Si8000m Multiple Dielectric Controlled Impedance Field Solver/
Si9000e insertion Loss Field Solver

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 207–2504

POLAR INSTRUMENTS LTD

COPYRIGHT

DISCLAIMER

1. Disclaimer of Warranties

2. Limitation of Liability.

TRADEMARKS

Personal Computer Requirements

Computer	IBM PC AT or compatible
Processor	Intel Pentium or compatible – 1GHz or better
Operating system	Microsoft™ Windows 11™ or later
System memory required	2GB recommended
Hard disk space required	100MB (min.)
Video standard	FHD (HD 1080) (1920 x 1080) minimum
Mouse	Microsoft compatible
Software key port	Parallel port/USB port
Spreadsheet	Microsoft™ Excel™ 2016 or later

Contents

Introduction to the Si8000m and Si9000e

Si8000m/Si9000e Field Solvers

Si8000m Controlled Impedance Field Solver

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, Si8000m provides rapid modelling for a wide range of microstrip, stripline and coplanar structures.

Si9000e Insertion Loss Field Solver

The Si9000e Insertion Loss Field Solver incorporates fast and accurate, both 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 include compensation for resin rich areas between traces.

Multiline crosstalk

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.

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. For differential structures, Monte Carlo Analysis supports constant pitch for both Uniform and Normal Distributions

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.

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 copper surface roughness. Surface roughness compensation methods include: Hammerstad, Groisse, Gradient, Huray and Simonovich-Cannonball.

Hammerstad, Groisse, Gradient methods

The Hammerstad, Groisse and Gradient methods use RMS roughness (Rq) values (usually obtainable in consultation with the board manufacturer) as input parameters.

Huray, Simonovich-Cannonball methods

The Huray and Simonovich-Cannonball methods provide for higher data rates and allow for more complex input parameters to be specified. Huray and Simonovich-Cannonball accept Rz, peak to valley height, or Scanning Electron Microscope (SEM) data if available. 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.

Causal Extrapolation of Er / TanD

The Si9000e accepts constant (frequency independent) values for Er and TanD. Using frequency independent permittivity is a source of non-causal time domain responses, so causal extrapolation of dielectric constant and loss tangent is implemented in 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.

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.

Single / multiple frequency loss tangent goal seek

The Si9000e simplifies the complexity of the process of estimating dielectric loss by allowing the designer to enter the total measured attenuation at either a single frequency or multiple frequencies, calculate an estimate of copper losses from cross section data and 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.

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, Z_0 for variations in H_1, Er, etc.)

When calculating differential structures, multiple impedances may be plotted on the 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.

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, conductor loss with roughness, attenuation with roughness,) inductance, capacitance, resistance, conductance, skin depth, Alpha, Beta and measured attenuation and effective Er.

Calculated data and graphing

Single ended structures include calculated data and graphing for 2-port s-parameters.

When differential structures are selected, Si9000e calculates impedance magnitude data and graphing for differential, even and odd modes, along with data and graphing for 4-port and mixed mode s-parameters, near and far end crosstalk (NEXT and FEXT) and coupling coefficient.

Frequencies of Interest

When displaying All Losses (conductor loss, dielectric loss and insertion loss, conductor loss with roughness, attenuation with roughness) against frequency, up to 10 single frequencies of interest users can be defined for display and the results exported to a text file or spreadsheet for analysis. Frequency of interest settings can be applied to the current structure or all structures within a project.

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

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 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.

Integration with CGen Coupon Generator

Controlled impedance structures may be copied directly to CGen Coupon Generator and added to a layer within the coupon stack using CGen's Add Impedance Structure.

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 into the Si9000e. Import data formats include

- Si8000m database
- Atlas SPP, SET2DIL or Delta-L
- Touchstone (S*n*P) S-parameter data

S-parameter graphing options include magnitude, phase and Smith chart. 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.

Export data formats include

- W-Element (.wlc) format
- S-Parameter (.txt) format

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.

Single / Multiple Frequency 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 single or multiple frequency 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.

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.

Data can be exported in Touchstone format in Real / Imaginary, Magnitude / Degrees or dB / Degrees Touchstone formats, with a user-defined number of frequency steps

Si9000e can export Touchstone format files for multiple line lengths in a single step. Line lengths may be specified directly or pasted in from a third party product.

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) cannot 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 25Ω to 120Ω.

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

The roughness of the copper surfaces

Making accurate impedance measurements on fine line PCB traces

PCB controlled impedance measurements commonly take the form of a waveform of impedance over the length of a PCB trace displayed by a TDR such as the Polar CITS880s.

PCB trace line width and associated DC resistance will affect TDR waveform slope. Before the use of fine line PCBs, the low DC resistance of a PCB trace produced little or no effect on a TDR impedance measurement waveform.

However, as trace widths and weight on modern PCBs reduce, the DC resistance of the traces becomes significant, and, combined with high frequency skin effects reducing the effective cross-sectional area of the track, results in an upward slope of the TDR waveforms on fine line traces giving the false impression of impedance rising over the length of the trace. Series DC resistance can be compensated for by adjusting the slope of the waveform by a specified number of ohms/horizontal unit. This cancels out the series resistance leaving the true characteristic impedance displayed. See *TRC Plus Track Resistance Calculator – TDR View – Supplying DC Resistance Compensation to the CITS880s*

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 micro strip 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.

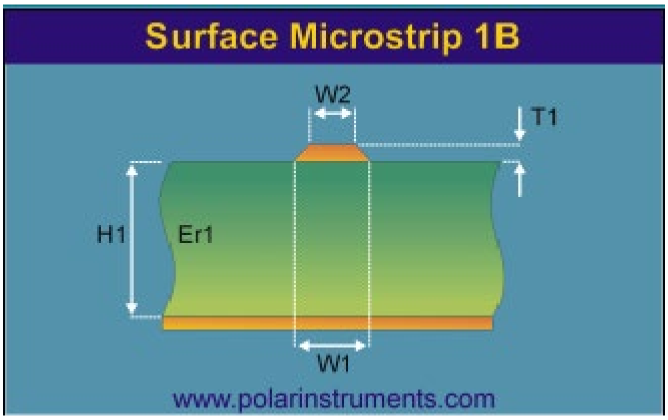
Single-ended Transmission Lines

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_1 refers to the trace width nearest the lower surface.

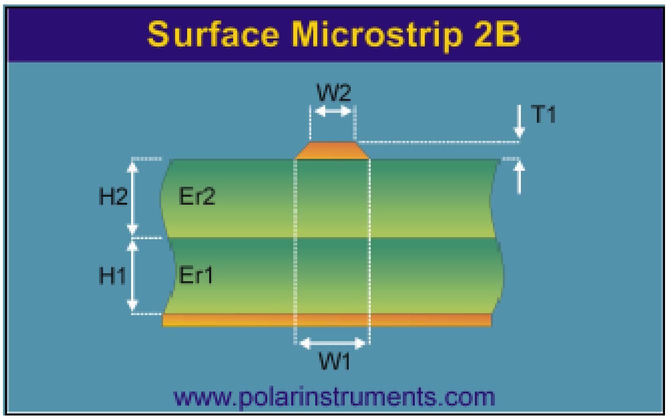
Surface Microstrip

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

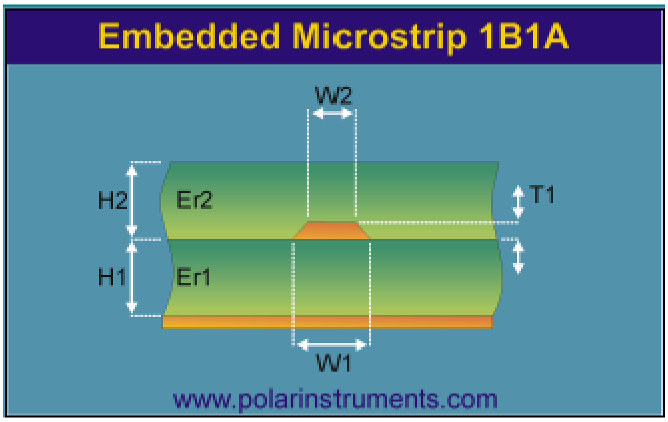
The diagram below shows the surface microstrip structure using two dielectric layers below the trace (designated 2B).



Surface microstrip with two dielectric layers below the trace

Embedded Microstrip

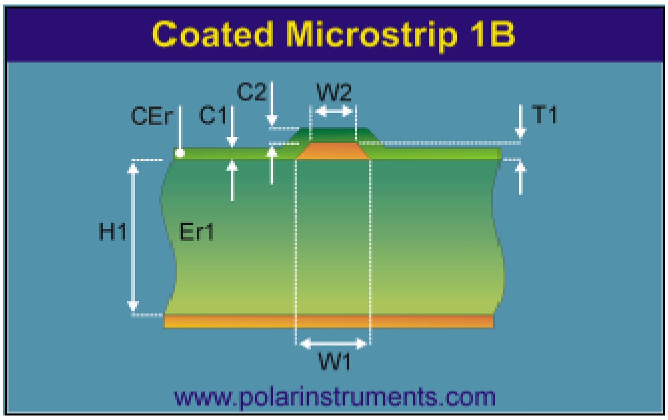
Embedded (or buried) microstrip is similar to surface microstrip, but with the signal line embedded between two dielectrics and located a known distance from the reference plane.



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

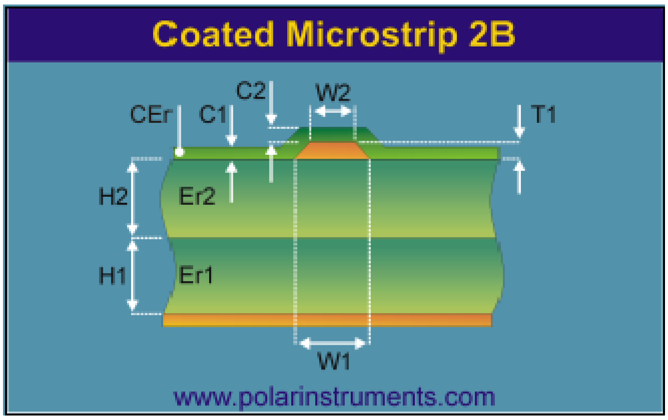
In this structure the two dielectrics are arranged one below and one above the trace (designated 1B1A). Embedding the signal line can lower the impedance by as much as 20% compared to an equivalent surface microstrip construction.

Coated Microstrip



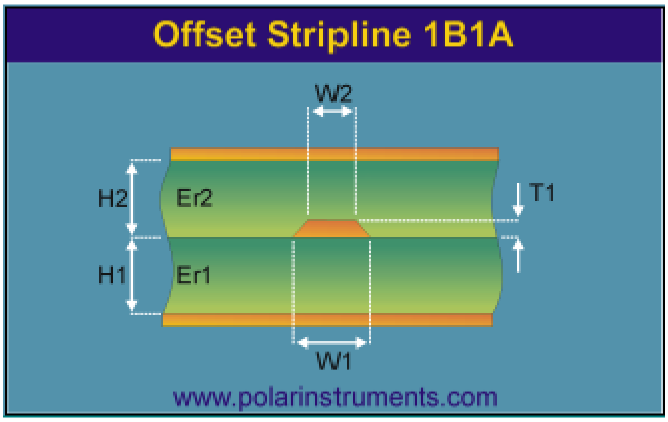
Coated microstrip with single dielectric below the trace

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.



Coated microstrip with two dielectrics below the trace

Offset Stripline



Offset Stripline with signal trace between two planes

In this configuration, also referred to as Dual Stripline, the signal trace is sandwiched between two planes – and may or may not be equally spaced between the two planes

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).

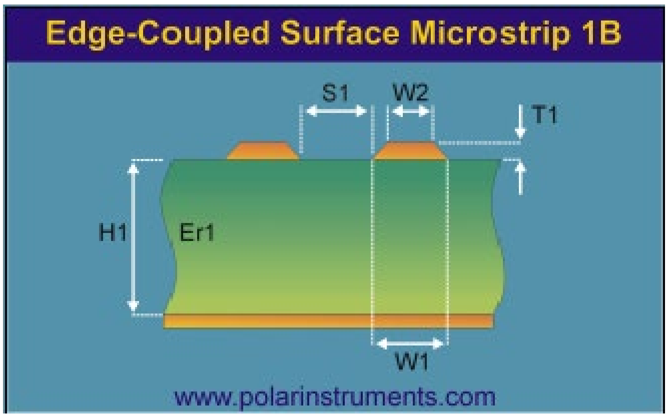
Differential Transmission Lines

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-moded 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.

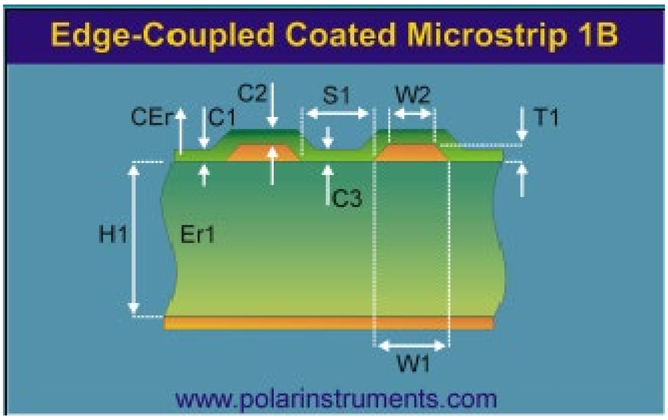
Edge-coupled Surface Microstrip



Edge-coupled surface microstrip with single dielectric below the trace

In this construction the gap between the traces, S1, 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

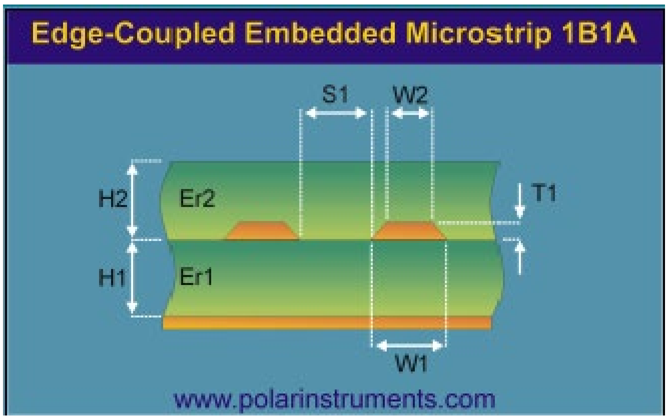


Edge-coupled coated microstrip with single dielectric below the trace

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.

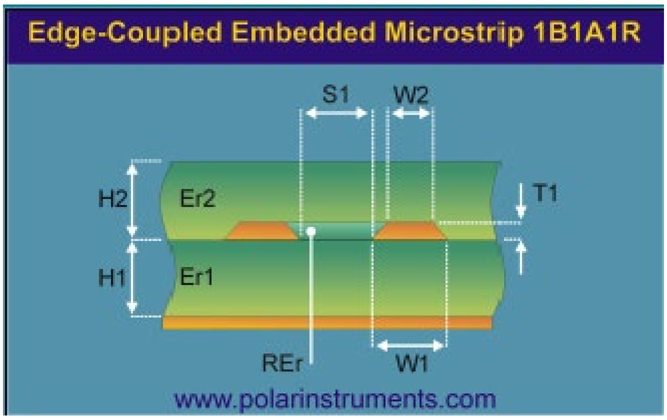
This construction is particularly sensitive to solder mask flooding with LPI (Liquid Photo Imagable) solder mask. This causes the dielectric constant in the edge coupling region to vary, depending on flood depth.

Edge-coupled Embedded Microstrip



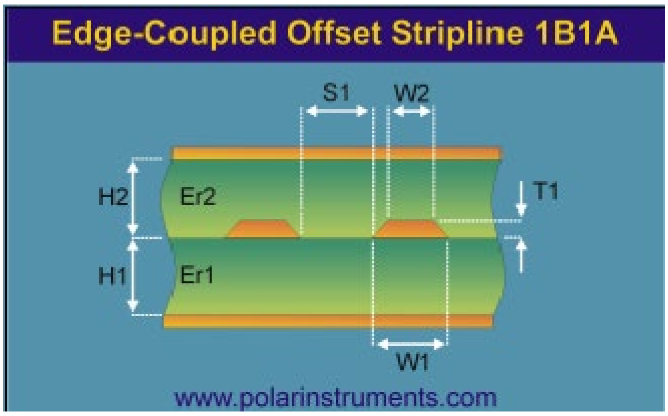
Edge-coupled embedded microstrip with one dielectric below and one above the traces

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

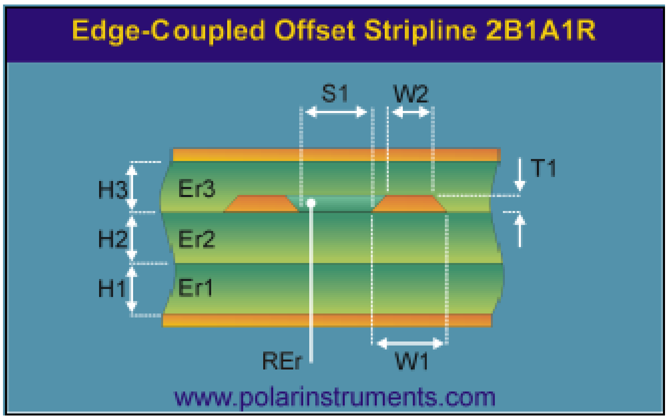
Edge-coupled Offset Stripline



Edge-coupled offset stripline – one dielectric below, one above the traces

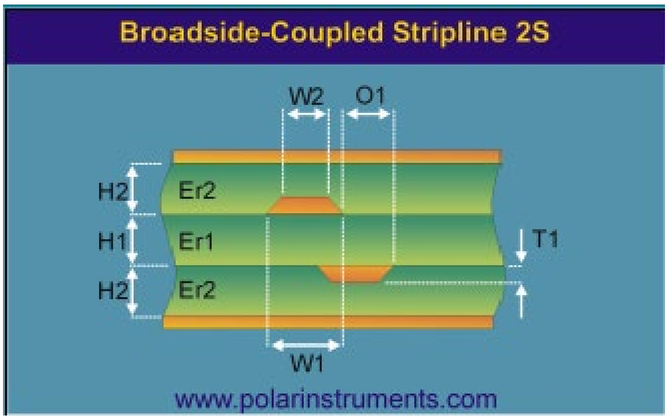
As in the case of the single-ended Offset Stripline construction this structure can be made up as a dual construction with a mirrored edge-coupled differential pair set a distance from the upper reference plane. The lower pair is routed orthogonal to the upper to minimise layer to layer coupling and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

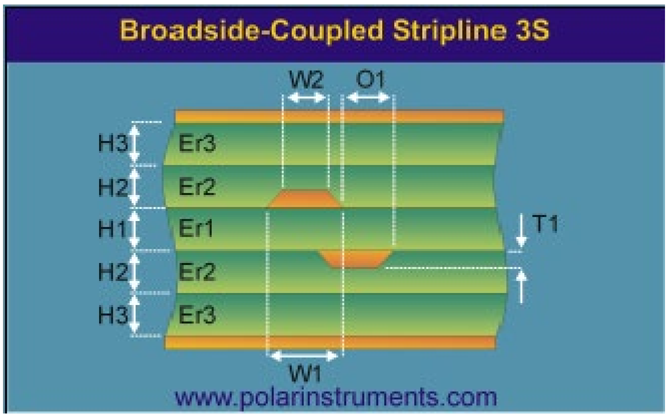


Broadside-coupled offset stripline with two substrate dielectrics, H1, H2

This apparently simple structure 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 misregistration 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, W_2 refers to the trace width nearest the surfaces, W_1 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.

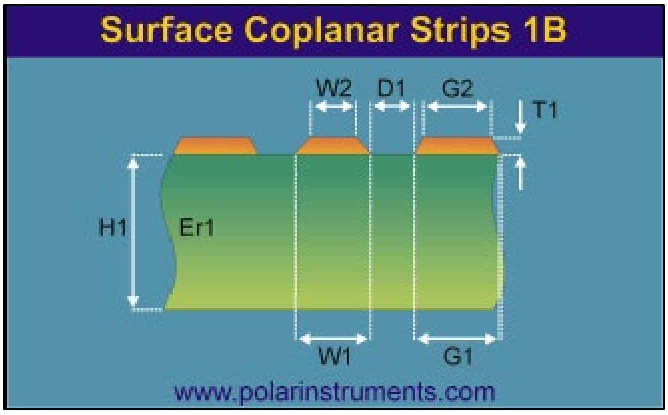
Components can be grounded on the same plane as the signal line; this means the coplanar configuration is ideally suited for surface mounted devices.

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

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

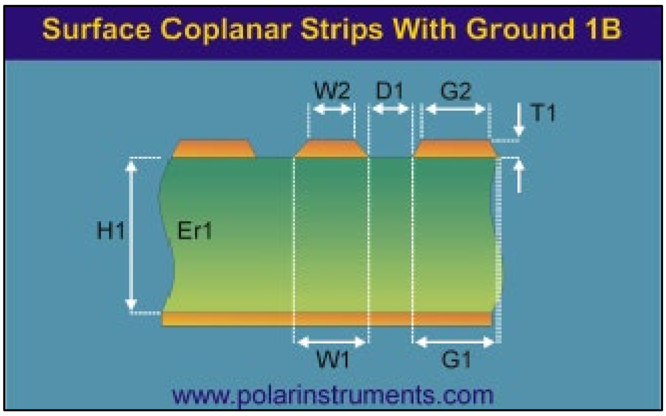
Coplanar lines may be constructed with or without a ground plane underneath the controlled impedance trace(s).

Surface Coplanar Strips



Surface Coplanar Strips without Ground

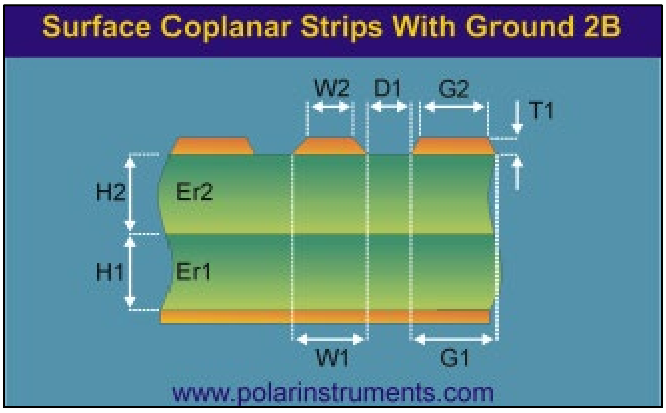
Surface Coplanar Strips with Ground



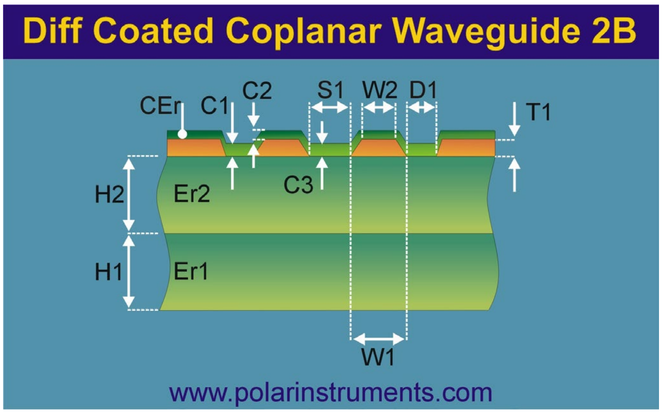
Surface Coplanar Strips with Ground

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



The diagram above shows a differential coated coplanar waveguide 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. Polar software products are based on FlexNet Publisher v11.19.0 or later – requires 32 or 64-bit Windows 11 or later or Windows Server 2016 or later (see Polar Application Note AP605 *System requirements for Polar software products*)

Upgrading from an earlier version of Polar software

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 new license to reactivate.

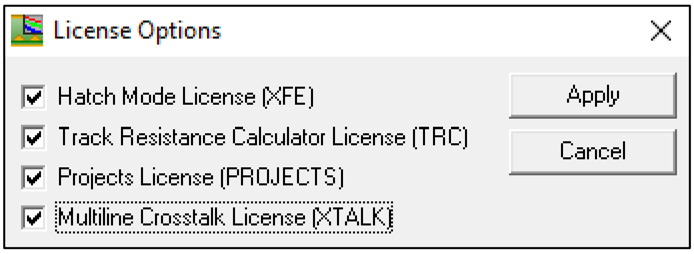
License types

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 using a hardware key (dongle) license 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.

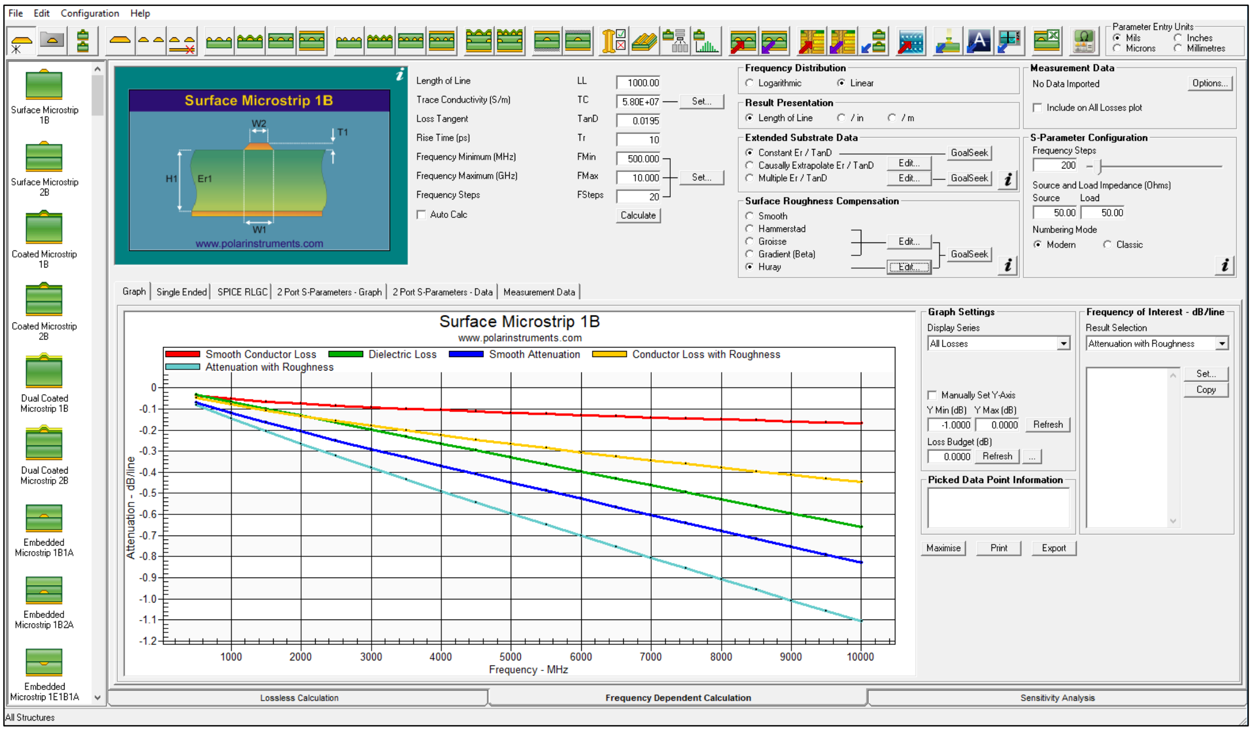


Uninstalling the software

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

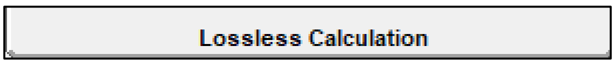
The Quick Solver interface



Si9000e Quick Solver

Startup Mode

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



Lossless Calculation tab

The Lossless Calculation Interface displays the selected structure graphic, structure parameters along with tolerances and calculation/goal-seeking results.

Frequency Dependent Calculation tab

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

Note that it will often be necessary to begin frequency dependent calculations by selecting the Lossless

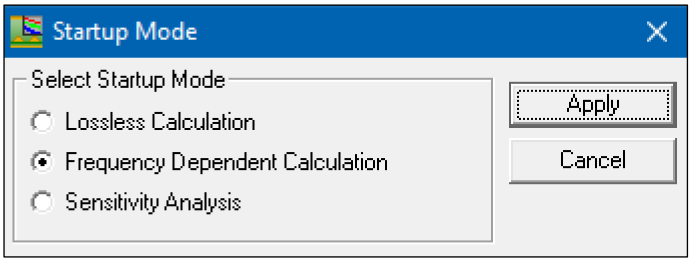
Calculation tab to enter structure parameters. In some cases it may be found convenient to change the Startup mode to Lossless Calculation – see *Specifying the Startup mode* below.

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.

Specifying the 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.

Quick Solver screen areas

The Quick Solver screen 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

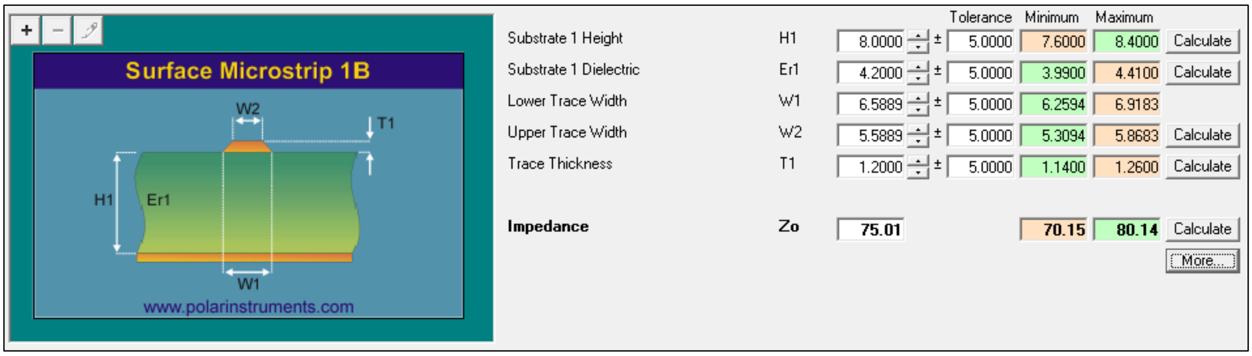
The Toolbar – containing all the commands and structure range select buttons

Structure Bar – displaying available structures within the selected range

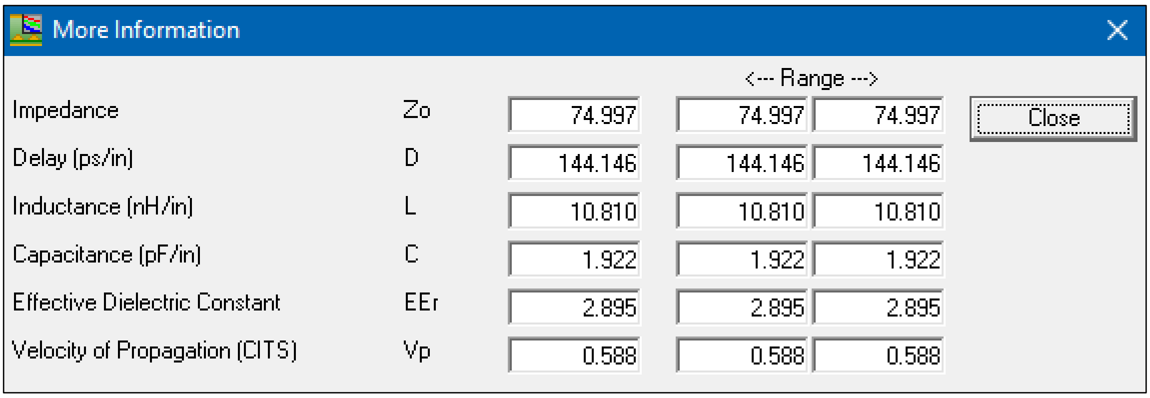
Structure Graphic – displaying the selected structure and associated parameters

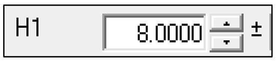
Lossless calculations

The Lossless Calculation Interface displays structure parameters and tolerances and calculation/goal-seeking results. The single ended structure interface is shown below.



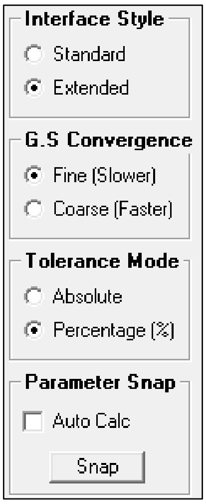
Click More… to display more results – as shown below. Single ended calculations include impedance, delay, inductance, capacitance, effective dielectric constant, and velocity of propagation.





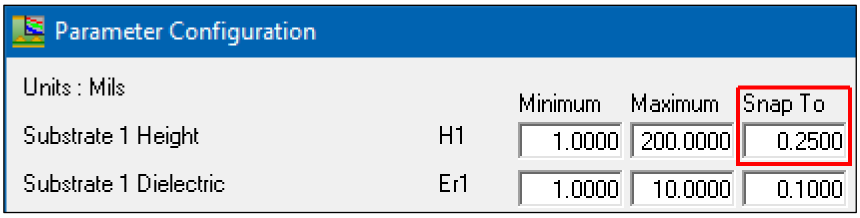
Parameter increment/decrement by snap value

Calculation Options allow the user to select parameter units, standard or extended interface style and goal seeking convergence (see **Field solving for board parameters**.) tolerance mode Absolute or Percentage) and parameter snap / auto calculate.

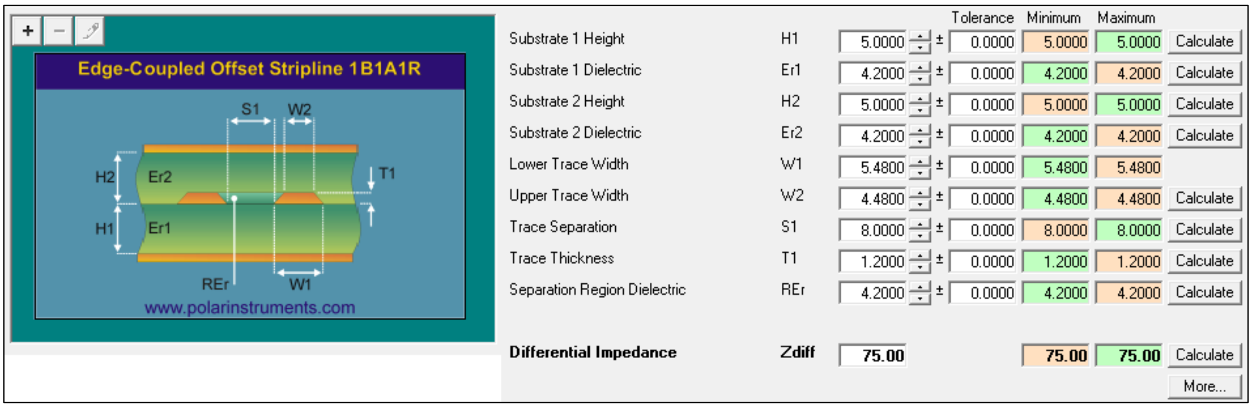


The Parameter Snap settings allow rounding parameter values by the Snap Value. Clicking the increment/decrement buttons adjusts the parameter values by the Snap Value.

The Snap Value for each parameter is set in the Configuration | Parameters dialog.

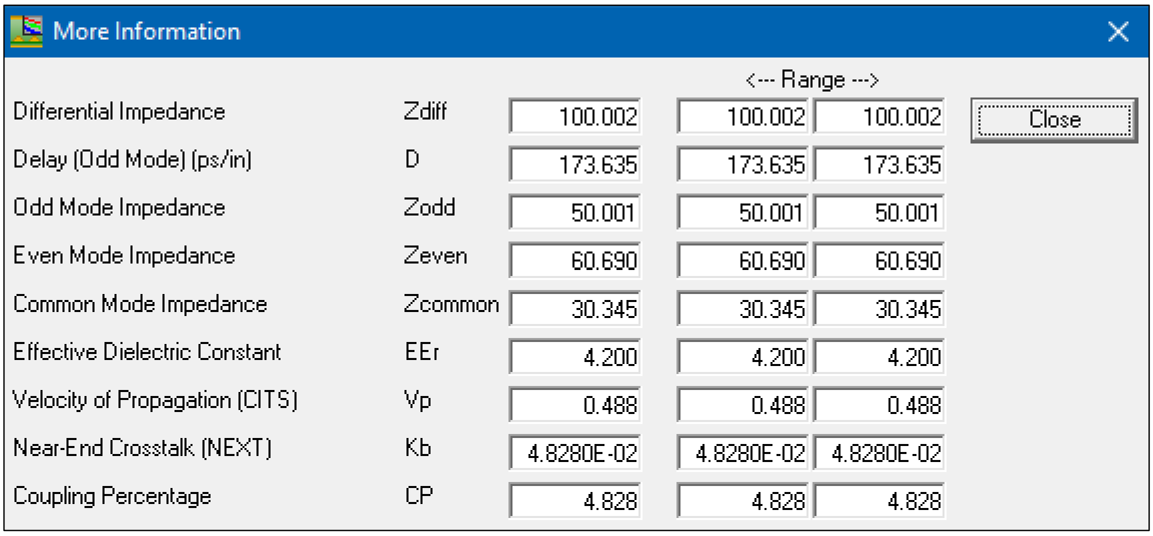


The interface for a differential structure is shown below.



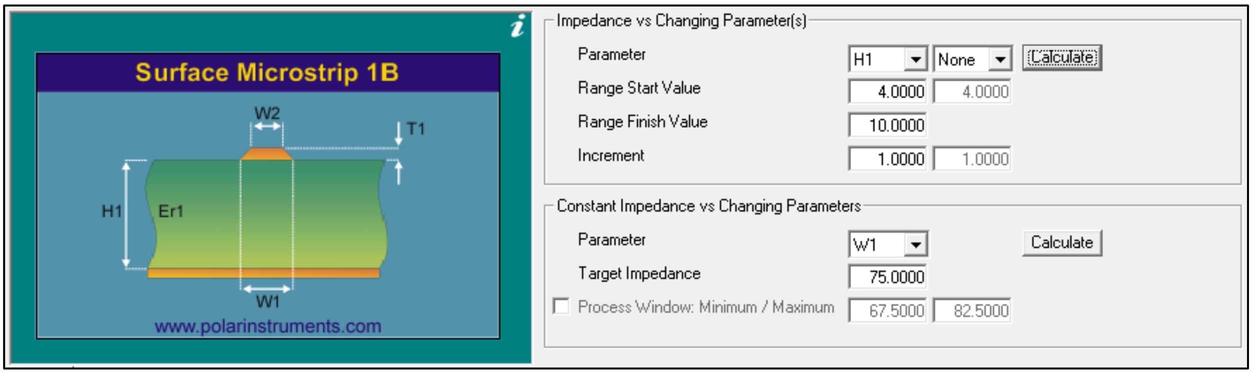
Differential calculations include differential impedance, odd mode and even mode impedance, common impedance, delay, inductance, capacitance, effective dielectric constant and velocity of propagation.

Click More…to view the More Information dialog

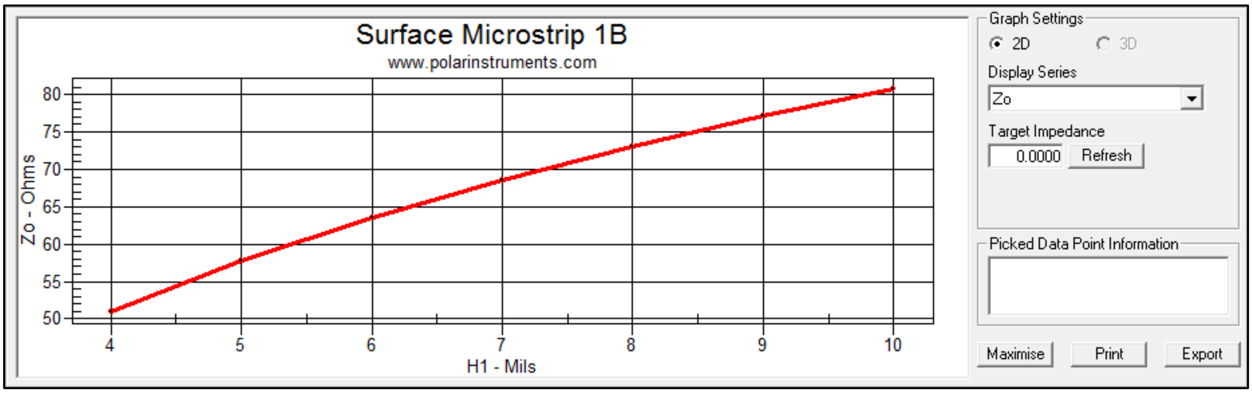


Sensitivity Analysis

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.

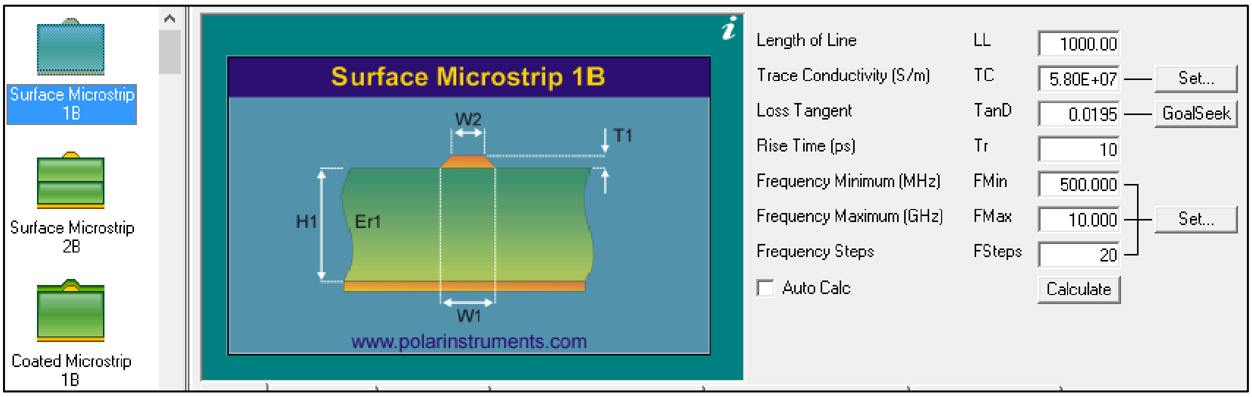
Graph | Results

H1	Er1	W1	W2	T1	Zo	Calc Success
4.0000	4.2000	7.0000	6.0000	1.2000	50.9353	Yes
5.0000	4.2000	7.0000	6.0000	1.2000	57.7437	Yes
6.0000	4.2000	7.0000	6.0000	1.2000	63.5424	Yes
7.0000	4.2000	7.0000	6.0000	1.2000	68.6230	Yes
8.0000	4.2000	7.0000	6.0000	1.2000	73.1059	Yes
9.0000	4.2000	7.0000	6.0000	1.2000	77.1310	Yes
10.0000	4.2000	7.0000	6.0000	1.2000	80.7856	Yes

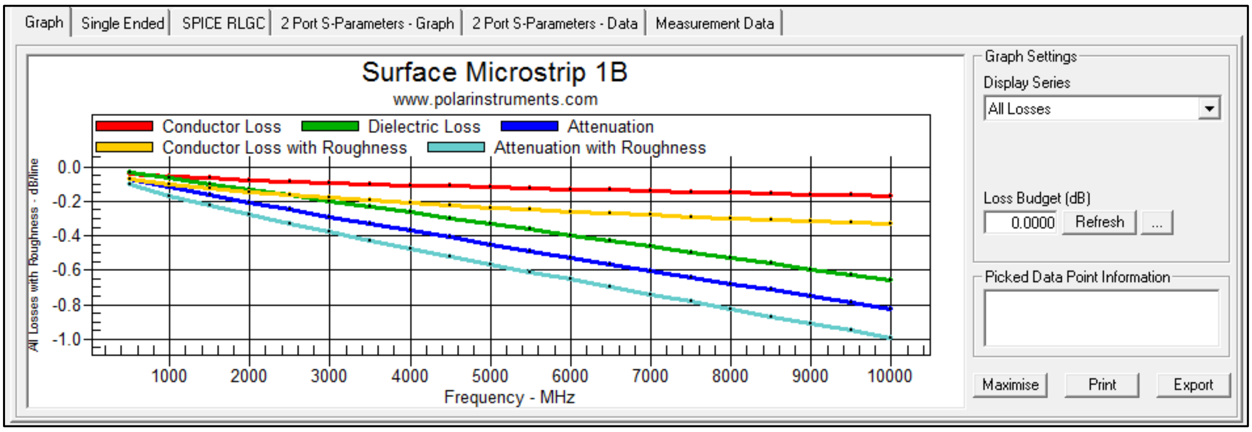
Alternatively, the graph may be exported to JPEG for easy and convenient inclusion in your documentation.

Frequency-dependent calculations (Si9000e only)

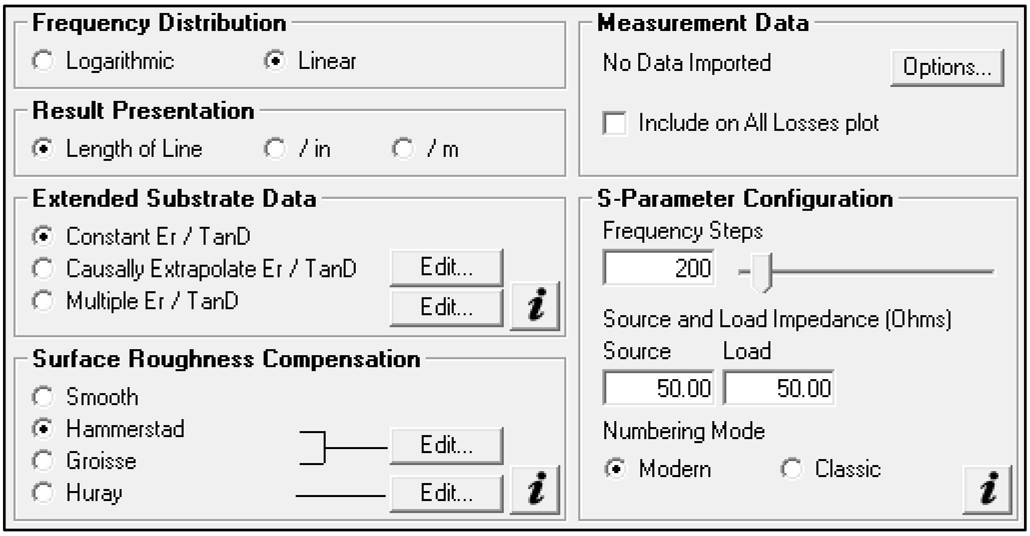
The Frequency-dependent Calculation Interface displays frequency-dependent and structure parameters.



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.



Via Checks

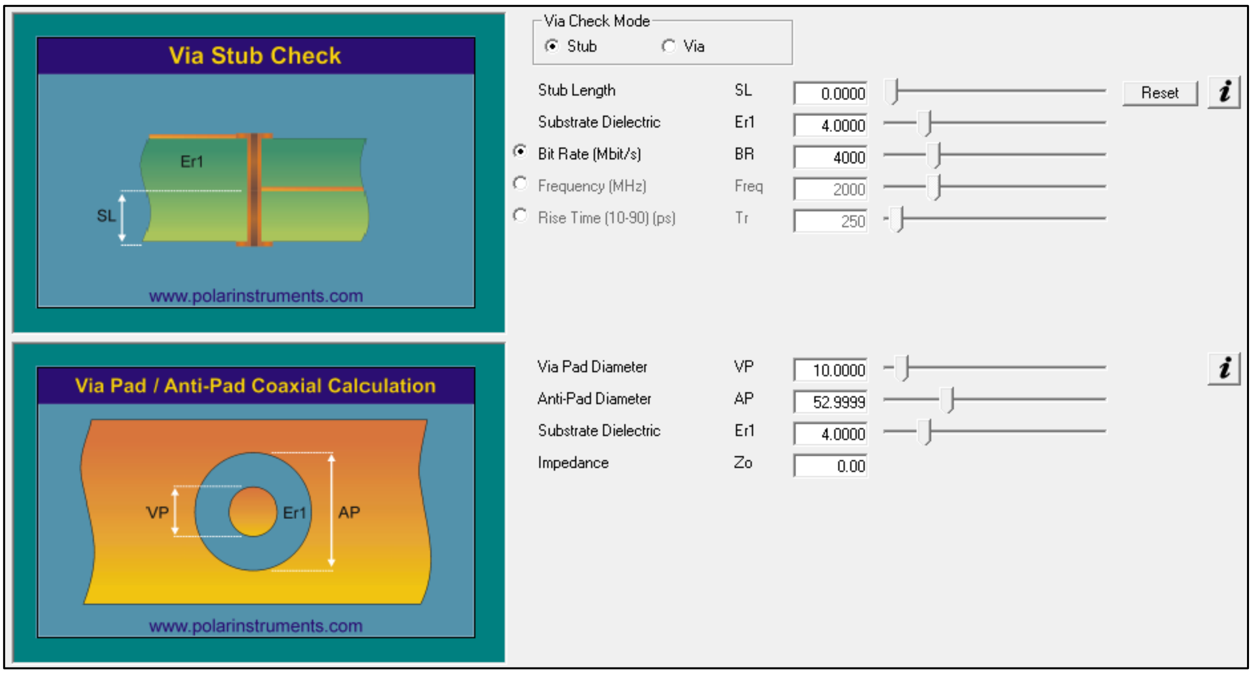
Via modelling

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/antipad coaxial calculation

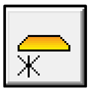
The Via Checks tab also includes via pad/anti-pad calculation. The anti-pad is the void area (shown as the blue annular ring in the diagram above) between the pad and the copper of the plane. It is generally designed so that it maintains the impedance of a transmission line as it passes through the plane.

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 / Paste buttons, the Atlas and CITS data log import and Track Resistance Calculator buttons.

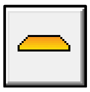
Display structures

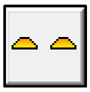
 Display all structures

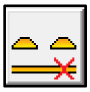
 Display user structures

 Si Project

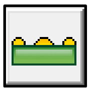
Single ended/differential structures

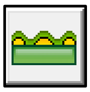
 Display single ended structures

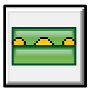
 Display differential structures

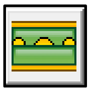
 Display groundless differential structures

Coplanar single ended structures

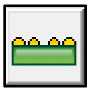
 Display surface coplanar structures

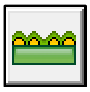
 Display coated coplanar structures

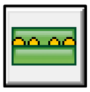
 Display embedded coplanar structures

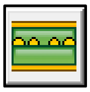
 Display offset coplanar structures

Coplanar differential structures

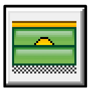
 Display surface coplanar differential structures

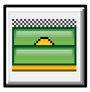
 Display coated coplanar differential structures

 Display embedded coplanar differential structures

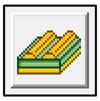
 Display offset coplanar differential structures

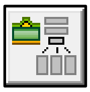
Toggle hatch plane

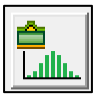
 Toggle lower hatch plane

 Toggle upper hatch plane

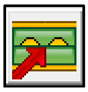
Process window

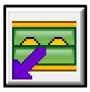
 Multiline Crosstalk

 Launch process window

 Monte Carlo Analysis

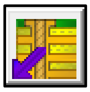
Copy/paste structure parameters

 Copy structure parameters

 Paste structure parameters

Copy/paste structures to/from Speedstack

 Copy structure to Speedstack

 Paste structure from Speedstack

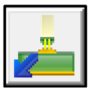
Copy structure to CGen Coupon Generator

 Copy structure to CGen Coupon Generator

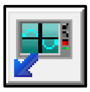
Paste from Speedstack into Si Project

 Paste structures from Speedstack into Si Project

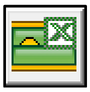
Import measurement data

 Import measurement data from CITS

Import measurement data from Atlas

 Import measurement data from Touchstone

Excel interface



Launch Si Excel interface

Track resistance calculator

Launch Track Resistance Calculator

Using menu commands

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.

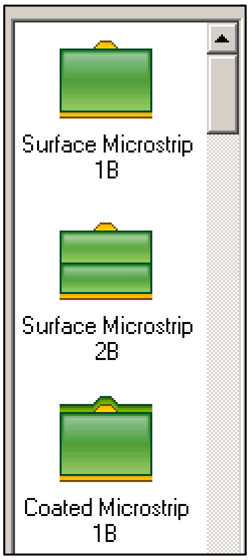
Use the Configuration menu to set structure parameter minimum and maximum values and goal seeking convergence settings used by the calculation engine.

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 button selects the associated range of controlled impedance structures (single-ended, differential, coplanar, etc.) for display in the Structure Bar.

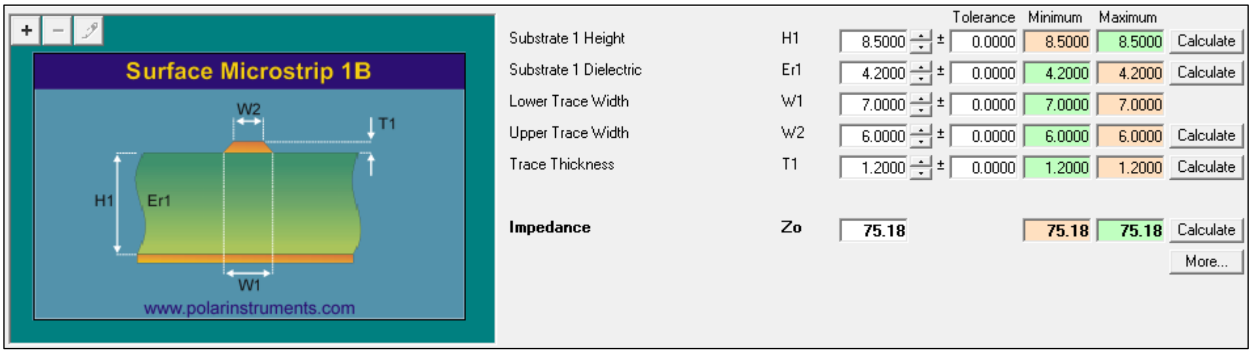
Use the Copy / Paste buttons to exchange controlled impedance information with Speedstack PCB Stackup Builder and copy structures to CGen Coupon Generator.

