver for processing. The units setting in Speedstack will replace the setting in Si9000e.

Solving for impedance

With the structure loaded into the Si9000e switch to the Lossless Calculation tab to display the structure graphic and lossless parameters.
Specify the target impedance then click the Calculate button for the parameter to be used in the goal seek (e.g. trace width); with the target impedance reached switch to the Frequency Dependent Calculation tab.

**Running frequency dependent calculations**

Enter the frequency dependent parameters, the extended substrate data settings, the surface roughness compensation method and values and click Calculate to refresh the results.

With all calculations complete click the Copy Structure to Speedstack to return the structure to the stackup in Speedstack.

The Paste Structure Properties dialog is displayed.
Choose which properties are to be updated and click Apply.
Rebuild and calculate the structure in Speedstack. The structure reflects the updated values.

Transferring multiple structures via Si Projects
To transfer all the structures in a stack use the Si Projects transfer function incorporated in Speedstack Si and Si9000e.
Si Projects allows for transfer of all controlled impedance structures along with all lossless and frequency dependent parameters from Speedstack Si into the Si9000e field solver.
Si Projects allows groups of structures to be saved and recalled in Si9000e and the updated structures pasted back into Speedstack.
The stackup in the example below contains four structures.

Use the To Si Project toolbar icon to copy the group of four structures from Speedstack Si and place them onto the clipboard; these structures can then be pasted directly into the Si9000e as a new project.

Switch to the Si9000e and use the Si9000e’s Paste from Speedstack into Si Project to paste the set of four structures into the Si9000e as a project.

The Si9000e and Speedstack should automatically switch to the units that were in use when the structure was copied. (For instance, if Speedstack is in Mils and Si9000e is in Microns and a structure is copied from Speedstack to Si9000e the Si9000e should automatically switch to Mils.)
The complete set of structures appears in the field solver’s Project window in the same order as shown in Speedstack.

The Si Project window lists the transferred structures in Speedstack’s display order, showing the order number and impedance value along with a thumb nail graphic indicating the structure configuration.

**Modifying structures**

Selecting each structure displays its associated graphic in a grey background.

With a structure selected the structure parameters can be modified as required and all values recalculated. The recalculated structures can be pasted back into Speedstack.

To paste a structure back into Speedstack select the target structure in Speedstack, switch to the Si9000e, select the structure for transfer and use the transfer icons to update the selected structure in Speedstack.

Click the Rebuild and Recalculate Displayed Structure to refresh the displayed structure.

Click the Rebuild and Recalculate All Structures to update all structures in the stack.
Analysing S-parameters with the LA9000

Loss results may be exported to the Polar LA9000 S-parameter Analyser for a single structure or as a suite of results for a project for comparison between the structures within the project.

The LA9000 S-parameter analyser

The LA9000 S-parameter analyser allows graphical comparison of S-parameters from a variety of sources; for example, the LA9000 will also allow you to compare S-parameters exported from Si9000e with 3rd party simulation tool S-parameters and Touchstone™ files from VNAs and other measurement systems.

Exporting a single structure

With the structure selected click the Launch LA9000 Loss Analysis icon. Name and save the data file with the La9 file extension.

The LA9000 is initiated and the loss data file opened and displayed in the LA9000. Choose the S-parameter for display.
Exporting results of a project

The SI9000e “projects” option can automatically export a bundled suite of s-parameter data along with the associated mechanical and electrical properties for the transmission lines being simulated. This can for example allow you to display differences in insertion loss for a project (i.e. a group of structures) which may be geometrically identical and yet deploy different performance base materials.

In this example, the SI Project below comprises six named models of the a stripline structure with increasing degrees of roughness, from 0 – 2.5 microns in 0.5 micron steps.

On the Frequency Dependent Calculation tab calculate each structure to display its associated loss.

To compare the six results, click the Launch LA9000 Loss Analysis icon.

Name and save the exported LA9000 data file with all the structures in the project.

The six models will be recalculated and exported to the LA9000.
Choose the S-parameter for display (in this case S_{12}) – the six project structures are displayed for easy graphical comparison with their associated names in the LA9000.

**Importing/exporting data**

**Importing/exporting data in Touchstone™ format**

The Si9000e includes the capability to import Touchstone™ data so that measured and modelled S-parameter data may be compared.

**Importing Touchstone files**

Import Touchstone files by using the File | Import Touchstone Format menu selection.

Designers are able to import a Touchstone file containing S-parameter data, with options to display just the Touchstone data or combine this data with the current selected structures S-parameter data.

Select the structure from the Si9000e structure bar

Select the File | Import Touchstone Format... command and select the .s2p or .s4p file and click Open.

The data will be displayed as a green dataset.
Select Overlay Calculated S-Parameter Data.

As the imported Touchstone file is likely to encompass a frequency range different from the current structure, if the structure frequency parameters need to be altered to match the Touchstone file the Si9000e offers to change the frequency range and then recalculate.

The current model will display as the red dataset.