Structure Bar

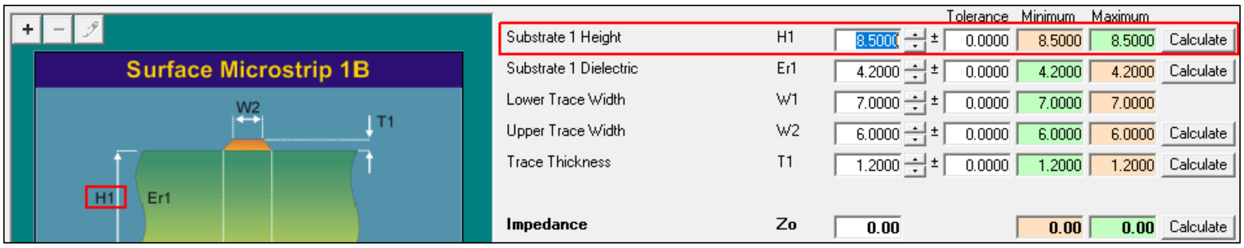


Use the Structure Bar to select a controlled impedance structure from the list of structures displayed. The range of structures displayed is controlled by the associated button on the Toolbar.

Structure graphics

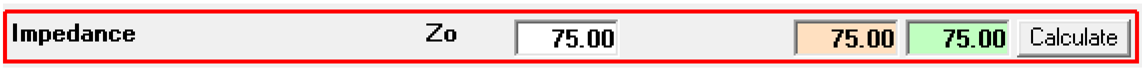


The Structure Graphic reflects the chosen controlled impedance structure. During lossless modelling, clicking the parameter "hotspot" (the parameter label in the graphic, H1, Er1, etc.) outlines the parameter in red and activates the associated parameter field for editing.



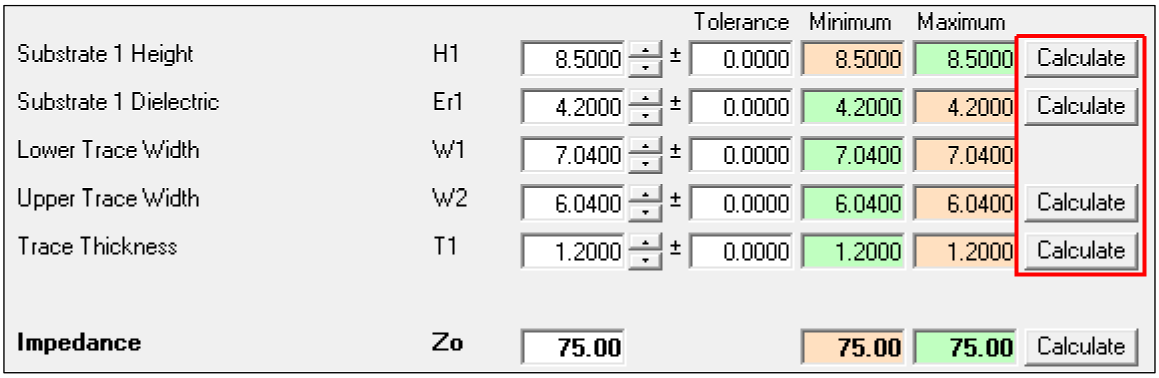
Calculation interface

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. Click the Impedance Calculate button to calculate Z_0 for all the currently displayed parameters.



Goal seeking a parameter

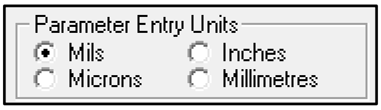
Choose an impedance for Z_0 and click the Calculate button against a parameter to goal seek a value for that parameter to achieve the target impedance.



See *Goal Seek Convergence* below for a discussion of coarse and fine convergence.

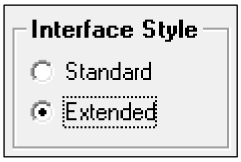
Parameter Entry Units

Use the Parameter Entry Units to specify the calculation units, mils, microns, inches or millimetres

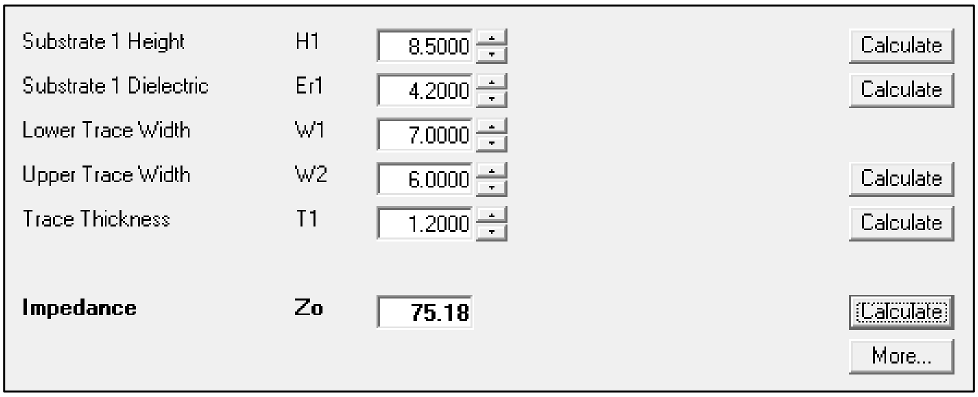


Interface Style – Standard/Extended Interface

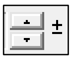
Choose between the Standard or Extended interface.



Use the Standard or Extended interface to specify the parameters associated with the selected structure.



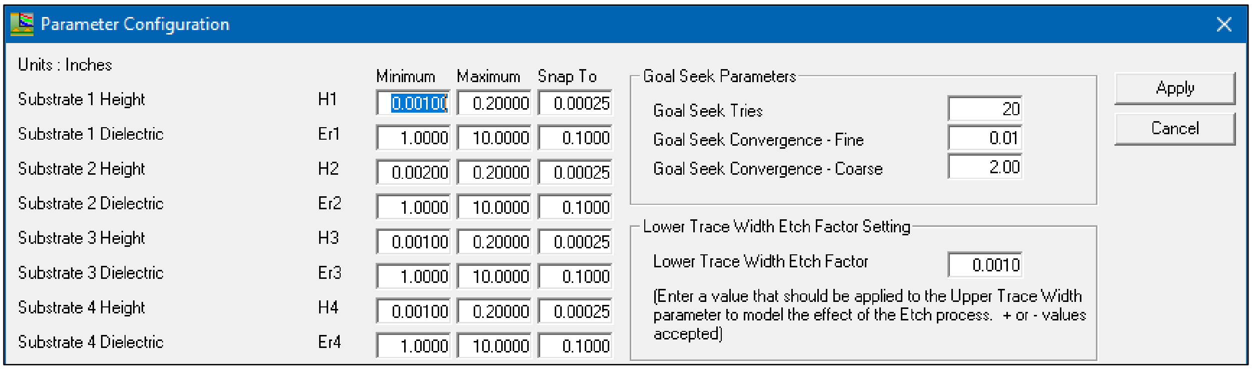
Standard Interface



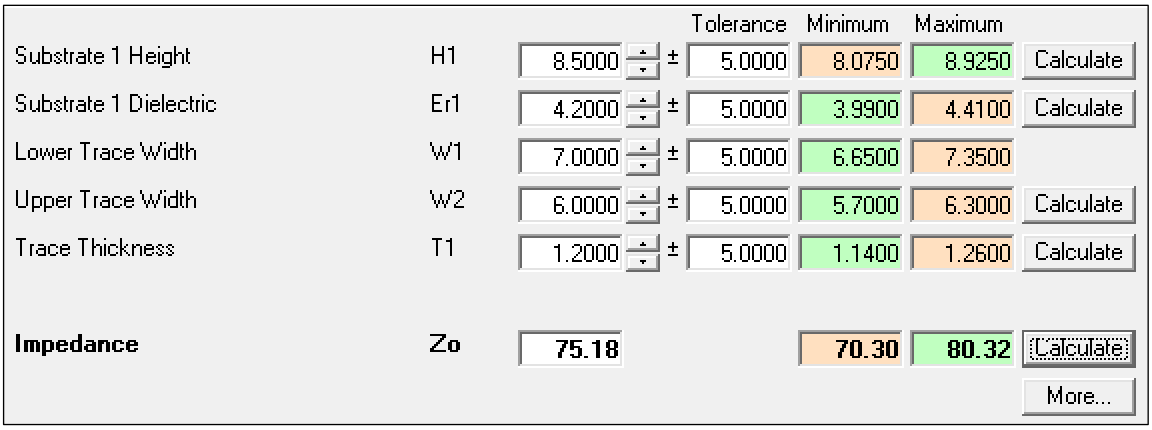
Snap To Button

Enter each parameter value directly or use the Snap To buttons to increment to the target parameter value.

Snap To button increments are specified in the Configuration | Parameters | Parameter Configuration dialog (below.)



Parameter Configuration dialog.



Extended Interface with tolerances and minimum and maximum parameter values

Use the Extended Interface to specify structure parameters and also apply tolerances to a calculation. The colored text fields indicate which parameter affects the minimum and maximum impedance values; for example, consider the green-colored fields in the graphic above – variations in the maximum value of H1 and the minimum value of Er1, W1, W2 and T1 affect the maximum value of impedance, Z_0. Similarly, the orange fields affect the minimum value of Z_0.

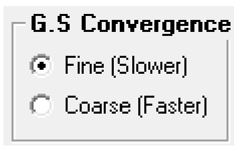
Goal Seek Convergence

Goal seeking parameters on controlled impedance structures

The Si8000m and Si9000e field solvers "goal seek" for parameter dimensions (core thickness H1, trace widths W1, W2, etc.) on controlled impedance structures using an iterative calculation process. Goal seeking continues until the convergence process brings the parameter within acceptable limits.

Using coarse and fine convergence

Some parameters will be variable but constrained within limits (for example, core thickness will vary but in discrete increments.) For these parameters, it is appropriate to goal seek using *Coarse* goal seeking convergence (see the G.S. Convergence dialog below) with its associated time savings to arrive at a final value.



.

Other parameters, however, such as trace widths W1, W2 and trace separation S1 can be regarded as infinitely variable. For these, to arrive at the correct values within fine limits use *Fine* convergence.

Quick Solver operating configuration

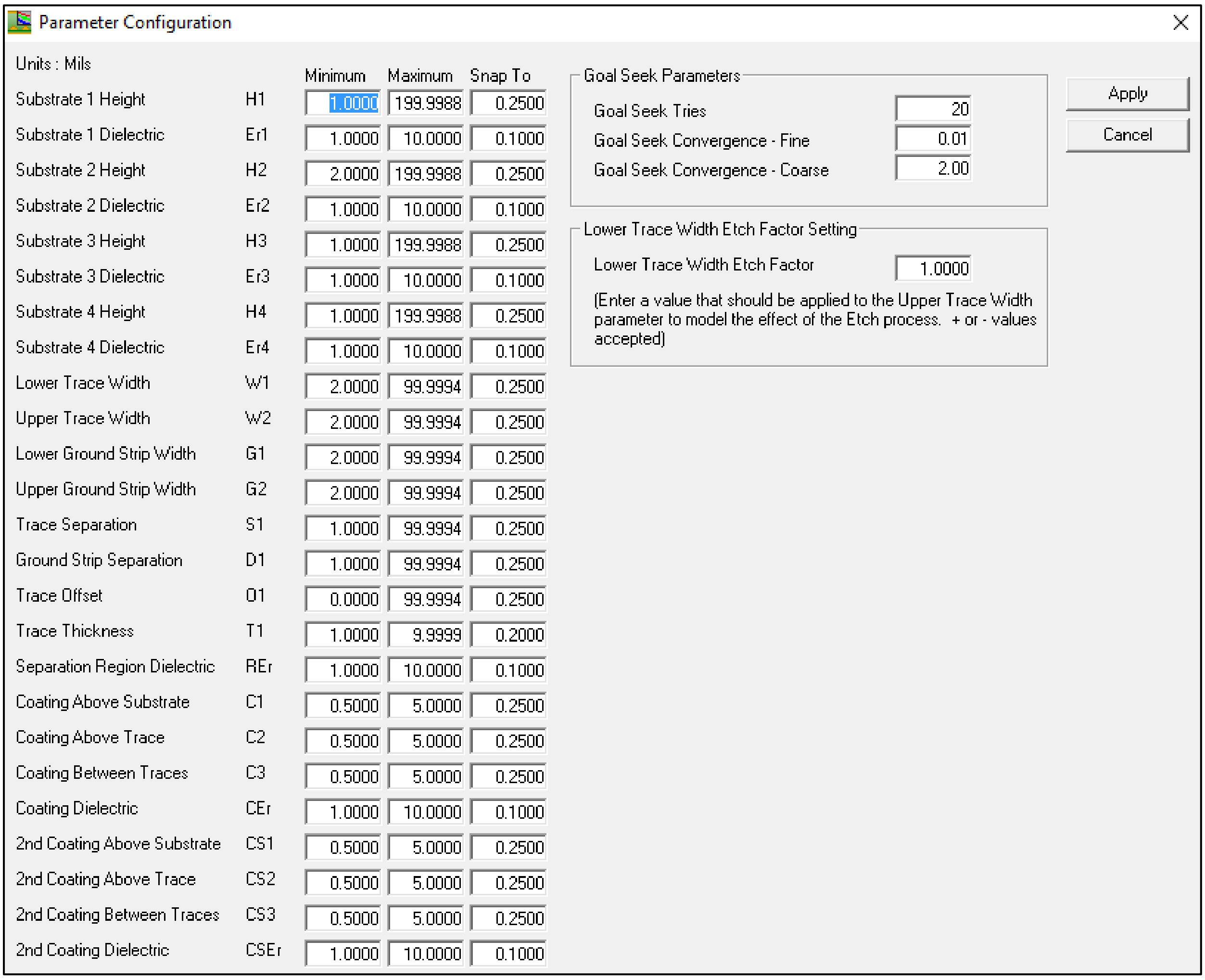
Parameter Configuration

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.

Calculation engine parameter values

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



Enter the values for minimum and maximum for each parameter.

Goal Seek parameters

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

Lower Trace Width Etch factor settings

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

Hatch configuration

The Quick Solver hatch plane/mesh module 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.

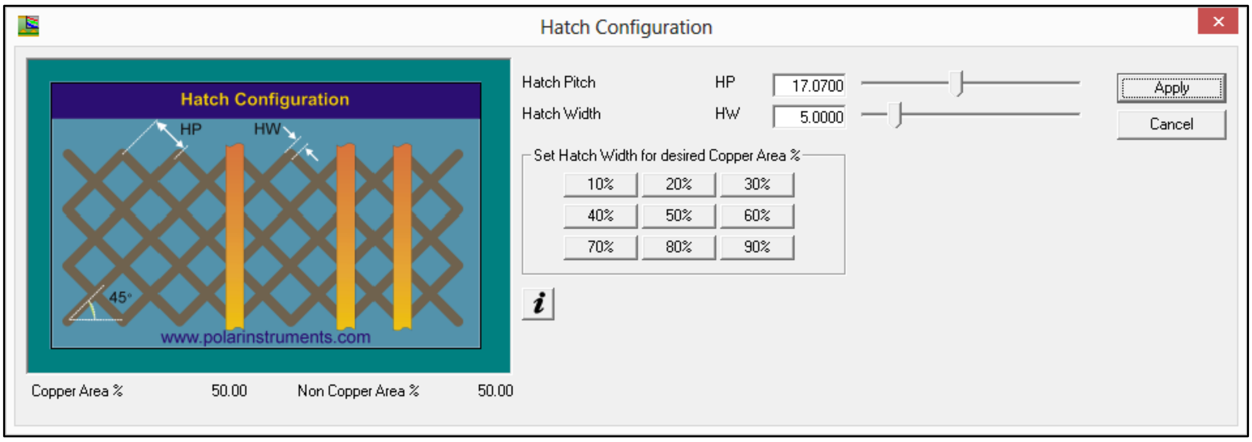
Modelling impedance on traces with hatch plane grounds

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

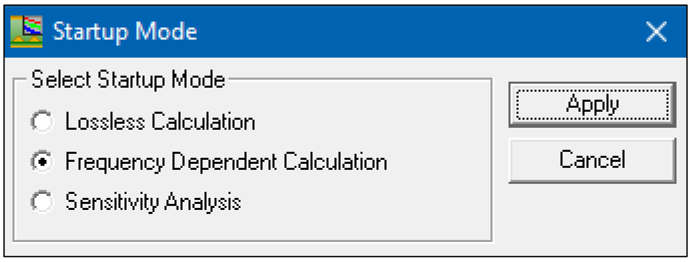
The Quick Solver's proprietary technique, XFE (Crosshatch Flex Enhancement) employs Polar's 2-D field solvers but uses a unique algorithm to correct for the effects of flex over a wide range of typical controlled impedance structures.



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 above. Hatch pitch and width may be specified either directly or by association with a choice of copper area settings. Set the pitch by 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

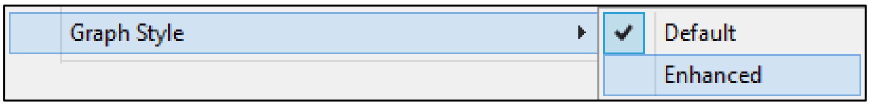
From the Configuration menu choose Startup Mode to specify the tab displayed when the program is started.



Choose the option and click Apply.

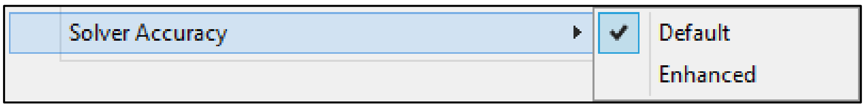
Graph Style

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.

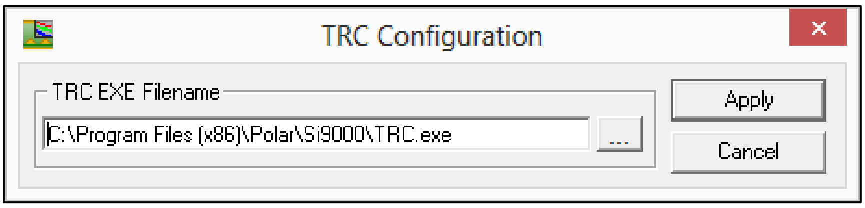


The Enhanced Mode is especially useful when calculating fine trace thickness geometries.

Note: Enhanced Mode will increase calculation time.

TRC configuration

If necessary, specify the location of the TRC Plus Track Resistance Calculator executable.



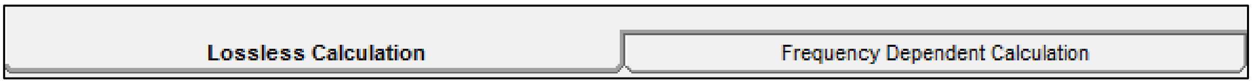
Lossless calculations

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



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

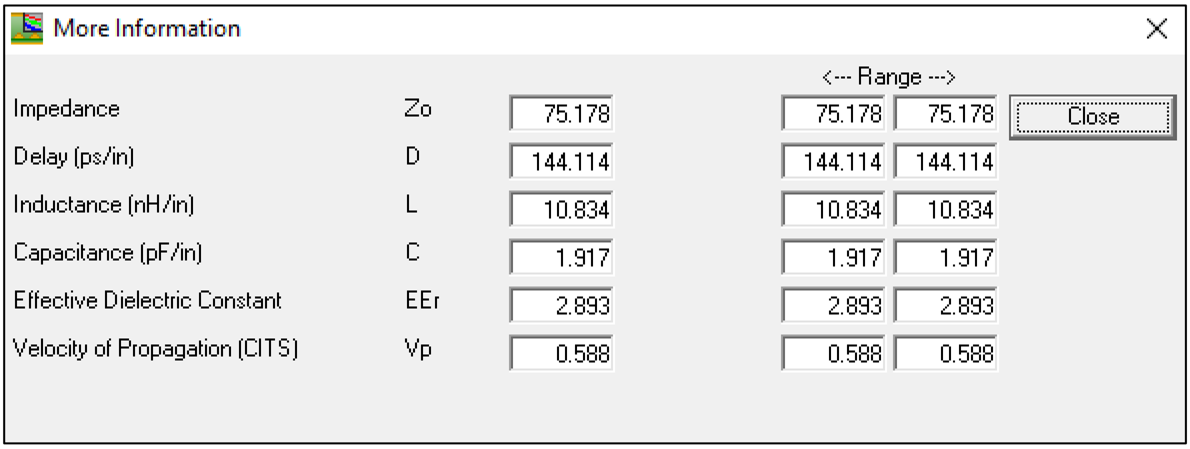
into the associated text boxes and press the Impedance **Calculate** button. The calculated impedance will appear in the Impedance (Z_o) box.

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.

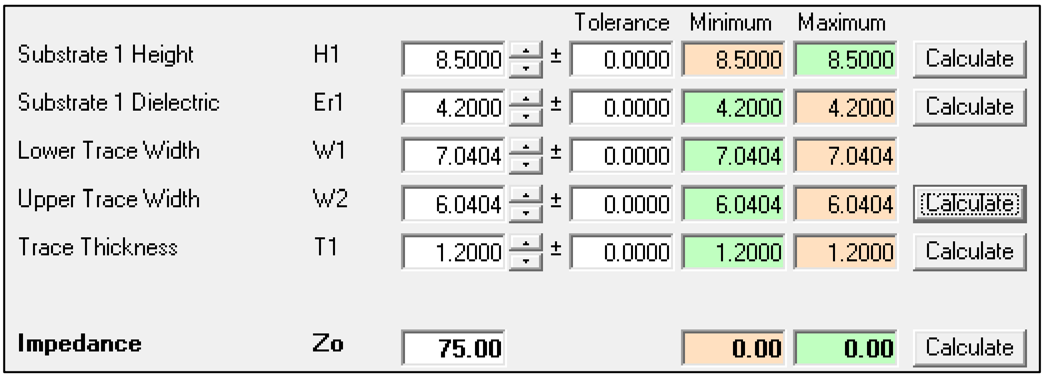
Enter the parameter values as described above into the text boxes and press the **More…** button. For the Standard Interface single ended results are shown below



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.

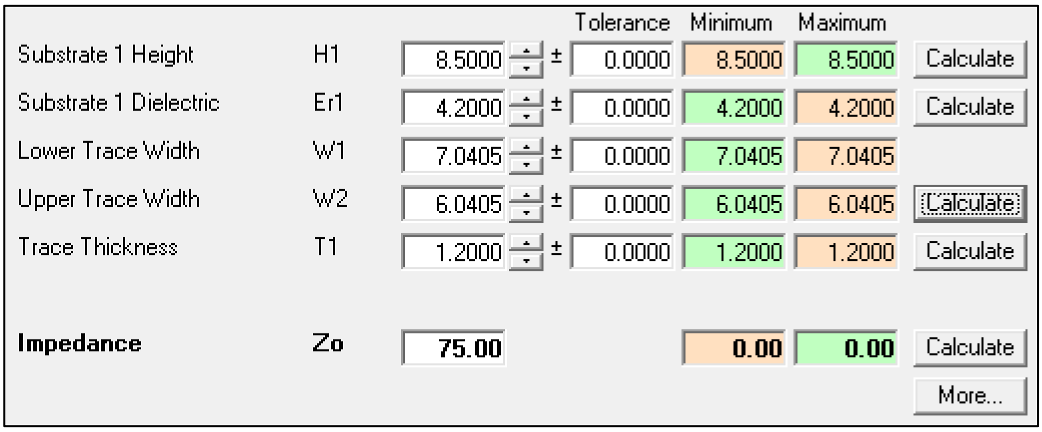
Specifying Goal Seeking convergence

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

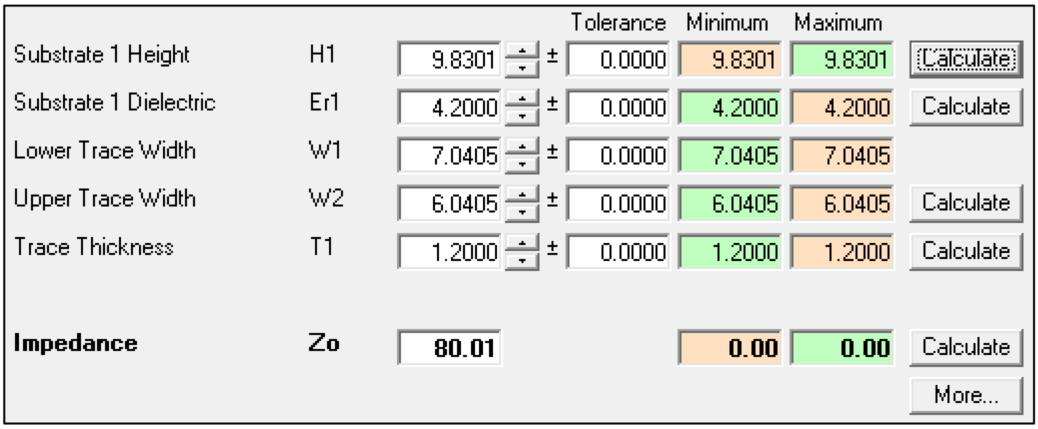
Selecting the Extended Interface Style displays additional fields, Tolerance, Minimum and Maximum allowing the user to specify a range of values for each parameter and observe the effect of manufacturing process variations.



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

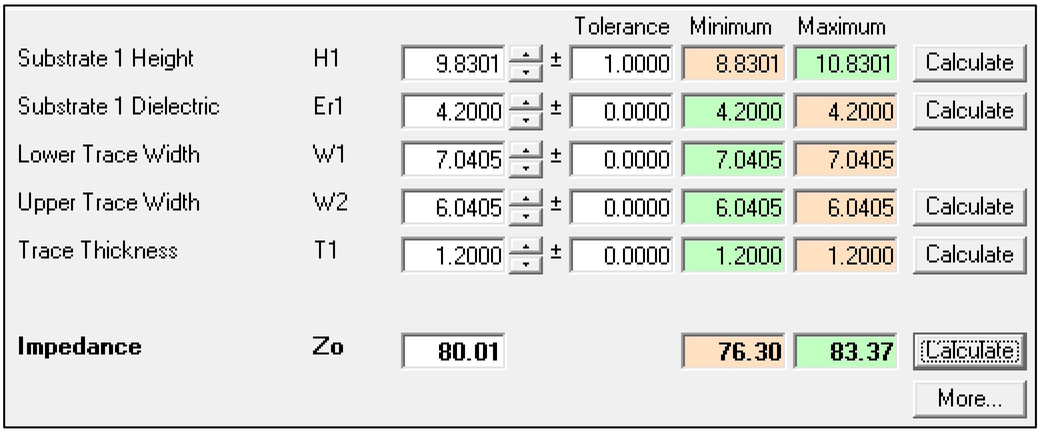
In this example we specify a nominal impedance value of 80 ohms and observe the effects on the nominal impedance of a manufacturing variation of ±1mil in the substrate height.

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



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 1mil 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.

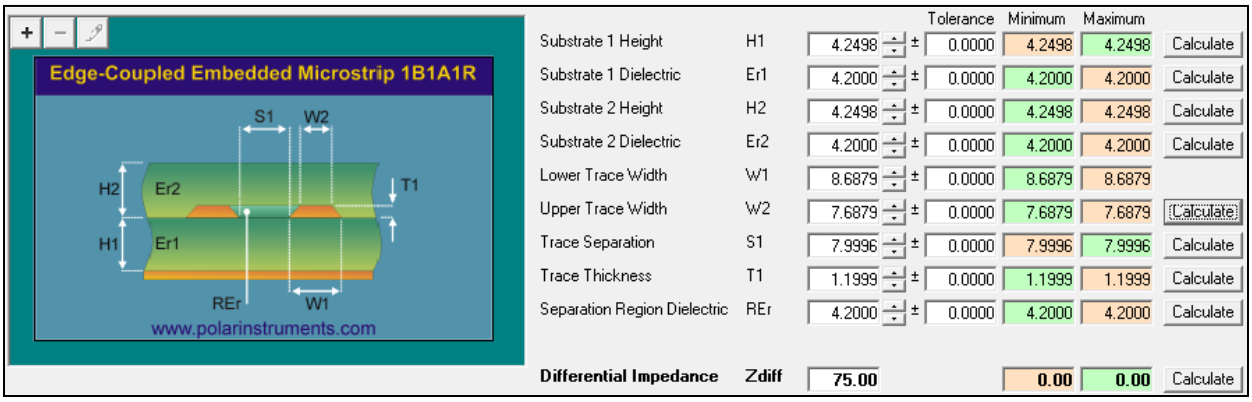
Using multiple tolerances

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.

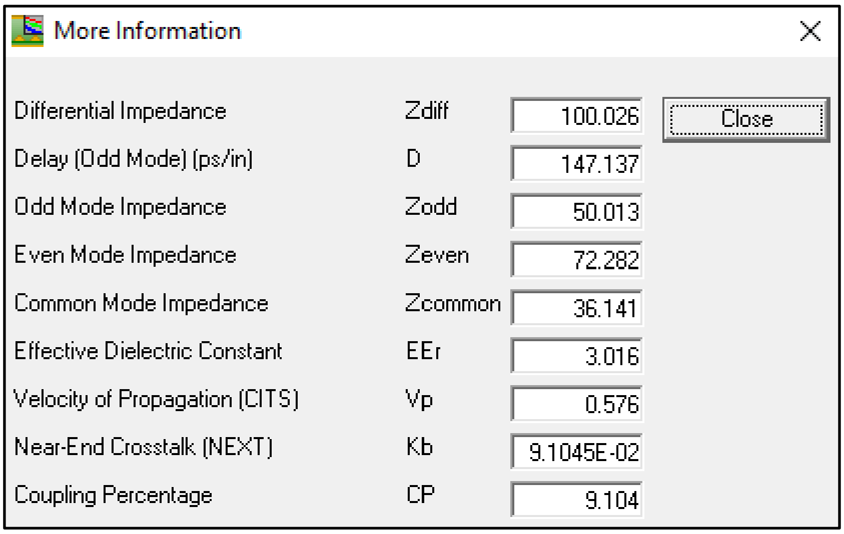


Enter the parameter values and tolerances if required into their respective fields and Click Calculate to calculate the resulting impedance.

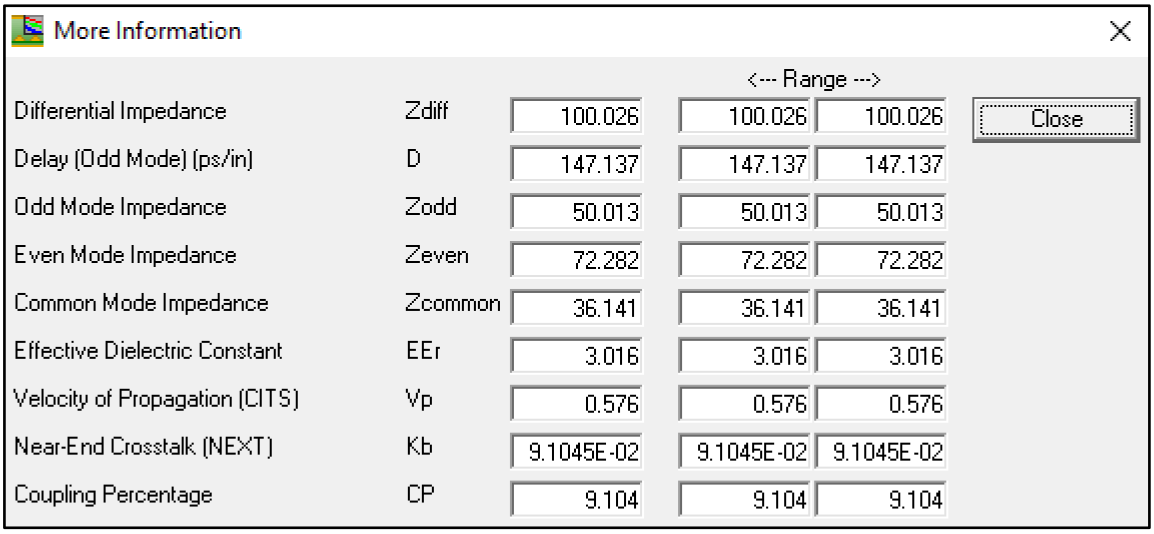
Use the other Calculate buttons to goal seek for the parameter values required to return a target impedance.

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.

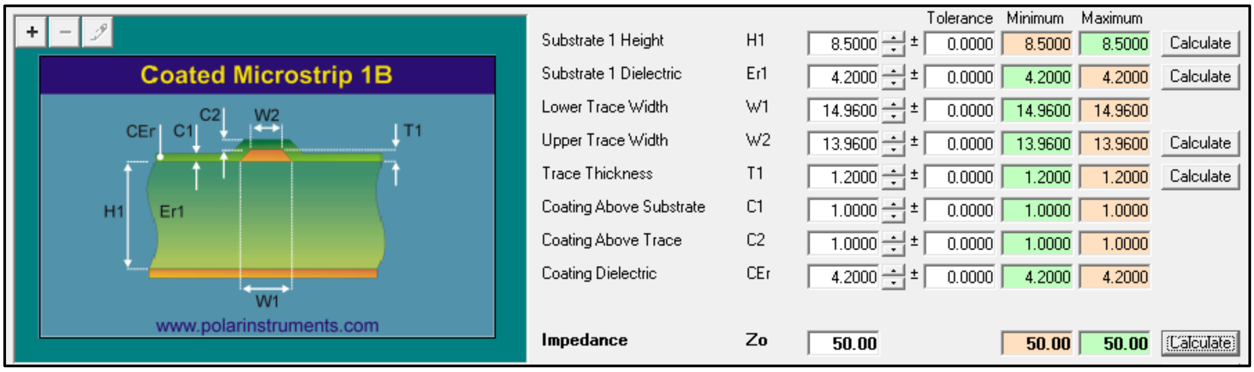
To recall a set of results choose **Open…** from the **File** menu and choose the desired results file and press **Open**.

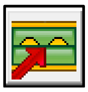
Copying and pasting parameters between structures

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.

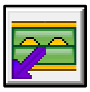
Select the single ended Coated Microstrip 1B structure and goal seek using trace width for 50 ohms.



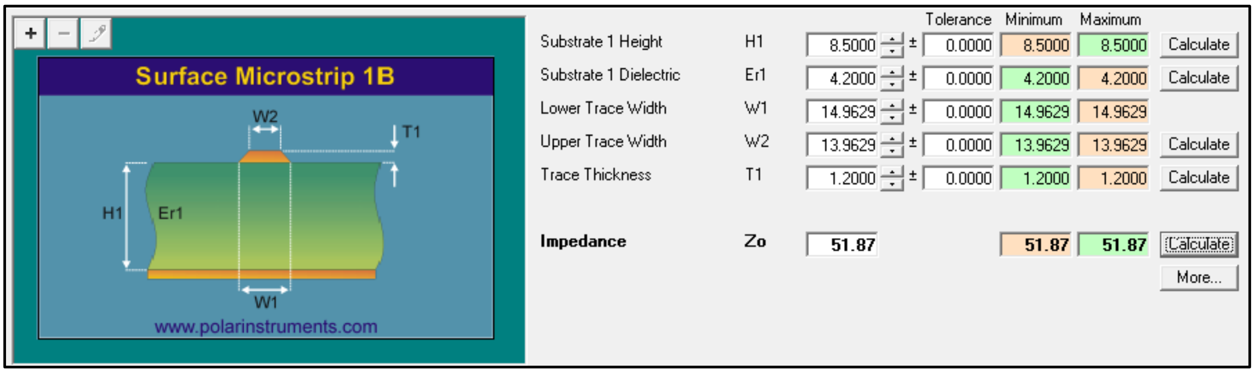


Copy the structure parameters to the clip board

Select the single ended Surface Microstrip 1B structure.



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

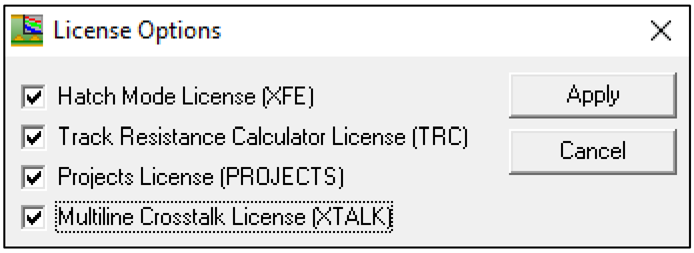


Si9000e calculates the impedance of the same structure prior to coating, illustrating the approximately 2 ohms difference.

Si Crosstalk – modelling multiline crosstalk

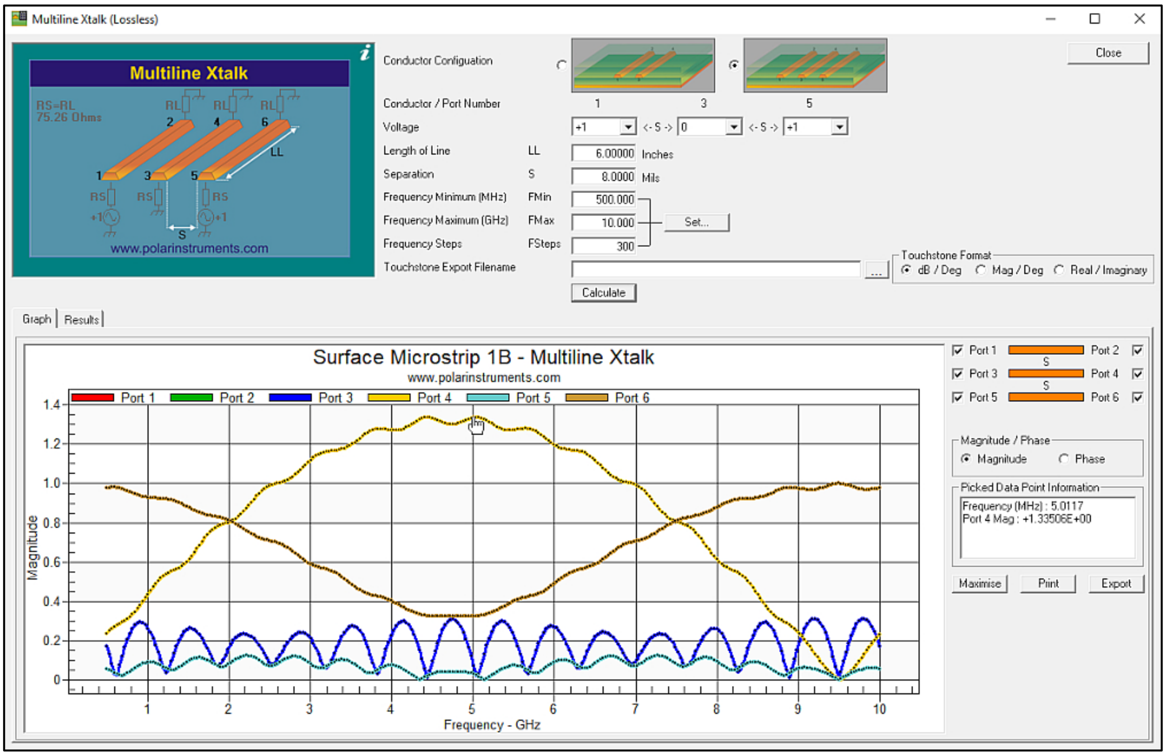
Note: Prior to performing multiline crosstalk modelling ensure that the Multiline Crosstalk License (XTALK) option is selected.

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

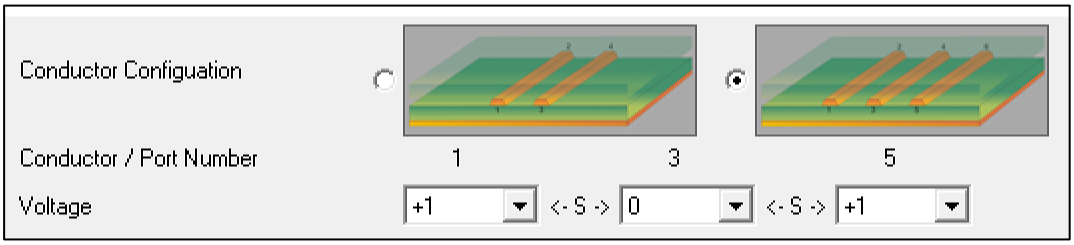
Forward, or *far-end*, crosstalk is energy that is coupled from the active signal line, the aggressor, onto a quiet passive victim line so that the transferred energy "travels forward" to the end of the victim line. *Forward*, or *far end*, crosstalk can be a problem if it is necessary, for example,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

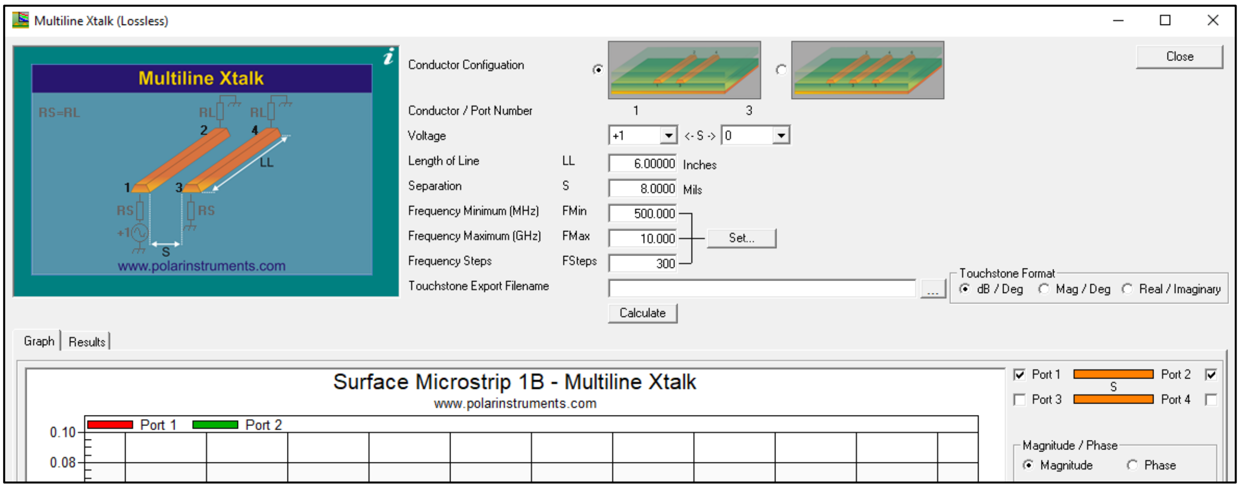
With the Si Crosstalk option, it is easy to illustrate, for example, how surface traces are much more prone to far end crosstalk than stripline traces.See Polar Application Note AP8164 *Introduction to forward and reverse crosstalk*.

Conductor configurations

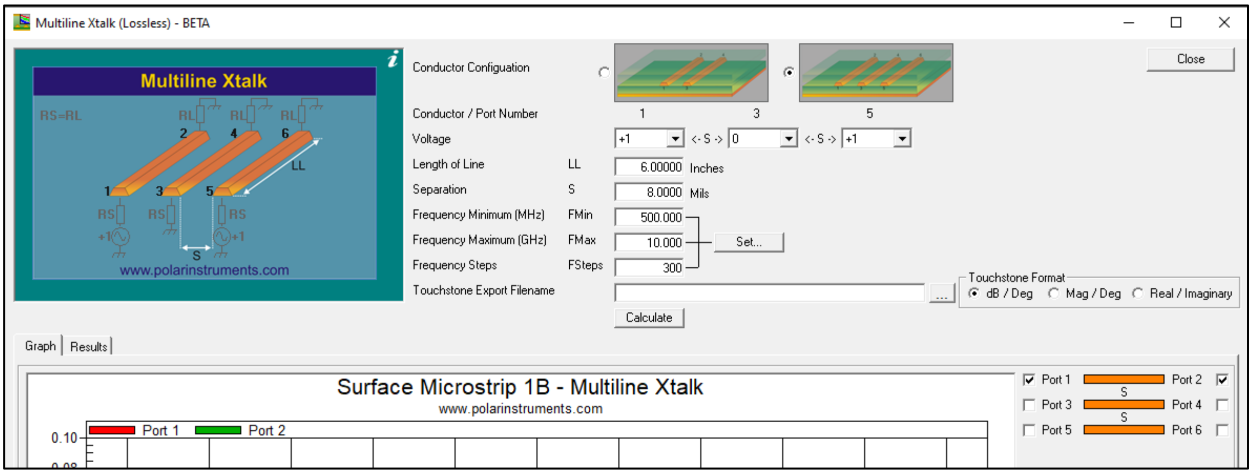
Modelling is provided for both *Aggressor – Victim* and *– Aggressor – Victim — Aggressor* conductor configurations.



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 conductor configuration

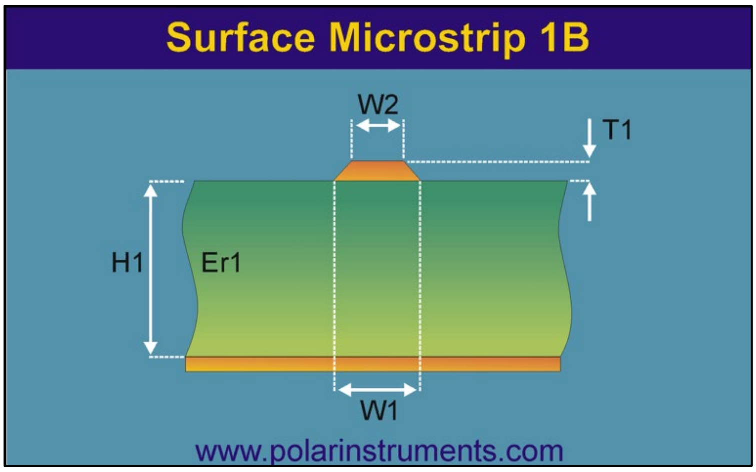


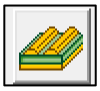
Aggressor – Victim – Aggressor conductor configuration

Modelling crosstalk on single ended microstrip traces

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

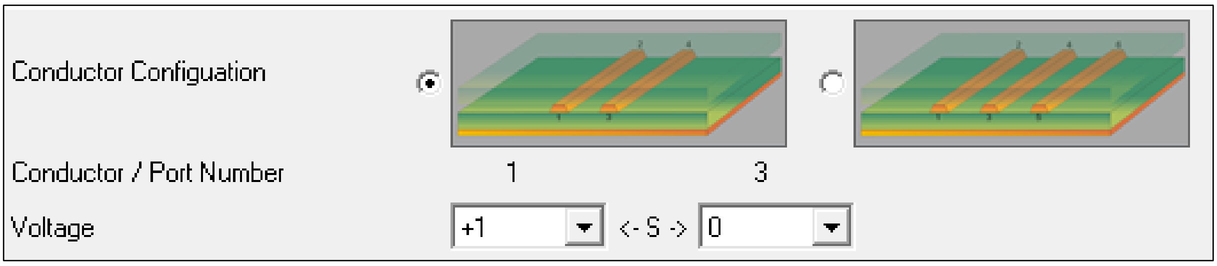
In the Lossless Calculation tab choose the single ended Surface Microstrip structure and enter the parameters for the target impedance.



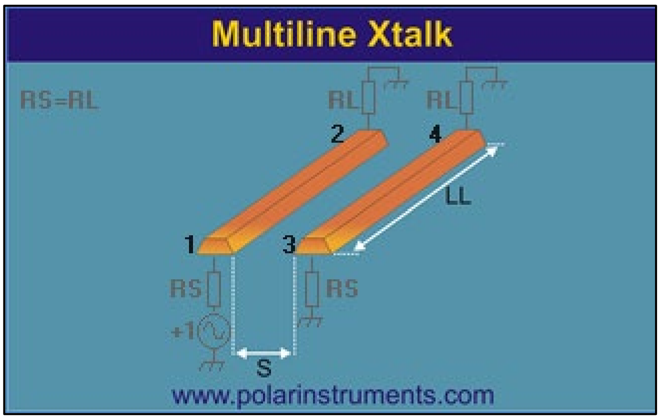


Multiline Crosstalk

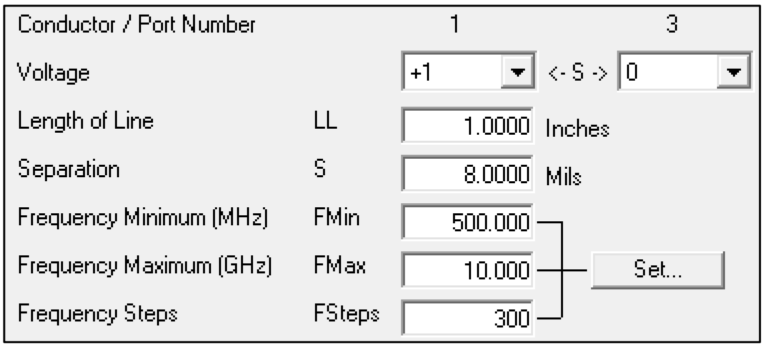
Click the Multiline Crosstalk icon to display the Multiline Crosstalk dialog – the Multiline Crosstalk option will run a pair of surface microstrips alongside each other and model the resulting crosstalk; i.e. it uses the selected structure and models a pair of those lines side by side.
For this example, an aggressor trace and a victim trace, choose the Aggressor – Victim conductor configuration.



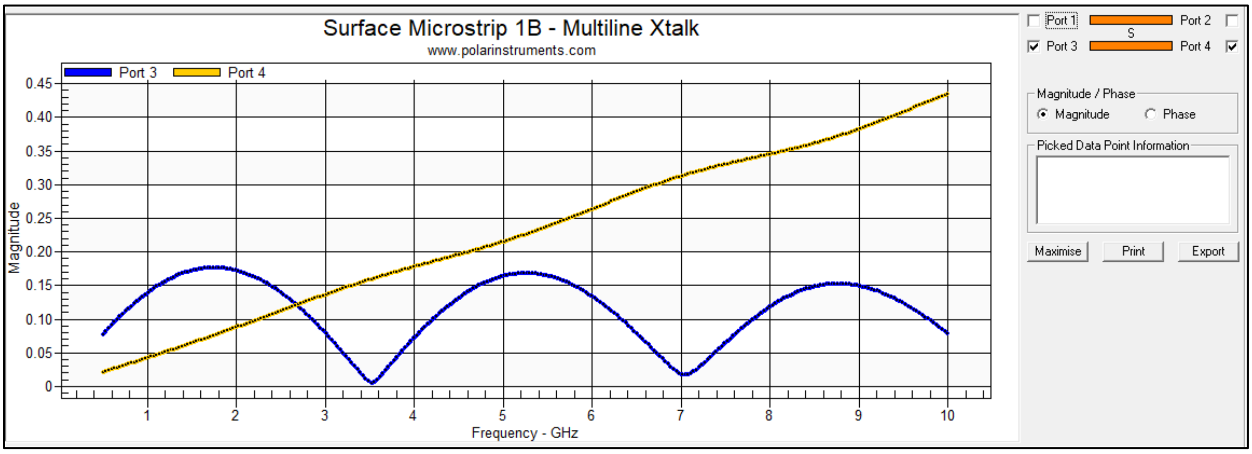
The Crosstalk graphic displays the two lines and four ports associated with the model.



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



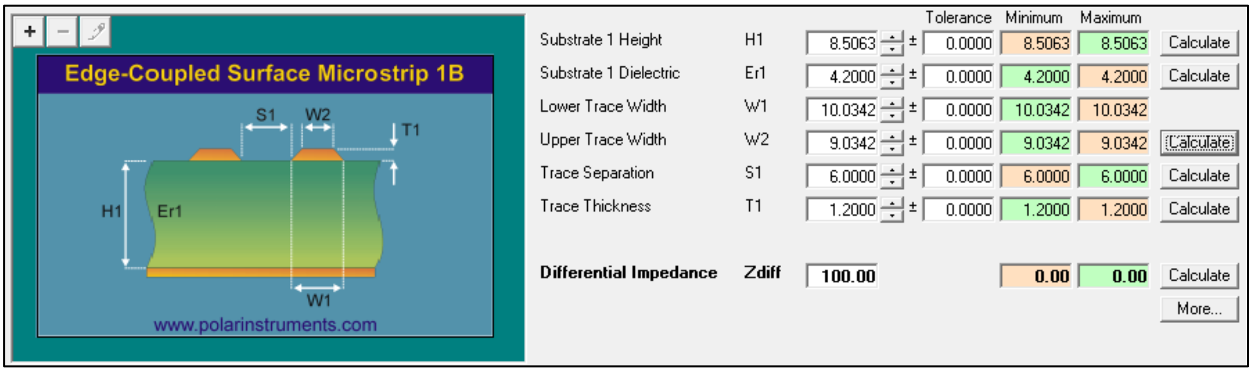
Check the ports to be displayed and click Calculate.

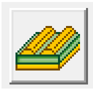


Change the, line length, separation, frequency, etc. to examine the effect of changing parameters on near and far end crosstalk.