To change the structure parameters it will be necessary to close the dialog, alter the parameters and calculate the structure and then return to the Import Touchstone option.
The imported data will be retained for the duration of the Si9000e session so it will not be necessary to import again, the red modelled S-Parameters will update accordingly.

**Exporting Touchstone files**

A Touchstone file may be exported from the Si9000e using the File | Export Touchstone Format option, this makes it possible to compare two sets of modelled data on the same graph.

When overlaying the two sets of data the software will automatically check that the frequency range of the calculated data matches that of the Touchstone data; if this isn’t the case an option will be displayed offering an adjust and recalculate function.

**Importing insertion loss data from Polar Atlas**

The Si9000e can import measurement data directly from the Polar Atlas Transmission Line Test System.

The designer can import insertion loss measurement data ($S_{21}$, $S_{DD21}$) acquired using all the test methodologies supported by Atlas, Delta-L, SPP and SET2DIL, allowing for easy comparison of modelled and measured results.

Data may be imported via the Windows clipboard where Atlas and Si9000e coexist on the same machine or via text files where data are transferred from a separate Atlas system.

Using the modelling capability of the Si9000e it is possible to fine tune the structure parameters based on the reality of measurement data.

For example, a designer is able to adjust the substrate height and trace width / separation geometries, goal seek the loss tangent and then model the effect of surface roughness on the conductor layers.

**Importing Atlas data via a text file**

To import Atlas data click the Atlas import button on the toolbar – choose data from the clipboard or text file.

Once imported, the Measurement Data frame on the Frequency Dependent interface updates, summarising key information about the data imported.
In this example, the data was imported was from a Delta-L test in dB loss per inch.

Measurement data may be optionally included or excluded from the All losses plot via the associated check box.

The imported measurement data may be overlaid onto modelled data for analysis. In the example below, the set of measured attenuation data for the Delta-L test is shown as an additional Measured Attenuation Delta-L data set on the All Losses plot.

Comparing the Attenuation with Roughness curve with the Measured Attenuation indicates the degree of correlation.

By altering the Frequency Minimum / Maximum settings for the structure it is possible to set the extents of the model, to the frequency of interest, in this case, to 15GHz.

It will be necessary to recalculate to reflect the new settings. Click Apply and then click the Calculate button.
The Measurement Data tab shows the imported data in table form.

<table>
<thead>
<tr>
<th>Frequency Hz</th>
<th>Measured Attenuation: dB Loss per 1.00 inches</th>
<th>Effective Er</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.00E+09</td>
<td>0.000</td>
<td>3.312</td>
</tr>
<tr>
<td>1.10E+09</td>
<td>0.000</td>
<td>3.306</td>
</tr>
<tr>
<td>1.20E+09</td>
<td>0.000</td>
<td>3.299</td>
</tr>
<tr>
<td>1.30E+09</td>
<td>0.000</td>
<td>3.299</td>
</tr>
<tr>
<td>1.40E+09</td>
<td>0.000</td>
<td>3.291</td>
</tr>
<tr>
<td>1.50E+09</td>
<td>0.000</td>
<td>3.279</td>
</tr>
<tr>
<td>1.60E+09</td>
<td>0.000</td>
<td>3.277</td>
</tr>
<tr>
<td>1.70E+09</td>
<td>0.000</td>
<td>3.271</td>
</tr>
<tr>
<td>1.80E+09</td>
<td>0.000</td>
<td>3.264</td>
</tr>
<tr>
<td>1.90E+09</td>
<td>0.000</td>
<td>3.252</td>
</tr>
<tr>
<td>2.00E+09</td>
<td>0.000</td>
<td>3.255</td>
</tr>
<tr>
<td>2.10E+09</td>
<td>-0.006</td>
<td>3.254</td>
</tr>
<tr>
<td>2.20E+09</td>
<td>-0.016</td>
<td>3.253</td>
</tr>
<tr>
<td>2.30E+09</td>
<td>-0.026</td>
<td>3.253</td>
</tr>
<tr>
<td>2.40E+09</td>
<td>-0.035</td>
<td>3.252</td>
</tr>
<tr>
<td>2.50E+09</td>
<td>-0.045</td>
<td>3.248</td>
</tr>
</tbody>
</table>

The measurement data table can be exported via the Windows clipboard for further analysis using other tools.

**Measured Attenuation and Measured Effective Er**

Measured Attenuation and Measured Effective Er display series have been adding to the graphing display series options.

Selecting the Measured Attenuation display series allows the measurement data to be plotted without the modelling data.

Notice that a line fit algorithm has been applied to the raw measurement data.
Measured Attenuation

Similarly, selecting the Measured Effective Er display series allows that measurement data to be plotted without the modelling data.

Measured Effective Er

As before, click on a plotted data point to query the Effective Er at a frequency of interest. In this example, at 20 GHz the Effective Er is 3.144

Measurement Data options

Use the Measurement Data Options (below) to:

Clear the previously imported measurement data from the currently selected structure
Auto-adjust the current structure to match the frequency range
Auto-adjust the line length of the imported measurement data.
The new settings will be applied after the dialog box is closed. It will be necessary to click Calculate to update results.

**Modelling Delta-L insertion loss with the Si9000e**

The Si9000e is suitable for modelling insertion loss on a wide range of PCB structures and stackups and the models are accurate boundary element field solved calculations of insertion loss that will correlate with a variety of insertion loss measurement techniques.

Techniques include Delta-L, SPP, SET2DIL and direct VNA measurements – provided the measurements are performed carefully with probes, cables and well-designed test vehicles. It should also be noted that Si9000e models the pure transmission line loss with the via effects fully de-embedded.

The Polar Si9000e can model the s-parameter loss characteristics of a PCB substrate measured with the Delta-L methodology.
**Delta-L measurement technique**

One of the benefits of Delta-L is the technique it uses to remove – SI engineers call this *de-embed* – the effects of the via and test system interconnect, leaving the pure loss of the PCB and its composite materials in the measurement.

The Delta-L measurement technique achieves this by measuring a short and a long transmission line structure and mathematically processing the results; as the short and long line structures both contain almost identical interconnect paths it is possible to "divide out" the interconnect artefacts from the measurement and leave only the losses of the line itself. (Note that the Si9000e is not able to model the intermediate stages of the process – i.e. the loss of the short line and the loss of the long line. The measurements of the short and long line are intermediate steps to gather raw data that need processing before the finished loss result is mathematically derived.)

Using the final result from a Delta-L based measurement system and a correctly configured Si9000e you should be able to establish good correlation between Delta-L and Si9000e.

**Notes**

Note that Delta-L will produce the loss per inch (typically) – this may depend on your system vendor. Si9000e will always produce s-parameters *per line length* so it will be necessary to set the length of line, LL, to 1 inch in the Si9000e to get a correlating result.

Note also that the Si9000e presents attenuation or 4 port s-parameters or mixed mode s-parameters; if you prefer to see the result in dB/inch and s-parameters you should select "Mixed mode" s-parameters as your graph.

(“Mixed mode” is the Si engineers’ terminology for "Differential s-parameters").
If you use 4 port in this situation you may see s-parameters that look very wavy; this is normal – simply select Mixed mode or look at the main attenuation graph instead.

If the Si9000e underestimates the loss it may be worthwhile inspecting the cross section of the measured trace and having the surface roughness estimated. This can be factored into the model with the surface roughness capability in the Si9000e.
Using the Si Excel Interface

Note: Si8000m/Si9000e is compatible with Microsoft™ Excel™ 2003 or later (32-bit only)

The graphics displayed in this section are based on Microsoft™ Excel™ 2013. Dialog box graphics from different versions of Microsoft™ Excel™ may display slight differences from those shown here.

The Field Solver functions for the Si8000m/Si9000e controlled impedance structures are built into the Microsoft Excel workbooks Si8000.xls and Si8000Expert.xls as user-defined functions. This allows rapid and convenient analysis of board trace characteristics such as impedance, propagation delay, inductance and capacitance against several varying board parameters.

In addition to the Field Solver functions, the Si8000.xls workbook includes a selection of the most popular pre-built sample data worksheets incorporating tables of functions and their associated parameters. Structure models not included can be built as required as described later in this section.

If the Si8000.xls workbook opens with the warning that the workbook contains macros (Visual Basic code), click the Developer tab of the ribbon and then the Macro Security command in the Code section to display the Trust Center Macro Settings to allow the field solver to perform calculations.

(See the discussion on security levels in Excel’s help.)

The workbook opens by default as read-only; this allows the operator to perform calculations but not save changes to the workbook.

The Si8000Expert.xls workbook includes the controlled impedance functions but not the sample worksheets.

Double click the Field Solver icon on the desktop; Microsoft Excel opens the Si8000.xls workbook at the index sheet.
Controlled impedance structure categories

The index sheet displays the structure categories:

- Single ended structures
- Differential structures
- Differential without ground
- Surface coplanar
- Coated coplanar
- Embedded coplanar
- Offset coplanar
- Differential surface coplanar
- Differential coated coplanar
- Differential embedded coplanar
- Differential offset coplanar

Each group of structures contains a selection of the associated models.

To select a structure, scroll to the category and click on its graphic, e.g., Surface Microstrip 1B. Excel activates the associated worksheet. Structure models not included in the workbook can be built as required as described later.
Each worksheet comprises the graphic associated with the chosen model, a table with predefined values and an embedded chart that uses the table as its data source (typically set to chart impedance against substrate height).

The chart source data can be redefined to show results for other columns.

![Surface Microstrip sample worksheet](image)

**Moving through the structure sheets**

Structure sheets may also be selected via the Tab Scrolling Buttons,

![Tab Scrolling Buttons](image)

Click the buttons to select the first, previous, next or last structure sheets.

Alternatively, use the Ctrl + Page Up/Ctrl + Page Down keys to move to the previous/next sheet.

To move directly to a structure, right click the Tab Scrolling Buttons to display the list of structure sheets.
Select the structure from the list. Scroll through the list to display all supplied structures.