Modelling differential pairs

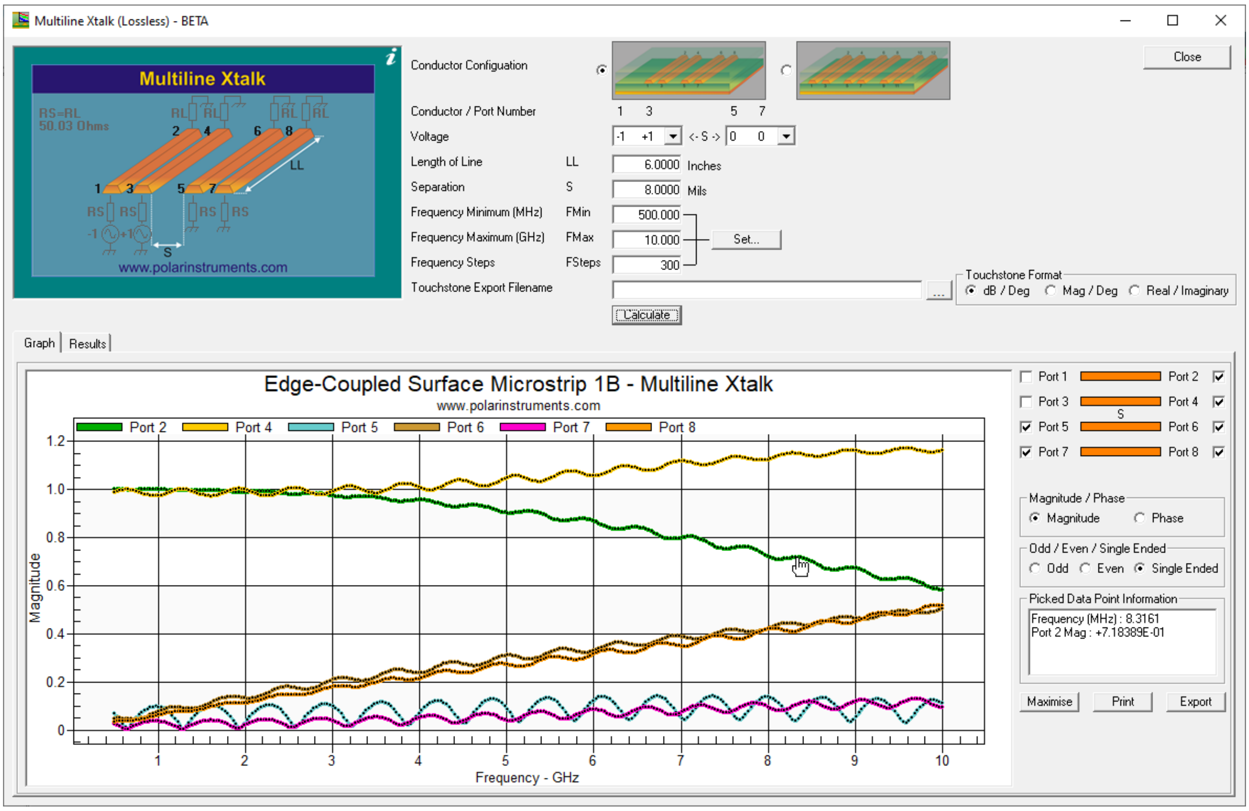
Select the Edge-Coupled Surface Microstrip structure and supply the parameters (goal seeking if necessary) for the target impedance.





Multiline Crosstalk

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.

Monte Carlo impedance analysis

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

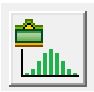
Si8000m and Si9000e include Monte Carlo simulation of printed circuit board impedance to provide a graphical mechanism for predicting and presenting the variation of PCB trace impedance for a production run of PCBs.

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

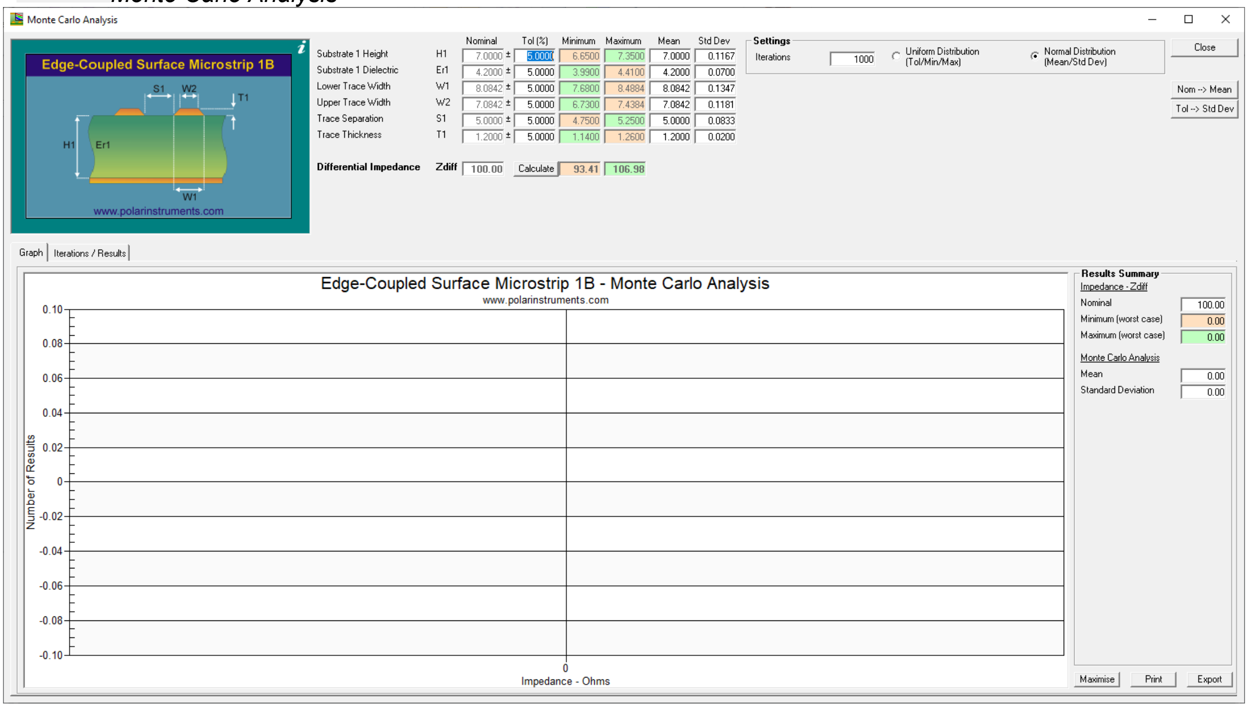
Choose the structure to be analysed.

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



Monte Carlo Analysis

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



Monte Carlo analysis with Constant Pitch

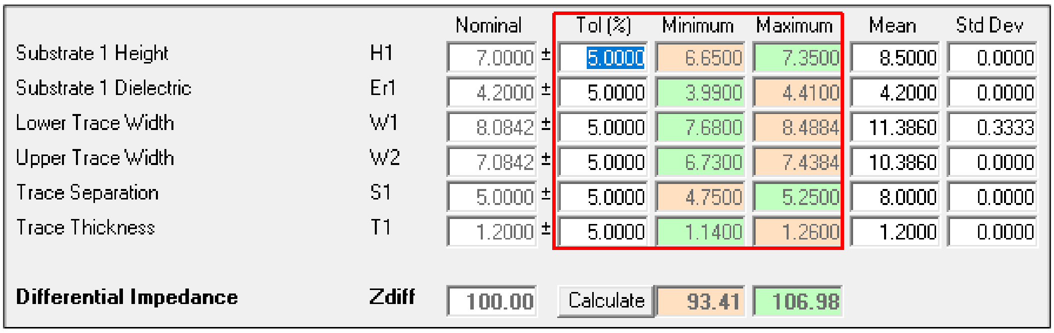
For differential structures, the Monte Carlo analysis supports constant pitch between traces; the field solver will build a bar graph of the variations in trace impedance for the range of parameter values; the Constant Pitch option supports both Uniform and Normal Distributions.

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.

Uniform Distribution

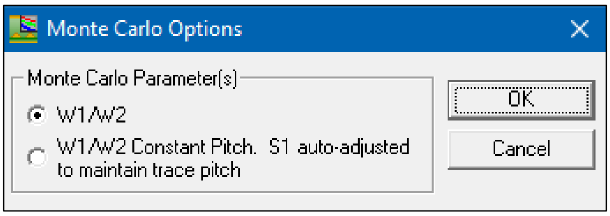
Choose Uniform Distribution to use the values specified in the Tolerance, Minimum and Maximum columns.



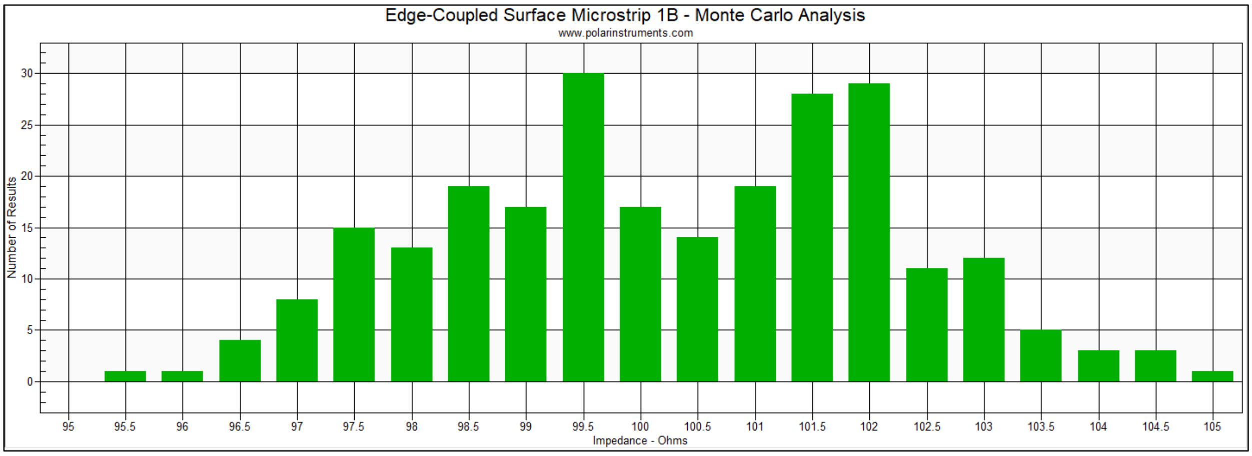
		Nominal		Tol (%)	Minimum	Maximum	Mean	Std Dev
Substrate 1 Height	H1	7.0000	±	5.0000	6.6500	7.3500	8.5000	0.0000
Substrate 1 Dielectric	Er1	4.2000	±	5.0000	3.9900	4.4100	4.2000	0.0000
Lower Trace Width	W1	8.0842	±	5.0000	7.6800	8.4884	11.3860	0.3333
Upper Trace Width	W2	7.0842	±	5.0000	6.7300	7.4384	10.3860	0.0000
Trace Separation	S1	5.0000	±	5.0000	4.7500	5.2500	8.0000	0.0000
Trace Thickness	T1	1.2000	±	5.0000	1.1400	1.2600	1.2000	0.0000
Differential Impedance	**Zdiff**	100.00		Calculate	93.41	106.98		

Click Calculate – the Monte Carlo Options dialog is displayed

Specify the Monte Carlo parameters, W1/W2.



The Uniform Distribution graph is displayed



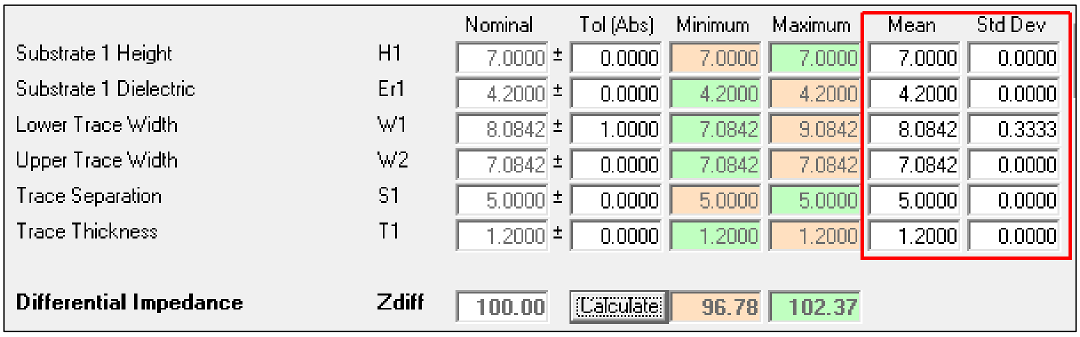
Monte Carlo Analysis with Monte Carlo Parameters W1/W2 option – Uniform Distribution

Note that if , the Nominal and Mean values will diverge so it is often useful to update the Mean values to match the Nominal. This can be achieved by either manually keying in the Mean values or by clicking the Nom -> Mean button.

Normal Distribution

Choose Normal Distribution to use the parameter values for Mean and Standard Deviation. Structure parameters will often have a tolerance; for example, the substrate heights (H*n*) for stack up materials will typically have a tolerance of 5% to 10%.

When using the Monte Carlo Normal Distribution option, the Mean and Standard Deviation (Std Dev) parameters are used, as illustrated below.

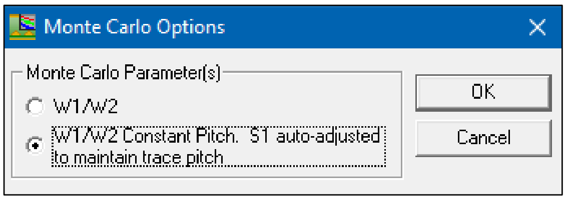


		Nominal		Tol (Abs)	Minimum	Maximum	Mean	Std Dev
Substrate 1 Height	H1	7.0000	±	0.0000	7.0000	7.0000	7.0000	0.0000
Substrate 1 Dielectric	Er1	4.2000	±	0.0000	4.2000	4.2000	4.2000	0.0000
Lower Trace Width	W1	8.0842	±	1.0000	7.0842	9.0842	8.0842	0.3333
Upper Trace Width	W2	7.0842	±	0.0000	7.0842	7.0842	7.0842	0.0000
Trace Separation	S1	5.0000	±	0.0000	5.0000	5.0000	5.0000	0.0000
Trace Thickness	T1	1.2000	±	0.0000	1.2000	1.2000	1.2000	0.0000
Differential Impedance	**Zdiff**	100.00		Calculate	96.78	102.37		

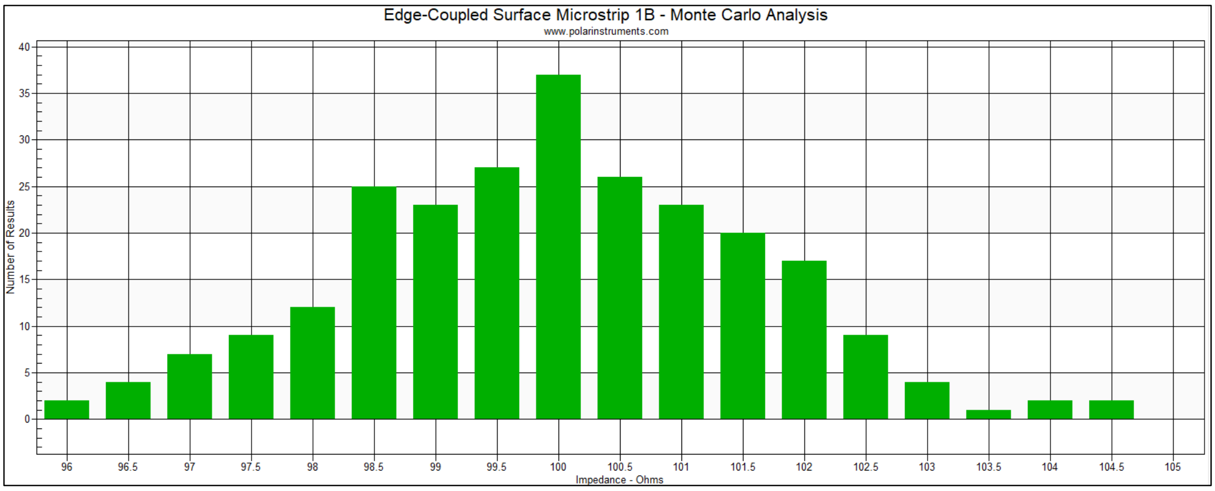
In the example above, W1 is assigned a tolerance of ±1 mil; other parameters remain unchanged.

Click the Tol -> Std Dev to use the Tolerance 3-sigma value to calculate Standard Deviation (sigma.)

Click Calculate – and specify the Monte Carlo parameters, W1/W2 Constant Pitch.



The Normal Distribution graph is displayed



Monte Carlo Analysis with Monte Carlo Parameters W1/W2 Constant Pitch option – Normal Distribution

Displaying Results Tables

Click Iterations/Results to inspect the W1/W2 results table: note the value of S1 – trace separation – is constant and the pitch varying.

Graph | Iterations / Results

H1	Er1	W1	W2	S1	Pitch	T1	Zodd	Zeven	Zdiff	Zcommon	Calc Success
8.5000	4.2000	6.9762	5.9762	8.0000	14.9762	1.2000	61.5292	88.1082	123.0585	44.0541	Yes
8.5000	4.2000	6.6423	5.6423	8.0000	14.6423	1.2000	62.6986	90.0426	125.3971	45.0213	Yes
8.5000	4.2000	6.4013	5.4013	8.0000	14.4013	1.2000	63.5819	91.5013	127.1637	45.7507	Yes
8.5000	4.2000	6.5219	5.5219	8.0000	14.5219	1.2000	63.1361	90.7652	126.2721	45.3826	Yes
8.5000	4.2000	6.9789	5.9789	8.0000	14.9789	1.2000	61.5203	88.0934	123.0405	44.0467	Yes
8.5000	4.2000	7.1399	6.1399	8.0000	15.1399	1.2000	60.9768	87.1935	121.9536	43.5968	Yes
8.5000	4.2000	6.6267	5.6267	8.0000	14.6267	1.2000	62.7547	90.1363	125.5095	45.0681	Yes
8.5000	4.2000	6.4952	5.4952	8.0000	14.4952	1.2000	63.2339	90.9268	126.4678	45.4634	Yes
8.5000	4.2000	7.4996	6.4996	8.0000	15.4996	1.2000	59.8122	85.2610	119.6244	42.6305	Yes
8.5000	4.2000	6.7852	5.7852	8.0000	14.7852	1.2000	62.1911	89.2034	124.3822	44.6017	Yes
8.5000	4.2000	7.1511	6.1511	8.0000	15.1511	1.2000	60.9395	87.1317	121.8790	43.5659	Yes
8.5000	4.2000	7.1574	6.1574	8.0000	15.1574	1.2000	60.9186	87.0972	121.8373	43.5486	Yes
8.5000	4.2000	7.0767	6.0767	8.0000	15.0767	1.2000	61.1885	87.5441	122.3771	43.7721	Yes
8.5000	4.2000	6.2418	5.2418	8.0000	14.2418	1.2000	64.1845	92.4955	128.3690	46.2477	Yes
8.5000	4.2000	6.9932	5.9932	8.0000	14.9932	1.2000	61.4714	88.0124	122.9427	44.0062	Yes

W1/W2 Results Table

For the W1/W2 Constant Pitch Results Table results table below, note the value of S1 – trace separation – is auto-adjusted to maintain constant pitch.

Graph | Iterations / Results

H1	Er1	W1	W2	S1	Pitch	T1	Zodd	Zeven	Zdiff	Zcommon	Calc Success
4.5952	4.2000	6.8279	5.8279	8.1721	15.0000	1.2000	50.7702	60.6544	101.5404	30.3272	Yes
4.5952	4.2000	6.8237	5.8237	8.1763	15.0000	1.2000	50.7892	60.6718	101.5783	30.3359	Yes
4.5952	4.2000	7.3659	6.3659	7.6341	15.0000	1.2000	48.4470	58.5449	96.8939	29.2725	Yes
4.5952	4.2000	7.5433	6.5433	7.4567	15.0000	1.2000	47.7170	57.8926	95.4340	28.9463	Yes
4.5952	4.2000	6.4140	5.4140	8.5860	15.0000	1.2000	52.6760	62.4130	105.3520	31.2065	Yes
4.5952	4.2000	7.1468	6.1468	7.8532	15.0000	1.2000	49.3721	59.3791	98.7443	29.6895	Yes
4.5952	4.2000	6.7802	5.7802	8.2198	15.0000	1.2000	50.9845	60.8511	101.9689	30.4255	Yes
4.5952	4.2000	7.0061	6.0061	7.9939	15.0000	1.2000	49.9808	59.9324	99.9616	29.9662	Yes
4.5952	4.2000	6.9307	5.9307	8.0693	15.0000	1.2000	50.3116	60.2342	100.6232	30.1171	Yes
4.5952	4.2000	7.1022	6.1022	7.8978	15.0000	1.2000	49.5643	59.5537	99.1285	29.7768	Yes
4.5952	4.2000	6.9142	5.9142	8.0858	15.0000	1.2000	50.3867	60.3032	100.7733	30.1516	Yes
4.5952	4.2000	6.8656	5.8656	8.1344	15.0000	1.2000	50.6022	60.5004	101.2043	30.2502	Yes
4.5952	4.2000	6.8796	5.8796	8.1204	15.0000	1.2000	50.5398	60.4433	101.0796	30.2216	Yes
4.5952	4.2000	7.1730	6.1730	7.8270	15.0000	1.2000	49.2601	59.2776	98.5202	29.6388	Yes
4.5952	4.2000	7.1849	6.1849	7.8151	15.0000	1.2000	49.2091	59.2315	98.4181	29.6157	Yes
4.5952	4.2000	6.3263	5.3263	8.6737	15.0000	1.2000	53.0962	62.8045	106.1924	31.4023	Yes

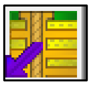
W1/W2 Constant Pitch Results Table

Nom --> Mean

Note: When Si8000m / Si9000e starts up the structure parameters for Nominal and Mean are set to the same values. During use, altering the nominal parameter values is quite common, either as a result of keying in new values or of goal seeking. In this case, the Nominal and Mean values will diverge so it is often useful to update the Mean values to match the Nominal. This can be achieved by either manually keying in the Mean values or by clicking the Nom -> Mean button.

Exchanging stackup structure information with Speedstack

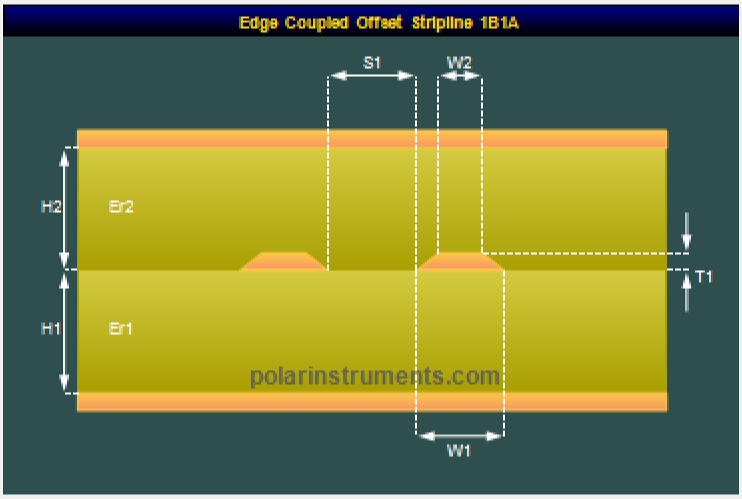
Polar field solvers exchange controlled impedance structures with the Speedstack Stackup Design System. Use the buttons shown below to import stackup layer data from Speedstack for use with the field solver goal seeking facility, and to export calculated values to Speedstack.

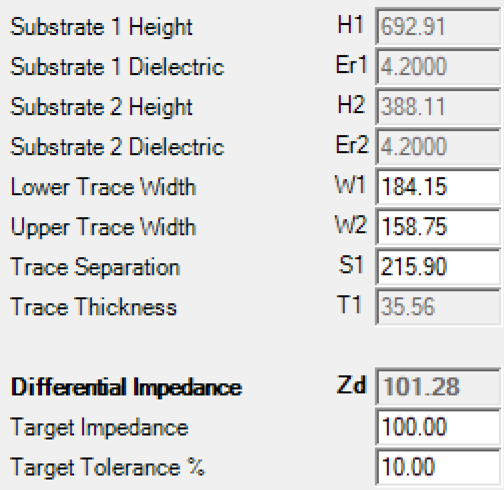


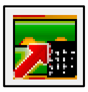
Paste structure from Speedstack

Copy structure to Speedstack

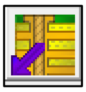
Select the structure within Speedstack.



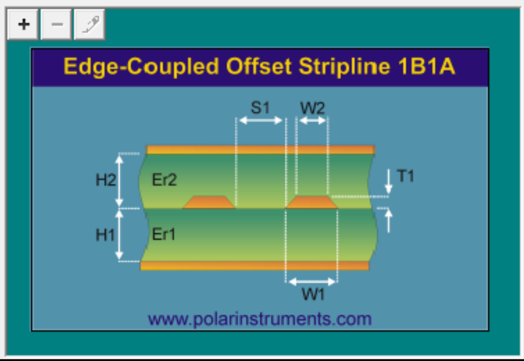




With the stackup parameters displayed in the Speedstack Controlled Impedance window, click the Speedstack **To Field Solver** button to transfer the current Speedstack parameters to the field solver.



Switch to the field solver and click the **Paste Structure From Speedstack** button to load the parameters into the associated fields.



				Tolerance	Minimum	Maximum	
Substrate 1 Height	H1	692.9100	±	0.0000	692.9100	692.9100	Calculate
Substrate 1 Dielectric	Er1	4.2000	±	0.0000	4.2000	4.2000	Calculate
Substrate 2 Height	H2	388.1100	±	0.0000	388.1100	388.1100	Calculate
Substrate 2 Dielectric	Er2	4.2000	±	0.0000	4.2000	4.2000	Calculate
Lower Trace Width	W1	184.1500	±	0.0000	184.1500	184.1500	
Upper Trace Width	W2	158.7500	±	0.0000	158.7500	158.7500	Calculate
Trace Separation	S1	215.9000	±	0.0000	215.9000	215.9000	Calculate
Trace Thickness	T1	35.5600	±	0.0000	35.5600	35.5600	Calculate
Differential Impedance	**Zdiff**	**101.28**			**101.28**	**101.28**	Calculate

Field solver parameter fields with Speedstack data loaded

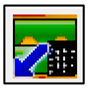
In the sample diagram above, the target impedance for the structure in the stack in Speedstack will be 100 Ohms. The designer has chosen to use just the values for Trace Width to goal seek for the target of 100 Ohms.

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.

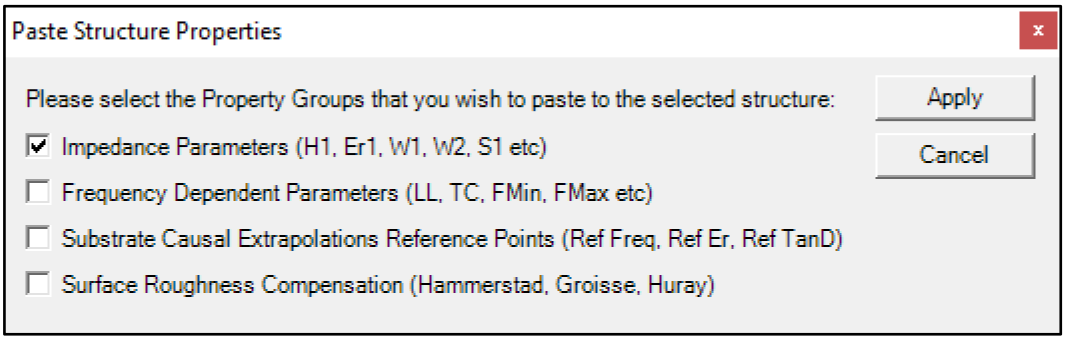
				Tolerance	Minimum	Maximum	
Substrate 1 Height	H1	692.9100	±	0.0000	692.9100	692.9100	Calculate
Substrate 1 Dielectric	Er1	4.2000	±	0.0000	4.2000	4.2000	Calculate
Substrate 2 Height	H2	388.1100	±	0.0000	388.1100	388.1100	Calculate
Substrate 2 Dielectric	Er2	4.2000	±	0.0000	4.2000	4.2000	Calculate
Lower Trace Width	W1	191.9693	±	0.0000	191.9693	191.9693	
Upper Trace Width	W2	166.5693	±	0.0000	166.5693	166.5693	Calculate
Trace Separation	S1	215.9000	±	0.0000	215.9000	215.9000	Calculate
Trace Thickness	T1	35.5600	±	0.0000	35.5600	35.5600	Calculate
Differential Impedance	**Zdiff**	**100.00**			**0.00**	**0.00**	Calculate

Solved values for impedance

Click the **Copy structure to Speedstack** button

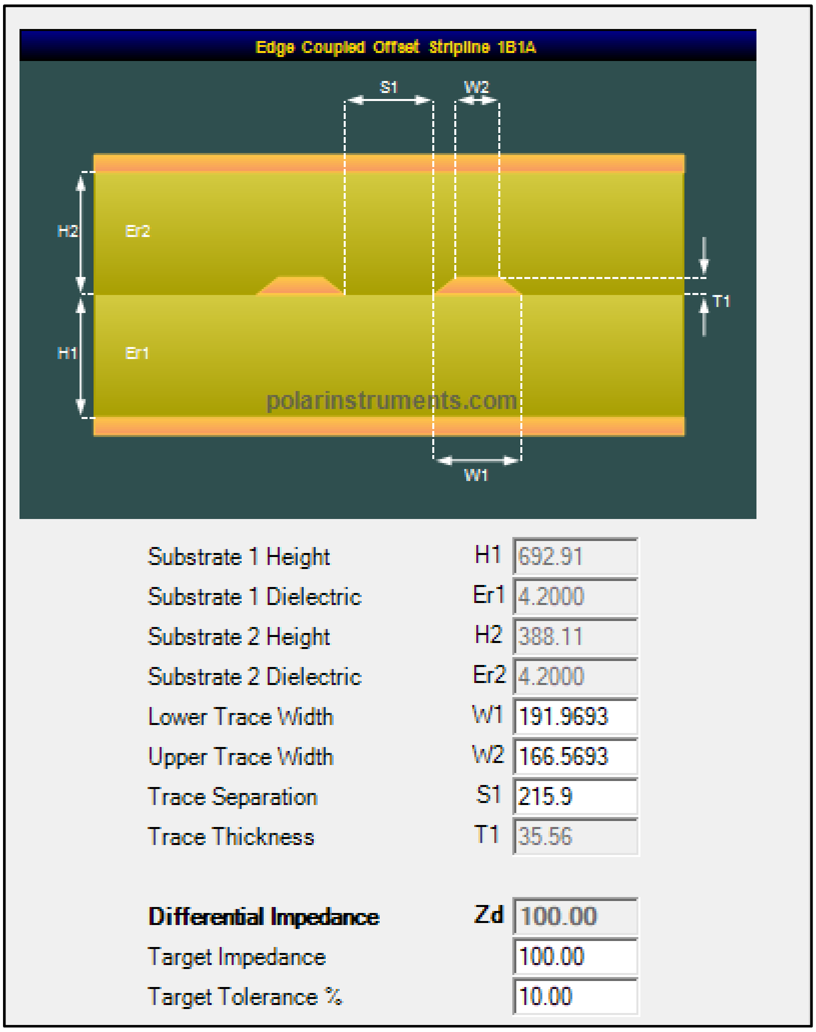


Switch to Speedstack and click the **From Field Solver** button – choose the properties to import and click Apply.



Selecting structure parameters to import to Speedstack

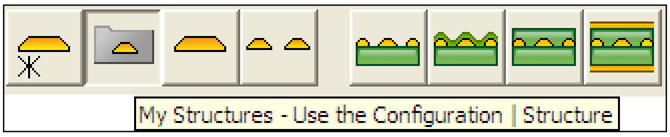
The structure is returned to Speedstack.



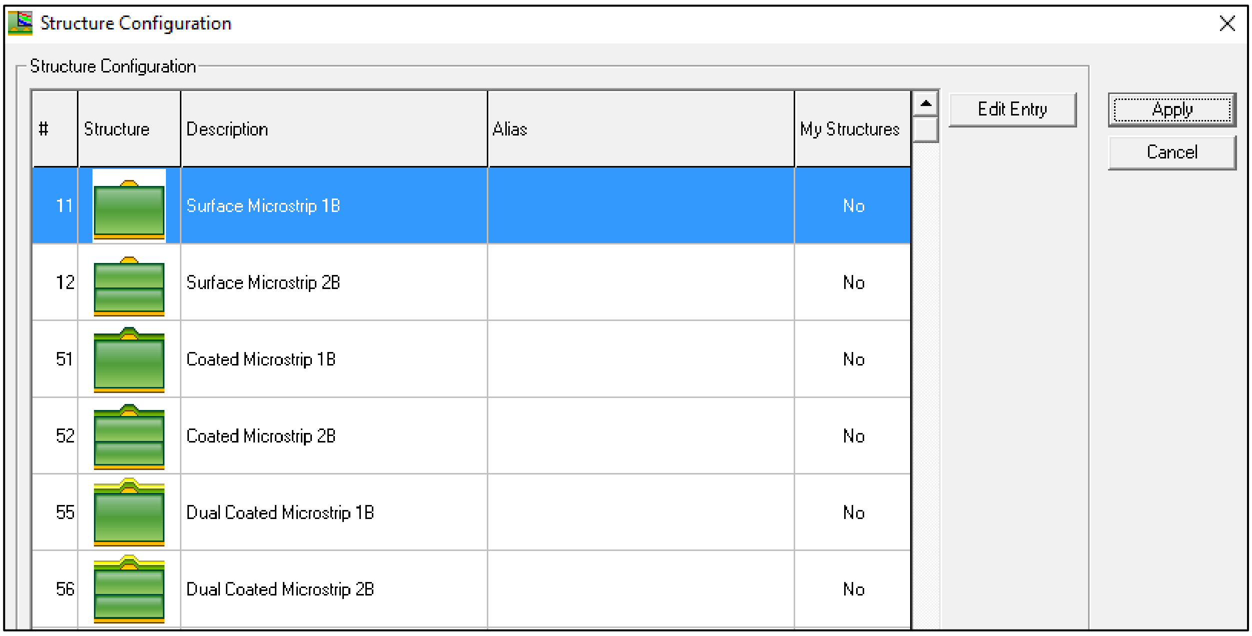
Rebuild and recalculate to refresh the stack. It may also be necessary to round some dimensions (e.g. dielectric heights) to the nearest practical values and recalculate the impedance

Creating a custom list of structures

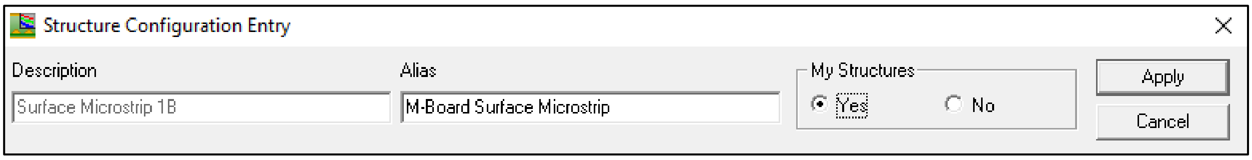
Use the My Structures function to create and edit a custom list of structures (the My Structures group).



Choose the Configuration | Structures command to display the complete list of structures – select the structure to be edited and click Edit:

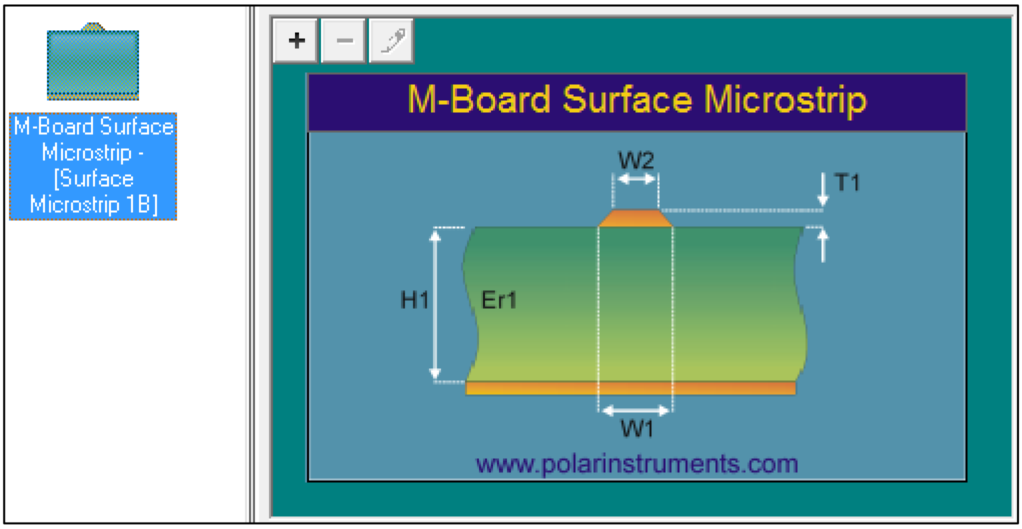


Add the descriptive text to provide a label for the structure in the Alias field:

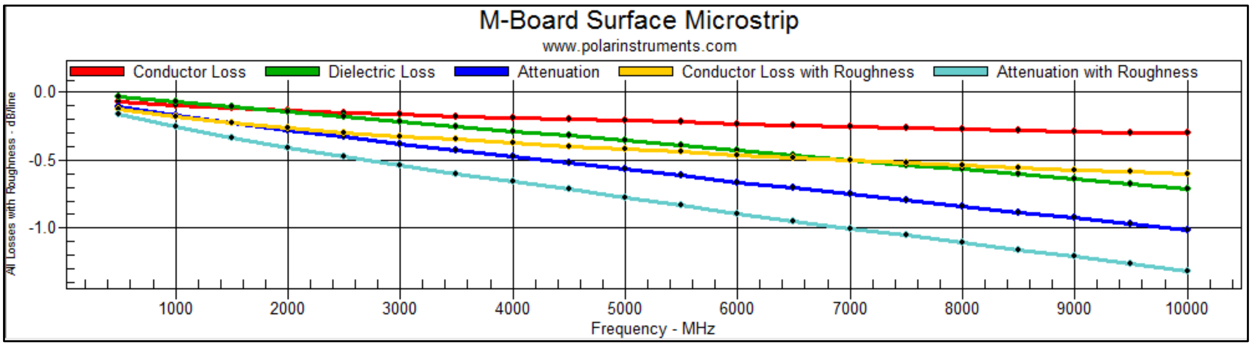


Click Yes to add to the My Structures list then click Apply.

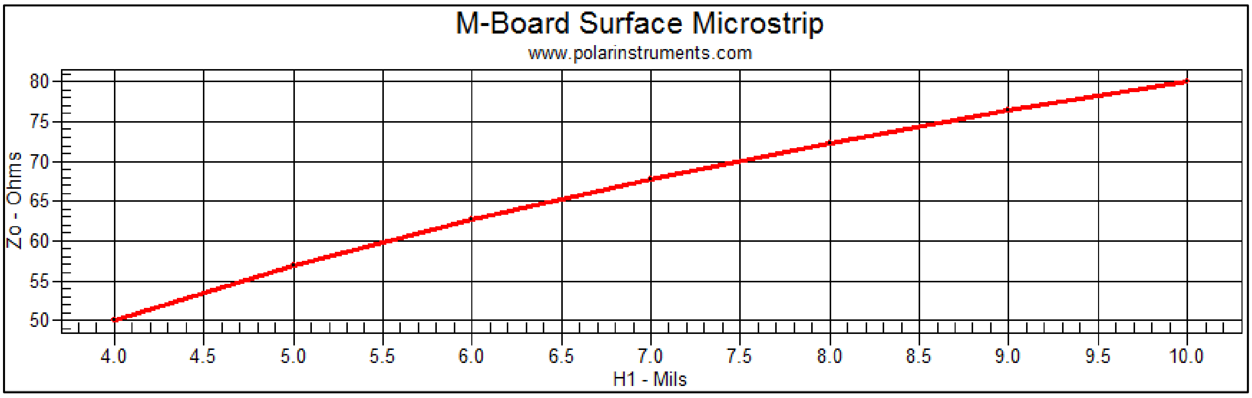
The structure is added to the list of custom structures and is displayed with the edited title.



The structure alias is applied to both structure image titles and graph titles in the frequency dependent tab



and the sensitivity analysis tab.



Printing results

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

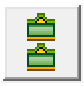
Using Si Projects

Si Projects allows the designer to store groups of related structures or rapidly to copy and paste all the impedance structures within a stackup 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.

Working with Si Projects in Speedstack

The Si 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.

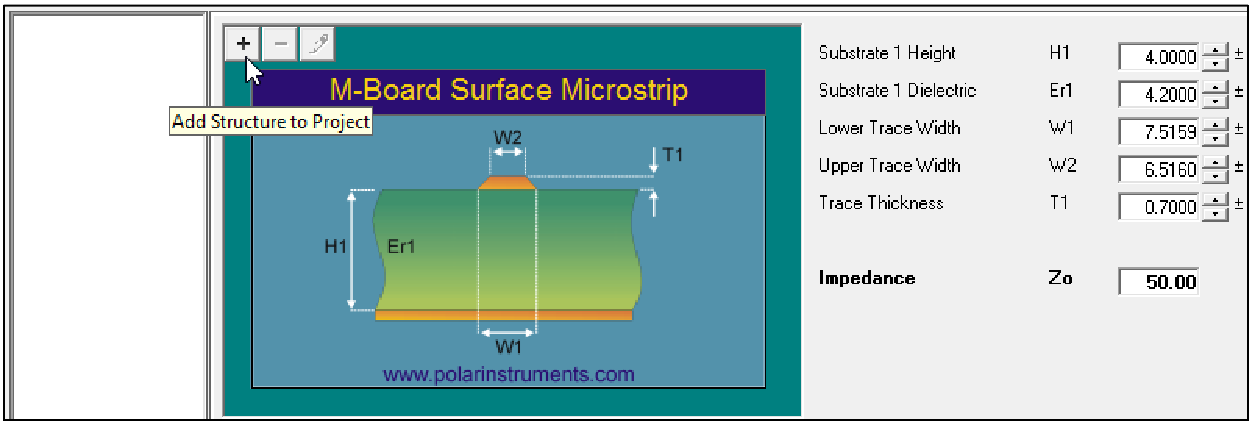
Creating new projects



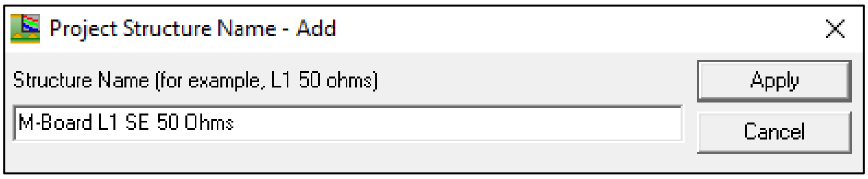
Projects button

Click the Projects button to create a new project.

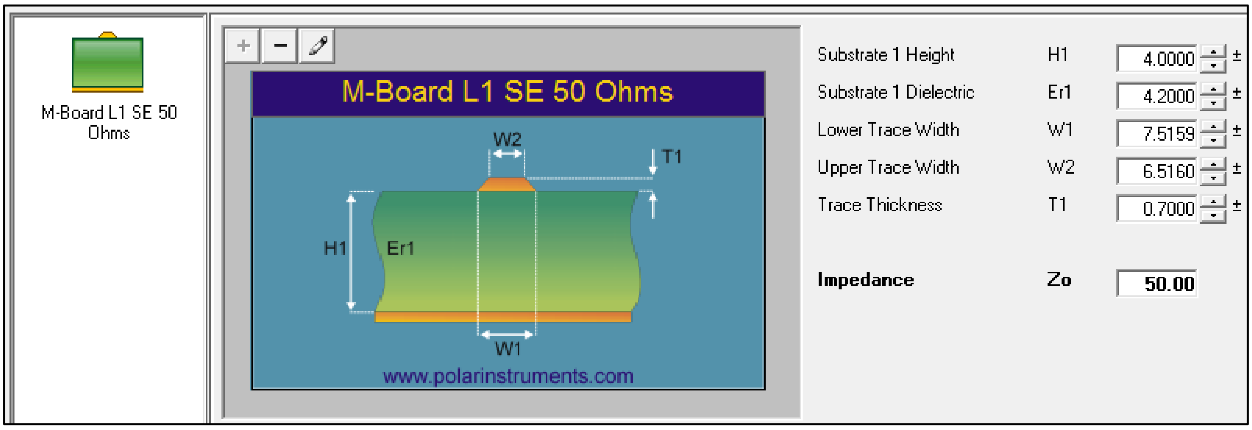
With the structure displayed, click the Add Structure to Project button



Supply a descriptive name for the project structure:



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

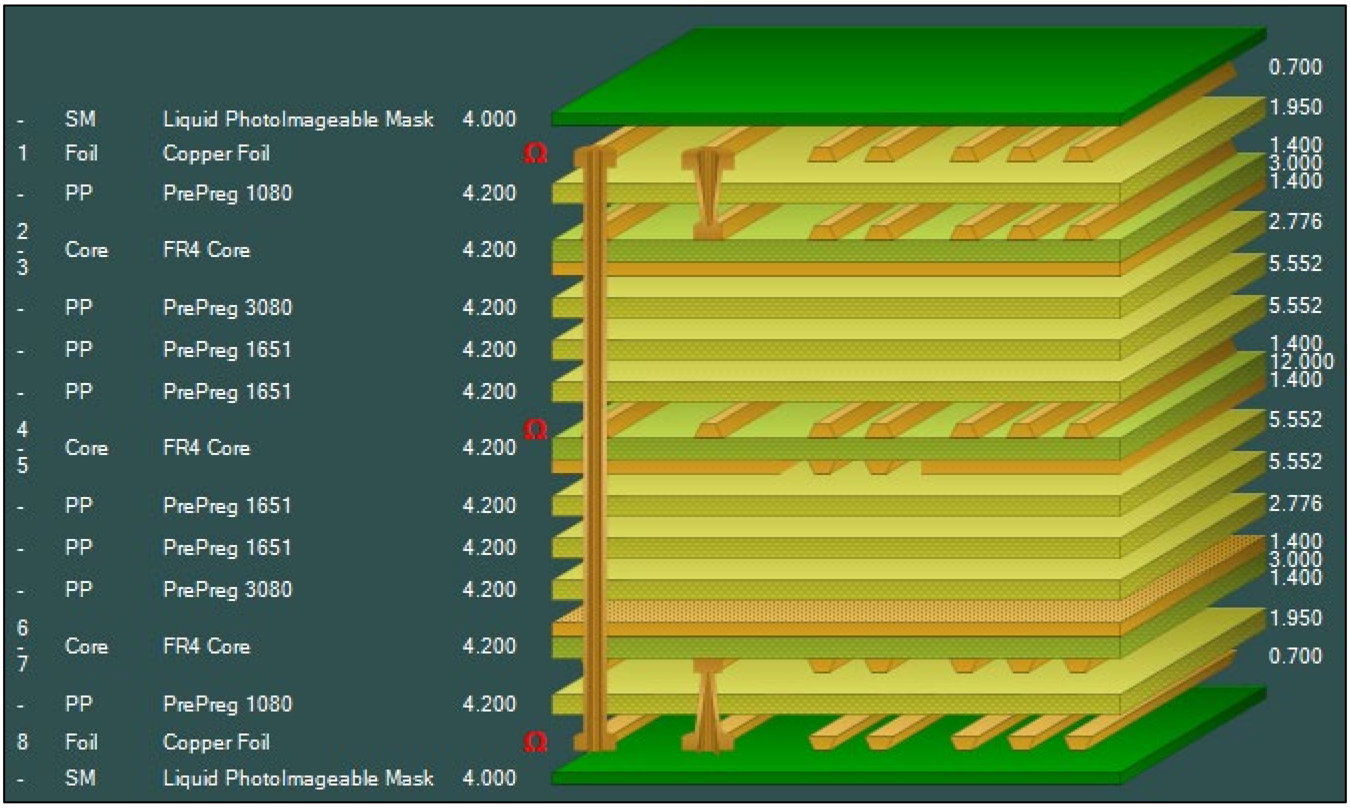
Speedstack / Field Solver data transfer via Si Projects

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.

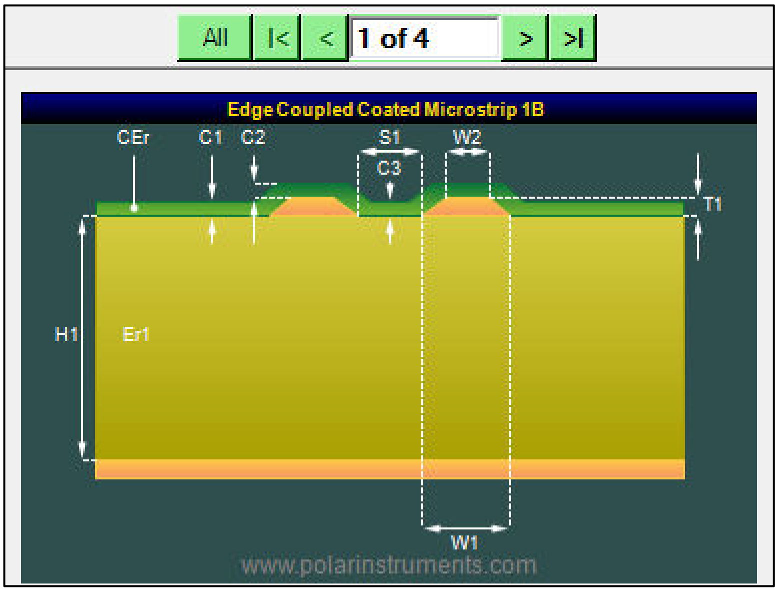
Si Projects allows groups of structures to be saved and recalled in Si8000m/Si9000e and entire stackups of structures to be pasted from Speedstack into the field solver with just a few clicks of the mouse – the toolbar option copies a group of structures from Speedstack and places them onto the clipboard,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

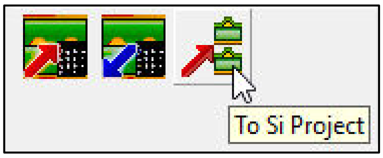


Click Speedstack's Controlled Impedance tab to display the structures.

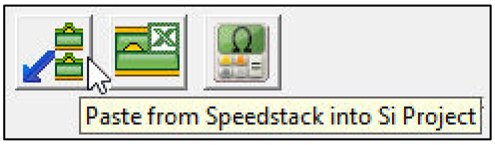


Use the toolbar buttons in the Speedstack and the field solver interfaces to transfer the structures to the field solver.

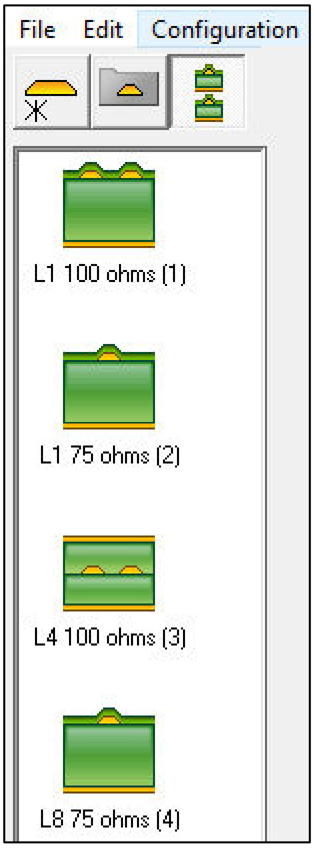
Within Speedstack click the To Si Project toolbar button:



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



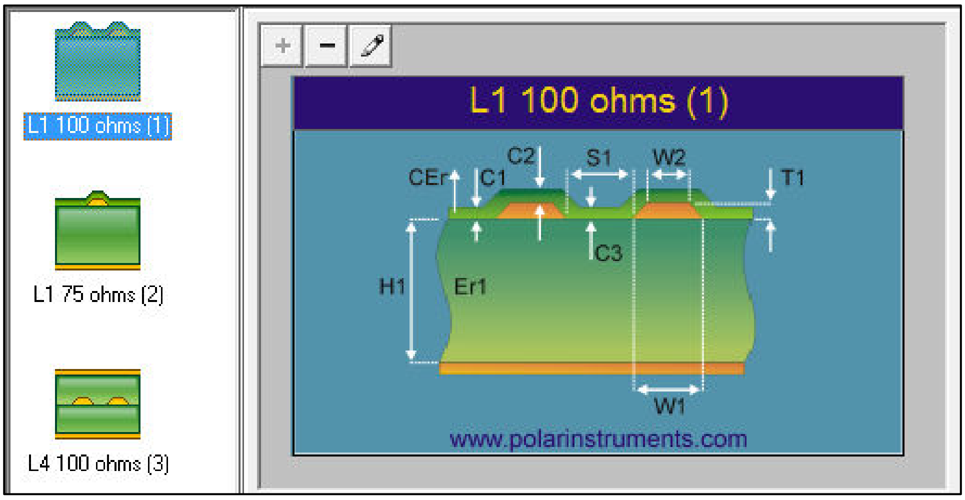
The set of structures appears in the Project window.



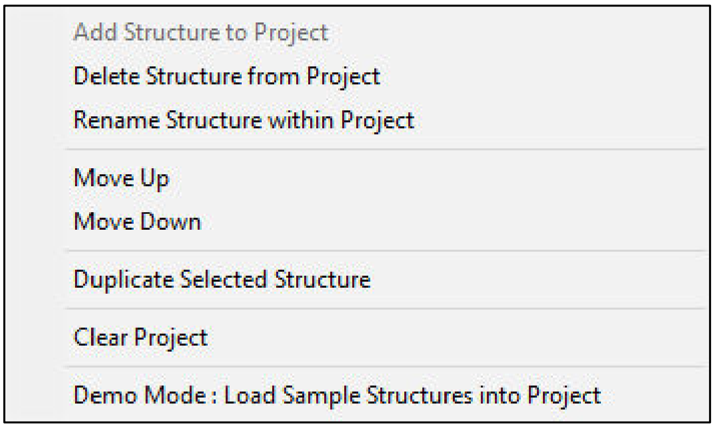
L*n* indicates the layer number in Speedstack.