**Calculating trace characteristics**

Each worksheet includes a pre-built sample application, incorporating a table of typical dimensions for use with the function associated with the structure and a chart displaying the change in impedance ($Z_0$), propagation delay ($D$), inductance ($L$), capacitance ($C$) or effective Er (EER) against structure dimensions (in the sample chart below $Z_0$ is shown against a varying Substrate Height ($H_1$) with other parameters fixed).

<table>
<thead>
<tr>
<th>$H_1$</th>
<th>$Er_1$</th>
<th>$W_1$</th>
<th>$W_2$</th>
<th>$T_1$</th>
<th>Calc Type</th>
<th>$Z_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.5</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>75.2</td>
</tr>
<tr>
<td>9.0</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td></td>
</tr>
<tr>
<td>9.5</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td></td>
</tr>
<tr>
<td>10.0</td>
<td>4.2</td>
<td>7</td>
<td>5</td>
<td>1.2</td>
<td>$Z_0$</td>
<td></td>
</tr>
<tr>
<td>10.5</td>
<td>4.2</td>
<td>7</td>
<td>5</td>
<td>1.2</td>
<td>$Z_0$</td>
<td></td>
</tr>
<tr>
<td>11.0</td>
<td>4.2</td>
<td>7</td>
<td>5</td>
<td>1.2</td>
<td>$Z_0$</td>
<td></td>
</tr>
<tr>
<td>11.5</td>
<td>4.2</td>
<td>7</td>
<td>5</td>
<td>1.2</td>
<td>$Z_0$</td>
<td></td>
</tr>
<tr>
<td>12.0</td>
<td>4.2</td>
<td>7</td>
<td>5</td>
<td>1.2</td>
<td>$Z_0$</td>
<td></td>
</tr>
<tr>
<td>12.5</td>
<td>4.2</td>
<td>7</td>
<td>5</td>
<td>1.2</td>
<td>$Z_0$</td>
<td></td>
</tr>
<tr>
<td>13.0</td>
<td>4.2</td>
<td>7</td>
<td>5</td>
<td>1.2</td>
<td>$Z_0$</td>
<td></td>
</tr>
<tr>
<td>13.5</td>
<td>4.2</td>
<td>7</td>
<td>5</td>
<td>1.2</td>
<td>$Z_0$</td>
<td></td>
</tr>
<tr>
<td>14.0</td>
<td>4.2</td>
<td>7</td>
<td>5</td>
<td>1.2</td>
<td>$Z_0$</td>
<td></td>
</tr>
</tbody>
</table>

Sample table with increasing values of $H$

The sheet opens with the single value of $Z_0$ calculated for the structure dimensions shown in the first row. The field solving function is located in the cell labelled $Z_0$, the
parameters for the function are contained in the associated cells labelled H1, Er1, W1, W2, T1, etc.

**Choosing the calculation type**

To calculate other characteristics for the selected parameters, enter the value D, L, C or EER in the associated cell in the Calculation Type column (labelled **Calc Type**), move to another cell and press the Calculate button. Re-label the results column if necessary. To see which characteristics are available for a structure, move the mouse over the Calc Type label to display the Note text box.

- **Single ended calculation types**

  Differential structures include other characteristics, e.g. Zeven, Zodd, Zcommon.

- **Differential calculation types**

  Enter the characteristic type in the Calc Type cells (e.g. Zeven, Zcommon, etc.) exactly as shown in the note.

**Charting against varying board parameters**

The structure sheet opens with the value of Zo against H1 for the structure dimensions shown in the first row and charted as shown below.
To chart the change in $Z_0$ (or $D$, $L$, $C$ or EER) as the height, $H_1$, changes over a range of values, use the Excel Fill Handle to copy the function formula down into the associated cells.

(To activate the Fill Handle, move the mouse to the lower right corner of the active cell. The mouse changes to a black plus sign. If the Fill Handle does not appear, select the File tab then Options | Advanced | Editing Options and tick the Enable fill handle and cell drag and drop check box.)

Use Excel’s Fill Handle to copy the formula down

Press the **Calculate** button to recalculate the worksheet. (The Si8000.xls workbook sets Excel’s Calculation mode to Manual; see the File tab, choose Options | Formulas | Calculation options.) Excel solves for the selected characteristic in all associated rows.
Using the Si Excel Interface

- The embedded chart is refreshed with the results of the calculation.

<table>
<thead>
<tr>
<th>H1</th>
<th>Er1</th>
<th>W1</th>
<th>W2</th>
<th>T1</th>
<th>Calc Type</th>
<th>Zo</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.5</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>75.2</td>
</tr>
<tr>
<td>9.0</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>77.1</td>
</tr>
<tr>
<td>9.5</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>79.0</td>
</tr>
<tr>
<td>10.0</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>80.8</td>
</tr>
<tr>
<td>10.5</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>82.5</td>
</tr>
<tr>
<td>11.0</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>84.1</td>
</tr>
<tr>
<td>11.5</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>85.7</td>
</tr>
<tr>
<td>12.0</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>87.1</td>
</tr>
<tr>
<td>12.5</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>88.6</td>
</tr>
<tr>
<td>13.0</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>90.0</td>
</tr>
<tr>
<td>13.5</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>91.3</td>
</tr>
<tr>
<td>14.0</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>92.6</td>
</tr>
<tr>
<td>14.5</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>93.9</td>
</tr>
<tr>
<td>15.0</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>12</td>
<td>Zo</td>
<td>95.1</td>
</tr>
</tbody>
</table>

Choosing other parameters

Z₀, D, L, C and Er can be plotted against any of the function parameters.

For example, to display Z₀ as Er₁ varies, in the example reset H₁ to a single value, e.g. 8.5, and plot Z₀ against changes of Er₁ between 3.8 and 4.35 in 0.05 increments.

Changing the parameters

Select the first value in the Height column and use the Fill Handle to fill down to row 16 with the value 8.

Change the first value in the Er₁ column to 3.8, change the second value to 3.85 then select both cells.

Use the Fill Handle to fill down to row 16; Excel detects the two cell values as an incrementing sequence and fills the column accordingly with values increasing at 0.05 intervals.

Plot of Z₀ as Height (H1) varies
Click the **Calculate** icon to refresh the $Z_0$ column.

<table>
<thead>
<tr>
<th>$H_1$</th>
<th>$E_{r1}$</th>
<th>$W_1$</th>
<th>$W_2$</th>
<th>$T_1$</th>
<th>Calc Type</th>
<th>$Z_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.5</td>
<td>3.8</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>78.4</td>
</tr>
<tr>
<td>8.5</td>
<td>3.85</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>78.0</td>
</tr>
<tr>
<td>8.5</td>
<td>3.9</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>77.6</td>
</tr>
<tr>
<td>8.5</td>
<td>3.95</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>77.1</td>
</tr>
<tr>
<td>8.5</td>
<td>4</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>76.7</td>
</tr>
<tr>
<td>8.5</td>
<td>4.05</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>76.3</td>
</tr>
<tr>
<td>8.5</td>
<td>4.1</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>75.9</td>
</tr>
<tr>
<td>8.5</td>
<td>4.15</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>75.6</td>
</tr>
<tr>
<td>8.5</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>75.2</td>
</tr>
<tr>
<td>8.5</td>
<td>4.25</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>74.8</td>
</tr>
<tr>
<td>8.5</td>
<td>4.3</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>74.4</td>
</tr>
<tr>
<td>8.5</td>
<td>4.35</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>$Z_0$</td>
<td>74.1</td>
</tr>
</tbody>
</table>

$Z_0$ against $E_{r1}$ with other parameters fixed

**Modifying the chart**

It will be necessary to modify the chart to reflect the new scales and Category axis.

Right click the chart area and choose Select Data…

From the Select Data Source dialog box, click the $Z_0$ Series and choose Edit; the Series page shows the source data cell ranges for the chart.
Click the Collapse Dialog button, , and select the new range of values of Zo.

In the Horizontal (Category) Axis Labels pane click Edit and select the range of Er values charted.

Click the button again to restore the dialog box and press OK.

Click the Category Axis Title label and replace the H with Er. Right click the horizontal (Er) axis and format as required. Right click the value (Zo) axis and choose Format Axis… Choose the Scale tab and change the values as necessary for Minimum and Maximum scale values.

The chart should appear as shown below.
Format the chart (color, scales, etc.) as required.
Repeat the procedure for other parameter values.
Using the controlled impedance functions in other workbooks

The controlled impedance functions supplied by the Si8000 workbooks, Si8000.xls or Si8000Expert.xls, are available for use as user defined functions in other workbooks.

Prior to using any of the functions it will be necessary to ensure the Si8000.xls workbook or Si8000Expert.xls is open. In this discussion the worksheet is assumed to refer to the Si8000.xls workbook.

The functions use the board parameters, H1, W1, Er1, etc. as arguments. Parameter values can be derived from existing data in worksheet cells or inserted into the Function Arguments dialog directly.

Begin and save a new workbook. It will be necessary to save the workbook as a macro-enabled workbook.

It is recommended that worksheets are prepared with labels and parameter values (as shown below) prior to inserting controlled impedance functions.

In the example below cells B3 – H3 contain the labels for a Surface Microstrip 1B structure. The parameter values for the Surface Microstrip structure are contained in cells B4 to G4. The Surface Microstrip 1B function will be inserted into cell H4 and reference cells B4 – G4.

Construct the model as show above and click the Insert Function button on the formula bar.

The Insert Function dialog box is displayed.
From the function category dropdown select the User Defined functions to display the structure functions.

If necessary, scroll to the controlled impedance structure functions; click the function associated with the surface microstrip structure (Si8000.xls!SurfaceMicrostrip1B in this example) and click OK: the Function Arguments dialog is displayed.

Using the Function Arguments dialog to enter formulas

Use the Excel Function Arguments dialog to enter function parameters. The Function Arguments dialog creates an edit box for each argument in the function.