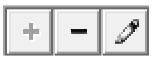
Adding/deleting and modifying structures

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.





Add/Remove Structure

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.

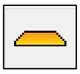
With a structure selected the structure parameters can be modified as required and the impedance recalculated.

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.

Project graphing (Si9000e only)

With a project created, the Project Graphing function calculates all the results for a group of structures contained in the project and then plots the selected data series on the same graph. This allows comparison of results from similar structures, especially with frequency dependent calculations where changing just one or two parameters can have significant impact. The example below will graph the loss curves for a single ended structure, a 50 Ohm surface microstrip, for different values of loss tangent

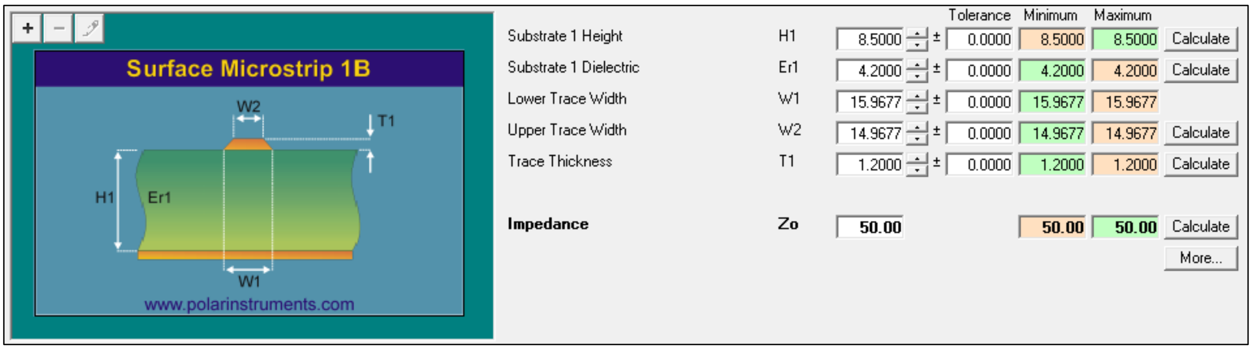
Creating the new project



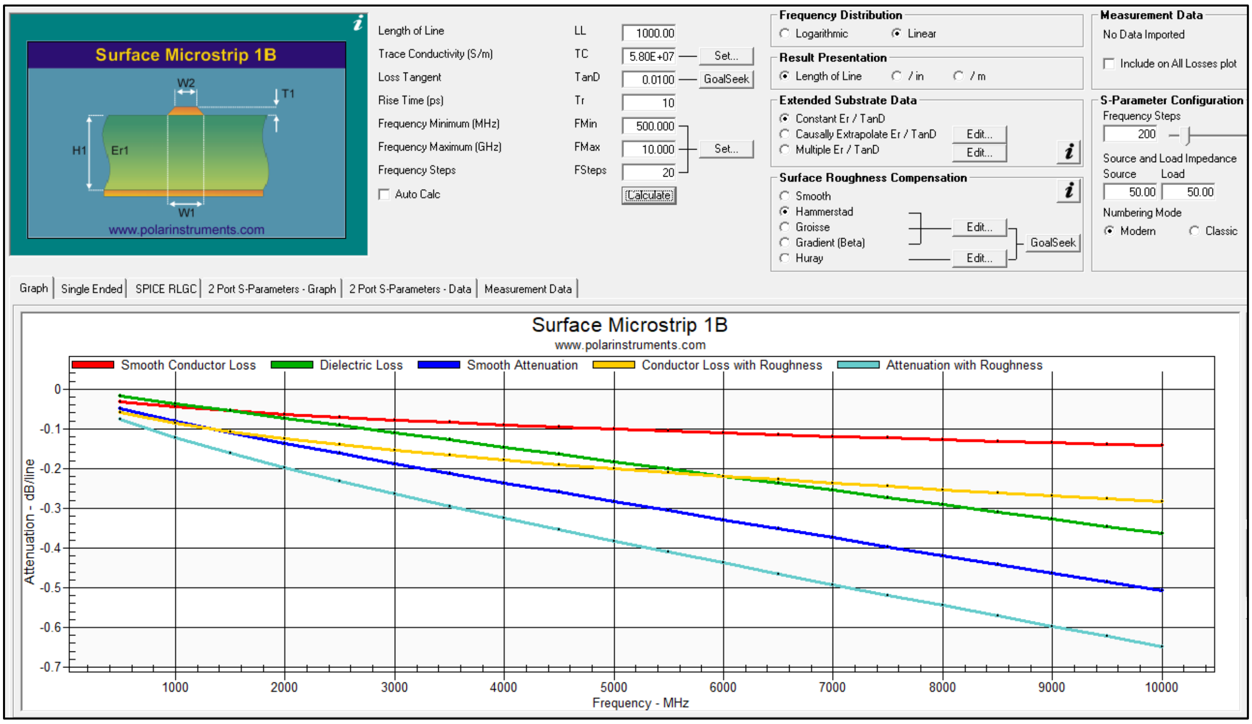
Single Ended Structures

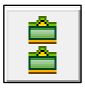
Choose Single Ended Structures and then from the Structure Bar choose Surface Microstrip 1B

Specify a nominal impedance of 50 Ohms and, if necessary, goal seek (for example on trace width) to achieve the target.

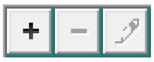


Switch to the Frequency Dependent Calculation tab and specify a value for TanD of 0.010. Click Calculate to display the losses for a Tan D of 0.010.





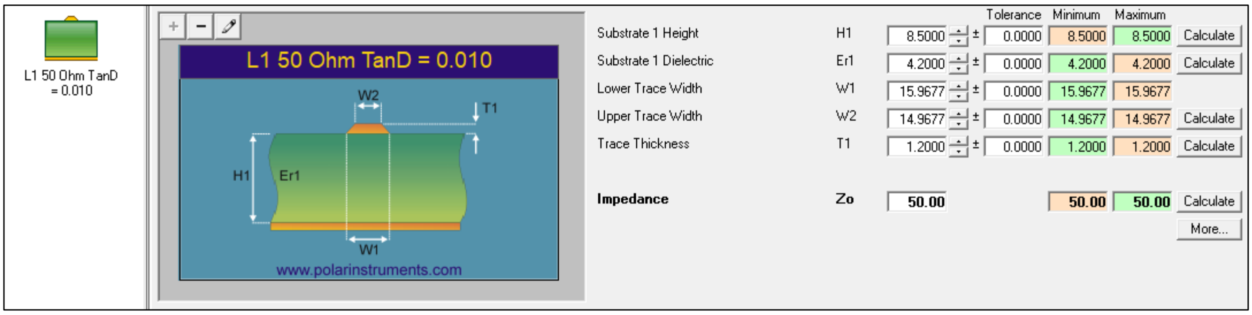
Project button



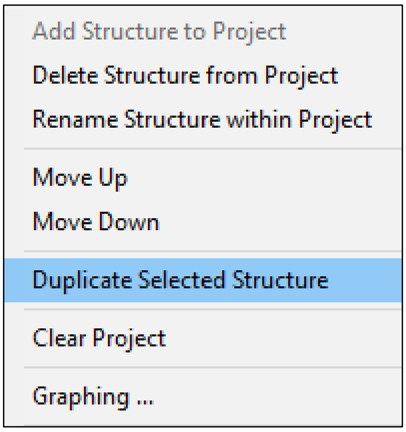
Add/Delete/Rename Structure

Click the Project button and add the structure to the project – either use the Add Structure to Project button or right click into the Structure Bar.

Supply a descriptive name for the structure within the project, in this example, *L1 50Ohm TanD = 0.010.* The structure is renamed and added to the project.

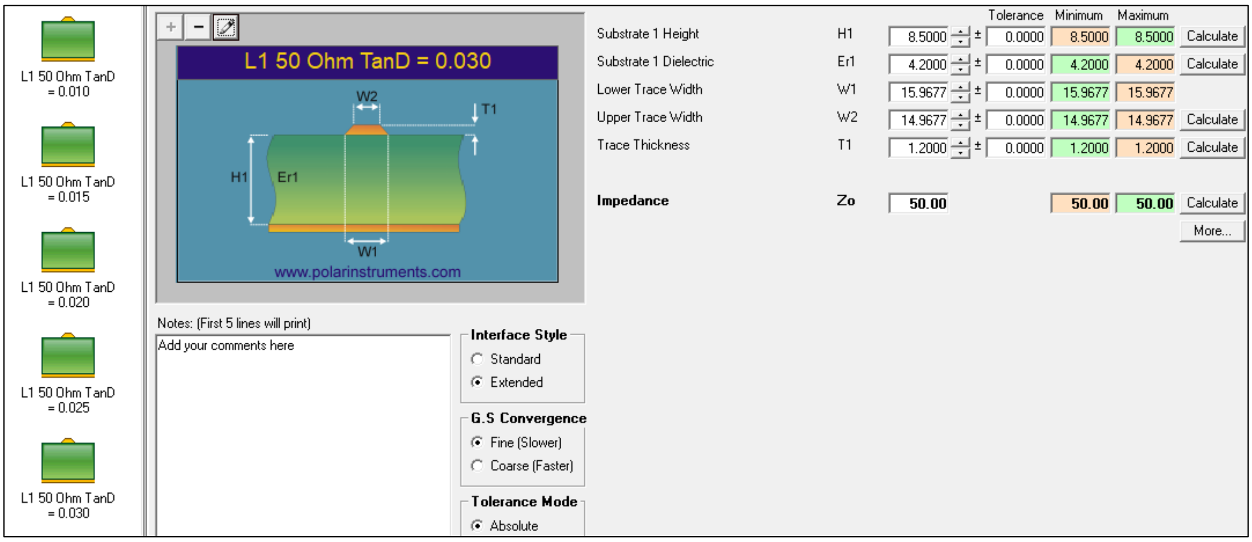


Save the project under a suitable name.
Right click the new structure within the project's Structure Bar and choose Duplicate Selected Structure.

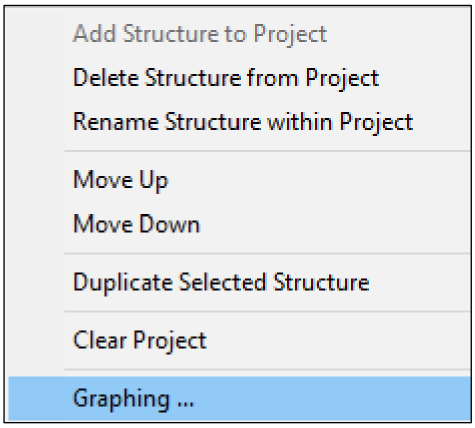


Rename the new structure *L1 50 Ohm TanD = 0.015.*

Repeat for the other structures for the series of values of TanD. The new structures are shown in the Structure Bar

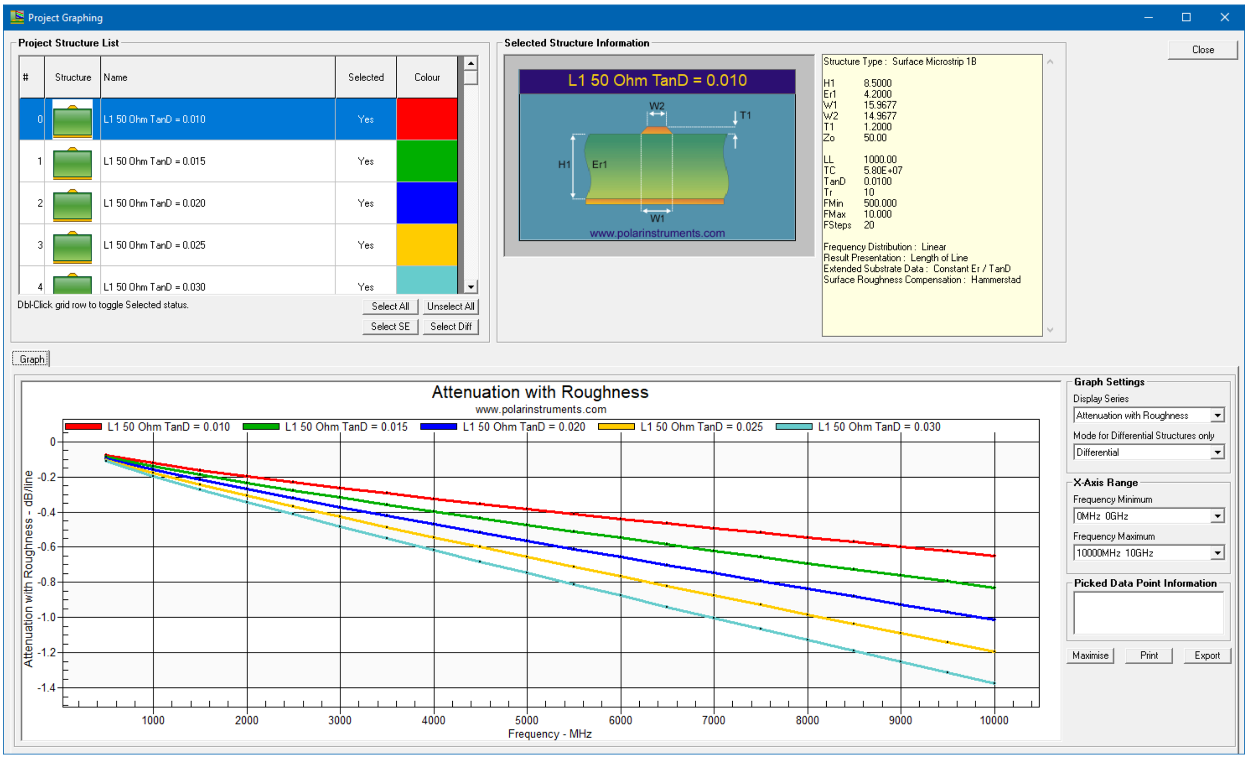


Right click the Structure Bar and choose Graphing…



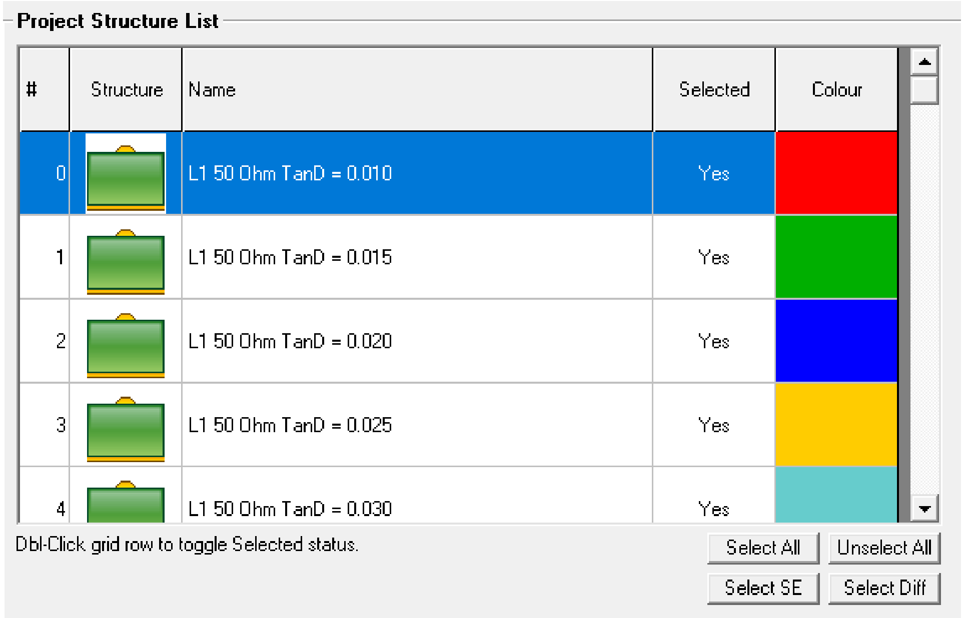
The Si9000e performs a full frequency recalculation of all structures within the project and displays the results for all the project’s structures on a single graph. Structure parameters are shown in the panel to the right of the structure graphic.

The structures are graphed in the colours indicated in the *Project Structure List*.



Project Structure List

Use the Project Structure List to choose which structures from the project are plotted. In the graphic below all the project structures are selected for display – indicated by Yes in the Selected column. Double-click the grid row to select / deselect each structure.

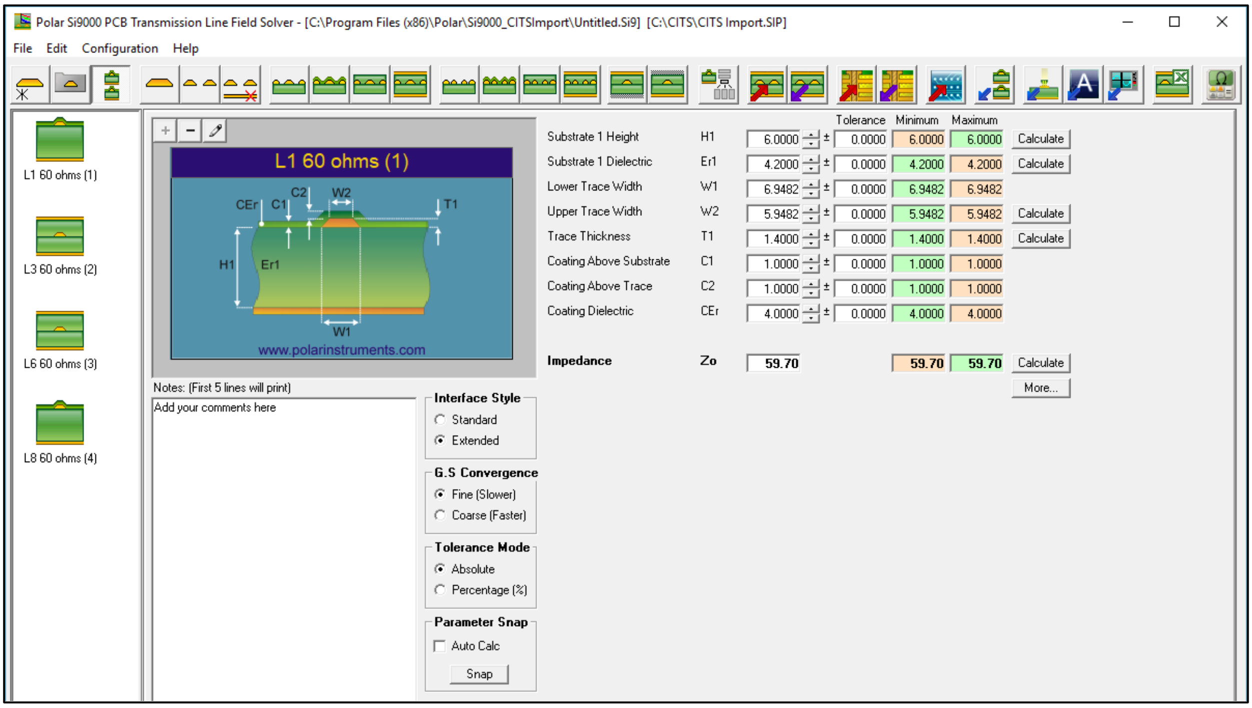


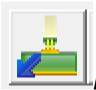
Click Select SE to display all single-ended structures; click Select Diff to display all differential structures.

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.

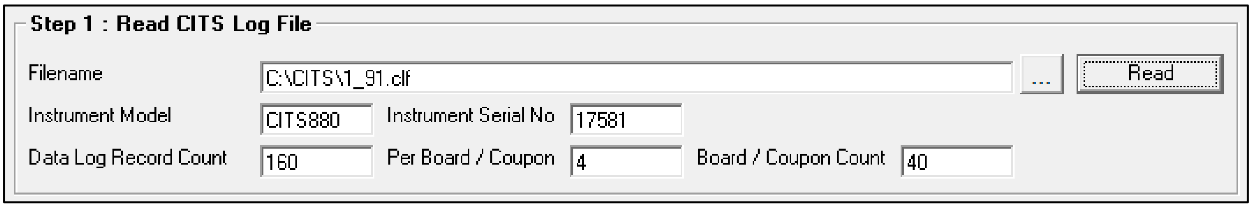



Import measurement data from CITS

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

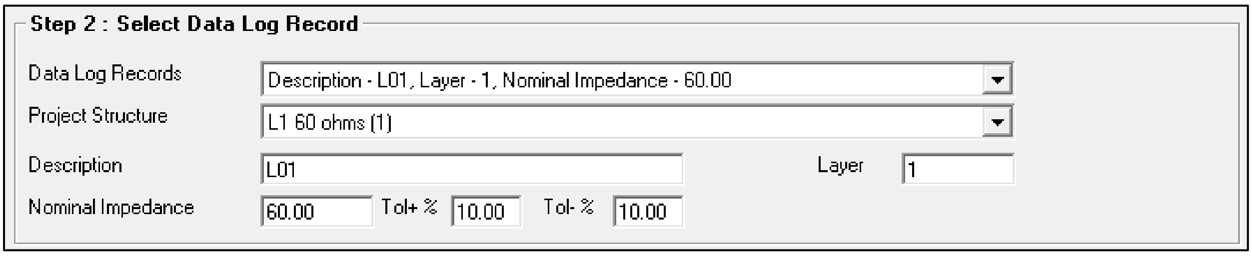


Summary data includes the CITS model and serial number along with the total number of data records, number of coupons per board and number of boards in the log file.

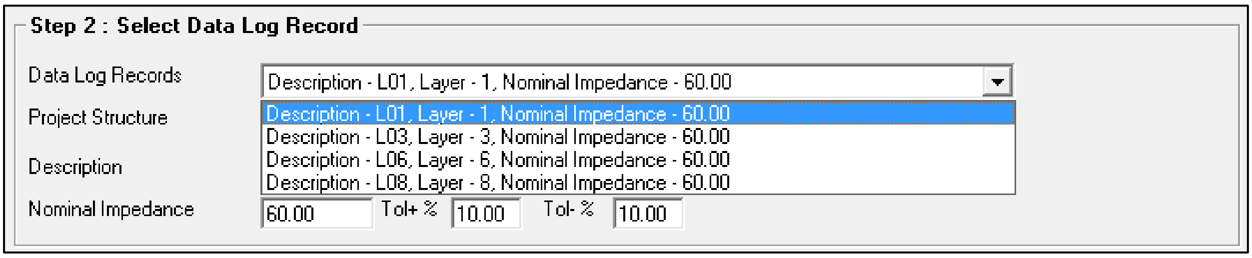
(In the example above the Data Log Record Count, 160, reflects the 40 coupons with 4 tests per coupon.)

Step 2 Selecting the Data Log record

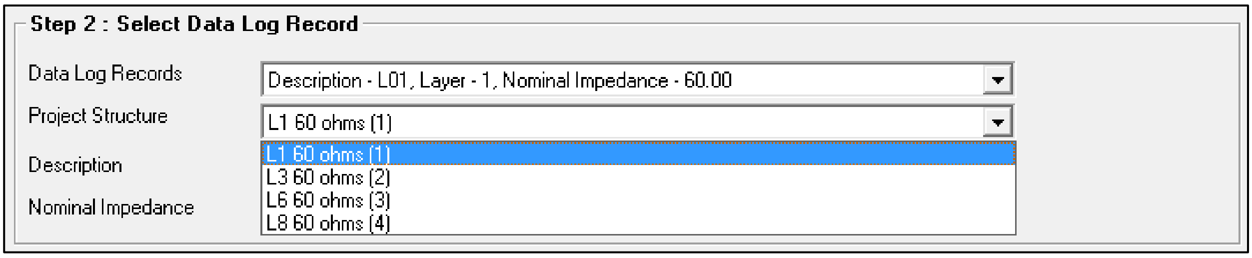
When the log file is read the measured data for each structure in the project may be selected for display and compared with the associated modelled impedance.



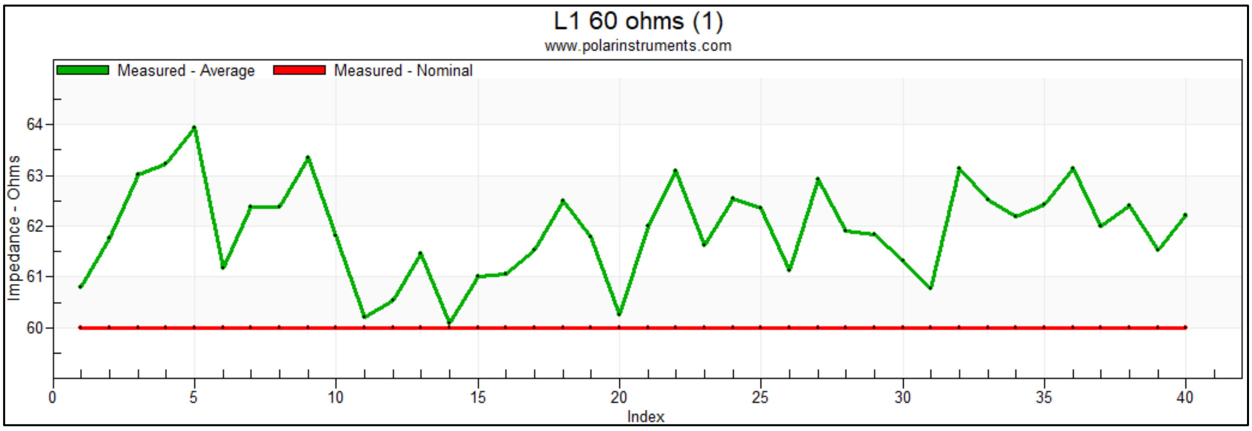
Click the Data Log Records drop-down to select the data log for each structure.



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

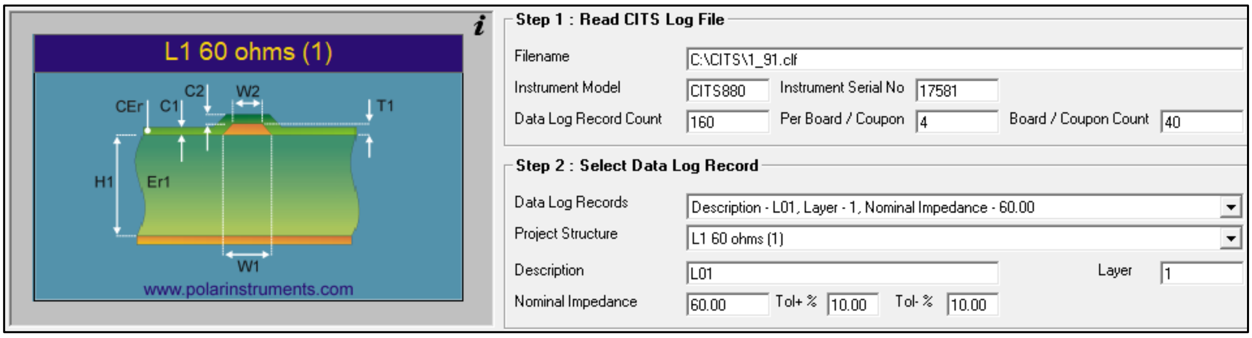


The selected structure graphic is displayed along with the resulting chart displaying the logged data for the structure.



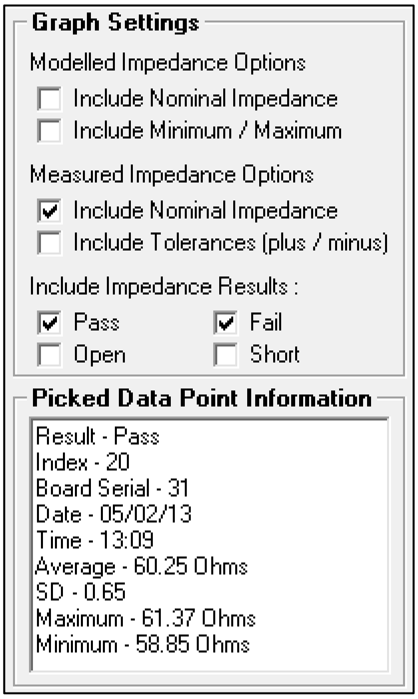
The chart above displays the logged measured data for each board along with the nominal impedance value.

The structure graphic for the project structure is shown alongside the log summary

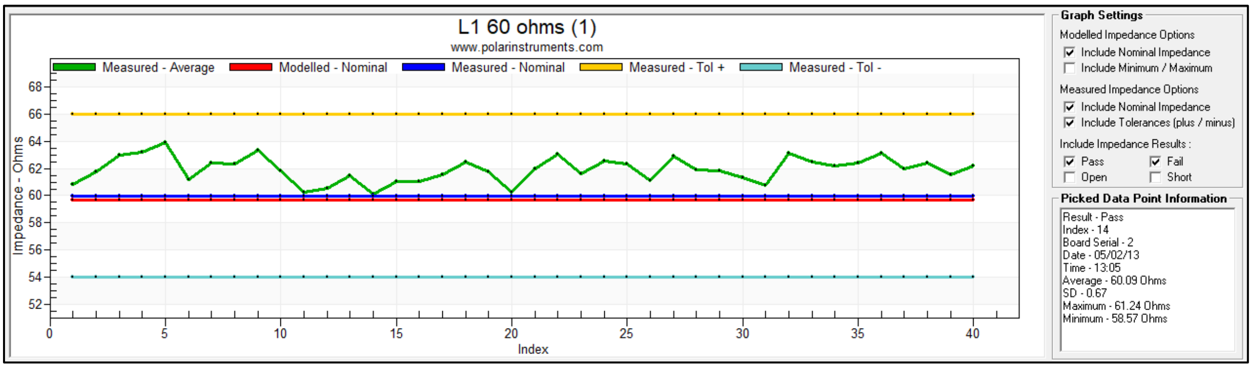


Graph settings

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

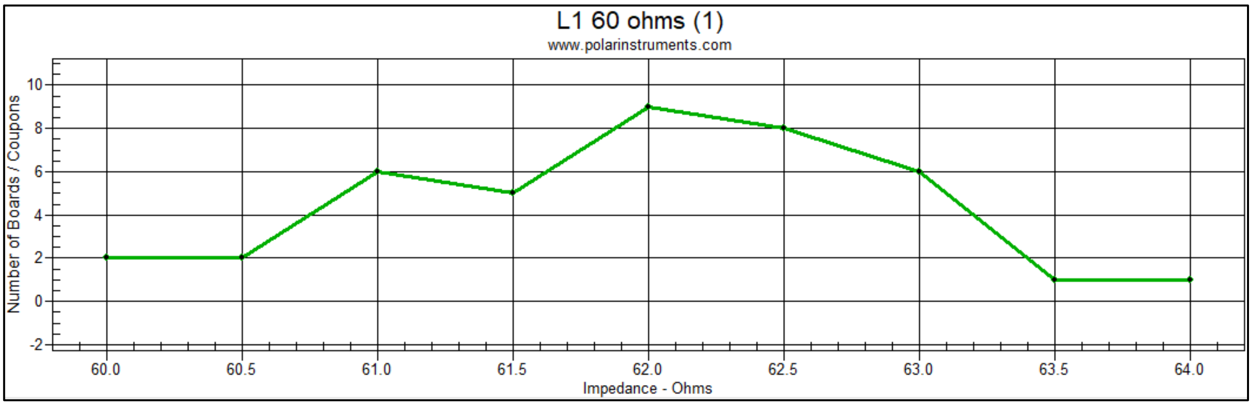
Outlying or invalid data values may be excluded from the chart; open or short circuit readings that occurred during testing will typically not be regarded as valid for logging so will generally be excluded.

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.)

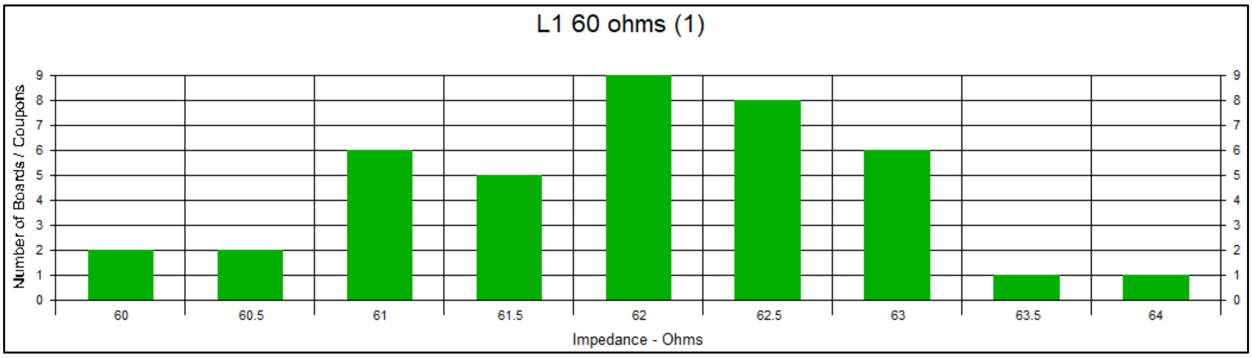
Ticking or clearing the Pass or Fail checkboxes will display or exclude PASS or FAIL data (i.e. results exceeding the tolerance limits) respectively. Displaying failed readings only would allow detailed analysis of the failed tests.

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:

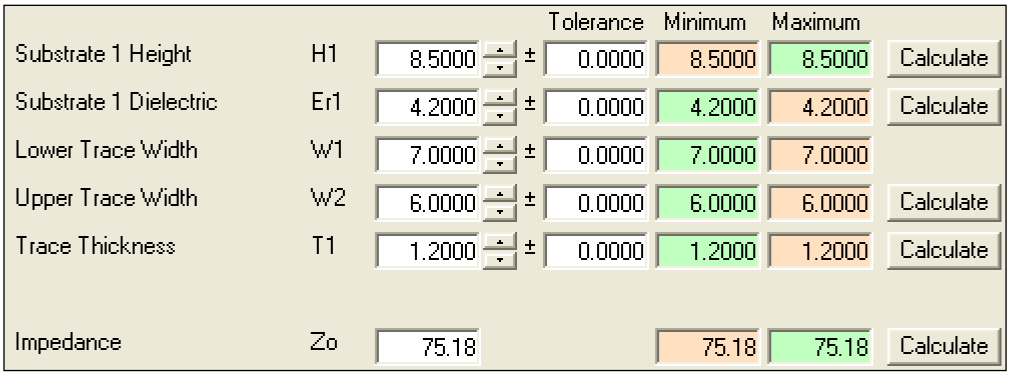
- Graphing impedance against any varying structure parameters
- Setting a target impedance line on the graphs
- Exporting the graph data to clipboard for use in Excel
- Graphing impedance for single-ended structures
- Graphing differential structures: Odd mode / Even / Differential / Common / All
- Exporting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

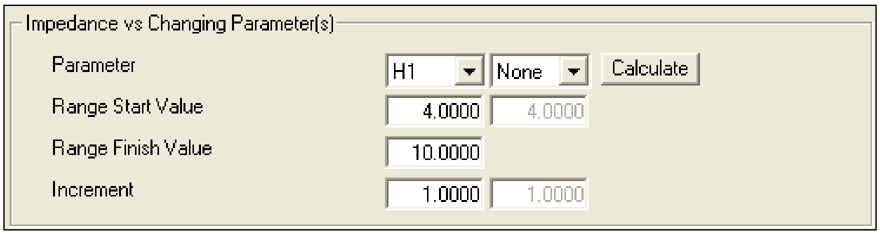
Varying a single parameter

Charting Z_0 as H_1 varies

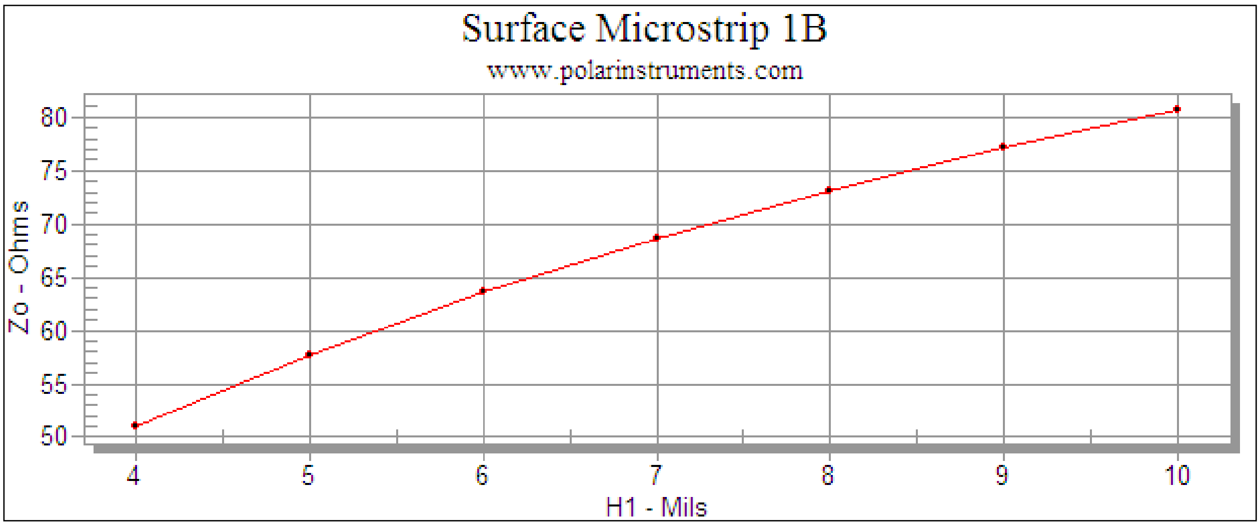
In this example, given the values below in the Lossless Calculation tab, switch to the Sensitivity Analysis tab (all values in mil.)



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.

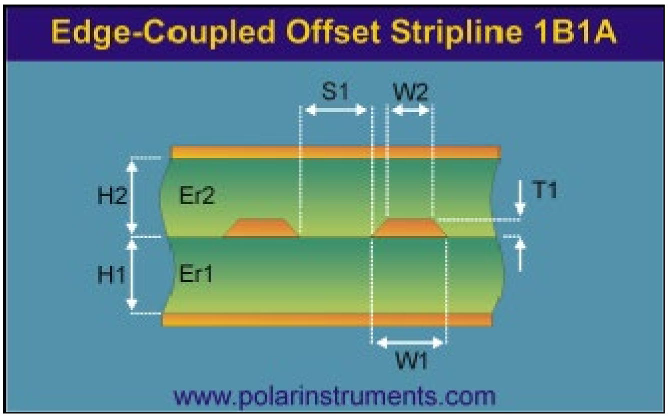


Varying multiple parameters

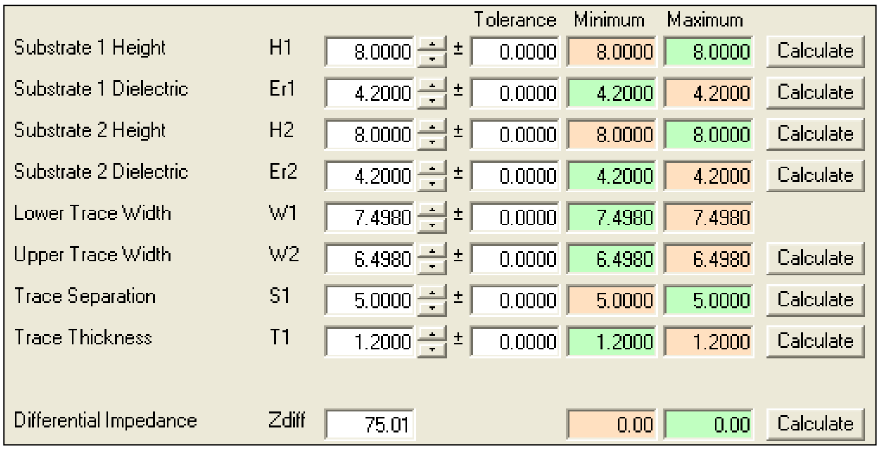
Charting Z_0 as H_1 and H_2 vary

For this example, choose an edge coupled stripline and chart the variation in Z_0 as the two substrate heights are varied.

Choose the structure Edge-Coupled Offset Stripline 1B1A

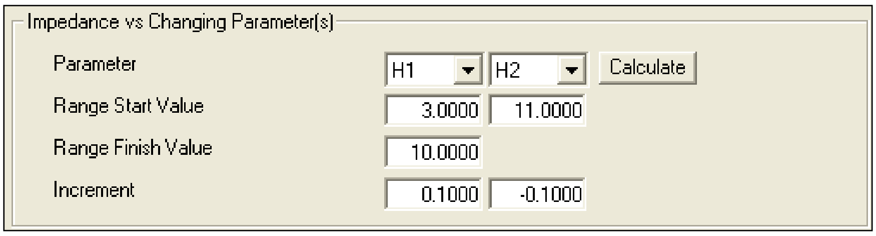


Enter the parameters below and goal seek on trace width for a differential impedance of 75 Ohms.

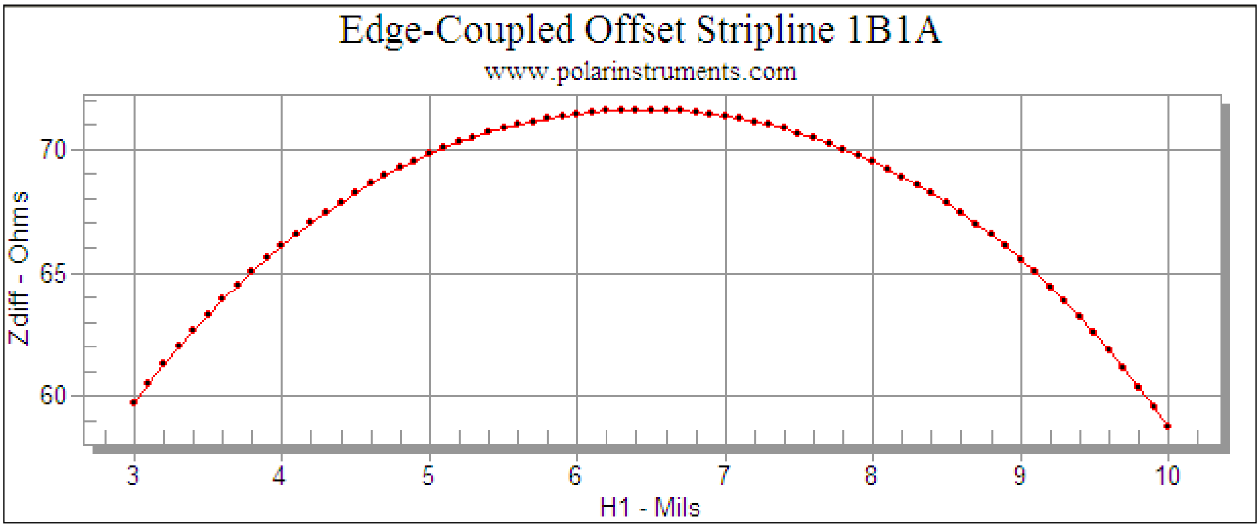


Note the nominal value for both H_1 and H_2 of 8 mil.

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)

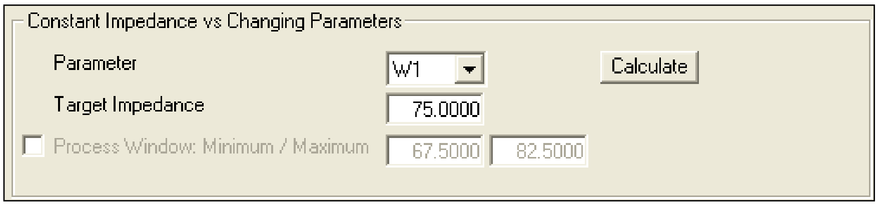


Click Calculate – the result is shown below

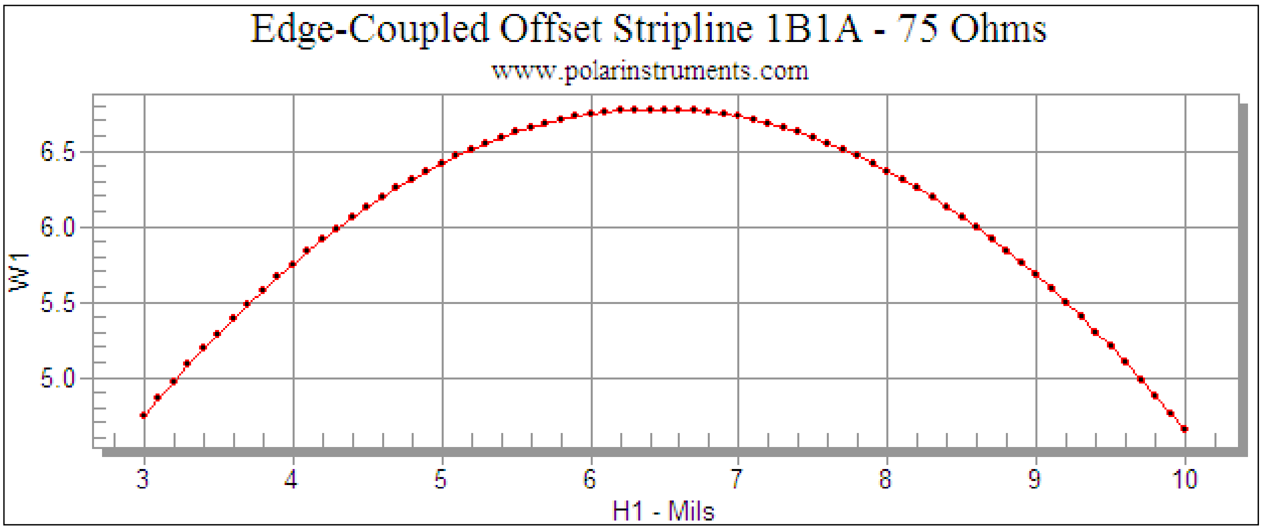


Constant impedance v changing parameters

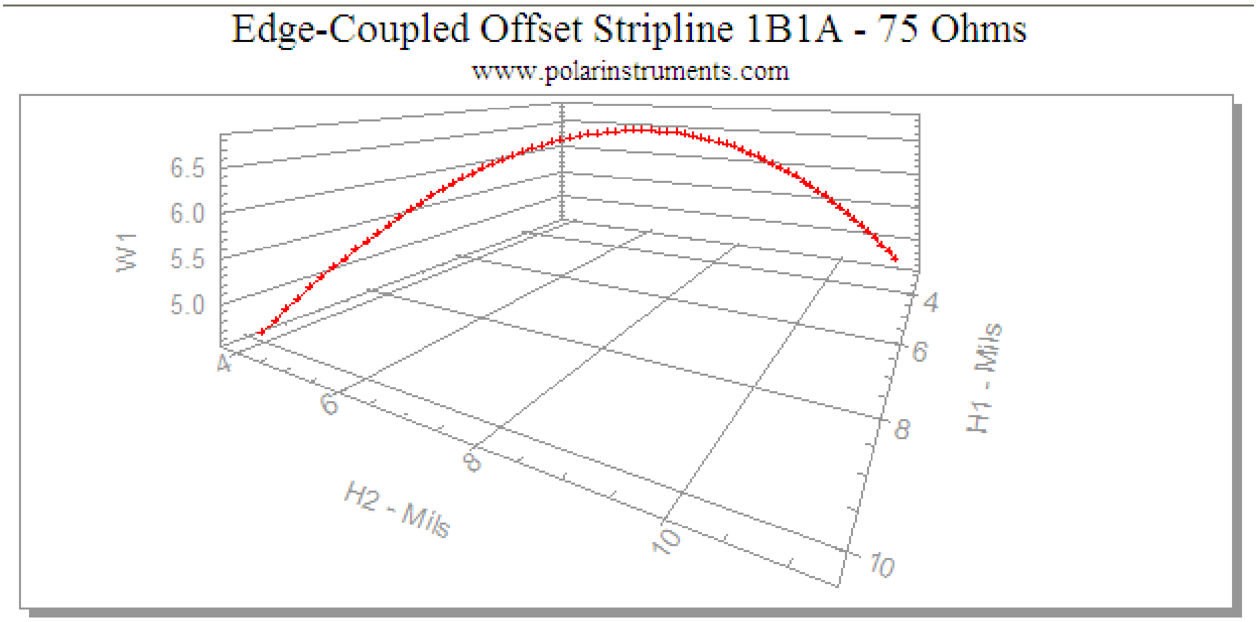
In this example, the differential impedance is held constant at 75 Ohms and the trace width, W_1, varied as H_1 and H_2 vary. Choose the varying parameter as W_1, set the target impedance as 75 Ohms and click Calculate below



The result is shown below.

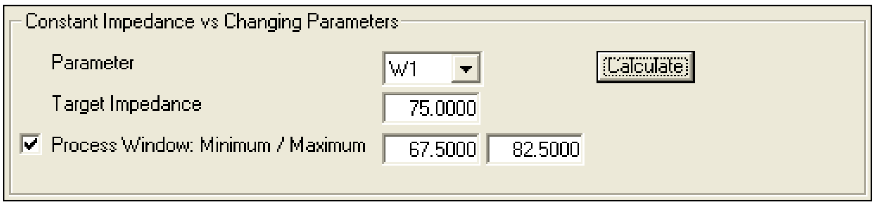


Choose the 3D option button to display the result in three dimensions, i.e. as all three parameters vary.

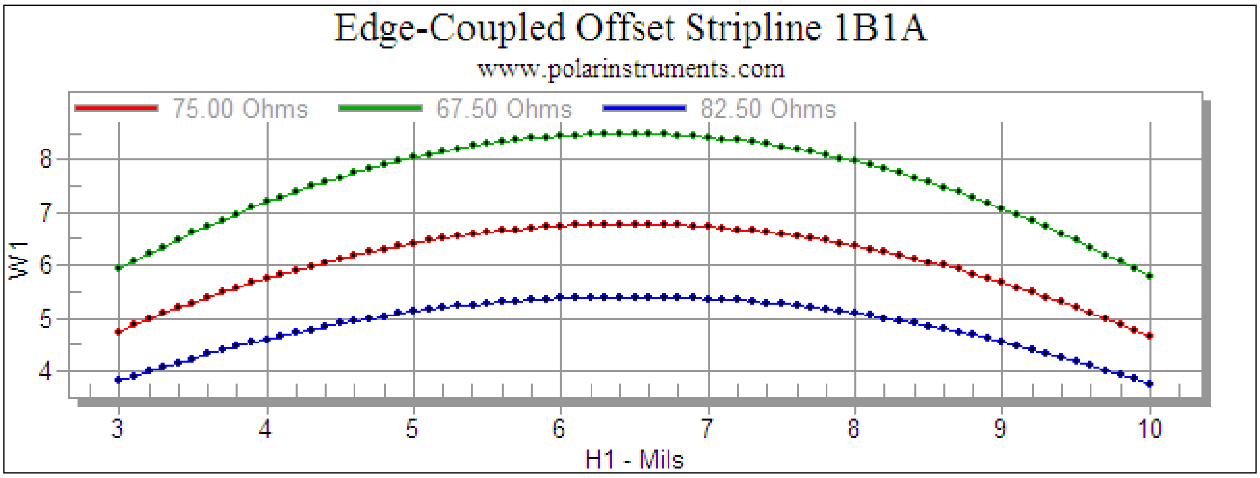


Process Window: Minimum / Maximum

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, W_1 as H_1 and H_2 vary. Choose the varying parameter as W_1, set the target impedance as 75 Ohms and the upper and lower limits as shown below and click Calculate



The results are shown below.



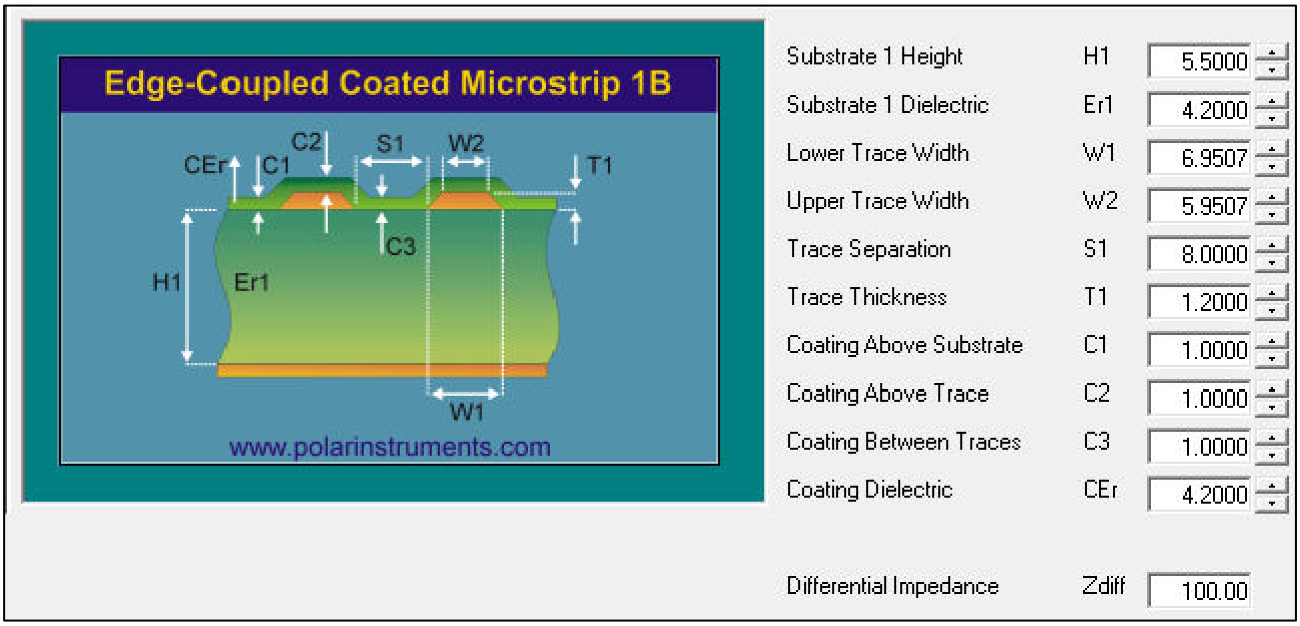
Using Sensitivity Analysis to graph crosstalk

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, Zdiff, should equal close to 100 ohms.