Click into each edit box and then into the worksheet cell containing the associated argument in turn (or use the Collapse Dialog button ( ) in the H1 edit box and select cell B4: click the button again. Tab through the other edit boxes and repeat the procedure for each value.) As the function is entered, the Function Arguments dialog displays the value of each of its arguments, the current result of the function, and the current result of the entire formula. When the last value is entered Excel calculates and displays the final result.
Press OK to close the Function Arguments dialog and complete the formula.

$Z_0$ calculated for a single set of values

To calculate $Z_0$ over a range of parameter values, select the data and formula (cells B4 to H4) and use the Fill Handle to copy down as necessary.

Select each column of cells as appropriate and enter the new parameter values.

*Hint: to fill a range of cells with a single value select the range, type the value and press Shift + Enter.*

Press Shift + F9 to recalculate the sheet.

If necessary use the Increase decimal/Decrease Decimal buttons

![Increase/Decrease Decimal buttons]

to select the required number of decimal places.

Format as required.
Z₀ calculated for changing Eᵣ

Charting results
Use the Excel Chart Wizard to chart the results.
Select the area to be charted: in this example the Eᵣ₁ and Z₀ ranges (to select non-adjacent ranges, press Ctrl while dragging the mouse over each range). If necessary, decrease decimal to the appropriate resolution.

From the Insert tab on the ribbon Click the Insert Scatter (X,Y) button

From the scatter type choose Scatter with Straight Lines and Markers
Excel charts the selected data.

Right click the chart and choose Select Data and check the Data Source.
Check the Chart Data Range: in this case the cell references are correct.

From the Design tab of the ribbon, click add Chart element or use the standard chart formats, and add titles for the chart and its axes and (optionally) remove the legend. Format the chart as required.

If necessary, right click the chart and choose Move Chart.
Using Move Chart dialog box to choose where Excel relocates the chart.

To place the chart on a new chart sheet, click New sheet: and type a name for the new chart sheet.

To embed the chart on the worksheet, click Object in:, select a sheet name from the list box, and click OK.

Drag and size the embedded chart as required on the worksheet.

To modify the data series (e.g. line weight, marker style etc.) right click the chart line and choose Format Data Series...

Change the series format as required.
Plotting multiple data series

**Plotting Z₀ for surface and coated microstrip**

*Inserting the first data series*

Supply the data and plot the data series for Surface Microstrip as described earlier.

<table>
<thead>
<tr>
<th>H1</th>
<th>Er1</th>
<th>W1</th>
<th>W2</th>
<th>T1</th>
<th>Calc Type</th>
<th>Zo</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>3.80</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>76.2</td>
</tr>
<tr>
<td>8</td>
<td>3.85</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>75.8</td>
</tr>
<tr>
<td>8</td>
<td>3.90</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>75.4</td>
</tr>
<tr>
<td>8</td>
<td>3.95</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>75.0</td>
</tr>
<tr>
<td>8</td>
<td>4.00</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>74.6</td>
</tr>
<tr>
<td>8</td>
<td>4.05</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>74.2</td>
</tr>
<tr>
<td>8</td>
<td>4.10</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>73.9</td>
</tr>
<tr>
<td>8</td>
<td>4.15</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>73.5</td>
</tr>
<tr>
<td>8</td>
<td>4.20</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>73.1</td>
</tr>
<tr>
<td>8</td>
<td>4.25</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>72.7</td>
</tr>
<tr>
<td>8</td>
<td>4.30</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>72.4</td>
</tr>
<tr>
<td>8</td>
<td>4.35</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>72.0</td>
</tr>
<tr>
<td>8</td>
<td>4.40</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>Z₀</td>
<td>71.7</td>
</tr>
</tbody>
</table>

In this example, plot Z₀ against Er.

To format any chart item, right click the item and change its properties via the short cut menu.

*Adding the second data series*

Supply the data for the Coated Microstrip structure as shown below.

<table>
<thead>
<tr>
<th>H1</th>
<th>Er1</th>
<th>W1</th>
<th>W2</th>
<th>T1</th>
<th>C1</th>
<th>C2</th>
<th>Cer</th>
<th>Calc Type</th>
<th>Zo</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>3.80</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>3.80</td>
<td>Z₀</td>
<td>72.2</td>
</tr>
<tr>
<td>8</td>
<td>3.85</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>3.85</td>
<td>Z₀</td>
<td>71.7</td>
</tr>
<tr>
<td>8</td>
<td>3.90</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>3.90</td>
<td>Z₀</td>
<td>71.3</td>
</tr>
<tr>
<td>8</td>
<td>3.95</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>3.95</td>
<td>Z₀</td>
<td>70.9</td>
</tr>
<tr>
<td>8</td>
<td>4.00</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>4.00</td>
<td>Z₀</td>
<td>70.5</td>
</tr>
<tr>
<td>8</td>
<td>4.05</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>4.05</td>
<td>Z₀</td>
<td>70.2</td>
</tr>
<tr>
<td>8</td>
<td>4.10</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>4.10</td>
<td>Z₀</td>
<td>69.8</td>
</tr>
<tr>
<td>8</td>
<td>4.15</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>4.15</td>
<td>Z₀</td>
<td>69.4</td>
</tr>
<tr>
<td>8</td>
<td>4.20</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>4.20</td>
<td>Z₀</td>
<td>69.0</td>
</tr>
<tr>
<td>8</td>
<td>4.25</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>4.25</td>
<td>Z₀</td>
<td>68.7</td>
</tr>
<tr>
<td>8</td>
<td>4.30</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>4.30</td>
<td>Z₀</td>
<td>68.3</td>
</tr>
<tr>
<td>8</td>
<td>4.35</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>4.35</td>
<td>Z₀</td>
<td>68.0</td>
</tr>
<tr>
<td>8</td>
<td>4.40</td>
<td>7</td>
<td>6</td>
<td>1.2</td>
<td>1</td>
<td>1</td>
<td>4.40</td>
<td>Z₀</td>
<td>67.6</td>
</tr>
</tbody>
</table>

Right click the chart and choose Select Data... from the menu. Click the Add to add another data series. Highlight the Zo column of the Coated Microstrip and add the series to the chart. Click Edit to rename the series if a legend is required.
Add the name Coated Microstrip to the Name text box, click the Z₀ series from the Series list and add the name Surface Microstrip; press OK.

Right click the Y axis and choose Format Axis..., choose the Axis Options bounds and specify a suitable value (in this case 67) as the minimum value.

The chart should appear similar to that shown below.
Plotting $Z_{\text{even}}$ and $Z_{\text{odd}}$ v trace separation

In this example we use the Edge Coupled Offset Stripline structure to examine the effects of decreasing trace separation on even and odd impedance.

Choose the Edge Coupled Offset Stripline structure from the Si8000.xls main index sheet.

Supply the values for H1 (copy the value 8 to all cells in the height column).

Supply the decreasing values for S (7.75 to 0.25 in 0.5 steps).

Change the Calc Type to $Z_{\text{odd}}$.

Change the Formula column heading to Zodd.

Fill down the formula column with the function.

Press the Calculate button to display the results.

Insert a column to the right of the formula column.

Select the formula cells, choose Copy and select the cell to the right of the Zodd label.

Paste the Zodd values into the column.

Change the Calc Type and the label of the formula column to $Z_{\text{even}}$.

Press the Calculate button. Partial results are shown below.

<table>
<thead>
<tr>
<th>H1</th>
<th>Er1</th>
<th>H2</th>
<th>Er2</th>
<th>W1</th>
<th>W2</th>
<th>S1</th>
<th>T1</th>
<th>Calc Type</th>
<th>Zeven</th>
<th>Zodd</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>10</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.3</td>
</tr>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>9.75</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.3</td>
</tr>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>9.5</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.3</td>
</tr>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>9.25</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.3</td>
</tr>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>9</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.3</td>
</tr>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>8.75</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.3</td>
</tr>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>8.5</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.3</td>
</tr>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>8.25</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.3</td>
</tr>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>8</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.3</td>
</tr>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>7.75</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.2</td>
</tr>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>7.5</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.2</td>
</tr>
<tr>
<td>3.00</td>
<td>4.2</td>
<td>3.00</td>
<td>4.2</td>
<td>7</td>
<td>6</td>
<td>7.25</td>
<td>1.2</td>
<td>Zeven</td>
<td>21.4</td>
<td>21.2</td>
</tr>
</tbody>
</table>

The associated chart should show the results for $Z_{\text{even}}$.

Drag the column of Zodd values onto the chart.

Modify the chart so the Source Data Category (X) axis labels, and Series refer to the S1, $Z_{\text{even}}$ and Zodd cell ranges.

Choose a suitable minimum value for the Value (Y) axis.

Format the axes and add text labels as required.

The results are shown below.
Using more complex models

Calculating the effect of etch back

In this example, the effect of PCB trace side-wall slope will be considered. The process includes charting the change in impedance due to variations in dielectric thickness and trace width. Choose the surface microstrip structure.

Begin by entering the parameter values for the surface microstrip structure in cells A2:F2.

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
<th>G</th>
</tr>
</thead>
<tbody>
<tr>
<td>H1</td>
<td>Er1</td>
<td>W1</td>
<td>W2</td>
<td>T1</td>
<td>Calc Type</td>
<td>Zo</td>
</tr>
<tr>
<td>8.0</td>
<td>4.2</td>
<td>7.0</td>
<td>6.0</td>
<td>1.2</td>
<td>73.1</td>
<td></td>
</tr>
</tbody>
</table>

The etch back factor will be a variable so assign W1 a value of 7.00, locate the etch back factor in cell D7 and define W as C2–D7 (i.e. W1 minus etch back factor). Assign a value of 0.3 as etch back factor and insert the Surface Microstrip function into cell F6:

=Si8000.xls!SurfaceMicrostrip(A2,B2,C2,D2,E2,F2)

Press Shift-F9 to calculate.

Calculating the effect of variations in Height

Next, chart the effect of varying height H1 in 0.05 steps. Create references to cells A2–F7 in cells A22–F22, add the surface microstrip function to G22 and change references B22–F22 to mixed as shown below (click into the formula and use the F4 key to change each reference).