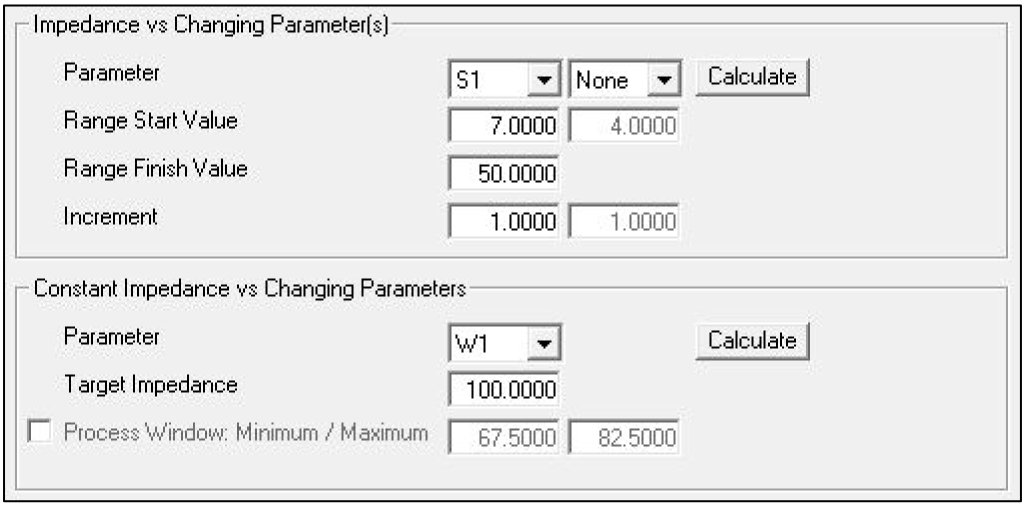


Using impedance v changing parameters

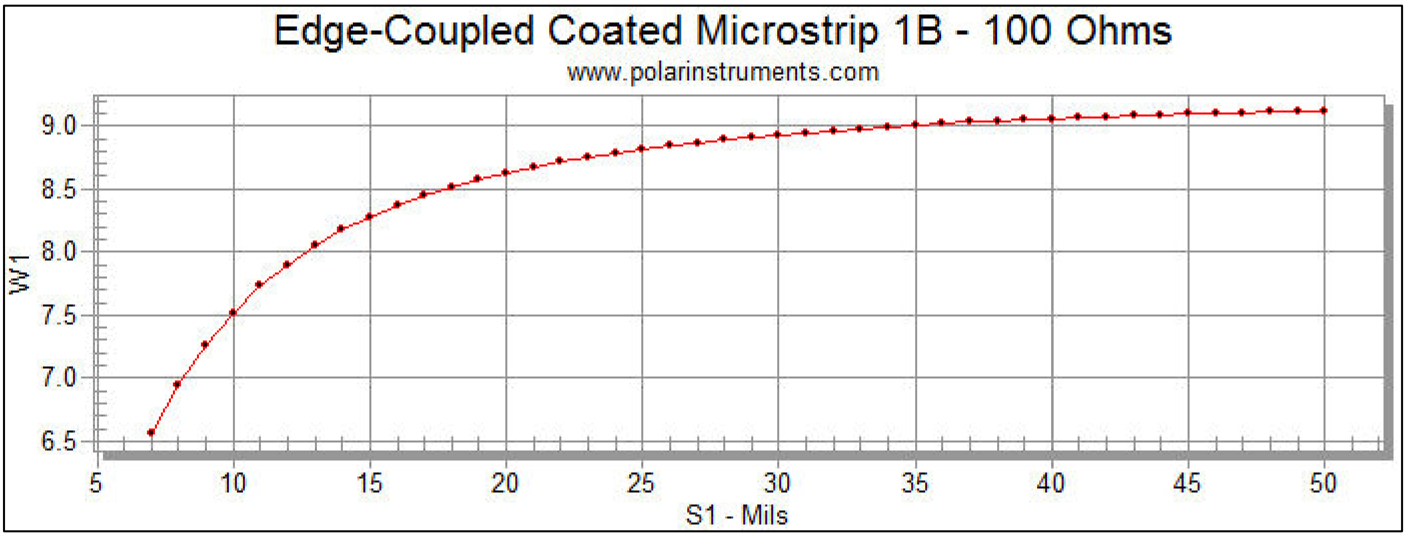
Switch to the Sensitivity Analysis tab.

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 v trace separation over the selected range of values of S1 while maintaining a constant value of 100 ohms differential impedance.



Viewing tabular results

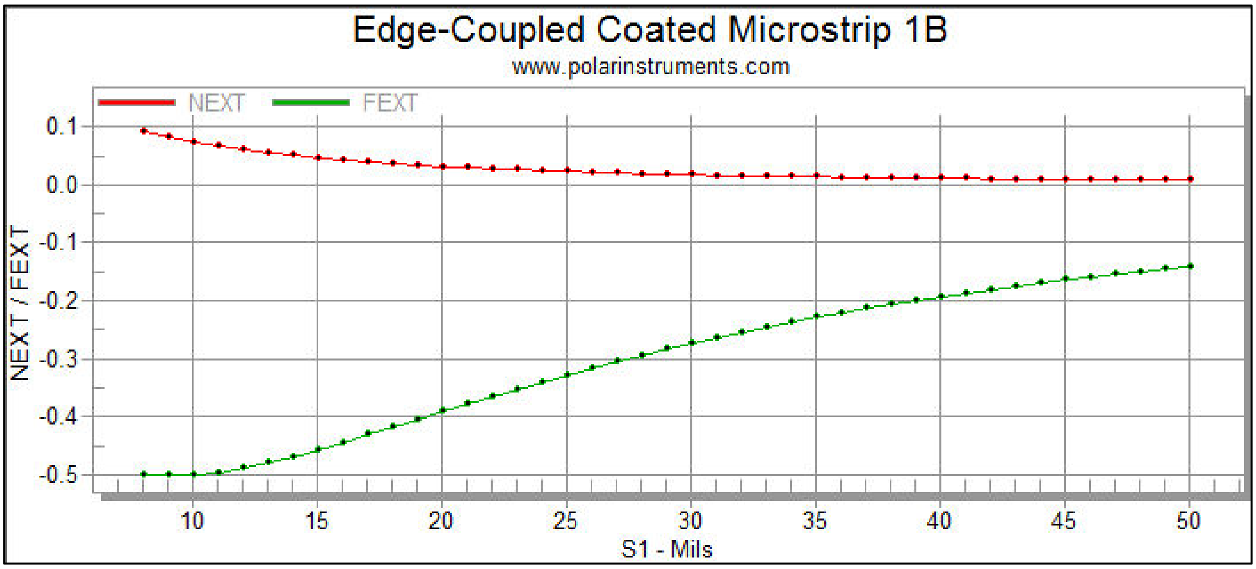
A subset of the results as trace width and separation vary is shown below The results tab shows W1 / W2 / S1 changing; the impedance, ZDiff, calculates to 100 ohms but the NEXT / FEXT values for each W1 / W2 / S1 combination of parameter values are displayed.

This data can be exported to other tools (for example, Microsoft Excel®) for further analysis.

W1	W2	S1	Zodd	Zeven	Zdiff	Zcommon	Kb (NEXT)	Kf s/in	FEXT
6.5549	5.5549	7.0000	49.9957	67.2289	99.9915	33.6144	7.3505E-02	-4.6179E-12	-4.6179E-01
6.9496	5.9496	8.0000	50.0024	64.3912	100.0048	32.1956	6.2892E-02	-4.6273E-12	-4.6273E-01
7.2637	6.2637	9.0000	50.0002	62.2024	100.0004	31.1012	5.4376E-02	-4.5454E-12	-4.5454E-01
7.5179	6.5179	10.0000	50.0035	60.5133	100.0070	30.2566	4.7548E-02	-4.4288E-12	-4.4288E-01
7.7272	6.7272	11.0000	50.0039	59.1502	100.0077	29.5751	4.1896E-02	-4.2780E-12	-4.2780E-01
7.9007	6.9007	12.0000	49.9983	58.0342	99.9965	29.0171	3.7192E-02	-4.1125E-12	-4.1125E-01
8.0472	7.0472	13.0000	50.0001	57.1256	100.0001	28.5628	3.3258E-02	-3.9400E-12	-3.9400E-01
8.1728	7.1728	14.0000	49.9968	56.3645	99.9935	28.1823	2.9935E-02	-3.7651E-12	-3.7651E-01
8.2775	7.2775	15.0000	49.9976	55.7251	99.9952	27.8625	2.7087E-02	-3.5920E-12	-3.5920E-01
8.3642	7.3642	16.0000	50.0017	55.1835	100.0033	27.5917	2.4632E-02	-3.4203E-12	-3.4203E-01
8.4450	7.4450	17.0000	49.9984	54.7128	99.9968	27.3564	2.2511E-02	-3.2548E-12	-3.2548E-01
8.5138	7.5138	18.0000	49.9977	54.3079	99.9953	27.1540	2.0662E-02	-3.0957E-12	-3.0957E-01
8.5736	7.5736	19.0000	49.9967	53.9542	99.9933	26.9771	1.9035E-02	-2.9428E-12	-2.9428E-01
8.6274	7.6274	20.0000	49.9962	53.6431	99.9925	26.8216	1.7594E-02	-2.7963E-12	-2.7963E-01
8.6723	7.6723	21.0000	50.0026	53.3797	100.0052	26.6899	1.6333E-02	-2.6606E-12	-2.6606E-01

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.



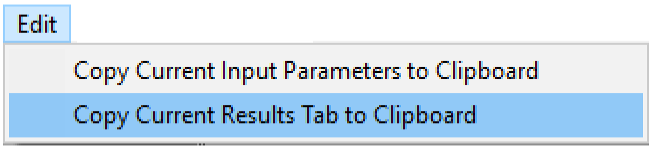
Copying Field Solver data to external programs

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.

Graph Results

H1	Er1	W1	W2	T1	Zo	Calc Success
4.0000	4.2000	3.0468	2.0468	1.2000	74.9959	Yes
5.0000	4.2000	3.9230	2.9230	1.2000	74.9960	Yes
6.0000	4.2000	4.8053	3.8053	1.2000	75.0003	Yes
7.0000	4.2000	5.6965	4.6965	1.2000	74.9909	Yes
8.0000	4.2000	6.5908	5.5908	1.2000	74.9989	Yes
9.0000	4.2000	7.4910	6.4910	1.2000	74.9953	Yes
10.0000	4.2000	8.3882	7.3882	1.2000	74.9991	Yes

From the Edit menu choose Copy Current Results Tab to Clipboard



The current results are copied from the field solver and located on the Windows clipboard.

The result tables may then be pasted to a suitable location in a spreadsheet or database (e.g. a Si8000m / Si9000e Excel workbook or Si8000m / Si9000e SiExcelExpert or Si8000m / Si9000e SiExcelExpert64 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 sufficient space exists on the target worksheet so that no important data are overwritten in the process.

The operator can use Excel's **Paste** Function command to insert the associated function using the pasted Quick Solver parameter values as arguments.

Copying the structure to CGen


Copy Structure to CGen Coupon Generator

Click the Copy Structure to CGen Coupon Generator button to

Using Sensitivity Analysis to graph multiple impedances

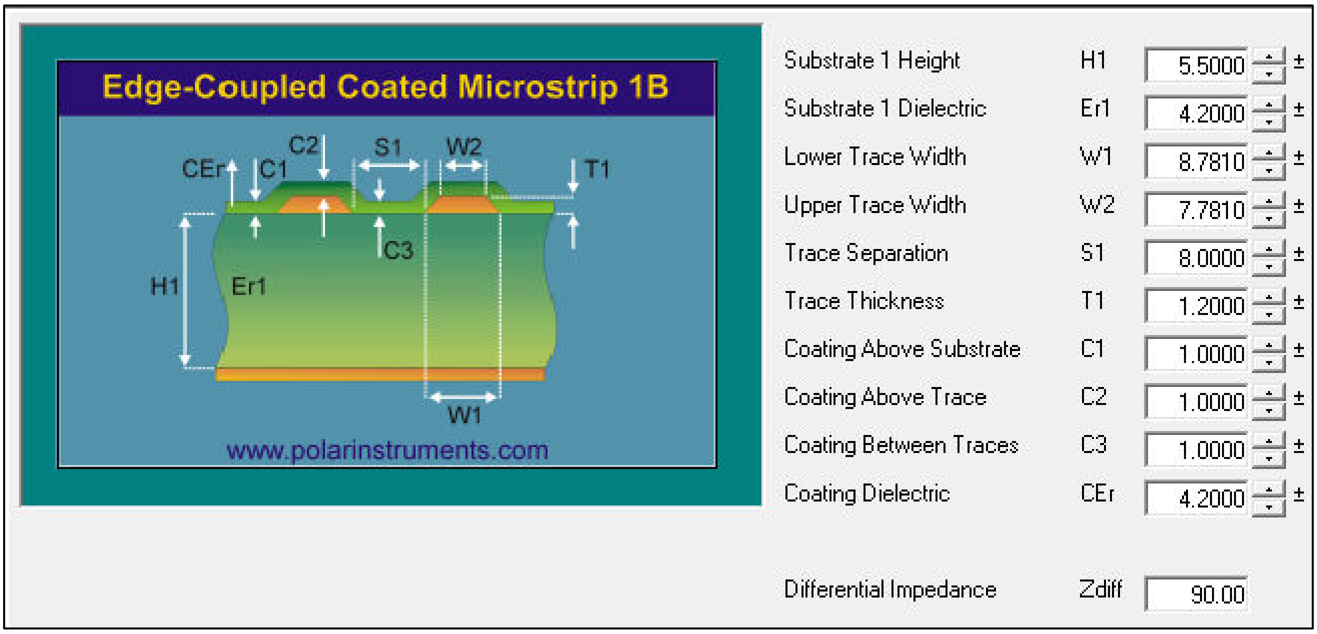
The Polar field solver Sensitivity Analysis function allows the designer to graph multiple impedances.

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.

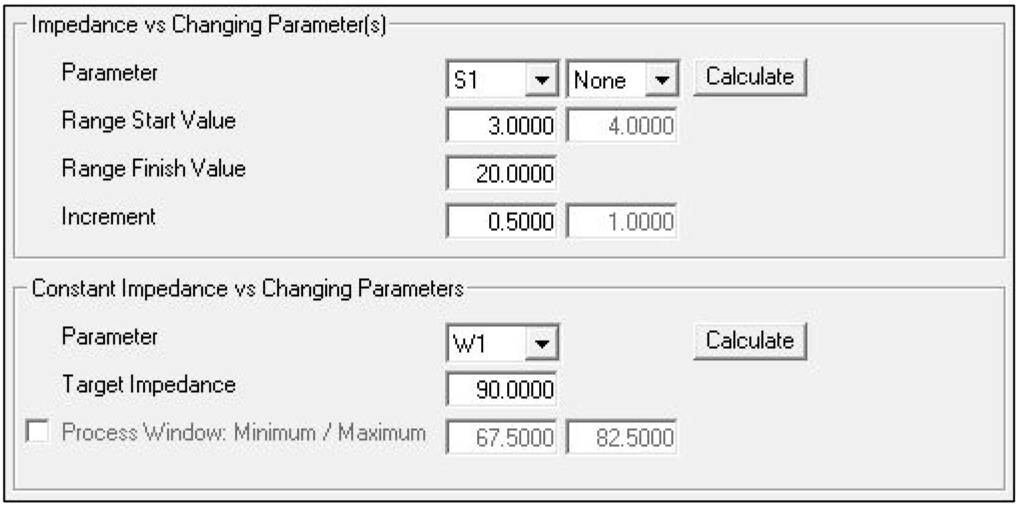


Switch to the Sensitivity Analysis tab.

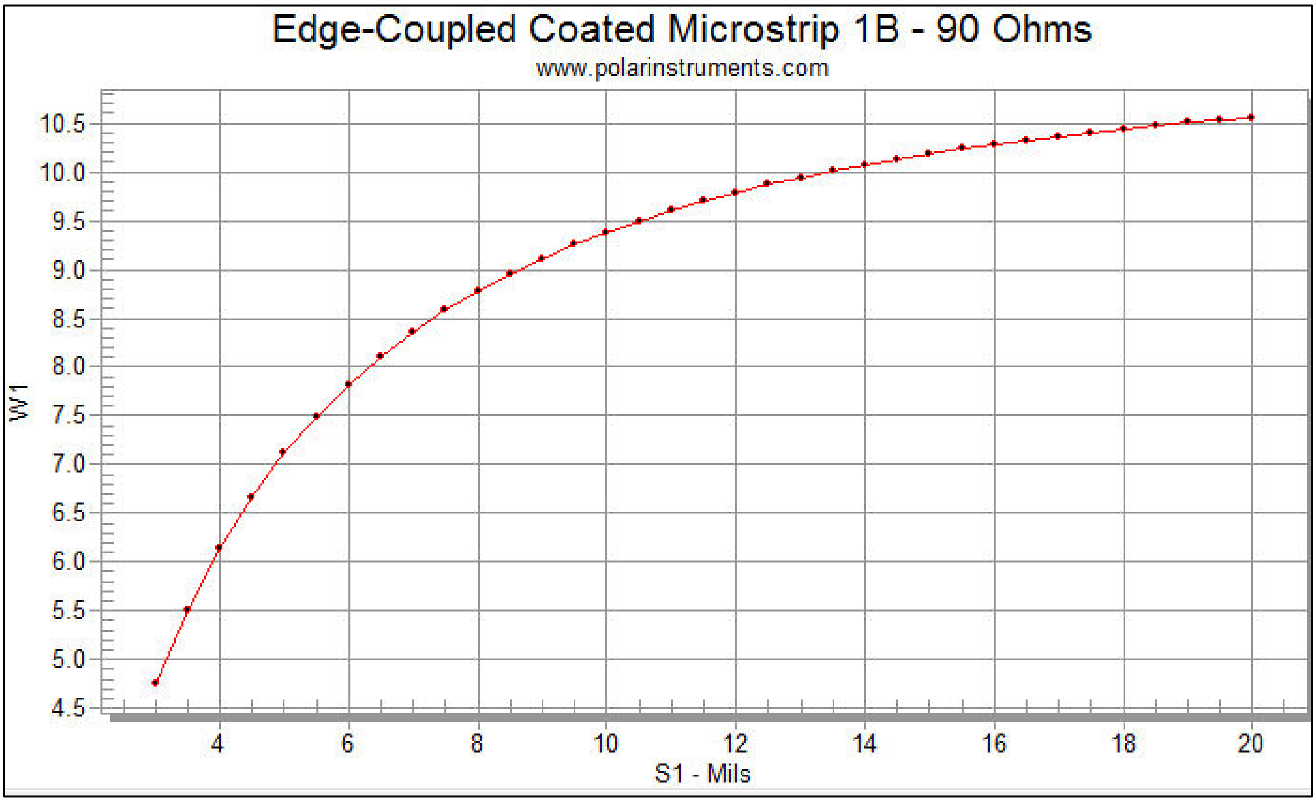
Under the Impedance vs Changing Parameter section set the Parameter to trace separation, S1, set the Range Start Value to 3 mils and the Range Finish Value to 20 mils – choose an increment of 0.5 mils.

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.

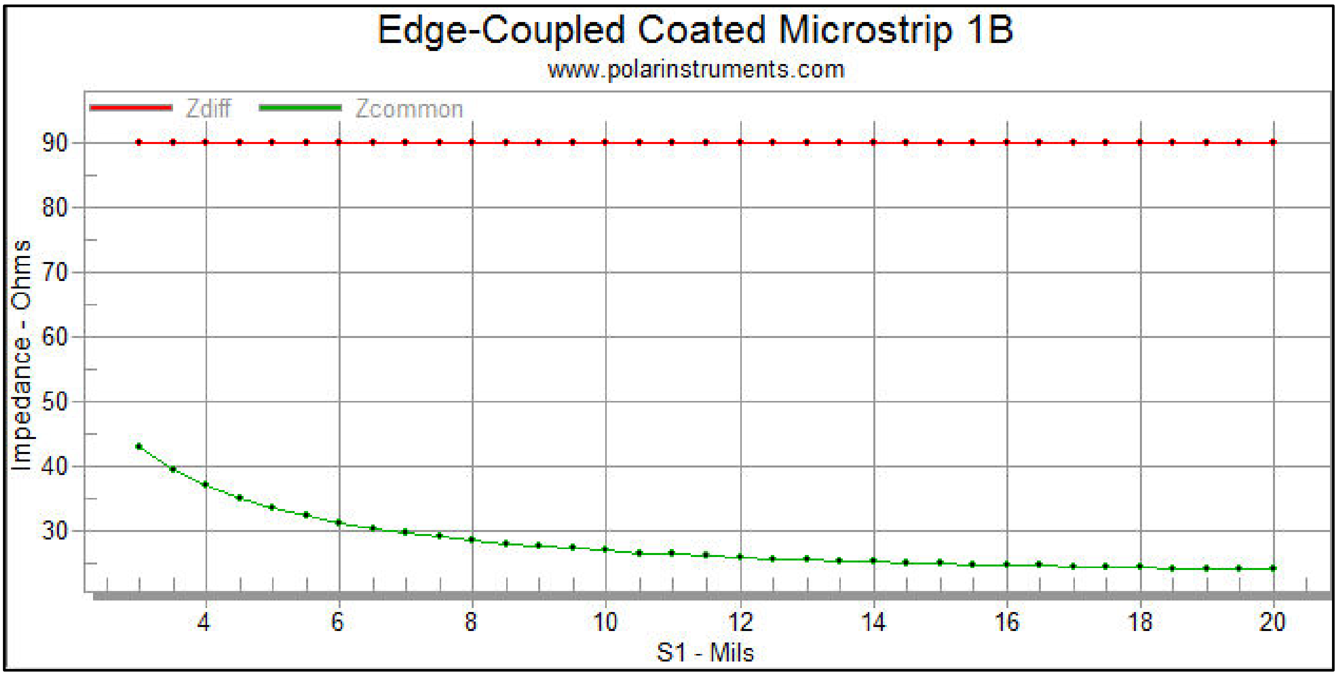


A subset of the sensitivity analysis results as trace width and separation vary is shown below.

W1	W2	S1	Zodd	Zeven	Zdiff	Zcommon	Kb (NEXT)	Kf s/in	FEXT	Calc Success
7.1141	6.1141	5.0000	44.9965	67.1007	89.9931	33.5503	9.8594E-02	-4.5595E-12	-4.5595E-01	Yes
7.4909	6.4909	5.5000	45.0050	64.6528	90.0100	32.3264	8.9587E-02	-4.6870E-12	-4.6870E-01	Yes
7.8229	6.8229	6.0000	44.9994	62.6161	89.9989	31.3081	8.1850E-02	-4.7615E-12	-4.7615E-01	Yes
8.1070	7.1070	6.5000	45.0021	60.9324	90.0041	30.4662	7.5189E-02	-4.8097E-12	-4.8097E-01	Yes
8.3612	7.3612	7.0000	44.9984	59.4907	89.9969	29.7453	6.9348E-02	-4.8197E-12	-4.8197E-01	Yes
8.5796	7.5796	7.5000	45.0042	58.2411	90.0083	29.1206	6.4104E-02	-4.7902E-12	-4.7902E-01	Yes
8.7799	7.7799	8.0000	45.0010	57.1536	90.0020	28.5768	5.9481E-02	-4.7540E-12	-4.7540E-01	Yes
8.9564	7.9564	8.5000	45.0013	56.2131	90.0026	28.1065	5.5386E-02	-4.7063E-12	-4.7063E-01	Yes
9.1149	8.1149	9.0000	44.9984	55.3778	89.9967	27.6889	5.1703E-02	-4.6476E-12	-4.6476E-01	Yes
9.2555	8.2555	9.5000	45.0017	54.6455	90.0033	27.3228	4.8390E-02	-4.5801E-12	-4.5801E-01	Yes
9.3811	8.3811	10.0000	45.0038	53.9914	90.0076	26.9957	4.5394E-02	-4.5056E-12	-4.5056E-01	Yes
9.5007	8.5007	10.5000	44.9979	53.3927	89.9958	26.6964	4.2661E-02	-4.4249E-12	-4.4249E-01	Yes
9.6054	8.6054	11.0000	44.9993	52.8636	89.9987	26.4318	4.0180E-02	-4.3412E-12	-4.3412E-01	Yes
9.7011	8.7011	11.5000	45.0001	52.3822	90.0003	26.1911	3.7902E-02	-4.2524E-12	-4.2524E-01	Yes
9.7878	8.7878	12.0000	45.0027	51.9499	90.0053	25.9749	3.5828E-02	-4.1641E-12	-4.1641E-01	Yes
9.8686	8.8686	12.5000	45.0019	51.5527	90.0038	25.7764	3.3923E-02	-4.0750E-12	-4.0750E-01	Yes
9.9433	8.9433	13.0000	45.0002	51.1888	90.0005	25.5944	3.2169E-02	-3.9851E-12	-3.9851E-01	Yes
10.0091	9.0091	13.5000	45.0029	50.8592	90.0057	25.4296	3.0546E-02	-3.8942E-12	-3.8942E-01	Yes
10.0749	9.0749	14.0000	44.9992	50.5524	89.9984	25.2762	2.9059E-02	-3.8043E-12	-3.8043E-01	Yes
10.1347	9.1347	14.5000	44.9955	50.2682	89.9910	25.1341	2.7674E-02	-3.7152E-12	-3.7152E-01	Yes

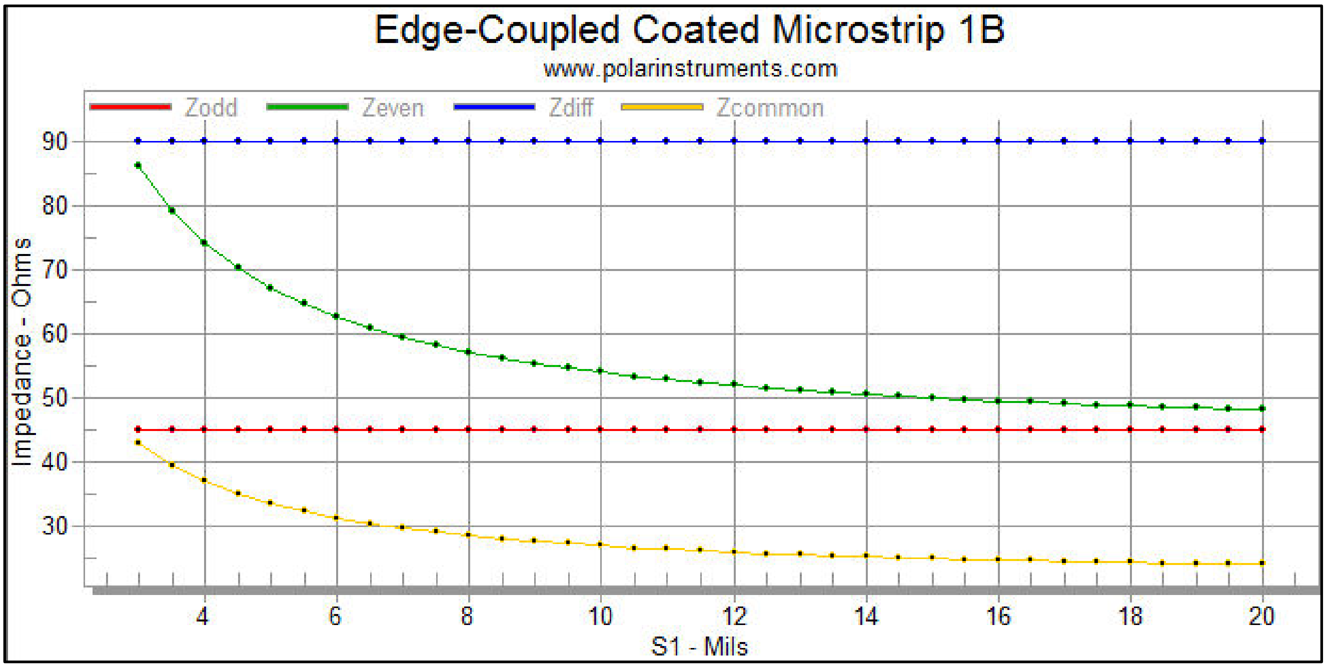
The Results tab shows W1 / W2 / S1 changing; the differential impedance, ZDiff, calculates to 90 ohms but the table indicates how the common impedance value, Zcommon, changes for each W1 / W2 / S1 combination of parameter values. This data can be exported to other tools (for example, Microsoft Excel®) 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

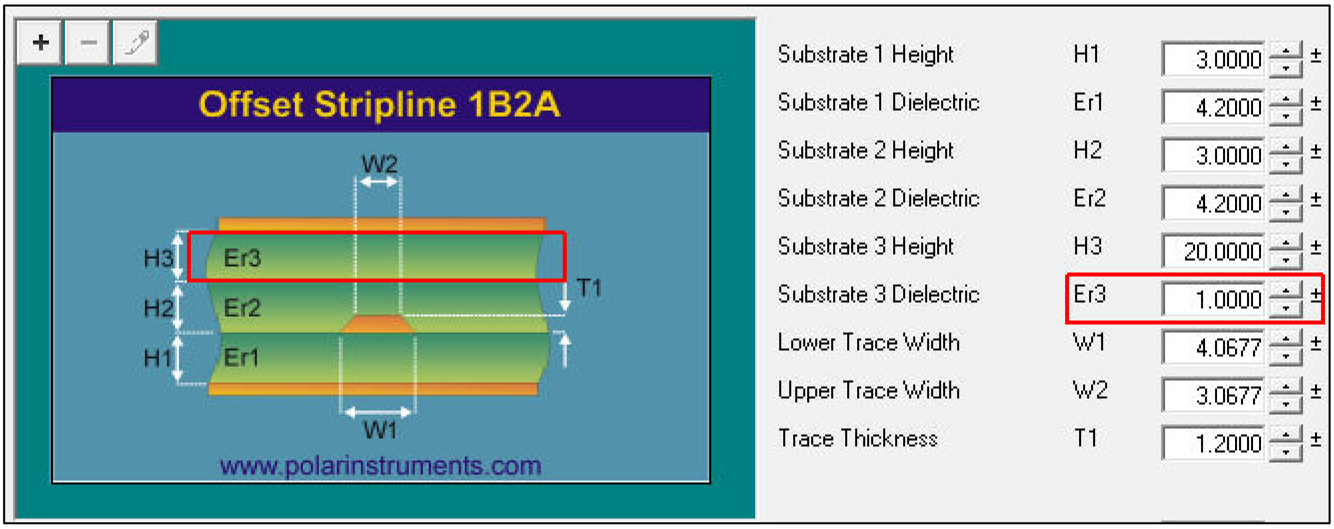
In this example, sensitivity analysis is used to predict the effects on impedance of a copper layer adjacent to a controlled impedance structure – in this example, interstitial copper leaves in bookbinder flex.

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

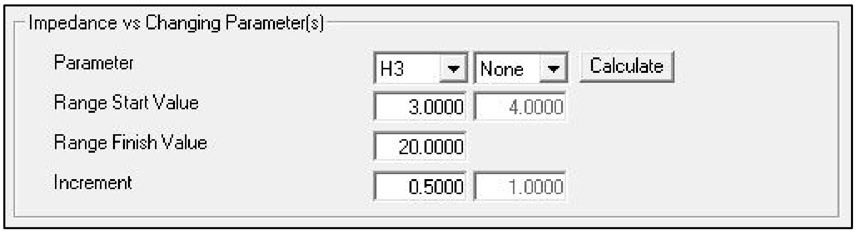
To model the effects of an interstitial copper layer, use one of the multi-dielectric controlled impedance structures in the field solver, setting the Er of one of the dielectrics in the multi-dielectric substrate to the approximate value for air, i.e. set Er = 1.0000.

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

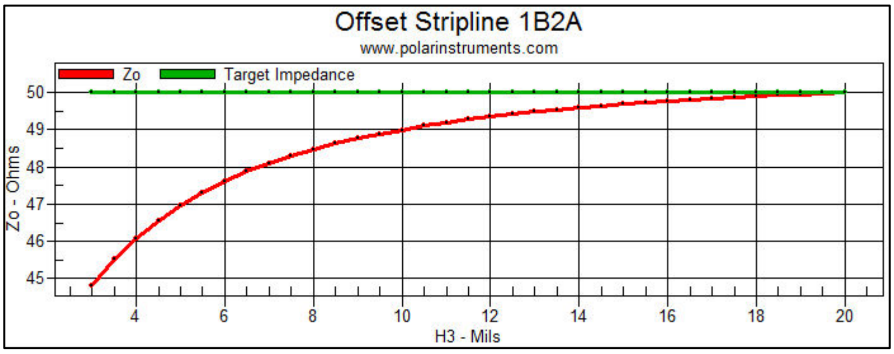


Set the structure parameters to their appropriate values, but set the value of Er3 to 1.0000. If necessary, goal seek on trace width to achieve the target impedance (In this example, 50 ohms.) Set the height of H3 to 20 mils.

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.



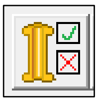
From the Impedance v Changing Parameters dialog (above), choose the H3 parameter and set its start value to 3 mils and the end value to 20 mils with a 0.5 mils increment. Click Calculate.



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%.

Via Checks



Via Checks

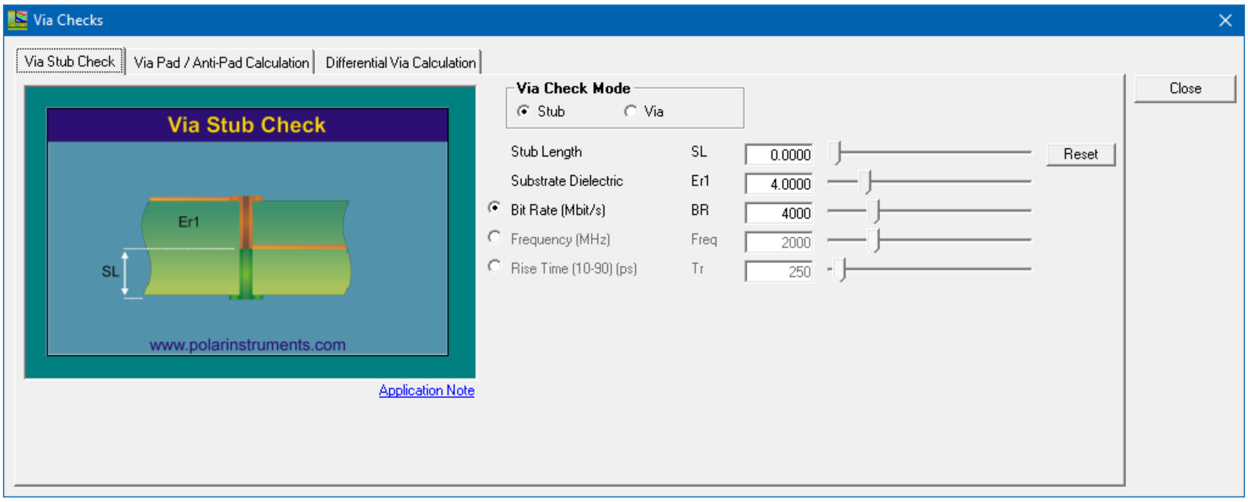
The Si8000m/Si9000e Via Checks incorporate

- Via Stub Check – that provides a simple colour coded go/no go check on the potential for signal distortion of a via stub.
- Via pad/antipad coaxial calculation – provides for modelling plated through hole (PTH) vias with respect to impedance and signal integrity
- Differential Via Calculation – for both horizontal oval anti-pad and round / oblong anti-pad styles

Via Stub Check

Click the Via Stub Check tab to calculate the effect of a 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.



Via Stub Check modes

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

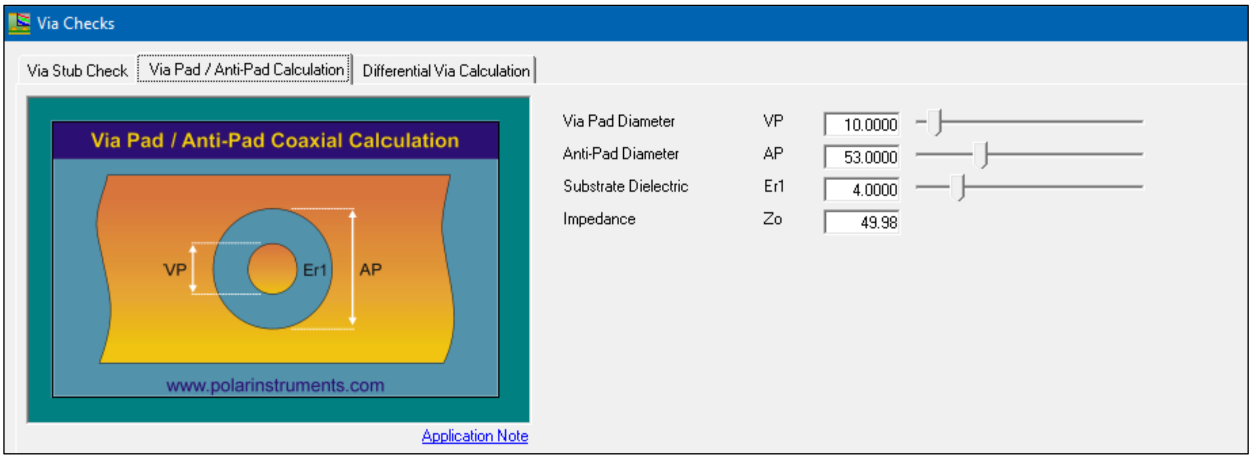
Click Bit Rate, Frequency or Rise time as appropriate and use the sliders to specify the stub length, Er value and your chosen parameter.

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

Via pad/antipad coaxial calculation

The Via Checks dialog includes via pad/anti-pad 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.



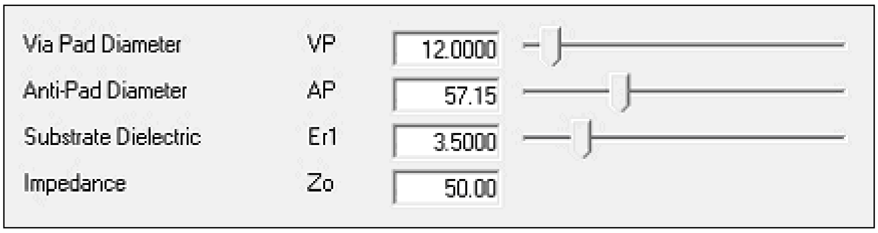
The anti-pad is the void area (shown as the blue annular ring in the diagram below) between the pad and the copper of the plane. It should be designed so that it maintains the impedance of a transmission line as it passes through the plane.

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 Er value in the immediate vicinity of the via will be lower than the bulk Er of the dielectric material as more resin will tend to flow into this type of region. In this example specify Er1 with a value of 3.5.

Enter the values of the via pad diameter, VP, of 12mil (0.3mm) and the Er1 of 3.5 into their respective fields. Move the slider bar for the anti-pad diameter, AP, until the Impedance (Zo) 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.

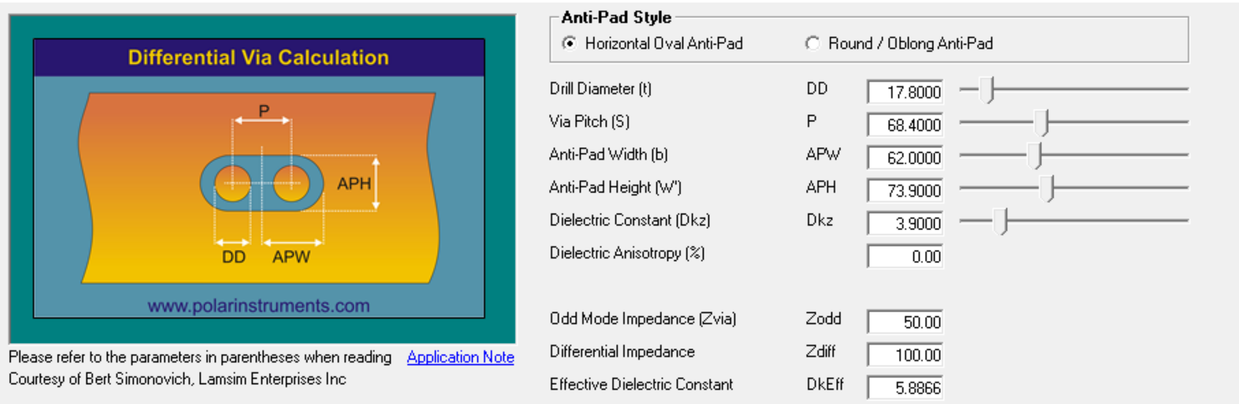
Differential via calculation

The Via Checks dialog includes *differential via calculation,* employing a simple and practical methodology for modelling differential vias. For an in-depth discussion of the modelling method employed, see Polar Instruments Application Note AP8204 *A practical alternative to 3D via modelling by Bert Simonovich, Lamsim Enterprises Inc.* https://www.polarinstruments.com/support/si/AP8204.pdf

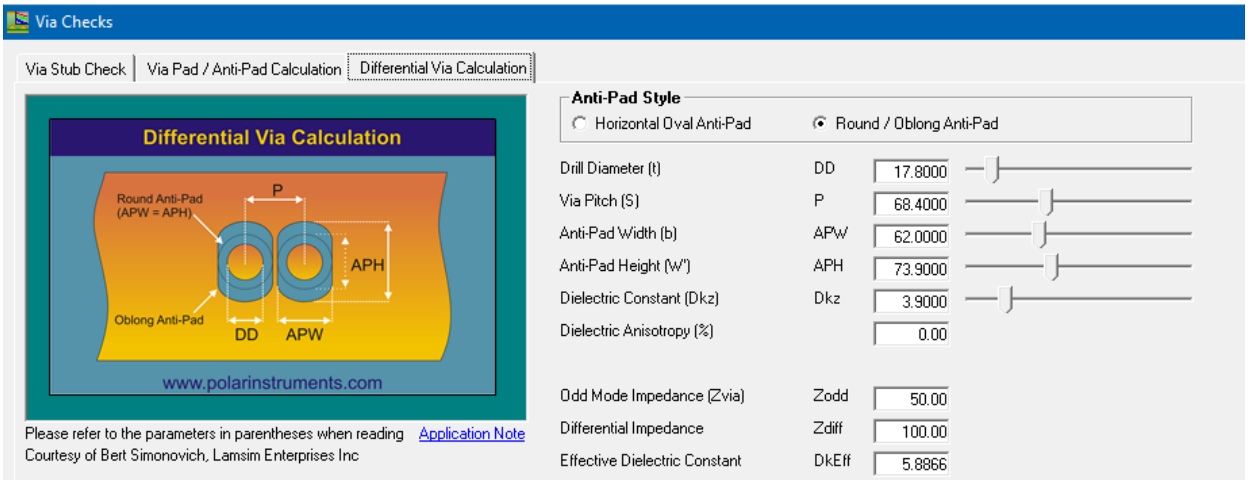
Anti-pad styles

Differential via calculation supports two anti-pad styles:

Horizontal oval anti-pad



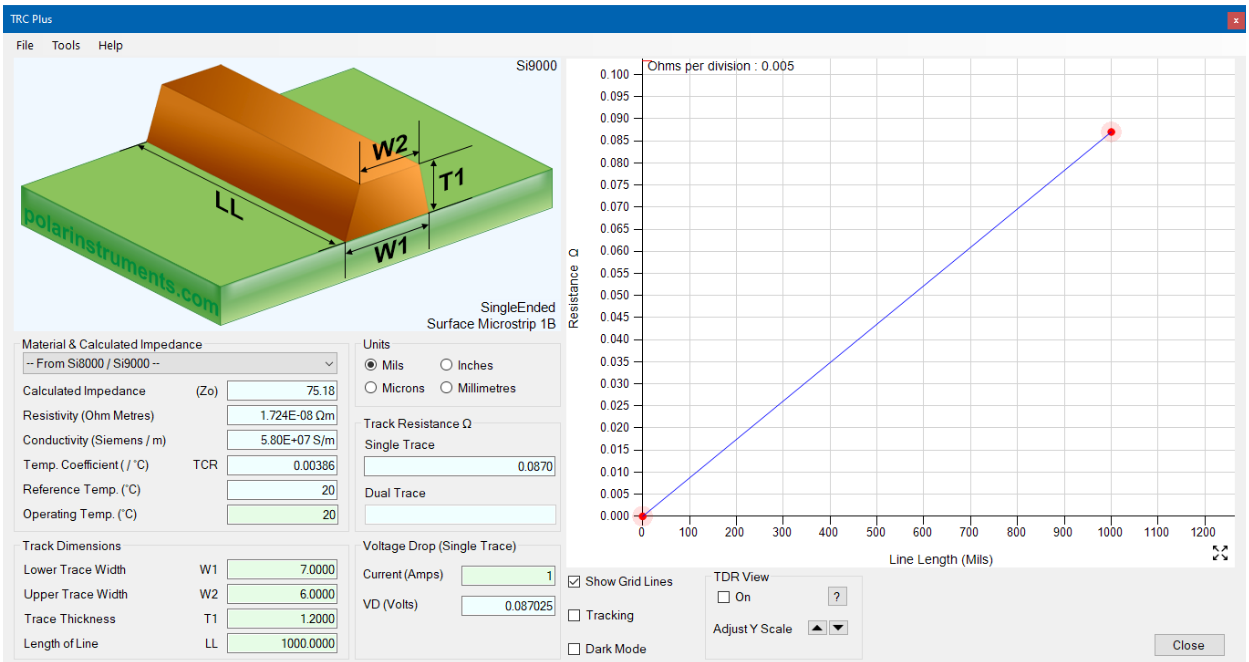
Round / oblong anti-pad



TRC Plus Track Resistance Calculator

The Si8000m/Si9000e include the Track Resistance Calculator (TRC Plus.) TRC Plus 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.

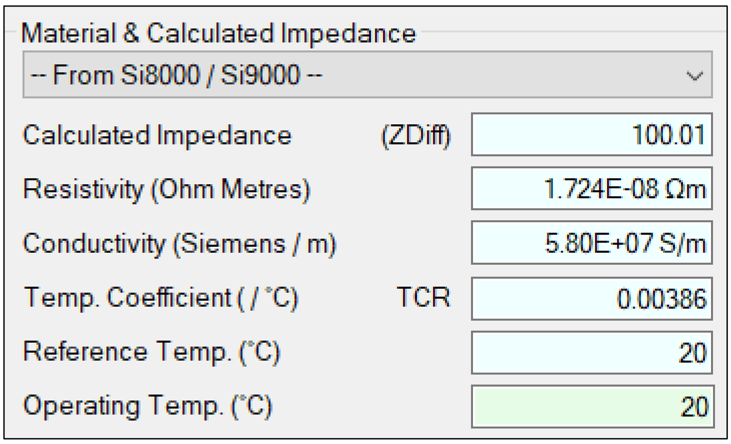
Calculating trace resistance will be found useful, for example, when working with fine geometry PCB tracks where series loss must be considered.



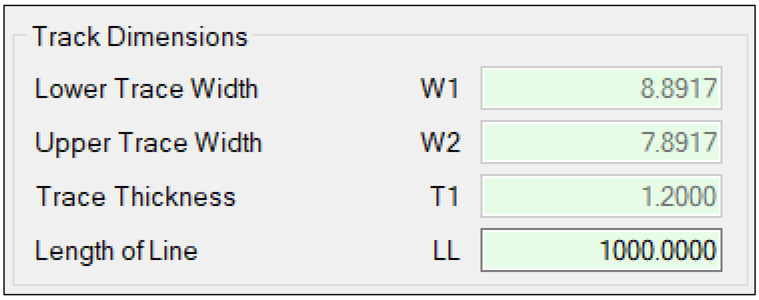
Calculating track resistance

Launch TRC / Refresh

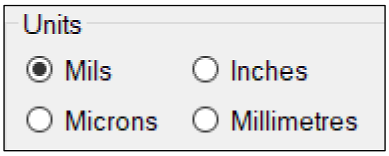
When TRC Plus is started, trace resistivity is automatically passed to TRC Plus alongside other parameters (upper and lower trace widths, W1, W2, trace thickness, T1, and the length of the line, LL) of the current structure specified in the Si8000m/Si9000e lossless calculation tab.



Track dimension values can also be typed in or changed directly.



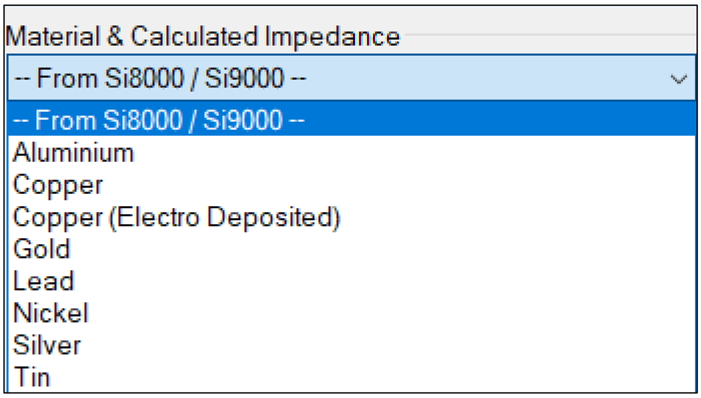
TRC Plus can work in all field solver units, Thou (mils), inches, microns (um) or millimetres.



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.

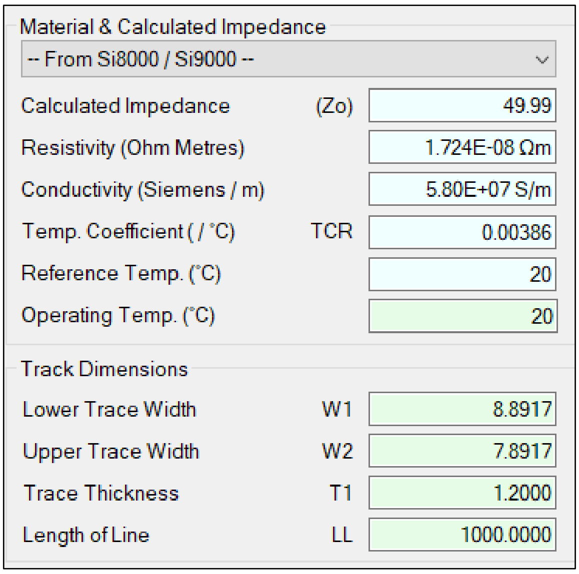


Choosing the **From Si8000 /Si9000** option will supply the values from the structure currently displayed in the field solver.


Launch TRC / Refresh

Clicking the Si8000m/Si9000e Launch Track Resistance Calculator/Refresh button will refresh the values derived from the field solver.

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

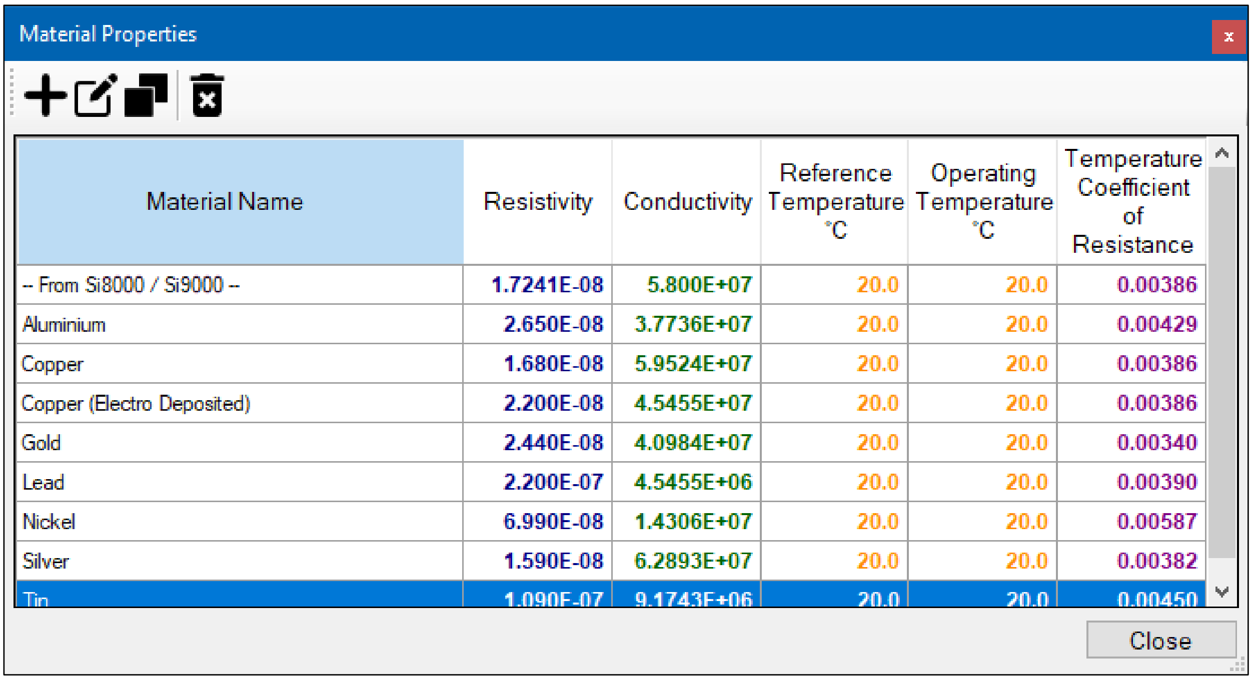


Adding and editing materials

TRC Plus maintains a table of materials and associated properties; material properties include resistivity, conductivity, reference temperature, operating temperature, and temperature coefficient of resistance.

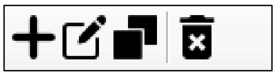
New materials can be added and existing materials edited, duplicated or deleted.

To add new materials or edit existing material values choose Tools|Edit Materials. TRC Plus displays the Material Properties dialog and associated toolbar.



Material Name	Resistivity	Conductivity	Reference Temperature °C	Operating Temperature °C	Temperature Coefficient of Resistance
-- From Si8000 / Si9000 --	1.7241E-08	5.800E+07	20.0	20.0	0.00386
Aluminium	2.650E-08	3.7736E+07	20.0	20.0	0.00429
Copper	1.680E-08	5.9524E+07	20.0	20.0	0.00386
Copper (Electro Deposited)	2.200E-08	4.5455E+07	20.0	20.0	0.00386
Gold	2.440E-08	4.0984E+07	20.0	20.0	0.00340
Lead	2.200E-07	4.5455E+06	20.0	20.0	0.00390
Nickel	6.990E-08	1.4306E+07	20.0	20.0	0.00587
Silver	1.590E-08	6.2893E+07	20.0	20.0	0.00382
Tin	1.090E-07	9.1743E+06	20.0	20.0	0.00450

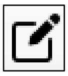
Use the Material Properties Toolbar to add, edit, duplicate and delete



Material Properties Toolbar

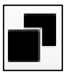
Add Material

Add New Material.



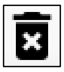
Edit selected material

Edit selected material properties



Duplicate selected material

Duplicate selected material and properties



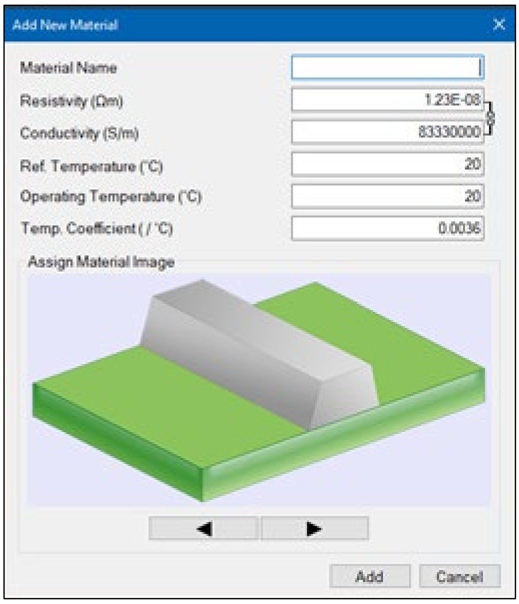
Delete selected material

Delete selected material

Adding a new material

Add New Material

To add a new material, click the Add New Material button and supply a descriptive name for the material and fill in the associated property fields.



When adding or editing materials, either the resistivity or conductivity 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.

TDR View

Making accurate impedance measurements on fine line PCB traces

PCB controlled impedance measurements are commonly shown as a waveform of the impedance over the length of a PCB trace displayed by a TDR such as the Polar CITS880s.

PCB trace line width and associated DC resistance will affect TDR waveform slope. The examples below show both single ended and differential models using 3 mil and 8 mil line widths. Until the arrival of fine line PCBs, on boards employing wide traces and thick copper, the impact of the DC resistance of a PCB trace was small enough to produce little or no effect on a controlled impedance measurement waveform of the impedance over the length of the trace as displayed by a TDR such as the Polar CITS880s.

However, as trace widths and weight on modern PCBs reduce, the DC resistance of the traces becomes significant, and, combined with high frequency skin effects reducing the effective cross-sectional area of the track, results in an upward slope of the TDR waveforms on fine line traces giving the false impression of impedance rising over the length of the trace. Series DC resistance can be compensated for by adjusting the slope of the waveform by a specified number of ohms/horizontal unit. This cancels out the series resistance leaving the true characteristic impedance displayed.

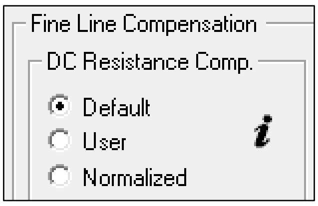
Note: Series DC resistance may be distinguished from the case of a slightly tapered track by testing from both ends of the trace. In the case of series DC resistance, the impedance waveform will appear to have the same rising slope when tested from both ends.

Supplying DC Resistance Compensation to the CITS880s

In order to produce a true picture of the impedance of a PCB trace the CITS880s provides for both automatic and manual DC resistance compensation for the series loss in the trace.

Default compensation

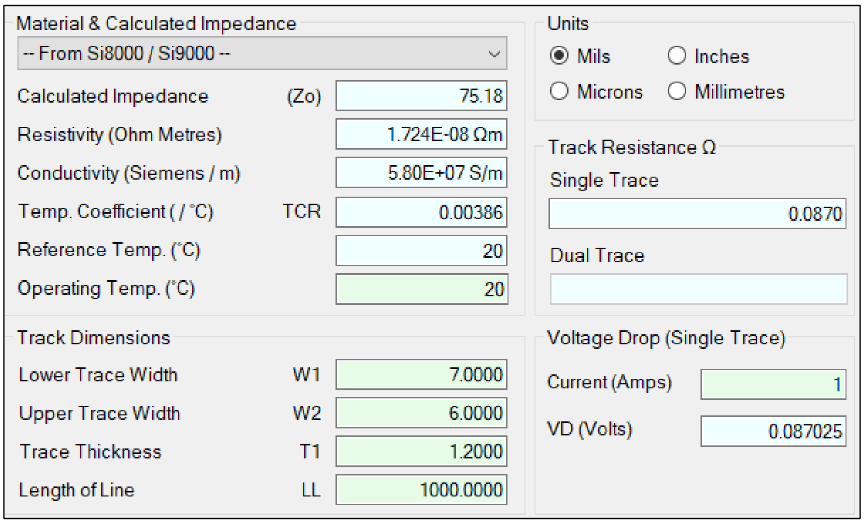
By default, no compensation is applied; i.e., the slope of the TDR's measured impedance waveform will not be altered.



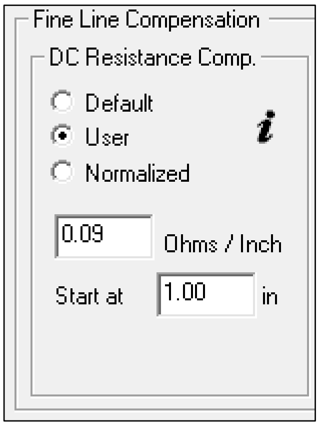
CITS880s DC Resistance Compensation

Applying user-defined compensation

To correct for a DC resistance produced slope in the impedance waveform manually, the TRC Plus calculated track resistance value can be copied to the CITS880s DC Resistance Compensation User field.



The trace resistance value (in Ohms/Inch) in the DC Resistance Compensation field may then be used to compensate for the series DC resistance in the track being tested. To apply user-defined compensation, in the CITS880s Test Editor DC Resistance Comp dialog select **User** from the compensation options then type in or copy and paste the TRC Plus calculated Ohms/inch track resistance value and CITS880s waveform start point In the DC Resistance Compensation text boxes. The Start at distance should be applied from the nominal start of the coupon or trace being tested.

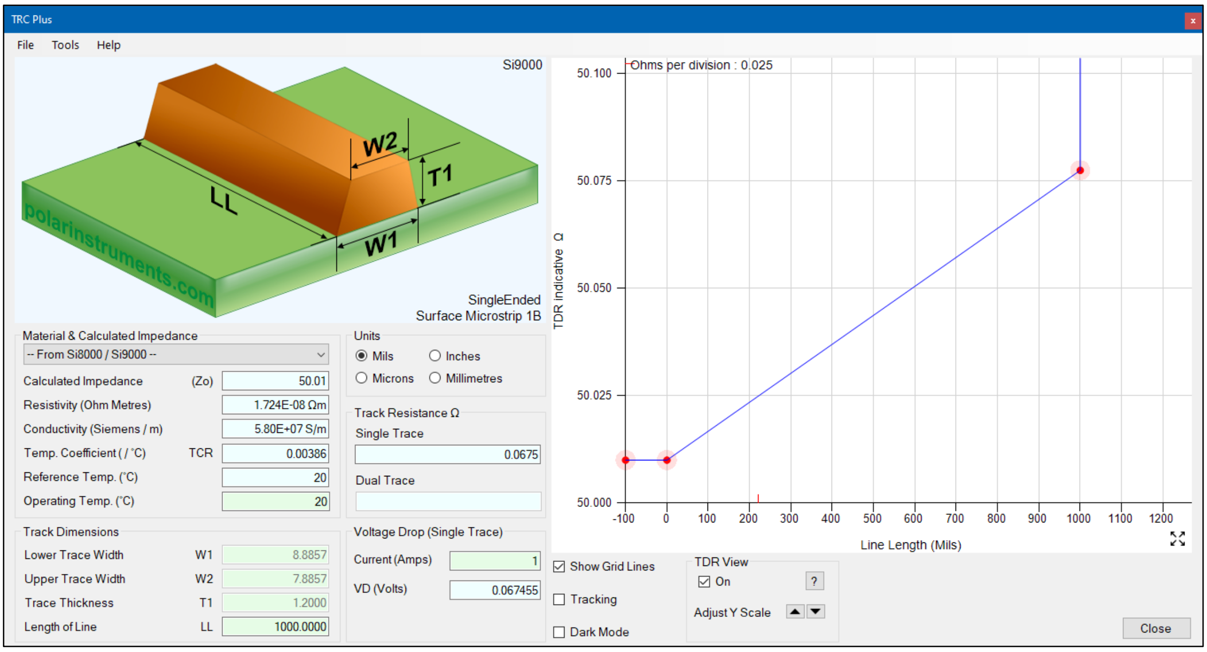


CITS880s User-defined DC Resistance Compensation

Compensation is applied on the CITS880s on a test by test basis. See the SPECIFICATIONS section of the CITS880s User Guide for the range of acceptable values.

Using TDR View

TRC Plus includes TDR View, providing a stylized, idealized view of a TDR trace on a coupon and illustrates how the measured impedance will be influenced by the distributed resistance along the trace in a PCB transmission line when tested on a TDR based test system such as the CITS880s.

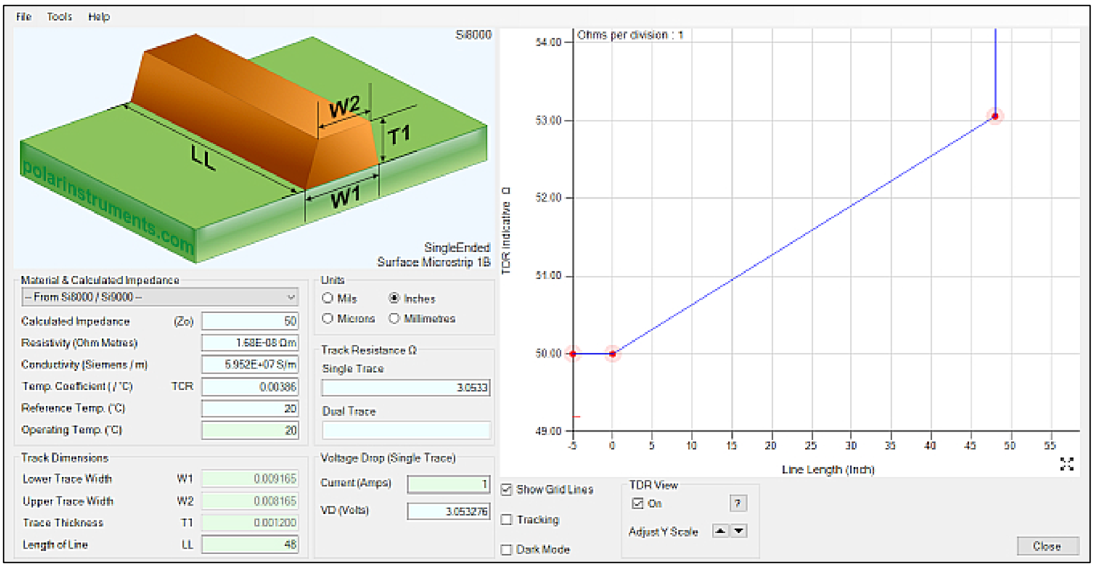


TDR View – idealized TDR waveform of 50 Ohm single ended trace showing effects of series resistance

TDR View provides an indicative impression of the effect of the distributed resistance in a PCB transmission line when tested on the Polar CITS880s. Used in conjunction with the Polar CITS880s, TRC Plus is designed to improve impedance correlation when measuring on fine line PCB traces, ensuring measured and modeled correlation.

Consider the traces below:

50 Ohm single ended trace on a fine line rigid flex design

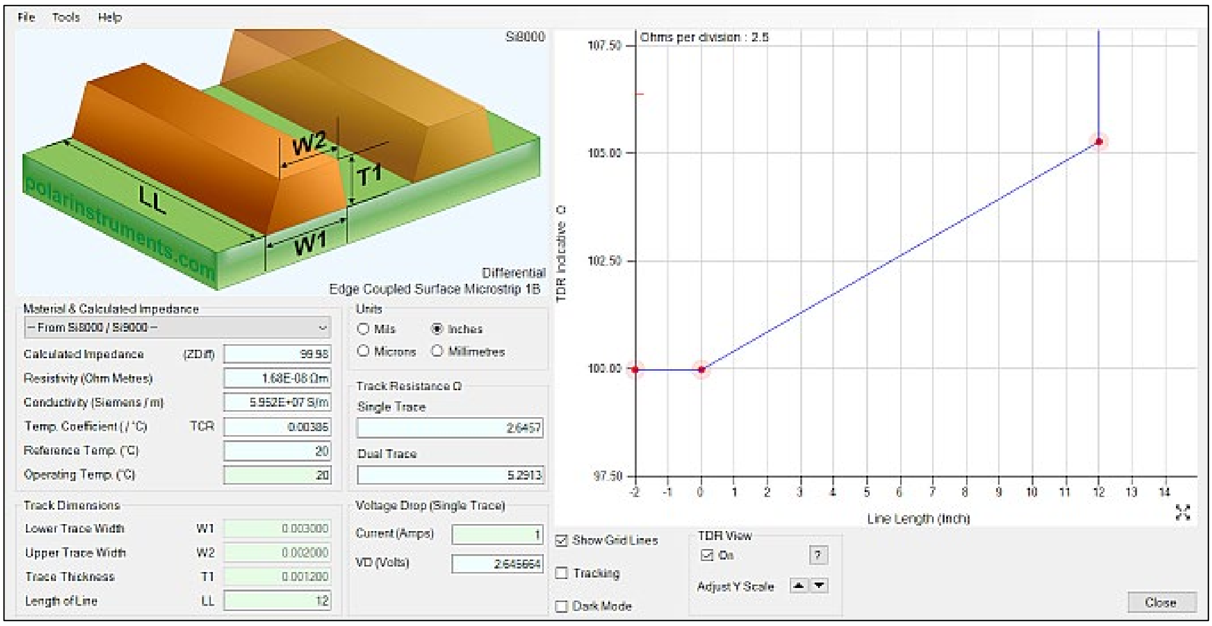


Example 1 – 3 mil line width 50 Ohm single ended trace with 3 Ohm series resistance

Example 1 shows a 50 Ohm single ended trace on a fine line (3 mil) rigid-flex design that exhibits significant DC trace resistance (3 Ohms.) This manifests itself as an upward slope over the length of the measured coupon.

100 Ohm differential pair with 3 mil line widths

Example 2 below shows a differential pair with 3.2 mil line widths with 5 Ohms DC resistance.



Example 2 – 3 mil line width 100 Ohm differential pair with 5 Ohms DC resistance

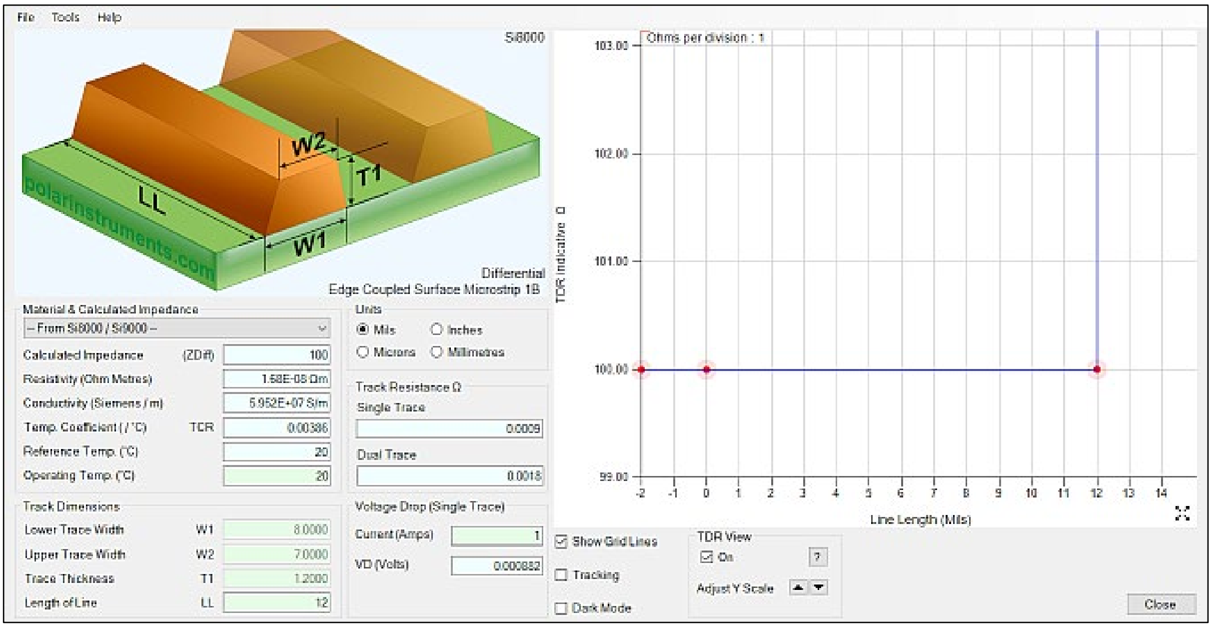
As with the single ended case above, the differential trace, also on a fine line rigid flex design, exhibits significant DC trace resistance – 5 Ohms over the length of the trace.

This is shown as an upward slope over the length of the measured coupon, illustrating how the distributed resistance component of the copper trace will cause the apparent impedance to rise over the length of the trace.

However, the characteristic impedance of the trace does not change; the graph shows the effects of the DC resistance accumulating in the reflection on the TDR.

In order to get an accurate measurement of characteristic impedance – which is independent of trace length – this trace resistance effect that will be observed on the TDR trace must be de-embedded (i.e. compensation applied.)

100 Ohm differential pair with 8 mil line widths



Example 3 – 8 mil line width 100 Ohm differential pair with low DC resistance

This differential pair with 8 mil line widths and heavier copper exhibits only a small amount of DC resistance over the line length – the rise in impedance is only a fraction of an ohm over the whole coupon length – small enough that it will not interfere with the correlation between measured and modeled impedance. With broader geometries such as this, it is safe to dismiss the effects of trace resistance on the impedance measurement

Note: Without de-embedding (i.e. applying compensation) the CITS880s TDR can appear to be measuring high. Some fabricators may therefore wrongly conclude that this high measurement has resulted because the supplied material datasheet value for Er is too high and so use a field solver to goal seek the "correct" Er.

This conclusion is incorrect as laminate suppliers are careful to provide accurate measurements for dielectric constant, and any field solver / TDR correlation should only be studied, a) if the DC resistance is small enough to be ignored, or b) the DC resistance has been de-embedded

Application Note AP8512 *Applying DC Resistance Compensation* on the Polar Instruments web site discusses applying DC resistance compensation for series loss.

Frequency-dependent calculations

Frequency-dependent calculations (Si9000e only)

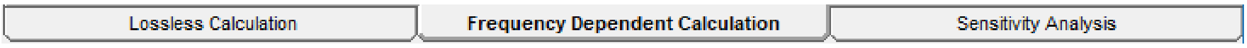
The Si9000e incorporates fast and accurate frequency-dependent PCB transmission line modelling, and extracts full transmission line parameters across its range of controlled impedance structures.

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.

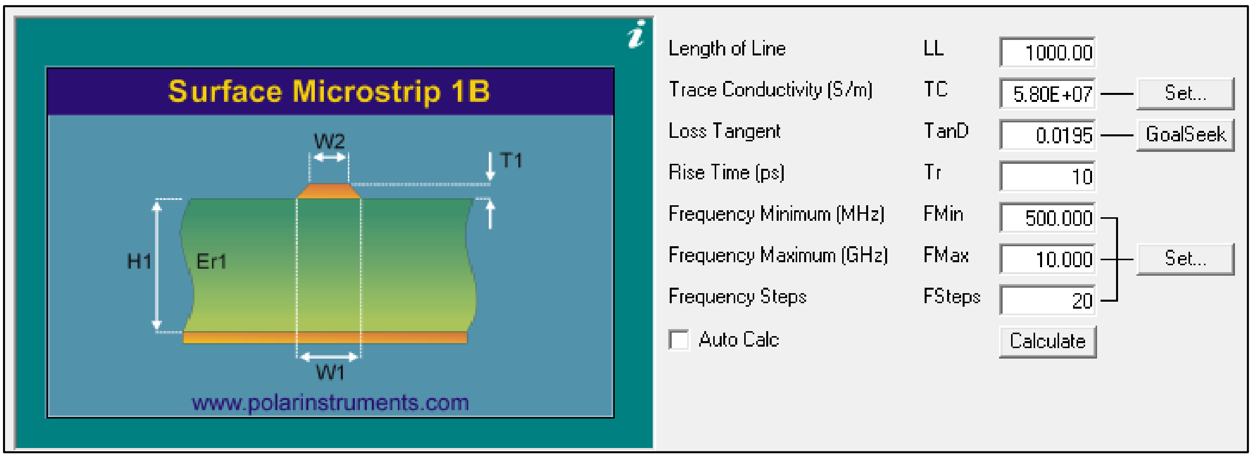
Frequency dependent calculation interface

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 calculation interface is displayed.

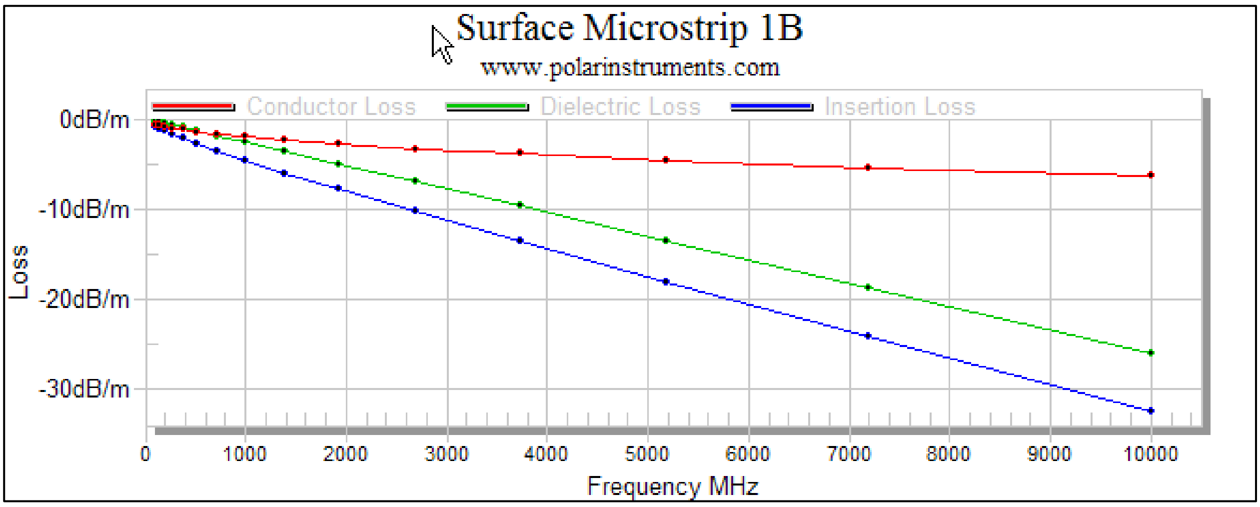


The frequency-dependent interface allows entry of the frequency-dependent calculation parameters for the selected structure, including line length and conductivity, loss tangent, minimum and maximum frequency, frequency steps, etc.; supply values for each field and click Calculate.



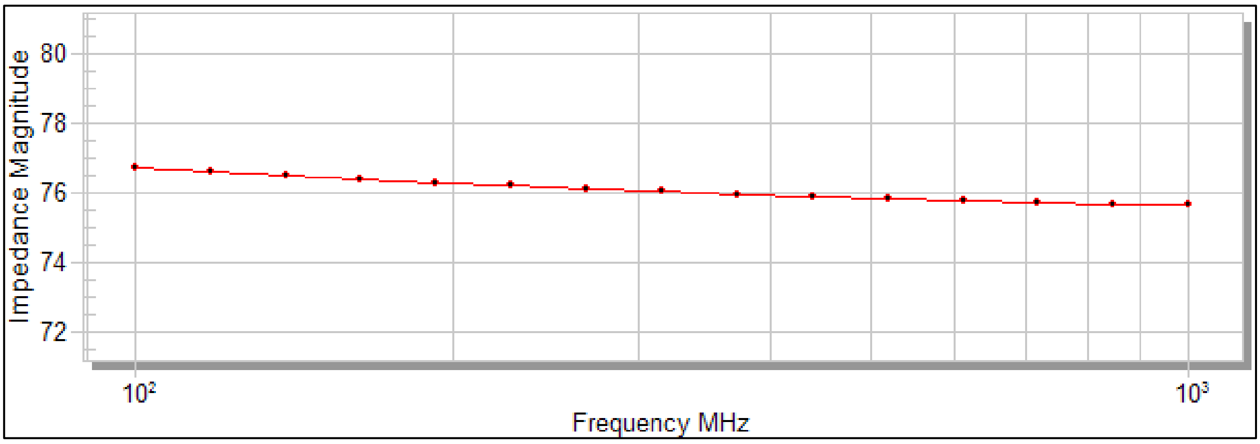
Click the Graph tab and select the data series from the Display Series dropdown. The Si9000e displays results over the specified frequency range.

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.

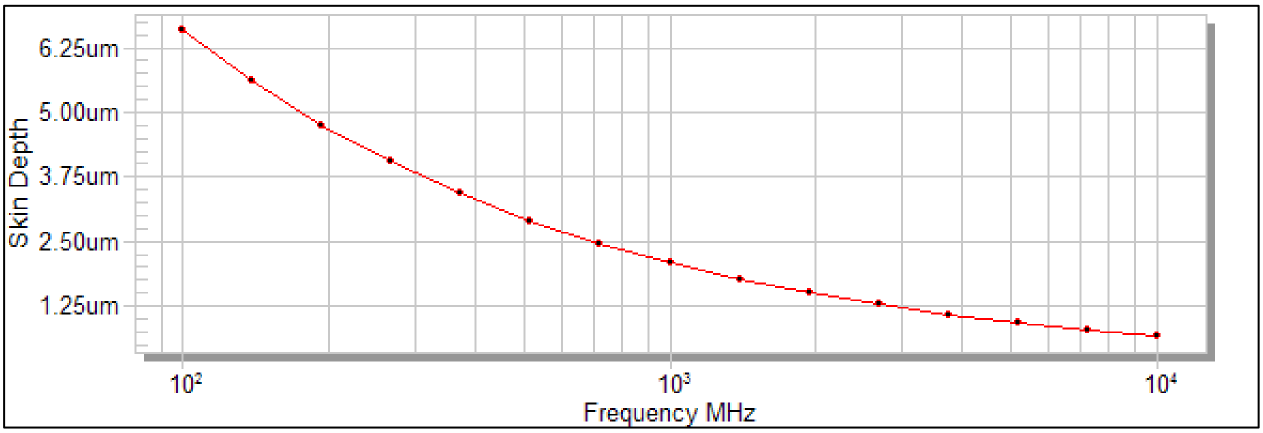


To change the structure parameters, switch to lossless mode and modify values as required.

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



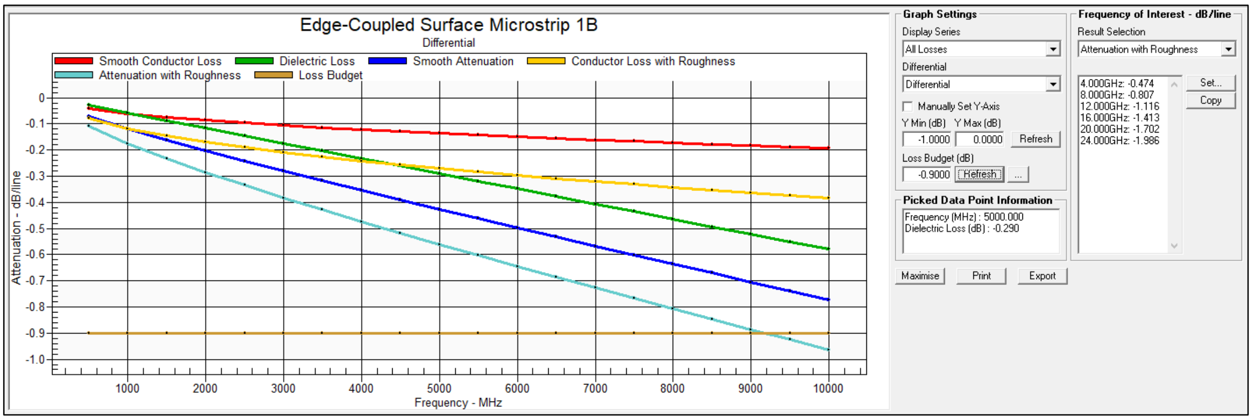
The graph below shows the variation in skin depth between 100MHz and 10 GHz.



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.



Setting the y-axis manually

To override the Si9000e graph auto scaling, click the Manually Set Y-Axis check box and specify values for Y-Min and Y-Max and click Refresh to redraw the graph at the new vertical scale.

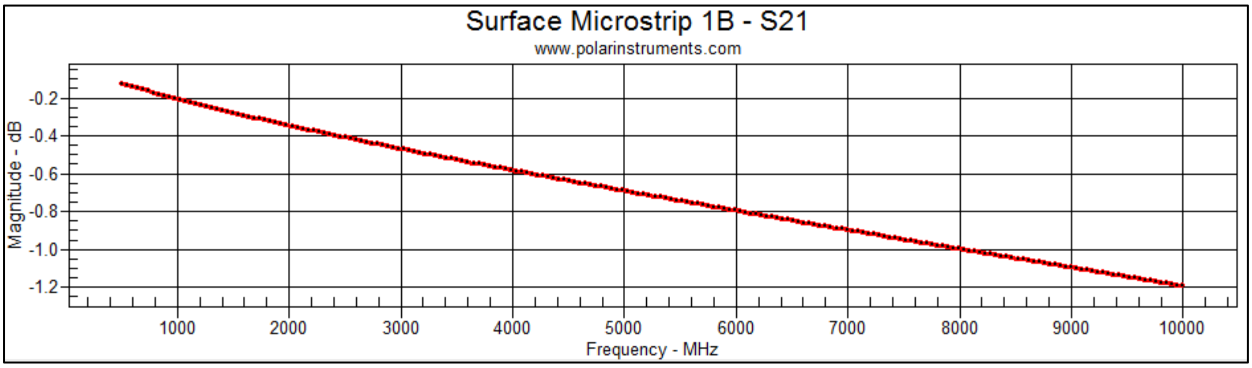
Viewing data in table form

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

Graph | Single Ended | SPICE RLGC | 2 Port S-Parameters - Graph | 2 Port S-Parameters - Data | Measurement Data

Frequency Hz	Impedance Real Ohms	Impedance Imaginary Ohms	Impedance Magnitude Ohms	Inductance H/line	Resistance Ohms/line	Capacitance F/line	Conductance S/line	Skin Depth in	Conductor Loss dB/line	Dielectric Loss dB/line
5.000E+08	5.069E+01	-2.288E-01	5.069E+01	7.697E-09	6.393E-01	2.996E-12	1.638E-04	1.164E-04	-5.477E-02	-3.607E-02
1.000E+09	5.049E+01	-3.341E-02	5.049E+01	7.637E-09	8.989E-01	2.996E-12	3.277E-04	8.228E-05	-7.731E-02	-7.185E-02
1.500E+09	5.040E+01	5.328E-02	5.040E+01	7.610E-09	1.097E+00	2.996E-12	4.915E-04	6.718E-05	-9.452E-02	-1.076E-01
2.000E+09	5.035E+01	1.004E-01	5.035E+01	7.596E-09	1.281E+00	2.996E-12	6.554E-04	5.818E-05	-1.105E-01	-1.433E-01
2.500E+09	5.032E+01	1.357E-01	5.032E+01	7.585E-09	1.432E+00	2.996E-12	8.192E-04	5.204E-05	-1.236E-01	-1.790E-01
3.000E+09	5.029E+01	1.617E-01	5.029E+01	7.577E-09	1.568E+00	2.996E-12	9.830E-04	4.750E-05	-1.354E-01	-2.147E-01

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.

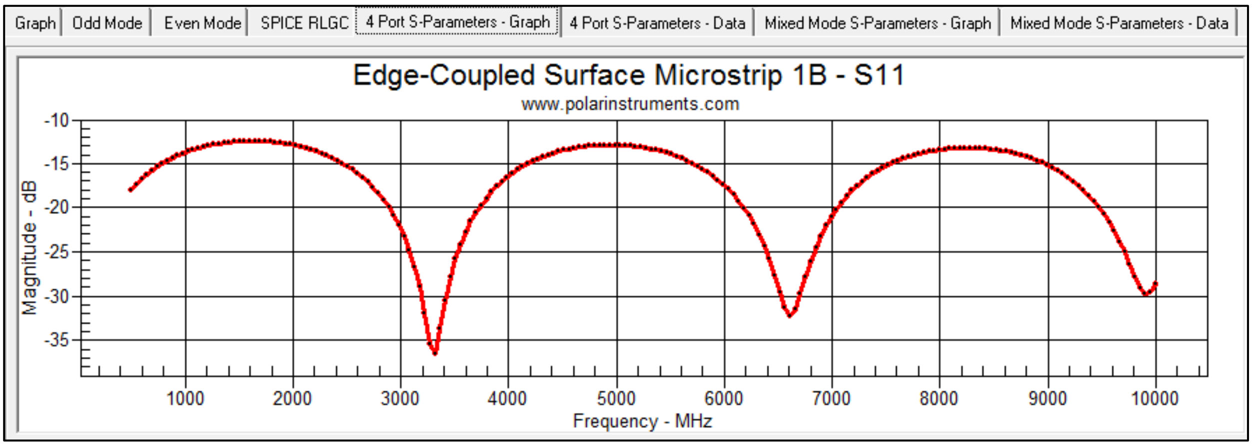
Graph | Odd Mode | Even Mode | SPICE RLGC | 4 Port S-Parameters - Graph | 4 Port S-Parameters - Data | Mixed Mode S-Parameters - Graph | Mixed Mode S-Parameters - Data

Frequency Hz	Impedance Real Ohms	Impedance Imaginary Ohms	Impedance Magnitude Ohms	Inductance H/line	Resistance Ohms/line	Capacitance F/line	Conductance S/line	Skin Depth in	Conductor Loss dB/line	Dielectric Loss dB/line	Attenuation dB/line
5.000E+08	5.089E+01	-4.857E-01	5.089E+01	6.915E-09	7.471E-01	2.671E-12	1.283E-04	1.164E-04	-6.376E-02	-2.835E-02	-9.211E-02
1.000E+09	5.063E+01	-2.321E-01	5.063E+01	6.845E-09	1.052E+00	2.671E-12	2.566E-04	8.228E-05	-9.026E-02	-5.642E-02	-1.467E-01
1.500E+09	5.051E+01	-1.196E-01	5.051E+01	6.814E-09	1.286E+00	2.671E-12	3.849E-04	6.718E-05	-1.106E-01	-8.443E-02	-1.950E-01
2.000E+09	5.044E+01	-5.250E-02	5.044E+01	6.795E-09	1.484E+00	2.671E-12	5.132E-04	5.818E-05	-1.277E-01	-1.124E-01	-2.402E-01
2.500E+09	5.039E+01	-6.675E-03	5.039E+01	6.783E-09	1.657E+00	2.671E-12	6.415E-04	5.204E-05	-1.428E-01	-1.404E-01	-2.832E-01
3.000E+09	5.036E+01	2.716E-02	5.036E+01	6.773E-09	1.815E+00	2.671E-12	7.699E-04	4.750E-05	-1.565E-01	-1.684E-01	-3.249E-01
3.500E+09	5.033E+01	5.347E-02	5.033E+01	6.766E-09	1.959E+00	2.671E-12	8.982E-04	4.398E-05	-1.690E-01	-1.963E-01	-3.654E-01
4.000E+09	5.031E+01	7.467E-02	5.031E+01	6.760E-09	2.094E+00	2.671E-12	1.026E-03	4.114E-05	-1.807E-01	-2.243E-01	-4.050E-01

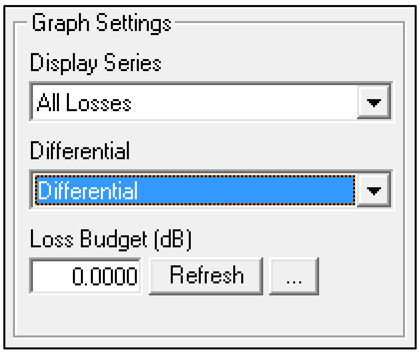
Kb (NEXT)	Kf s/in	FEXT
1.2653E-01	-8.3438E-12	-5.0000E-01

Near and far end crosstalk values

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

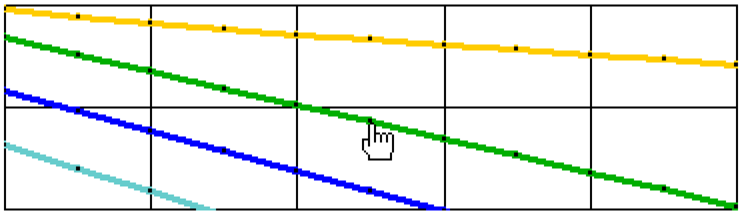


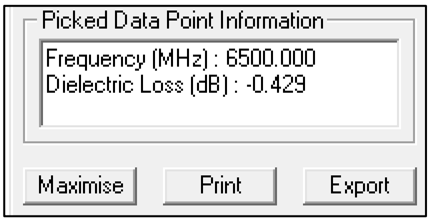
Choose from the Graph Settings Display Series drop-down list to choose results, including those for loss, impedance, inductance, resistance, capacitance, conductance, skin depth and attenuation and effective Er.



Viewing detailed data point information

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



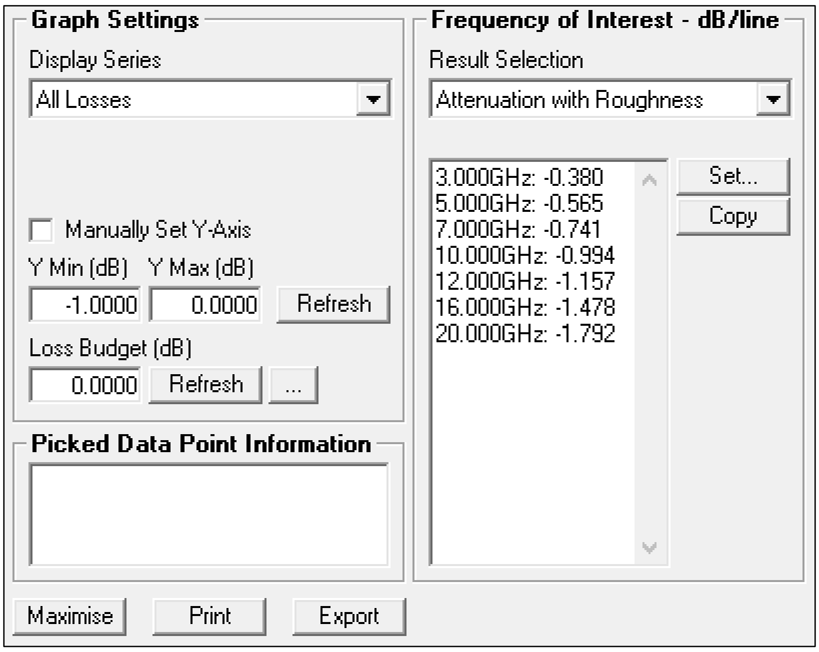


Creating and using frequencies of interest

Frequency of Interest

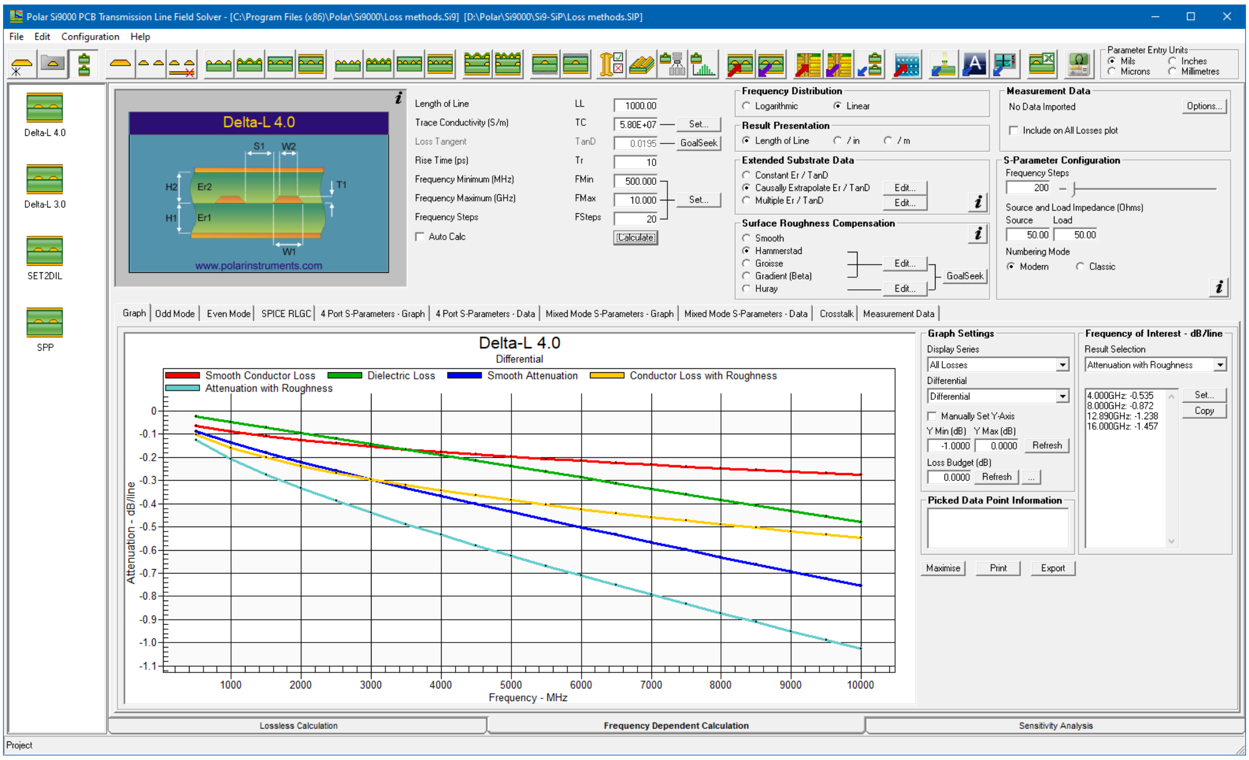
When displaying All Losses against frequency (All Losses include conductor loss, dielectric loss and insertion loss, conductor loss with roughness, attenuation with roughness), up to 10 single frequencies of interest can be defined for tabular display alongside the loss graph. The Frequency of Interest pane (shown below) displays a table of insertion loss at specific nominated frequencies.

The results may be exported in tabular form to a text file or spreadsheet for analysis.



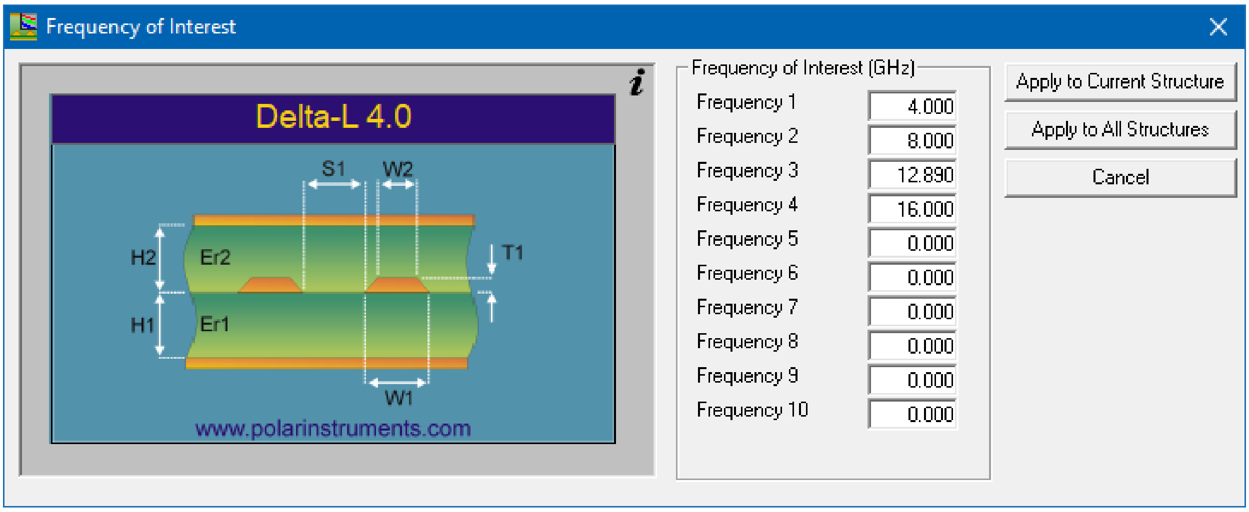
Frequency of interest settings can be applied to the current structure or, for example, all the structures within a project.

For example, the graphic below demonstrates a project containing four project structures – comprising an edge-coupled offset stripline with results from four test methods.



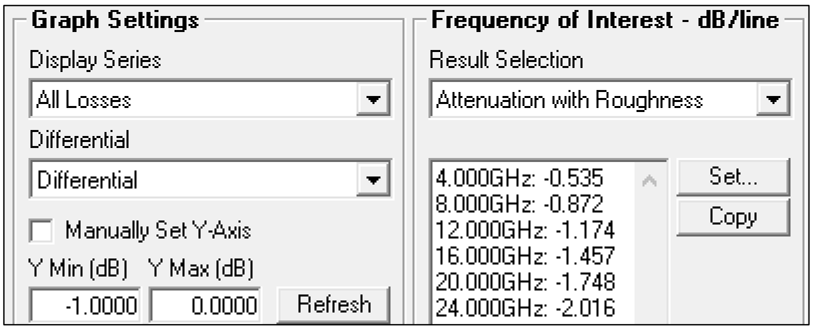
A set of frequencies of interest may be defined for each of the four structures.

Click on a structure (in this example, the Delta-L 4.0 loss method structure) and click the Set… button within the Frequency of Interest pane to open the Frequency of Interest dialog. Add a frequency in GHz for each frequency of interest; up to 10 frequency values per structure may be defined, so each of the four method structures in the project above may have up to 10 frequencies of interest. *Note that Frequency of Interest values can be outside the frequency range defined by the graph Frequency Min / Max settings.*



With all frequencies of interest defined, click on Apply to Current Structure (or Apply to All Structures to apply all frequencies of interest to all structures within the project.)

Step through the structures in the structure bar to display the graphed losses along with the listed insertion loss at each frequency of interest for each structure. In the example below, six frequencies have been defined



Selecting loss components for display

Use the Result Selection drop-down (shown below) to select the loss component to be displayed:

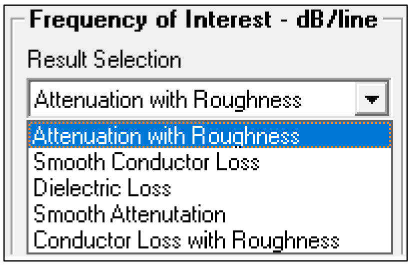
Attenuation with Roughness,

Smooth Conductor Loss,

Dielectric Loss,

Smooth Attenuation,

Conductor Loss with Roughness.

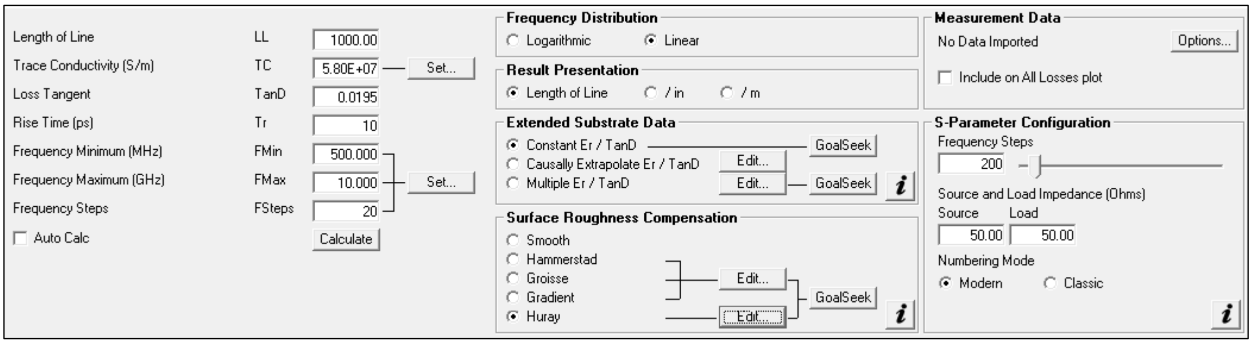


The graphic above shows Attenuation with Roughness selected. (If necessary, click Calculate to refresh results.)

Click the Copy button to copy the readings to the clipboard and paste them into, for example, a spreadsheet for subsequent analysis.

Frequency-dependent calculation interface

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



Use the Frequency Distribution options to choose between logarithmic or linear graphing.

Use the Result Presentation options to specify the vertical chart axis; choose between dB/line length, dB/inch or dB/m.

Use the Extended Substrate Data options 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 permittivity

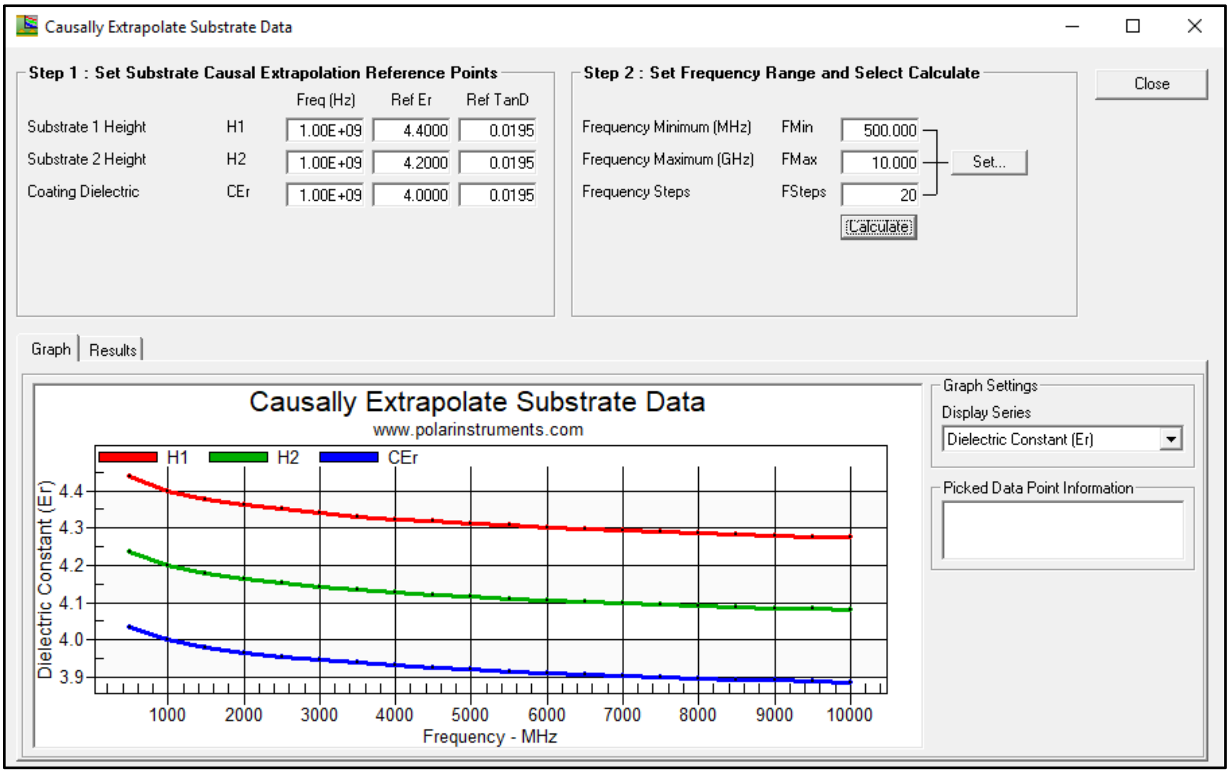
Using frequency independent permittivity is a source of non-causal time domain responses so causal interpolation of dielectric constant is implemented in the Si9000e via the Causally Extrapolate Er / TanD option from the Extended Substrate Data option group; this applies Svensson-Djordjevic modelling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

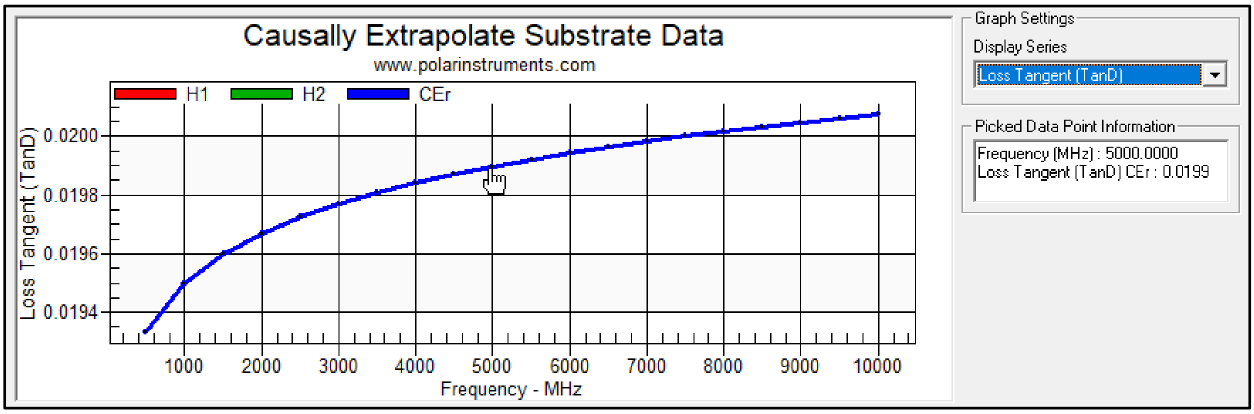
Click Edit… the Causally Extrapolate Substrate Data dialog is displayed.