= Si8000.xls!SurfaceMicrostrip1B(A22,B$22,C$22,D$22,E$22,F$22)
Copy the formula in G22 up to cell G17 and down to G27 as shown above.

Create a step value of 0.05 in D14, enter the equation \( =A22+D D14 \) in cell A21 and fill it up to A17.

Enter the equation \( =A22-D D14 \) in cell A23 and fill it down to A27.

Use the Auditing Toolbar Trace Precedent and Dependent arrows to check references are as shown above. Press Shift-F9 to recalculate.

Select ranges H1(var) A17:A27 and Z0(H1) G17:G27 and chart; the chart should appear as below.

**Charting trace width error**

Next, chart the effect of varying the trace width with a fixed trace side slope.

Create references to A2:F2 in cells A41:F41.
Enter the formula
=Si8000.xls!SurfaceMicrostrip1B(A$41,B$41,C41,D41,E$41,F$41)
in cell G41 and copy it up to G36 and down to G46 as shown. (Note that C41 and D41 are left as relative references.)

Create a step value of 0.10 in cell D33.

Enter formula =D41-$D$33 in cell C40 and fill up to C36.
Enter formula =C40-$D$33 in cell D40 and fill up to D36.
Enter formula =C42-$D$33 in cell D42 and fill down to D46.
Enter formula =C41+$D$33 in cell C42 and fill down to C46.

Use the auditing arrows to check cell precedents and dependencies.

Recalculate.

Select ranges C36:C46 and G36:G46 and chart.
The trace width error chart should appear as shown below.

![Charting etch back error](chart.png)

Charting etch back error

Finally, chart the effect of etch back error.

Create references to cells A2:F2 in cells A61:F61.
Enter the function
=Si8000.xls!SurfaceMicrostrip1B(A$61,B$61,C$61,D61,E$61,F$61)
in cell G61. (Note the relative reference to cell D61.) Fill up to G56 and down to G66.

Create a step value of 0.10 in cell C53
Enter the formula \(=D61-C53\) in cell D60 and fill up to D56.

Enter the formula \(=D61+C53\) in cell D62 and fill down to D56.

Insert cells E55:E66 and add label

Audit the precedents and dependencies.

Select cell ranges Etchback (E56:E66) and Zo (H56:G66) and chart.

The chart for an etch back error of 0.1 appears below.

Change the etch back factor cell value in cell C53 and recalculate to observe the change in impedance of a different trace side slope.
### Terms used in this manual

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AC</td>
<td>Alternating Current</td>
</tr>
<tr>
<td>CMOS</td>
<td>Complementary Metal Oxide Silicon</td>
</tr>
<tr>
<td>DC</td>
<td>Direct Current</td>
</tr>
<tr>
<td>ECL</td>
<td>Emitter Coupled Logic</td>
</tr>
<tr>
<td>EMI</td>
<td>Electromagnetic Interference</td>
</tr>
<tr>
<td>FR-4</td>
<td>Epoxy Glass Dielectric Material</td>
</tr>
<tr>
<td>TDR</td>
<td>Time domain Reflectometry</td>
</tr>
<tr>
<td>TTL</td>
<td>Transistor-Transistor Logic</td>
</tr>
<tr>
<td>$Z_0$</td>
<td>Characteristic Line Impedance</td>
</tr>
<tr>
<td>$Z_0'$</td>
<td>Characteristic Line Impedance (Loaded)</td>
</tr>
<tr>
<td>$E_r$</td>
<td>Relative Permittivity (homogeneous dielectric materials)</td>
</tr>
<tr>
<td>$E'_r$</td>
<td>Effective Relative Permittivity (non-homogeneous dielectric materials)</td>
</tr>
</tbody>
</table>
References

IPC-2141 – Controlled Impedance Circuit Boards and High-Speed Logic Design, April 1996

Cohn, Seymour B. – Characteristic Impedance of the Shielded-Strip Transmission Line
IRE Trans MTT-2 July 1954 pp52–57

Abramowitz, Milton and Irene A Stegun – Handbook of Mathematical Functions, Dover, New York 1965

Hilberg, Wolfgang – From Approximations to Exact Relations for Characteristic Impedances.
IEE Trans MTT-17 No 5 May 1969 pp259–265


Harrington, Roger F – Field Computation by Moment Methods, Pub: MacMillan 1968


Paris, Federico and Canas, Jose – Boundary Element Method: Fundamentals and Applications
Pub: Oxford University Press 1997

Kobayashi, Masanor – Analysis of the Microstrip and the Electro-Optic Light Modulator

Bogatin, Eric; Justice, Mike; DeRegoo, Todd and Zimmer, Steve – Field Solvers and PCB Stack-
up Analysis: Comparing Measurements and Modelling
IPC Printed Circuit Expo 1998 paper 505–3

Li, Keren and Fujii, Yoichi – Indirect Boundary Element Method Applied to Generalised
Microstrip Analysis with Applications to Side-Proximity Effect in MMICs
IEE Trans MTT-40 No 2 February 1992 pp237–244