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.

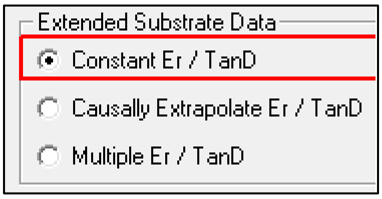
Select Dielectric Constant or Loss Tangent for display.



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.

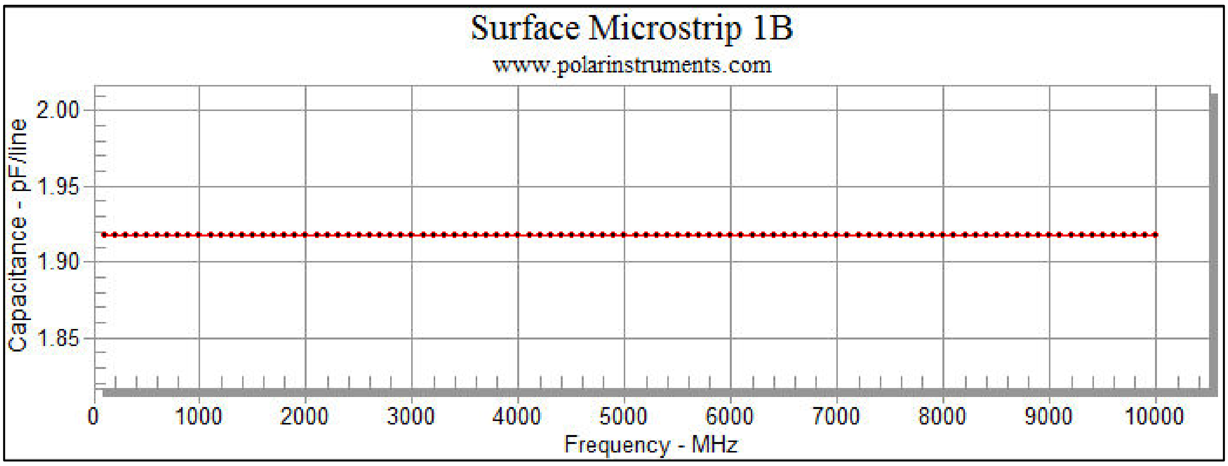
Frequency Hz	Dielectric Constant Er : H1	Loss Tangent TanD : H1	Dielectric Constant Er : H2	Loss Tangent TanD : H2	Dielectric Constant Er : CEr	Loss Tangent TanD : CEr
5.00E+08	4.437861	0.019334	4.236140	0.019334	4.034419	0.019334
1.00E+09	4.400000	0.019500	4.200000	0.019500	4.000000	0.019500
1.50E+09	4.377853	0.019599	4.178859	0.019599	3.979866	0.019599
2.00E+09	4.362139	0.019669	4.163860	0.019669	3.965581	0.019669
2.50E+09	4.349950	0.019724	4.152225	0.019724	3.954500	0.019724
3.00E+09	4.339992	0.019770	4.142719	0.019770	3.945447	0.019770
3.50E+09	4.331572	0.019808	4.134682	0.019808	3.937792	0.019808
4.00E+09	4.324278	0.019841	4.127720	0.019841	3.931162	0.019841
4.50E+09	4.317844	0.019871	4.121579	0.019871	3.925313	0.019871
5.00E+09	4.312089	0.019898	4.116085	0.019898	3.920081	0.019898
5.50E+09	4.306883	0.019922	4.111116	0.019922	3.915348	0.019922

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.

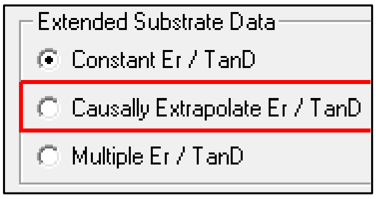
From the drop-down choose (for this example) Capacitance and click Calculate – the graph of capacitance is shown below.



Note: Using frequency independent permittivity is a source of non-causal time domain responses.

Causal interpolation of dielectric constant is implemented in the Si9000e Insertion Loss Field Solver by employing the Extended Substrate Data options.

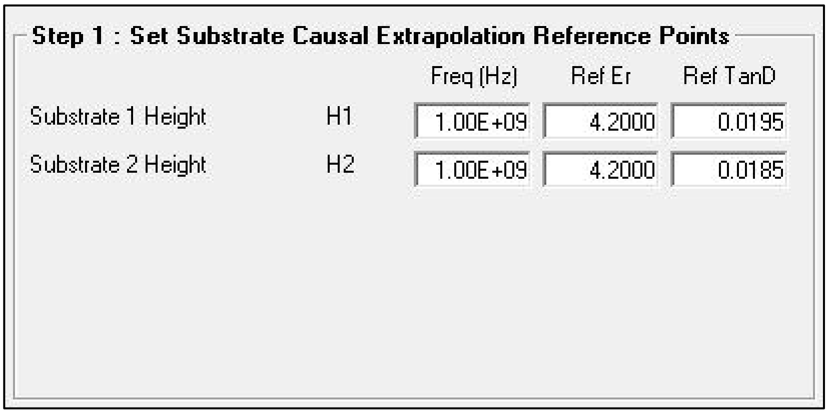
Causally extrapolating substrate data



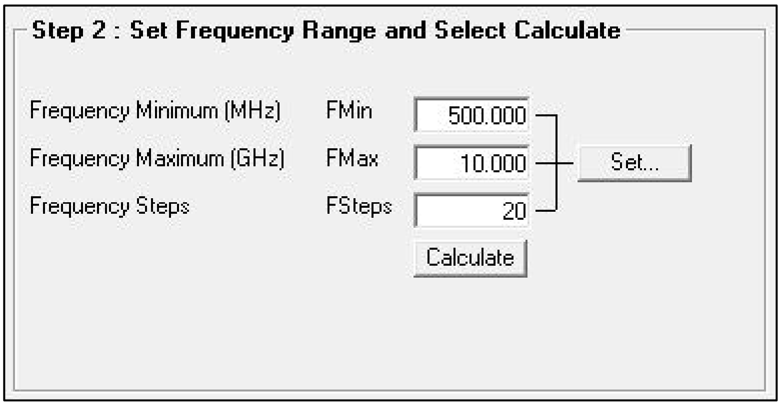
For this example, choose the Causally Extrapolate Er / TanD option from the Extended Substrate Data option group to apply Svensson-Djordjevic modelling to each dielectric layer in the current controlled impedance structure.

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

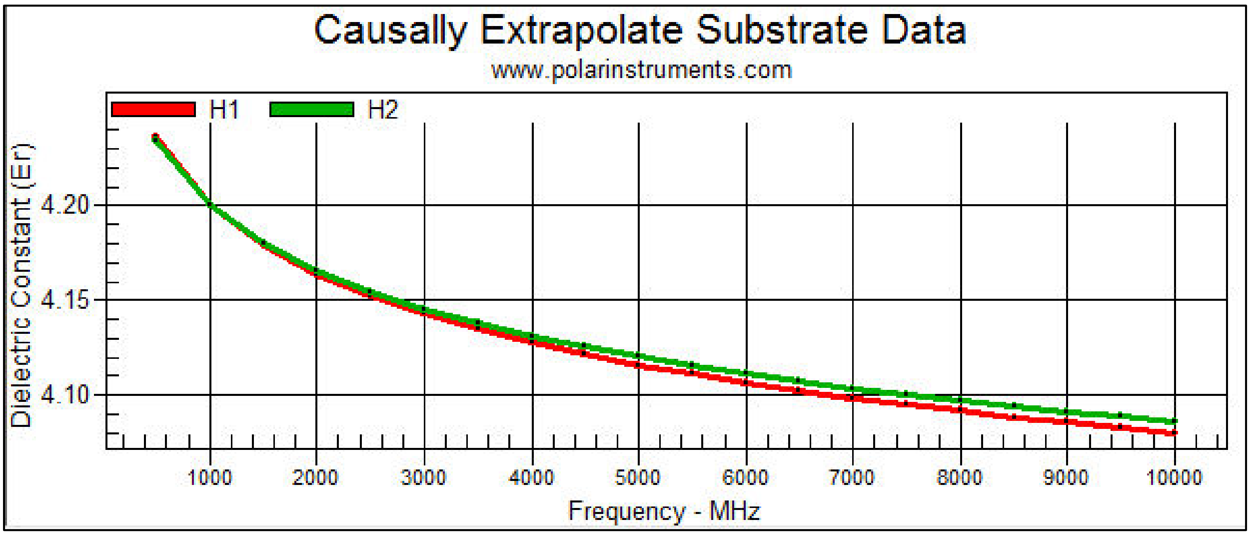
The Causally Extrapolate Substrate Data dialog is displayed with an entry for each dielectric. Use the dialog to set substrate causal extrapolation reference points: values for frequency, Er and TanD.



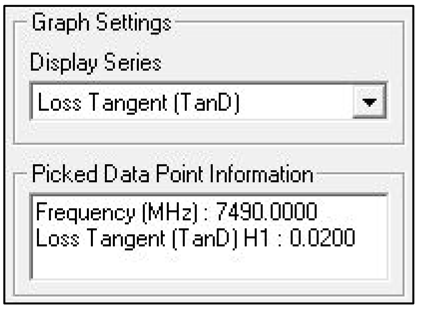
Set the frequency range and number of steps (or frequency increments.)



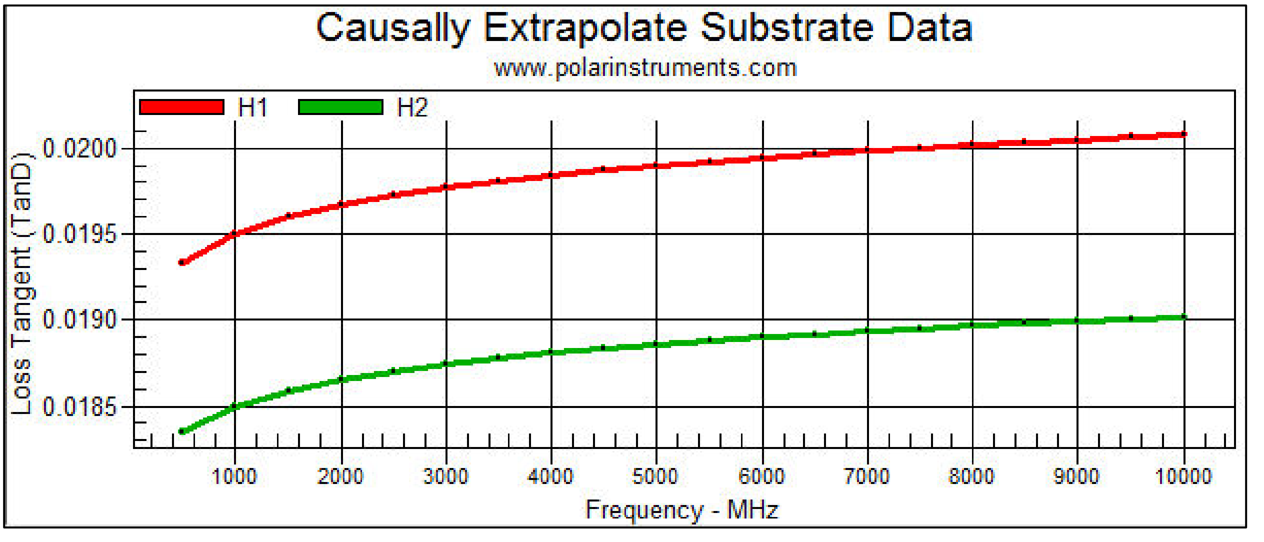
Click Calculate: Si9000e charts Dielectric Constant (Er) v frequency for each dielectric.



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

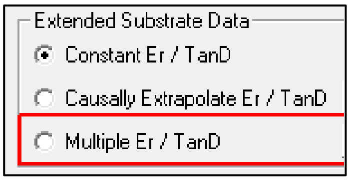
Si9000e frequency-dependent calculations can be refined using *extended substrate data*.

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

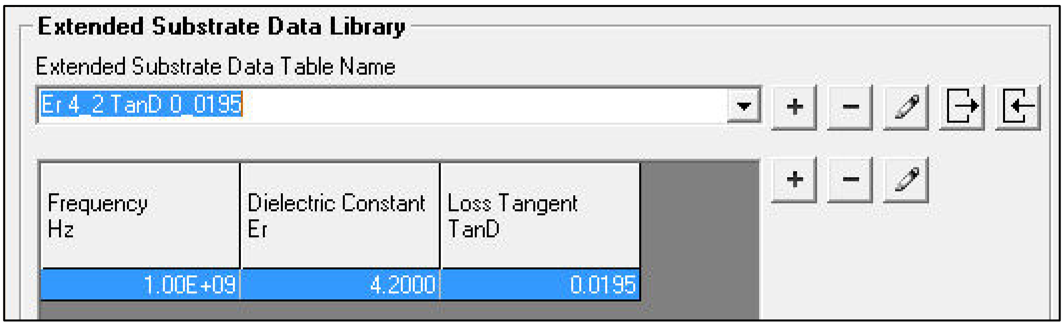


Using the Multiple Er / TanD option the Si9000 can accept tables of multiple values of dielectric constant and loss tangent or use a single value to enable Svensson-Djordjevic frequency dependent permittivity modelling.

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

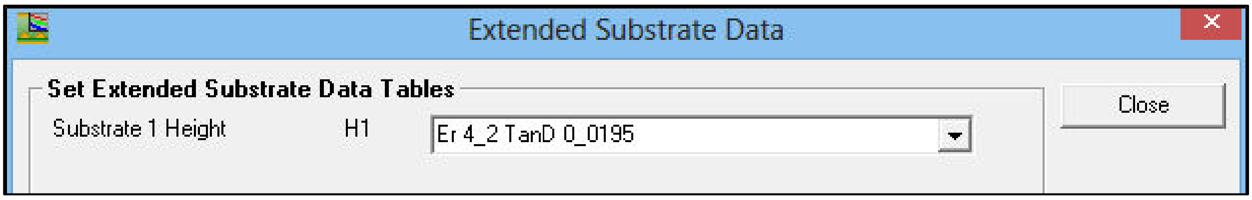
Creating a single-entry table

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.



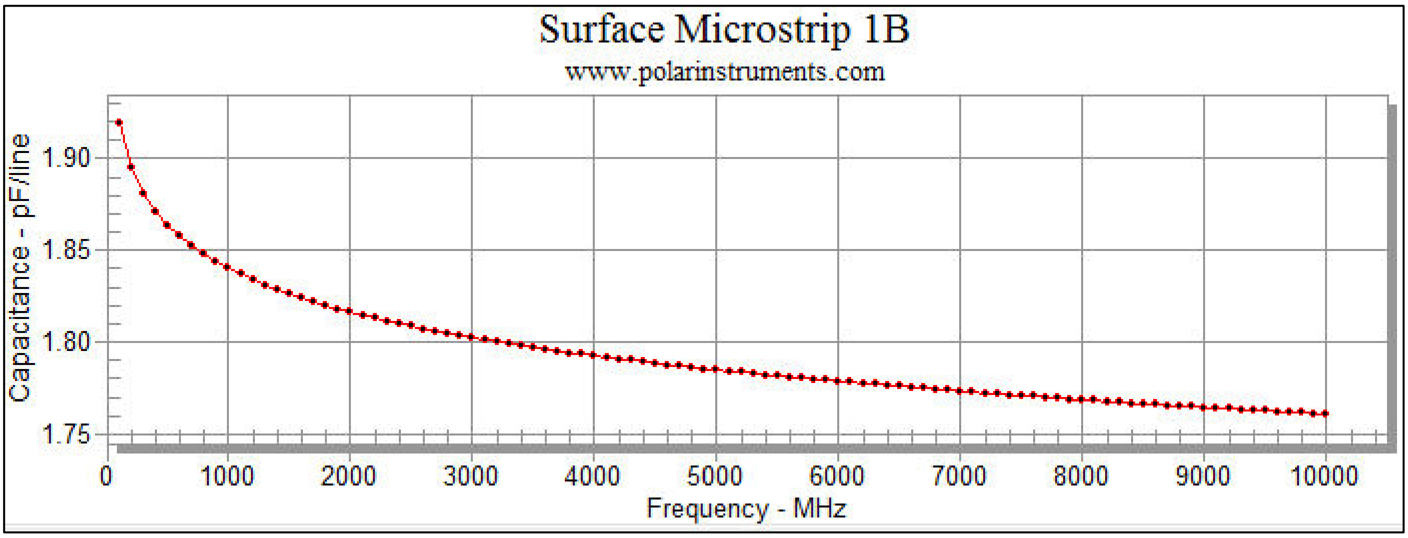
Choosing the table

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.

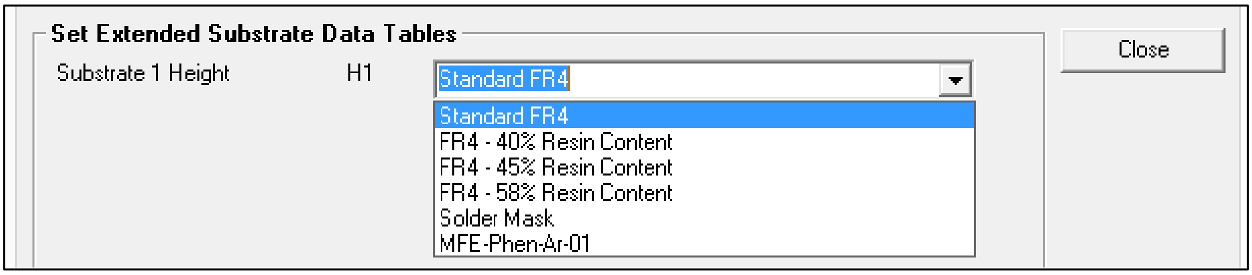
For this example, choose Capacitance – note the variation in capacitance with frequency. Compare with the frequency independent modelling above.



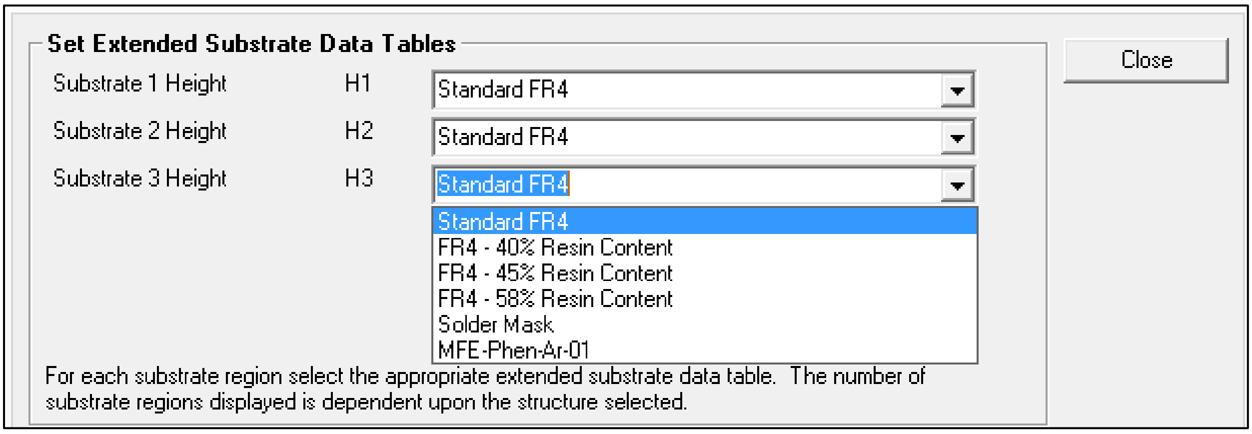
Note that the Si9000e applies frequency dependent permittivity modelling even though Er and TanD are specified with single values (i.e. as constants.)

Choosing a dielectric layer frequency profile

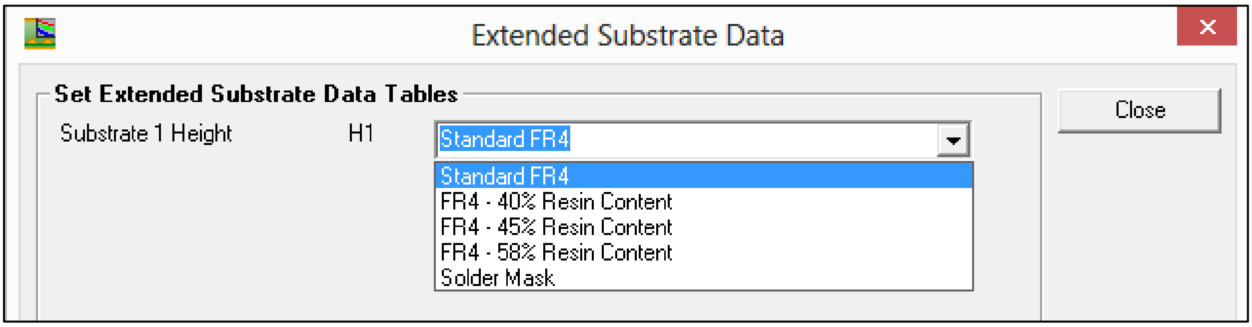
To choose a dielectric layer frequency profile, click the Edit button in the Extended Substrate Data screen area; the Extended Substrate Data dialog is displayed. A frequency profile table may be specified for each dielectric layer.



Single dielectric layer



Multiple dielectric layers



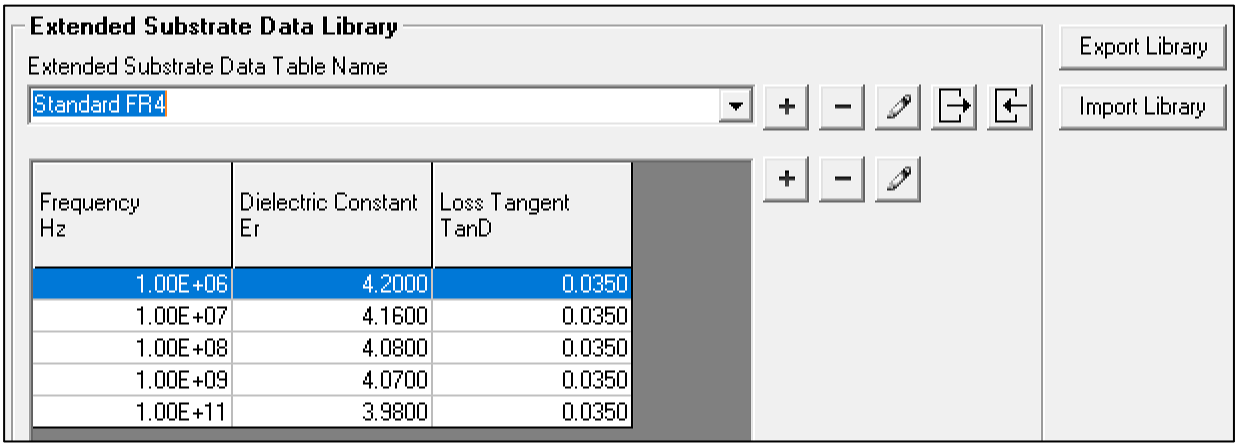
Click the dropdown list box arrow to display the list of available tables. For each dielectric layer choose a layer profile. Click Close.

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.

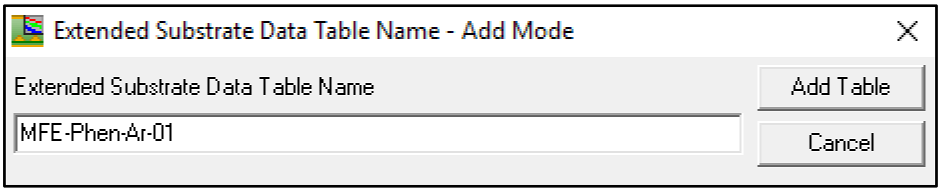
Tables may be added and edited as described below or imported and exported in pipe-delimited .ESL format or in comma-separated .CSV format, suitable for editing, for example, in Microsoft Excel®.



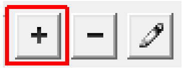
Adding a table

Add Table button

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.

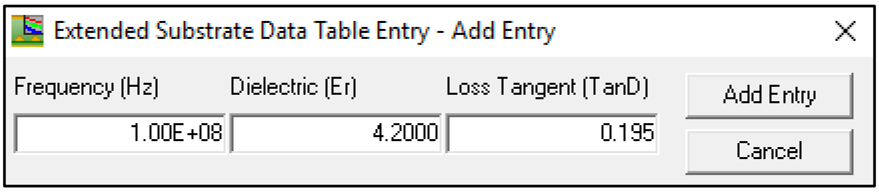


Adding data to the table

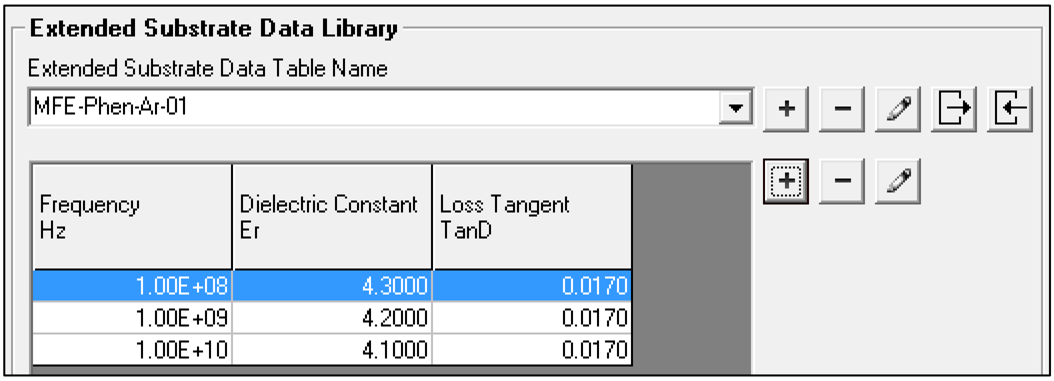


Add Entry

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.

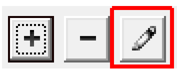


Each band is added to the table in ascending order of frequency. In this example the dielectric constant, Er decreases with frequency, but loss tangent, TanD remains constant.



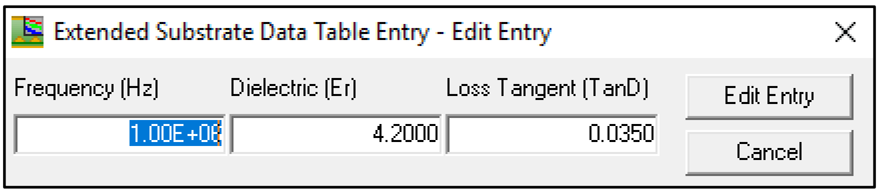
Editing and deleting table data

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

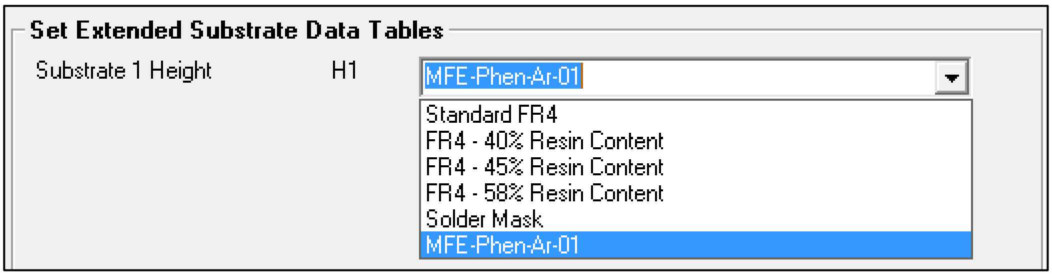


Edit entry button

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.

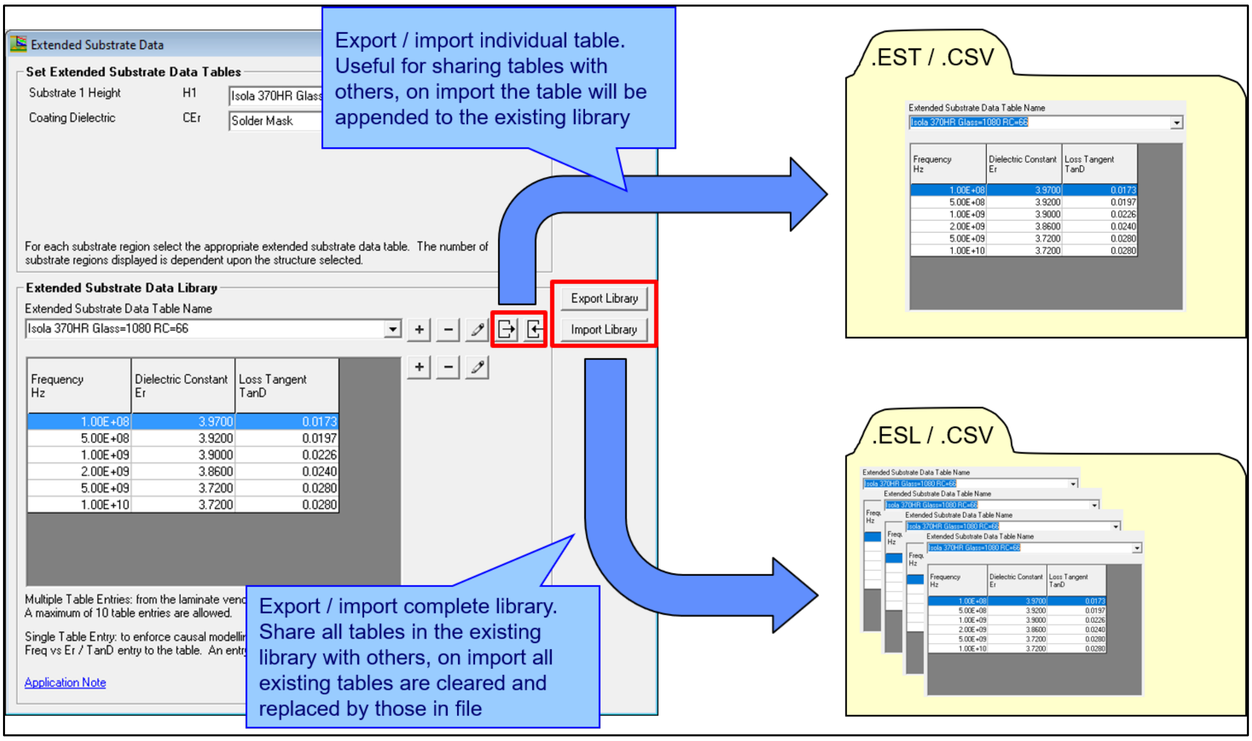


To use the new table, select the table from the dropdown list in the Set Extended Substrate Data Tables section of the dialog.



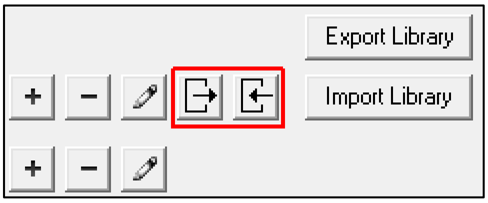
Importing and exporting material tables

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

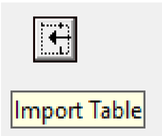


Use the Import and Export Table controls to read in or export existing tables. Tables can be exported into .EST (pipe delimited) or .CSV format suitable for editing in Microsoft Excel.®

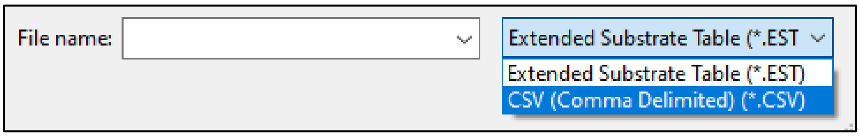
Libraries of tables can be imported or exported as .ESL files or as .CSV files



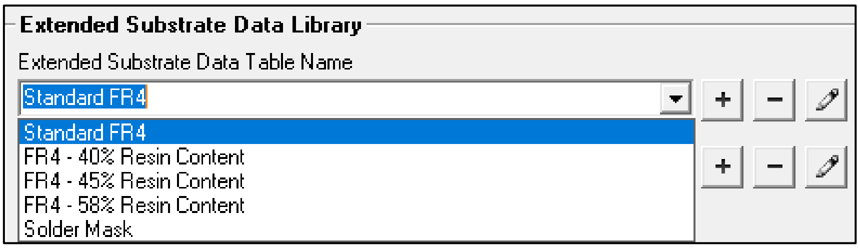
Importing individual tables



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.

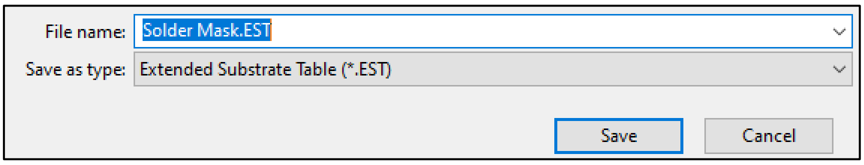


The table will be appended to the Library and may be selected via the Extended Substrate Table Name drop-down.



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®.

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
1	Polar Instruments Extended Substrate Table			Version	2											
2	Table Name	Freq 1	Er 1	TanD 1	Freq 2	Er 2	TanD 2	Freq 3	Er 3	TanD 3	Freq 4	Er 4	TanD 4	Freq 5	Er 5	TanD 5
3	FR4 - 45% Resin Content	1000000	3.89	0.01	10000000	3.81	0.01	100000000	3.79	0.01	1000000000	3.76	0.01	1E+11	3.71	0.01
4																
5																

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

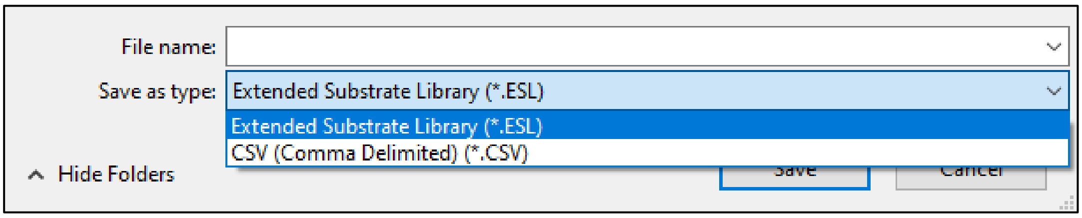
Note: The layout and format of the extended substrate table must be preserved when editing – alterations may prevent a successful subsequent import.

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.



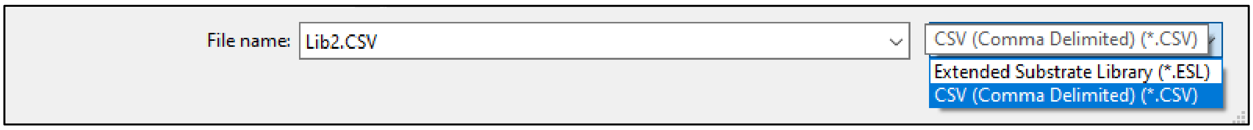
Open the file in a text editor or spreadsheet; a typical library export is illustrated below – each table is shown as a row in the spreadsheet.

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
1	Polar Instruments Extended Substrate Library				Version	2										
2	Table Name	Freq 1	Er 1	TanD 1	Freq 2	Er 2	TanD 2	Freq 3	Er 3	TanD 3	Freq 4	Er 4	TanD 4	Freq 5	Er 5	TanD 5
3	Standard FR4	1000000	4.2	0.035	10000000	4.16	0.035	100000000	4.08	0.035	1000000000	4.07	0.035	1.00E+11	3.98	0.035
4	FR4 - 40% Resin Content	1000000	4	0.01	10000000	3.96	0.01	100000000	3.92	0.01	1000000000	3.91	0.01	1.00E+11	3.82	0.01
5	FR4 - 45% Resin Content	1000000	3.89	0.01	10000000	3.81	0.01	100000000	3.79	0.01	1000000000	3.76	0.01	1.00E+11	3.71	0.01
6	FR4 - 58% Resin Content	1000000	3.59	0.01	10000000	3.57	0.01	100000000	3.51	0.01	1000000000	3.5	0.01	1.00E+11	3.41	0.01
7	Solder Mask	1000000	4	0.03	1E+11	4	0.03	0	0	0	0	0	0	0	0	0

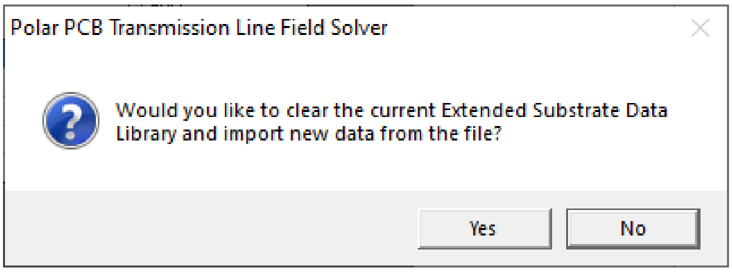
Note: If the library table is edited, the layout and format of the table must be preserved when editing – alterations may prevent a successful subsequent import.

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.

Graph | Single Ended | SPICE RLGC | 2 Port S-Parameters - Graph | 2 Port S-Parameters - Data | Measurement Data

Frequency Hz	R Matrix Ohms/in	L Matrix H/in	G Matrix S/in	C Matrix F/in
5.000E+08	1.389E+00 0.000E+00 0.000E+00 1.389E+00	8.992E-09 0.000E+00 0.000E+00 8.992E-09	2.127E-04 0.000E+00 0.000E+00 2.127E-04	3.473E-12 0.000E+00 0.000E+00 3.473E-12
1.000E+09	1.996E+00 0.000E+00 0.000E+00 1.996E+00	8.899E-09 0.000E+00 0.000E+00 8.899E-09	4.255E-04 0.000E+00 0.000E+00 4.255E-04	3.473E-12 0.000E+00 0.000E+00 3.473E-12
1.500E+09	2.458E+00 0.000E+00 0.000E+00 2.458E+00	8.859E-09 0.000E+00 0.000E+00 8.859E-09	6.382E-04 0.000E+00 0.000E+00 6.382E-04	3.473E-12 0.000E+00 0.000E+00 3.473E-12
2.000E+09	2.846E+00 0.000E+00 0.000E+00 2.846E+00	8.835E-09 0.000E+00 0.000E+00 8.835E-09	8.510E-04 0.000E+00 0.000E+00 8.510E-04	3.473E-12 0.000E+00 0.000E+00 3.473E-12
2.500E+09	3.186E+00 0.000E+00 0.000E+00 3.186E+00	8.818E-09 0.000E+00 0.000E+00 8.818E-09	1.064E-03 0.000E+00 0.000E+00 1.064E-03	3.473E-12 0.000E+00 0.000E+00 3.473E-12
3.000E+09	3.492E+00 0.000E+00 0.000E+00 3.492E+00	8.806E-09 0.000E+00 0.000E+00 8.806E-09	1.276E-03 0.000E+00 0.000E+00 1.276E-03	3.473E-12 0.000E+00 0.000E+00 3.473E-12

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.

Graph | Odd Mode | Even Mode | SPICE RLGC | 4 Port S-Parameters - Graph | 4 Port S-Parameters - Data | Mixed Mode S-Parameters - Graph | Mixed Mode S-Parameters - Data

Frequency Hz	Impedance Real Ohms	Impedance Imaginary Ohms	Impedance Magnitude Ohms	Inductance H/line	Resistance Ohms/line	Capacitance F/line	Conductance S/line	Skin Depth in	Conductor Loss dB/line	Dielectric Loss dB/line	Attenuation dB/line
5.000E+08	5.089E+01	-4.857E-01	5.089E+01	6.915E-09	7.471E-01	2.671E-12	1.283E-04	1.164E-04	-6.376E-02	-2.835E-02	-9.211E-02
1.000E+09	5.063E+01	-2.321E-01	5.063E+01	6.845E-09	1.052E+00	2.671E-12	2.566E-04	8.228E-05	-9.026E-02	-5.642E-02	-1.467E-01
1.500E+09	5.051E+01	-1.196E-01	5.051E+01	6.814E-09	1.286E+00	2.671E-12	3.849E-04	6.718E-05	-1.106E-01	-8.443E-02	-1.950E-01
2.000E+09	5.044E+01	-5.250E-02	5.044E+01	6.795E-09	1.484E+00	2.671E-12	5.132E-04	5.818E-05	-1.277E-01	-1.124E-01	-2.402E-01
2.500E+09	5.039E+01	-6.675E-03	5.039E+01	6.783E-09	1.657E+00	2.671E-12	6.415E-04	5.204E-05	-1.428E-01	-1.404E-01	-2.832E-01
3.000E+09	5.036E+01	2.716E-02	5.036E+01	6.773E-09	1.815E+00	2.671E-12	7.699E-04	4.750E-05	-1.565E-01	-1.684E-01	-3.249E-01
3.500E+09	5.033E+01	5.347E-02	5.033E+01	6.766E-09	1.959E+00	2.671E-12	8.982E-04	4.398E-05	-1.690E-01	-1.963E-01	-3.654E-01
4.000E+09	5.031E+01	7.467E-02	5.031E+01	6.760E-09	2.094E+00	2.671E-12	1.026E-03	4.114E-05	-1.807E-01	-2.243E-01	-4.050E-01
4.500E+09	5.029E+01	9.224E-02	5.029E+01	6.756E-09	2.220E+00	2.671E-12	1.155E-03	3.879E-05	-1.917E-01	-2.522E-01	-4.439E-01
5.000E+09	5.028E+01	1.071E-01	5.028E+01	6.752E-09	2.340E+00	2.671E-12	1.283E-03	3.679E-05	-2.021E-01	-2.802E-01	-4.823E-01
5.500E+09	5.026E+01	1.199E-01	5.026E+01	6.748E-09	2.453E+00	2.671E-12	1.411E-03	3.508E-05	-2.120E-01	-3.081E-01	-5.201E-01
6.000E+09	5.025E+01	1.311E-01	5.025E+01	6.745E-09	2.562E+00	2.671E-12	1.540E-03	3.359E-05	-2.214E-01	-3.360E-01	-5.574E-01
6.500E+09	5.024E+01	1.409E-01	5.024E+01	6.742E-09	2.666E+00	2.671E-12	1.668E-03	3.227E-05	-2.305E-01	-3.640E-01	-5.944E-01
7.000E+09	5.023E+01	1.497E-01	5.023E+01	6.740E-09	2.766E+00	2.671E-12	1.796E-03	3.110E-05	-2.392E-01	-3.919E-01	-6.311E-01
7.500E+09	5.023E+01	1.575E-01	5.023E+01	6.738E-09	2.863E+00	2.671E-12	1.925E-03	3.004E-05	-2.476E-01	-4.198E-01	-6.674E-01
8.000E+09	5.022E+01	1.647E-01	5.022E+01	6.736E-09	2.957E+00	2.671E-12	2.053E-03	2.909E-05	-2.557E-01	-4.477E-01	-7.034E-01
8.500E+09	5.021E+01	1.712E-01	5.021E+01	6.734E-09	3.047E+00	2.671E-12	2.181E-03	2.822E-05	-2.636E-01	-4.757E-01	-7.392E-01
9.000E+09	5.020E+01	1.771E-01	5.021E+01	6.733E-09	3.135E+00	2.671E-12	2.310E-03	2.743E-05	-2.712E-01	-5.036E-01	-7.748E-01
9.500E+09	5.020E+01	1.826E-01	5.020E+01	6.731E-09	3.221E+00	2.671E-12	2.438E-03	2.669E-05	-2.787E-01	-5.315E-01	-8.102E-01
1.000E+10	5.019E+01	1.876E-01	5.019E+01	6.730E-09	3.304E+00	2.671E-12	2.566E-03	2.602E-05	-2.859E-01	-5.594E-01	-8.453E-01

Differential mode data

Graphing impedance variation with frequency

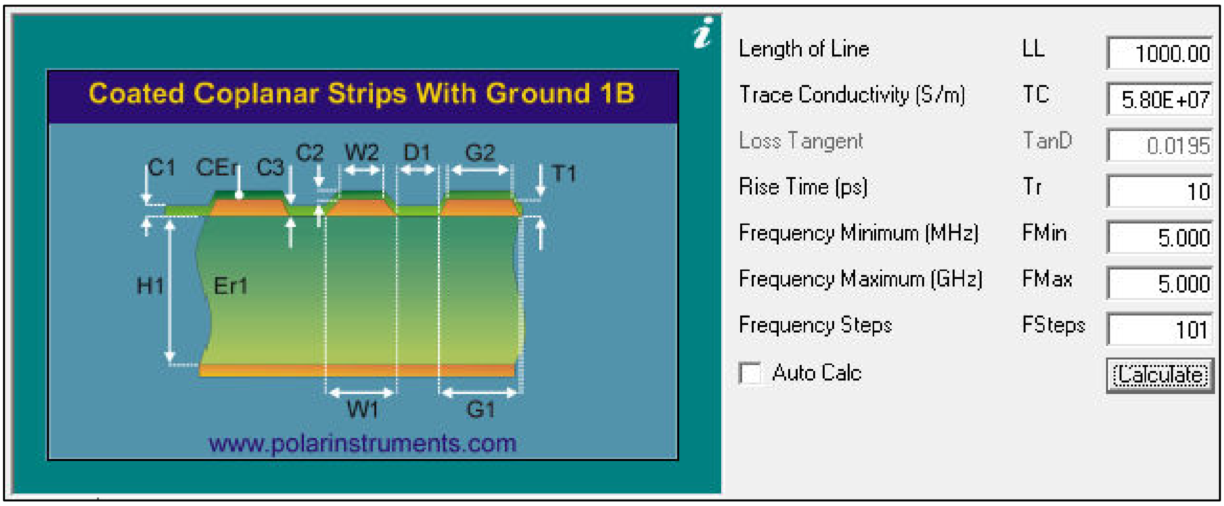
Transmission line impedance

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

$$Z_o = \sqrt{\frac{L}{C}}$$

(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.

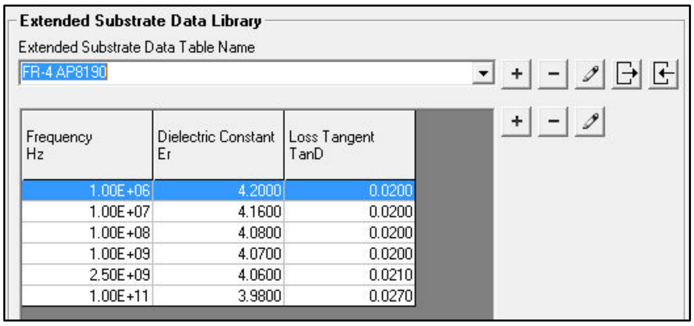
Choose the Coated Coplanar Strips with Ground 1B structure.



Frequency-dependent calculations

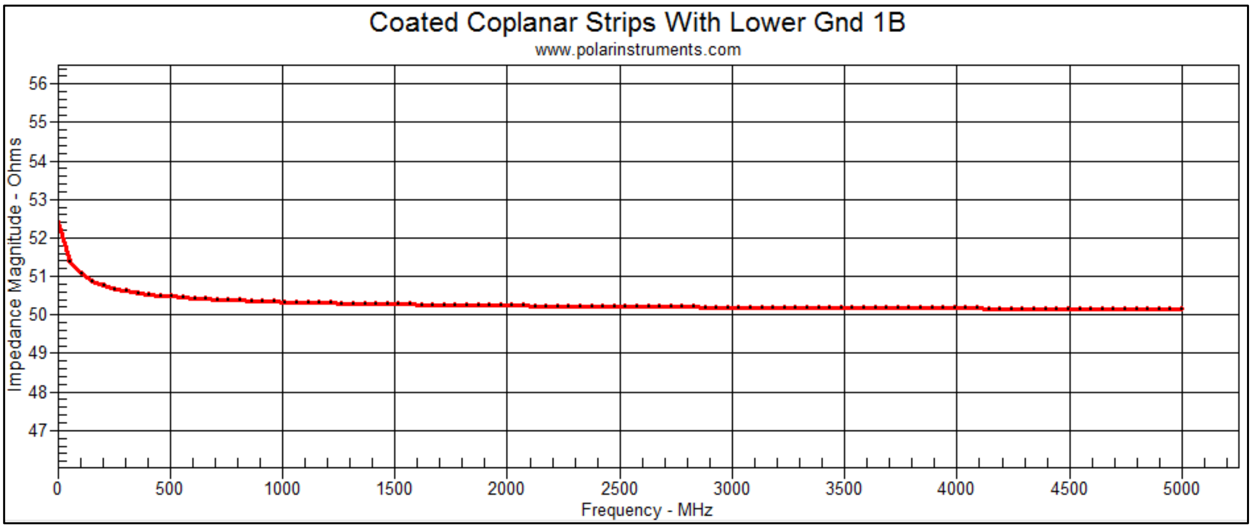
Using the Extended Substrate Data table Edit function, add a table describing the frequency-dependent behaviour of the substrate material.

For this example, supply the values in the table below. Er and Loss Tangent are defined for the frequencies of interest; for this material Er decreases with frequency.



Frequency Hz	Dielectric Constant Er	Loss Tangent TanD
1.00E+06	4.2000	0.0200
1.00E+07	4.1600	0.0200
1.00E+08	4.0800	0.0200
1.00E+09	4.0700	0.0200
2.50E+09	4.0600	0.0210
1.00E+11	3.9800	0.0270

Graphing Z_0 against frequency in the Si9000e shows that as the Er decreases the impedance, Z_0, also decreases.



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

Frequency Hz	Impedance Magnitude Ohms	Inductance H/line	Resistance Ohms/line	Capacitance F/line	Conductance S/line	Skin Depth in	Conductor Loss dB/line	Dielectric Loss dB/line	Attenuation dB/line
5.000E+06	5.274E+01	7.982E-09	7.503E-02	2.995E-12	1.684E-06	1.164E-03	-6.244E-03	-3.776E-04	-6.622E-03
5.600E+08	5.116E+01	7.660E-09	5.110E-01	2.927E-12	1.839E-04	1.099E-04	-4.338E-02	-4.086E-02	-8.424E-02
1.115E+09	5.104E+01	7.618E-09	7.191E-01	2.924E-12	3.671E-04	7.792E-05	-6.118E-02	-8.138E-02	-1.426E-01
1.670E+09	5.100E+01	7.599E-09	8.790E-01	2.921E-12	5.595E-04	6.367E-05	-7.485E-02	-1.239E-01	-1.988E-01
2.225E+09	5.098E+01	7.588E-09	1.014E+00	2.919E-12	7.582E-04	5.516E-05	-8.635E-02	-1.679E-01	-2.542E-01
2.780E+09	5.097E+01	7.580E-09	1.132E+00	2.918E-12	9.560E-04	4.935E-05	-9.649E-02	-2.116E-01	-3.081E-01
3.335E+09	5.095E+01	7.574E-09	1.240E+00	2.918E-12	1.149E-03	4.505E-05	-1.057E-01	-2.542E-01	-3.598E-01
3.890E+09	5.094E+01	7.570E-09	1.338E+00	2.917E-12	1.342E-03	4.172E-05	-1.141E-01	-2.968E-01	-4.109E-01
4.445E+09	5.093E+01	7.566E-09	1.430E+00	2.917E-12	1.535E-03	3.902E-05	-1.220E-01	-3.396E-01	-4.616E-01
5.000E+09	5.092E+01	7.564E-09	1.517E+00	2.917E-12	1.730E-03	3.679E-05	-1.293E-01	-3.825E-01	-5.119E-01

The Inductance column shows that at the lower frequency end the inductance is changing as indicated above – and as Er 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 Er will have more predominant effect.

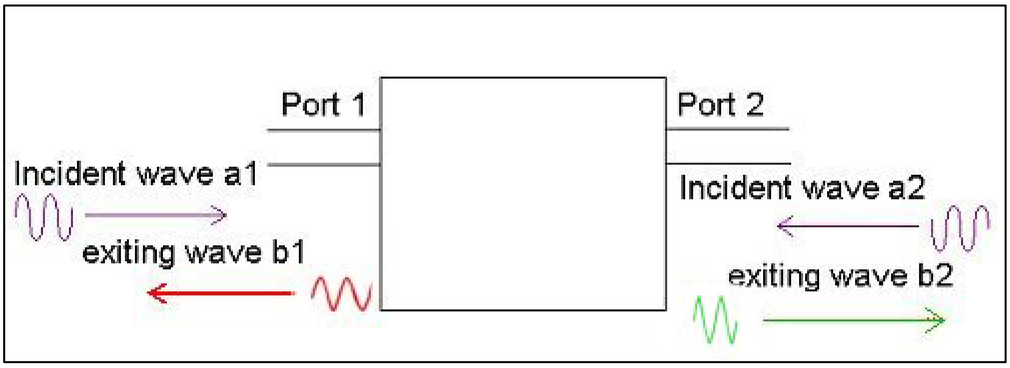
However, as the frequency rises the change in Er also slows down, so the Z_0 reduction that is observed as the frequency rises to fully develop the skin effect is expected behaviour.

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, b1 and b2, exiting from each port in terms of incident waves, a1 and a2,



where s-parameters:

S_{11} = b1 / a1

S_{12} = b1 / a2

S_{21} = b2 / a1

S_{22} = b2 / a2.

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.

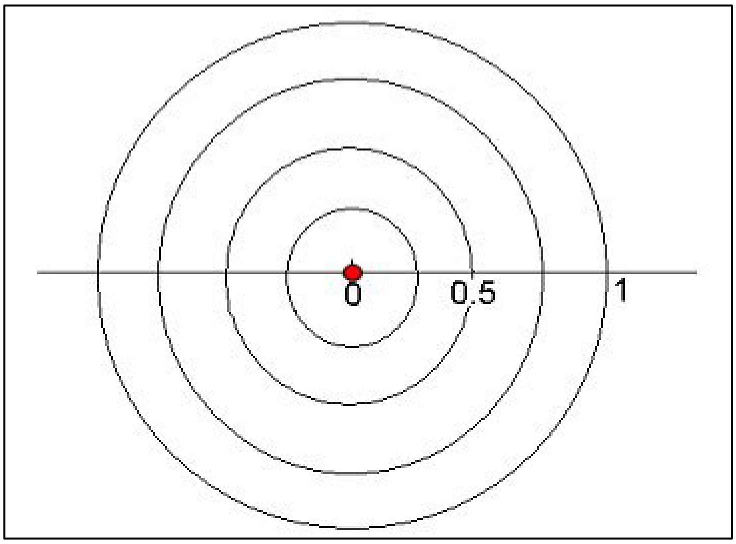
S-parameters can be graphed in several ways; one option is to use two graphs (magnitude v frequency and phase v frequency) to represent one s-parameter.

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

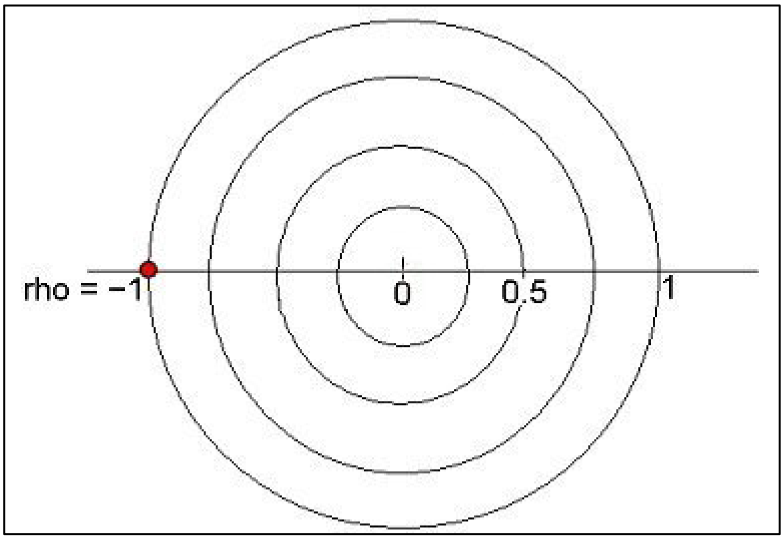
Smith Charts

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.

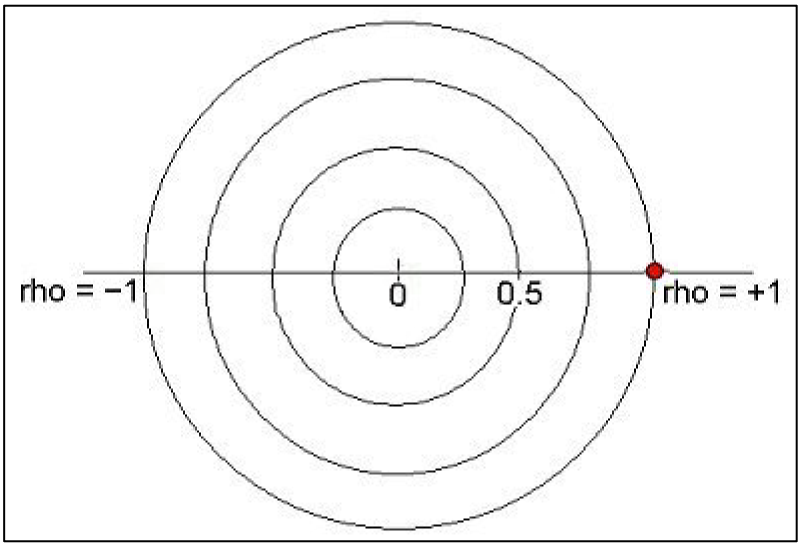
Consider the following graphs of reflection coefficient, *rho*. Smith Charts are constructed within the circle described when rho is unity.



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.

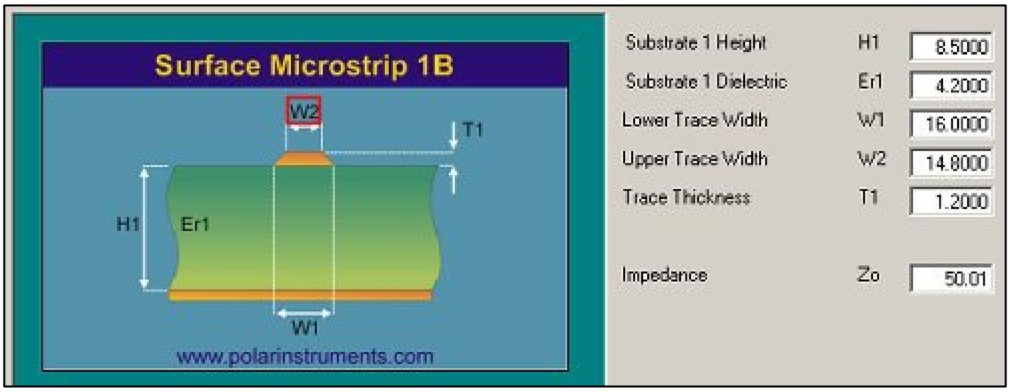


Similarly, a point plotted at right-hand edge shows 100% +ve reflection, i.e. unity reflection with no phase change, implying a transmission line terminated with an open circuit.

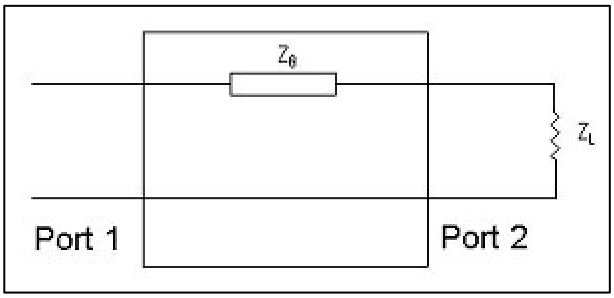
A point plotted anywhere on this circle, a reflection coefficient of unity, shows a perfect reflection at different phase angles.

Plotting s-parameters on the Si9000e Smith chart

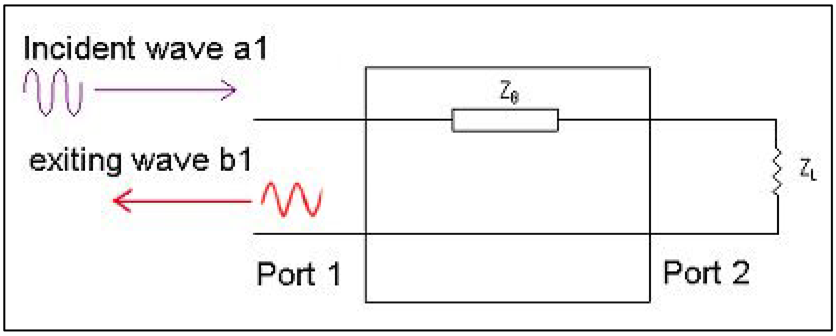
Consider a 50 Ohm structure



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.



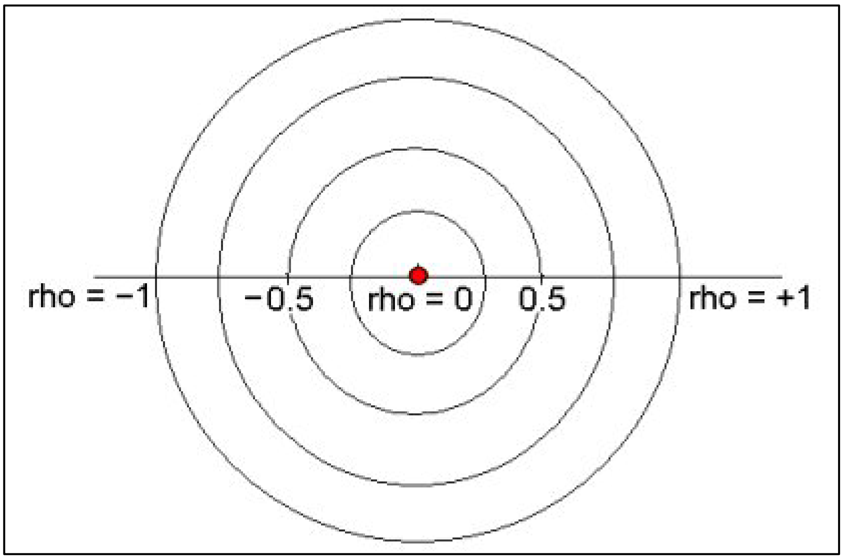
Plotting reflection coefficient



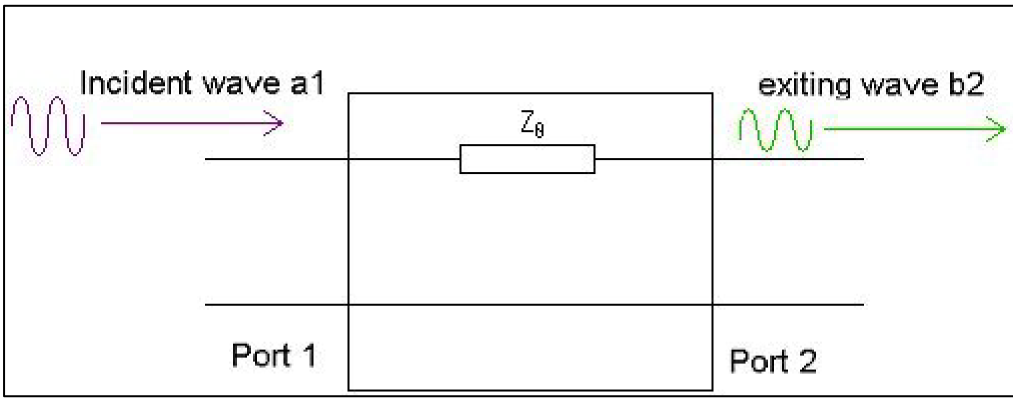
S-parameter S_{11} = b1 / a1

In a perfect system $Z_0 = Z_L$ = 50 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



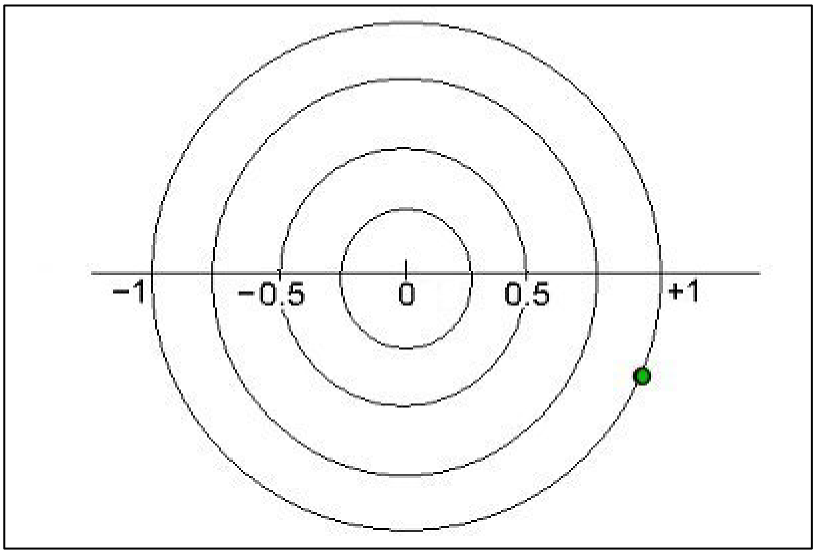
Plotting transmission coefficient



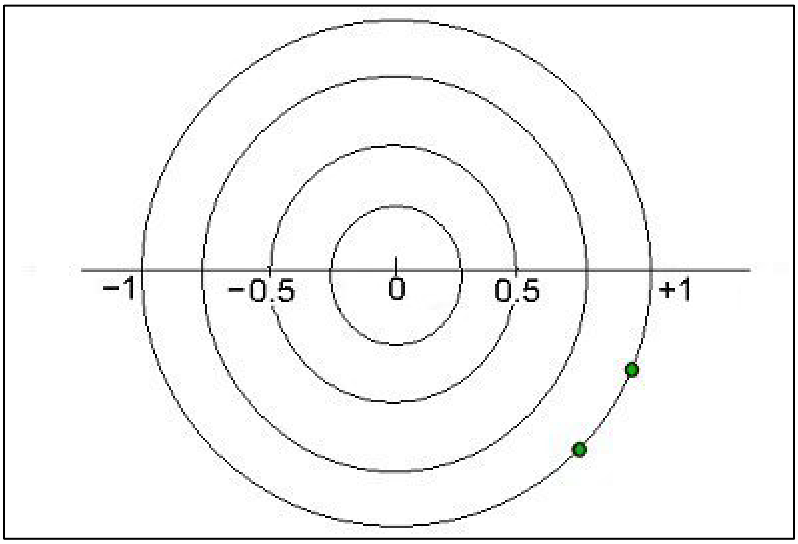
S-parameter S_{21} = b2 / a1

In a perfect system there is no loss and the signal passes through the transmission line unattenuated:
the magnitude of b2 = magnitude of a1 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.

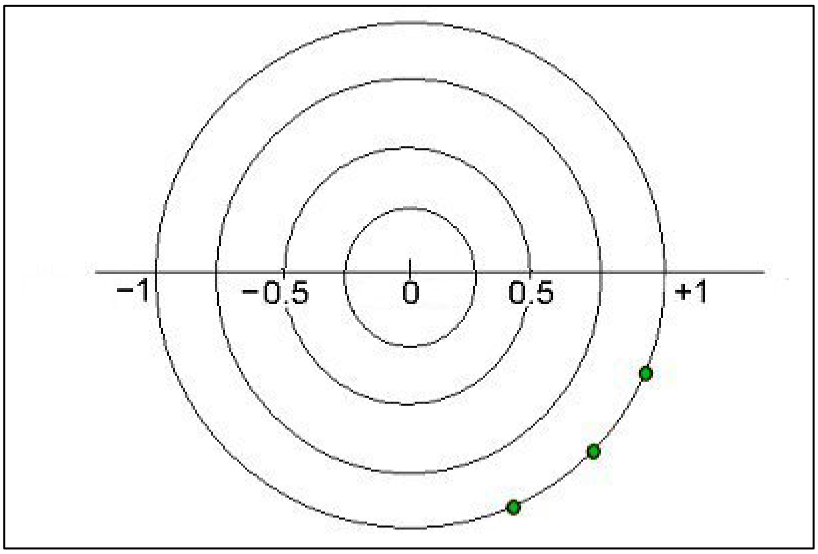


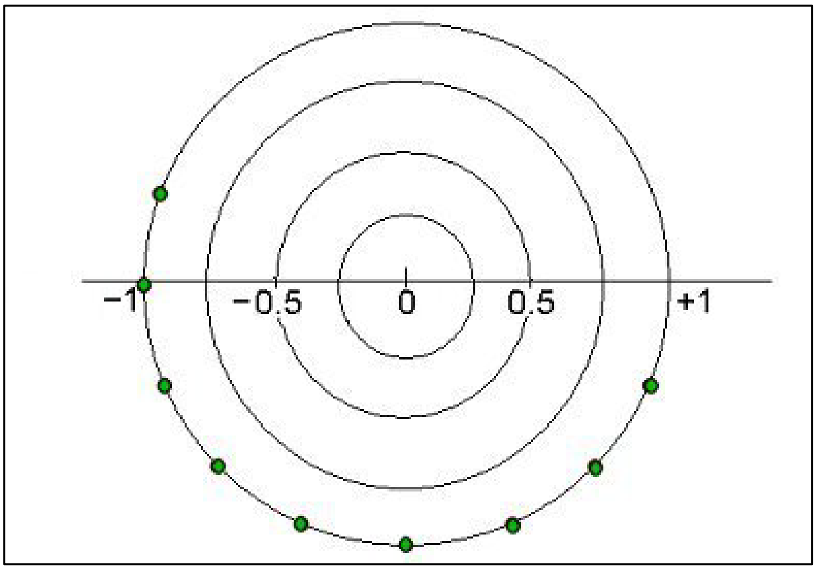
Where this dot is plotted would depend on the phase shift through the transmission line. If the frequency were increased, and other S_{21} frequency readings obtained, the magnitude of S_{21} would still be 1 but the phase shift would change.



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.



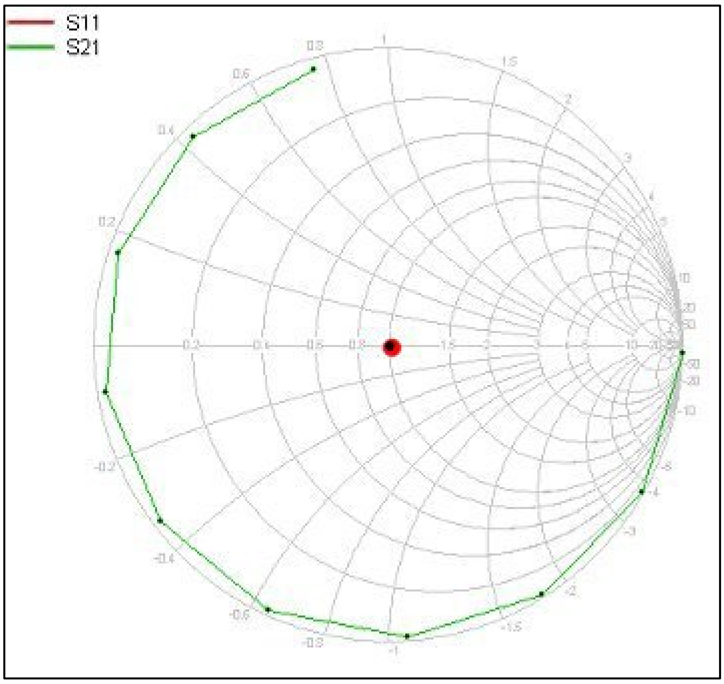


S-parameters can thus be used to completely characterise a network; the values of the s-parameters change with frequency and can be plotted on 2 conventional graphs, magnitude v frequency & phase v frequency or on a Smith Chart.

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.

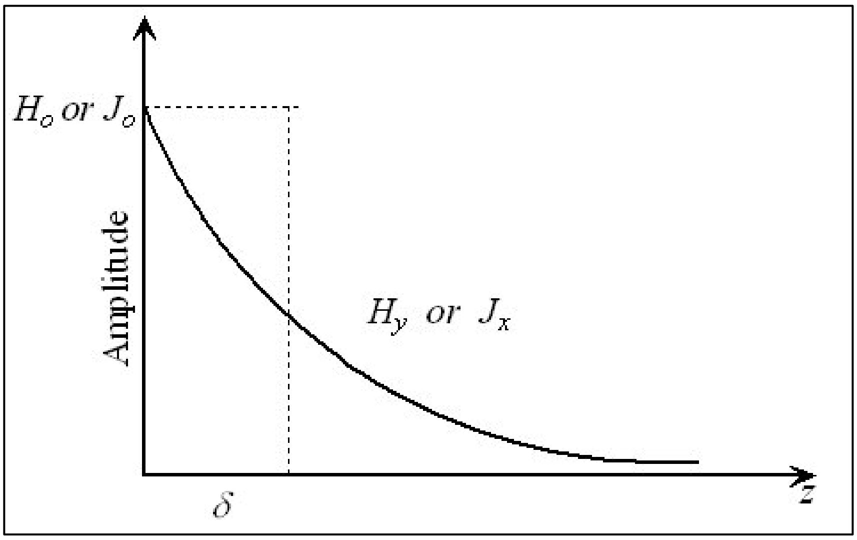
Surface roughness effect on PCB trace attenuation / loss

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



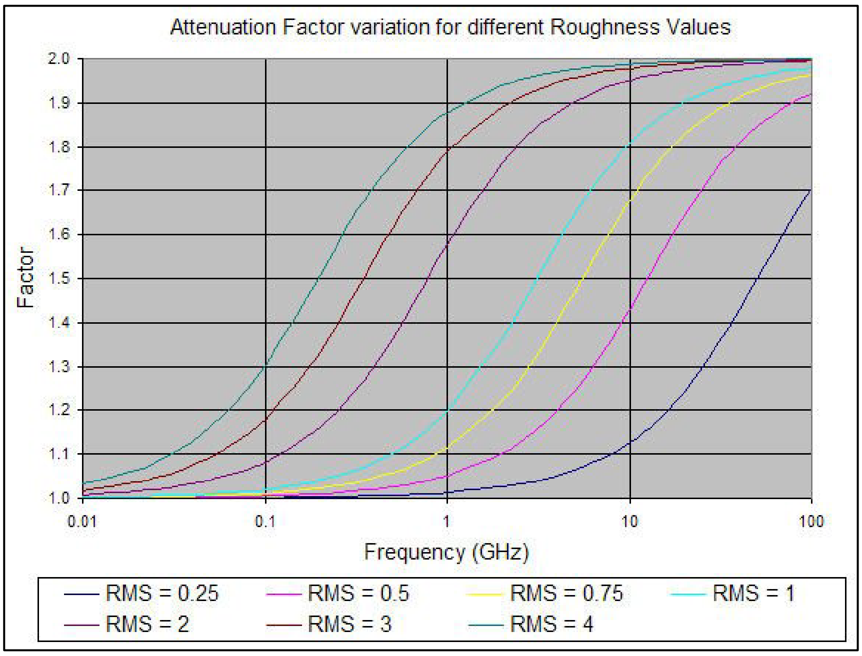
The diagram above graphs the amplitude of magnetic field against depth (*z*) into a conductor and shows the variation of the amplitude of magnetic field H_y in the z-direction where

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 δ is the value of z at which $|J_x| = J_{0/e}$. This is also the same value at which the rectangular area δJ_0 in the diagram equals the area under the exponential curve. δ is known as the Skin Depth.

Surface roughness

At very high frequencies (where skin depth δ 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)

Hammerstad modelling

Groisse modelling

Gradient modelling

Huray / Simonovich-Cannonball modelling

The *Smooth* copper option provides for no compensation for copper loss.

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

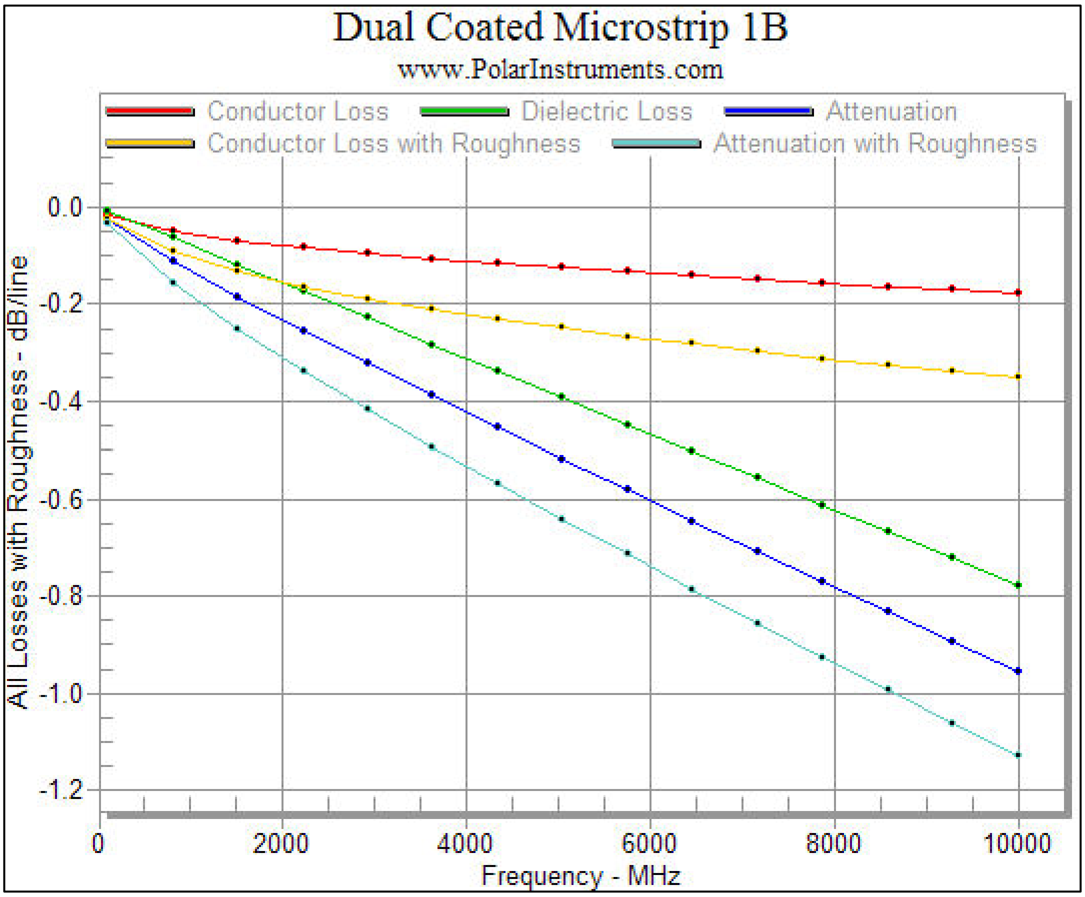
Gradient modelling models microscopic roughness as a continuous transition of conductivity perpendicular to the conductor surface, i.e., a conductivity gradient.

Huray and Simonovich-Cannonball modelling extends the roughness modelling validity up to 40 to 50GHz and possibly higher, but is more demanding in terms of input.

However, as a rule of thumb, if the detailed SEM measurement information needed for Huray is not available, 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, Groisse, Gradient modelling

Using the modified Hammerstad, Groisse or Gradient 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.

Choosing the right Surface Roughness Measurement

Many surface roughness compensation models exist to help designers and fabricators model the effect that copper roughness has on insertion loss. The Si9000e provides several methods to meet your – or your OEM's – requirements. Hammerstad, Groisse and Gradient methods accept RMS roughness (Rq.) Hammerstad, Groisse are legacy methods which are only valid up to a few GHz. Huray (Cannonball) accepts Rz or SEM data if available. *Choosing the correct surface roughness measurement for the model input is as important as choosing the correct model.* The commonly used measurements are Ra, Rq, and Rz

- Ra is the absolute average of the profile values
- Rq is the root mean square (RMS) of the profile values.
- Rz is the peak to valley height – see below.

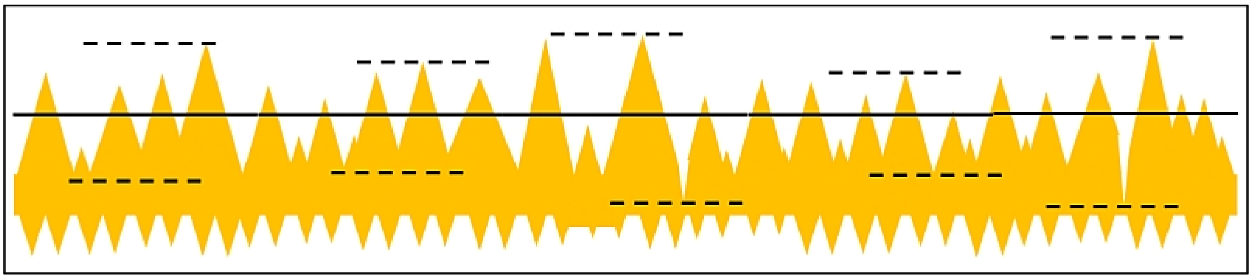
Different surface roughness compensation models will require different inputs. Some require Rz while others require Rq. It is important to note that these numbers come from different methods of summarizing a surface profile.

Any numerical conversion from one method to another should be handled with care and with the understanding that the conversion is only an approximation and can contribute to garbage ingoing to the model.

In addition, there are different ways to calculate Rz. The most common methods of calculating Rz come from the German Institute of Standards (DIN), Japanese Industrial Standards (JIS), and International Organization of Standardization (ISO.)

Each organisation employs its own methodology. The illustration below shows an example of a cross section profile for a roughness measurement and demonstrates the existence of various summarizing methods.

- Rz (DIN) uses an absolute average of the five highest peaks and five lowest valleys over a sample length.
- Rz (JIS) utilizes an absolute average of the five highest peaks and five lowest valleys over five sample lengths.
- Rz (ISO) is the maximum peak to valley distance over a sample length.



Cross section roughness profile

The graphic above shows an illustration of a cross section profile, illustrating why multiple Rz methodologies exist for summarizing peak to valley roughness.

In general, Rz DIN and Rz JIS are comparable, with Rz JIS always being smaller than Rz DIN, as Rz JIS incorporates more data points.

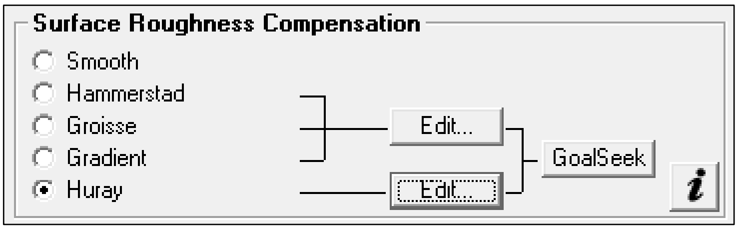
Rz ISO is not recommended as the number of data points is small.

Applying Surface Roughness Compensation

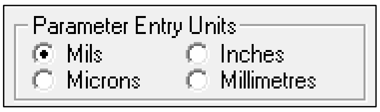
Hammerstad, Groisse or Gradient modelling

For the structure in use, in the Surface Roughness Compensation panel click the Hammerstad, Groisse or

Gradient option and Click Edit… to specify the RMS values for trace and plane roughness.

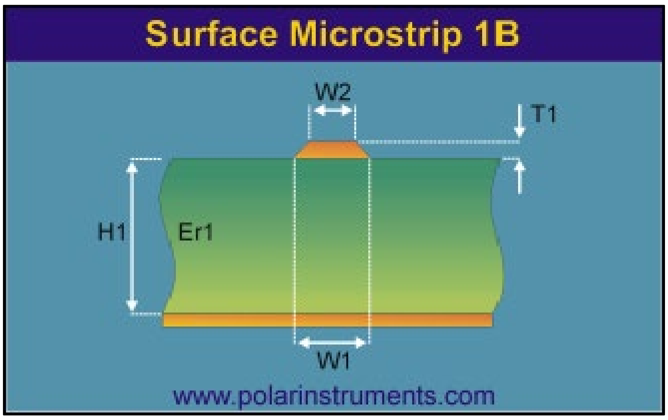


Values for surface roughness (obtainable in consultation with the board manufacturer) are specified in the currently chosen units.

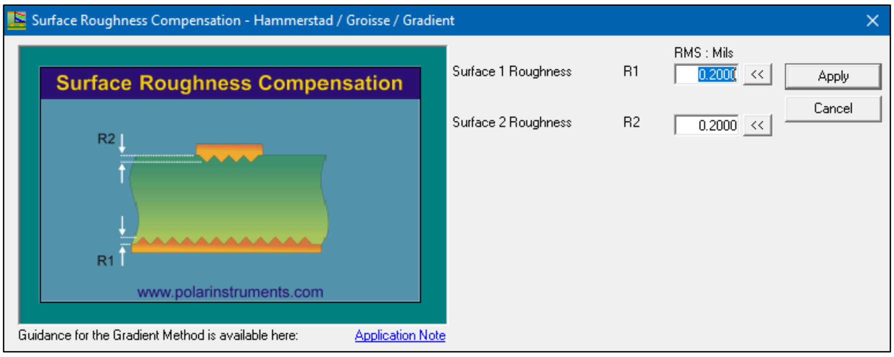


The Surface Roughness Compensation graphic reflects the current structure.

For example, for a Surface Microstrip structure:

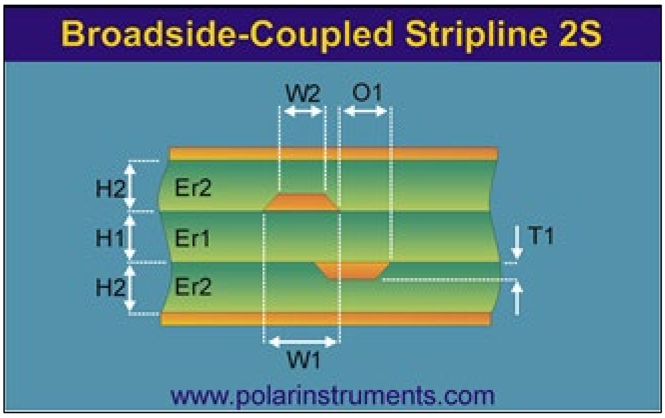


The Surface Roughness Compensation dialog allows the roughness values for the surfaces to be specified – as shown below.

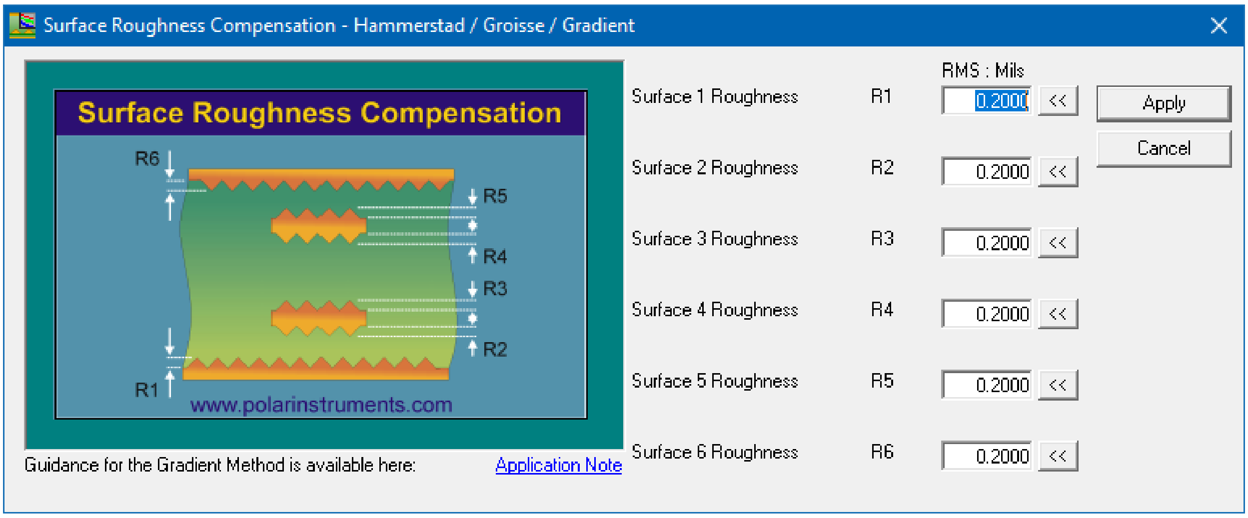


Roughness values may be set for each surface in the structure – *two surfaces* in the surface microstrip structure above.

The example below includes settings for all the surfaces of a broadside-coupled stripline structure.

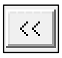


Note the *six surfaces* for the broadside-coupled stripline structure, and their associated values for roughness.

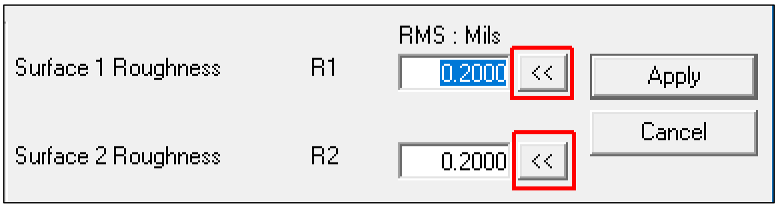


Values for surface roughness (obtainable in consultation with the board manufacturer) are specified in the currently chosen units. Typical values for RMS roughness could be 0.8 µm (0.03mils) for stripline, 1.6µm (0.06mils) for surface microstrip. The Si9000e assumes losses on all sides of a copper trace.

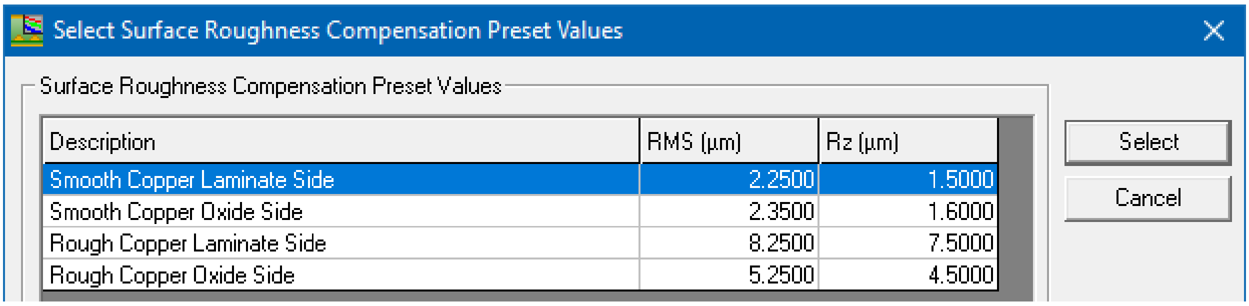
Selecting Surface Roughness Compensation Preset Values



Click the Preset Value button to use pre-stored roughness values



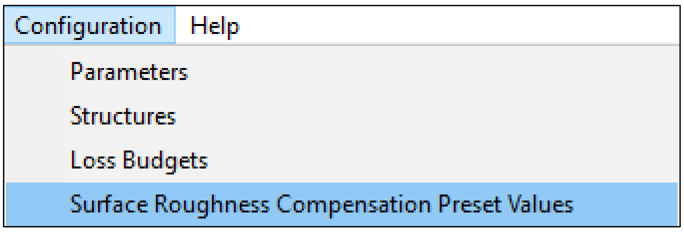
The table of preset values for the associated surface is displayed.



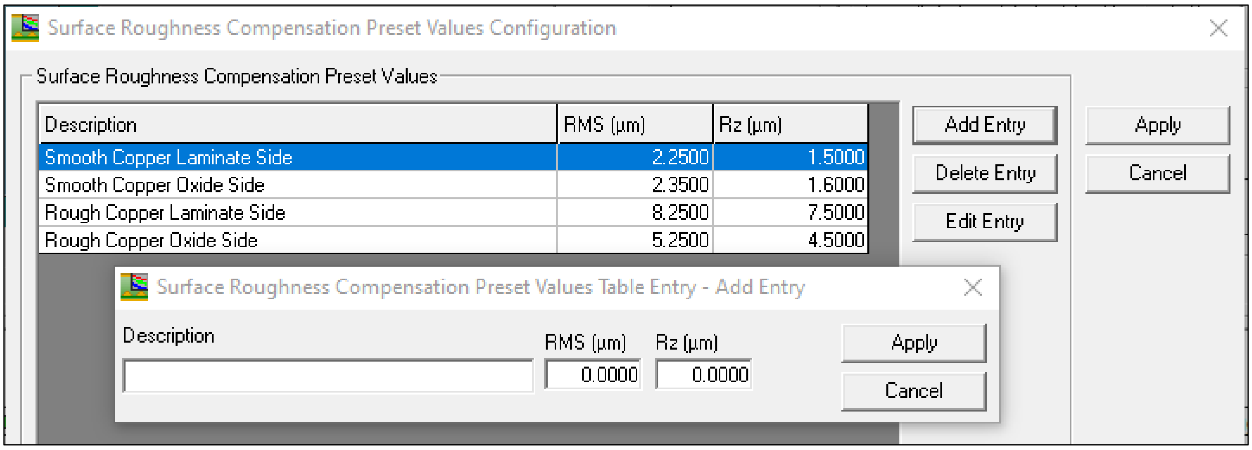
Note that the roughness values in the table are shown in microns regardless of the parameter entry units chosen.

Choose the value as appropriate from the table.

The table may be edited via the Configuration menu – Surface Roughness Compensation Preset Values.

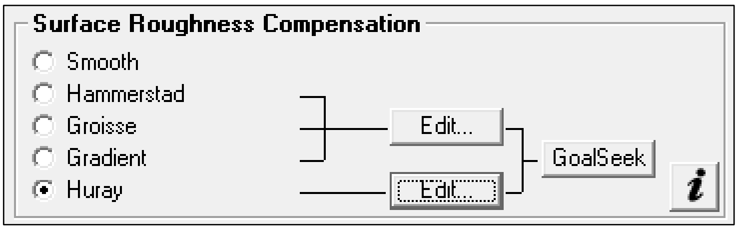


Edit the table, adding, modifying or deleting preset roughness values – click Apply to finish.



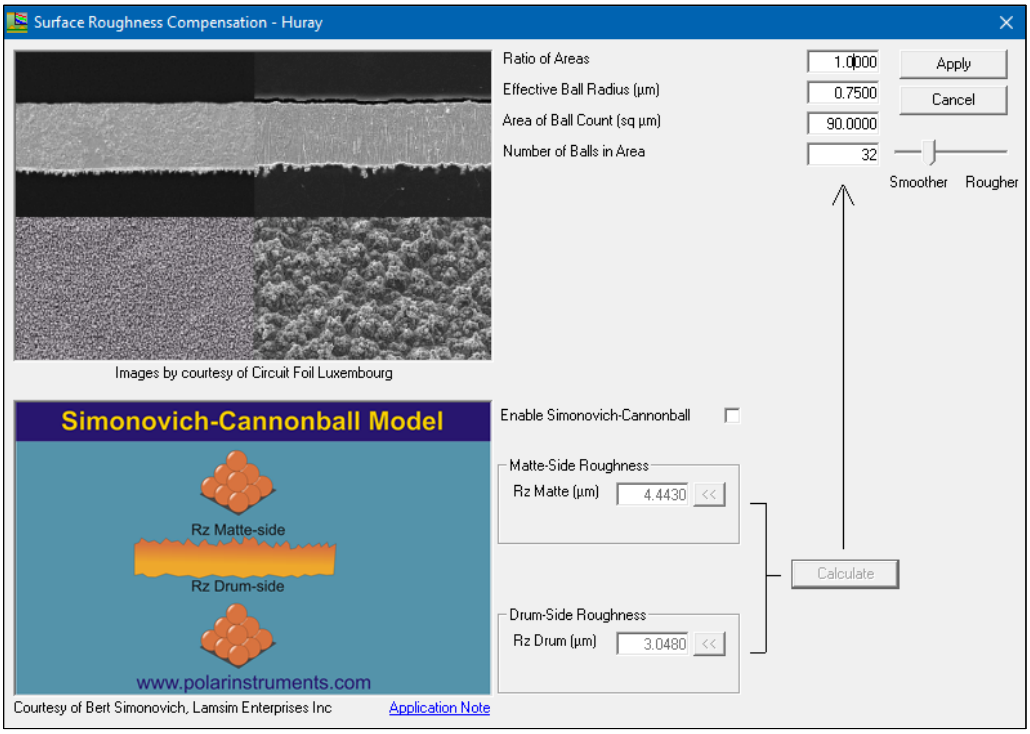
Huray modelling

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

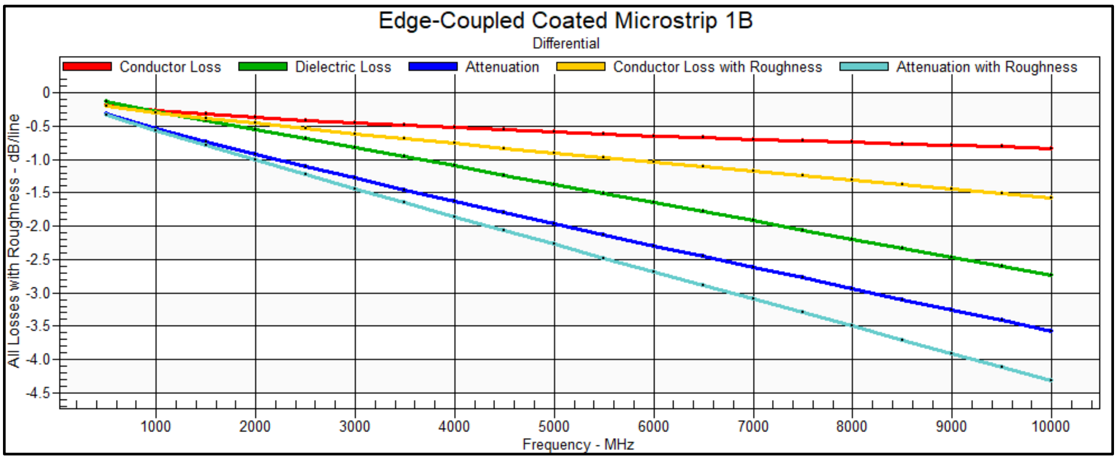


Huray modelling extends the roughness modeling validity up to 40 to 50GHz (and possibly beyond).

Click the Huray Edit button and specify the parameters for the Huray spheres (designated Balls In the dialog below.)



Supply the values in the associated fields and click Apply. If the Huray values are not available, click Enable Simonovich-Cannonball 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 Application Note AP553. This note contains links to two white papers by Bert Simonovich of LamSim Enterprises:

Practical Method for Modeling Conductor Surface Roughness Using The Cannonball Stack Principle

Heuristic Modeling of Transmission Lines due to Mixed Reference Plane Foil Roughness

Using Si9000e Loss Tangent Goal Seek

Goal seeking is a powerful method for seeking an unknown quantity; however, it should be used and approached with care. Should the input data be uncertain, goal seeking can lead to erroneous results. You should always bear in mind the result you expect to get before seeking and consider the results output against "sanity" tests – such as, "Do the results obtained obey the laws of physics?" The loss tangent you obtain with goal seeking, for example, should be similar to that of the material vendors data sheet. There should not be any sudden discontinuities in the loss tangent at a specific frequency. Should you see anomalies, that is a pointer to go back and look at the measurement technique – both the system, the de-embedding math and the design of the test vehicle.

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.

Isolating the copper and dielectric loss components

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

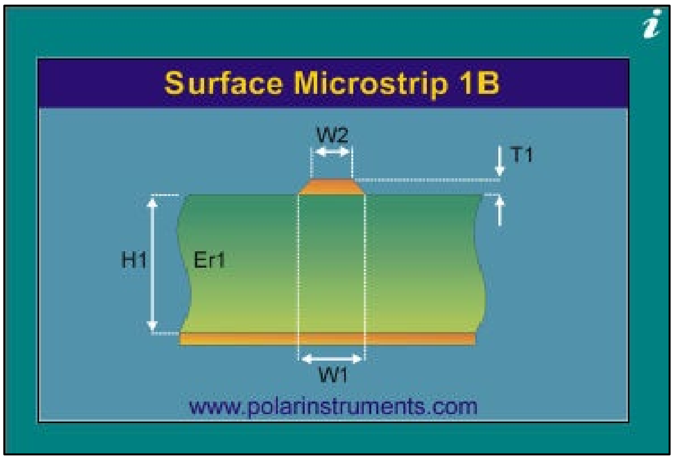
- enter the total measured attenuation
- calculate an estimate of copper losses from cross section data
- remove the copper loss from the total attenuation to leave the losses from the substrate alone.

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.

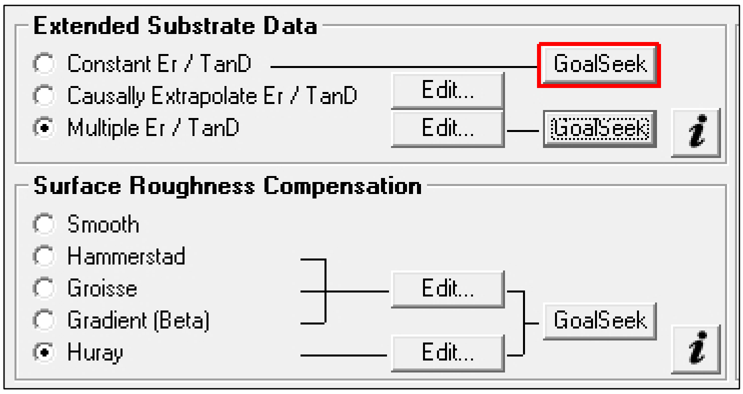
Si9000e can goal seek for loss tangent for both a single frequency and multiple frequencies.

Single Frequency Loss Tangent Goal Seek

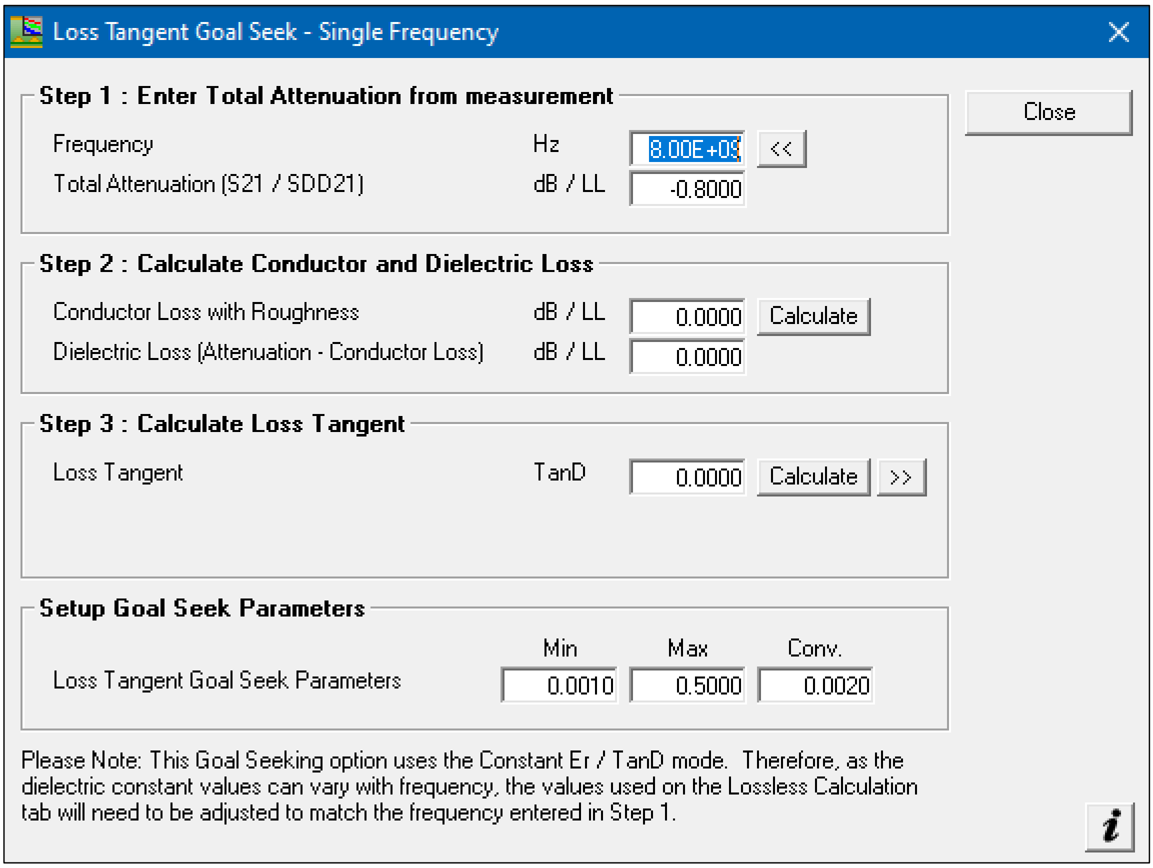
Select Surface Microstrip 1B structure with the default parameters:



From the Frequency Dependent tab select the Constant Er / TanD Loss Tangent Goal Seek option.

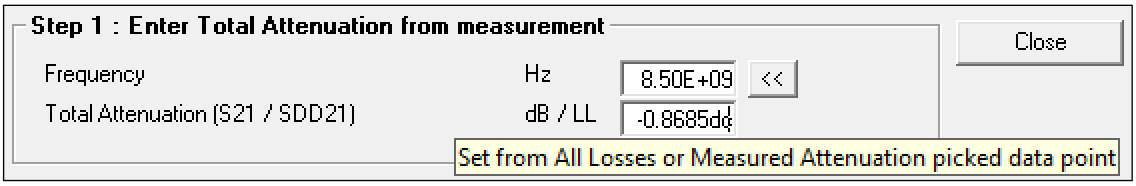


The Single Frequency Loss Tangent Goal Seek dialog is displayed.



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.

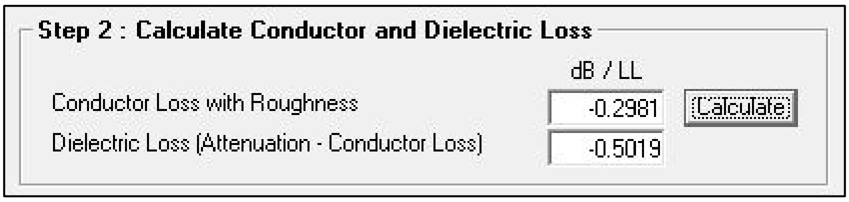


Note: The values for total attenuation can also be loaded from the currently picked data point on the displayed graph.

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

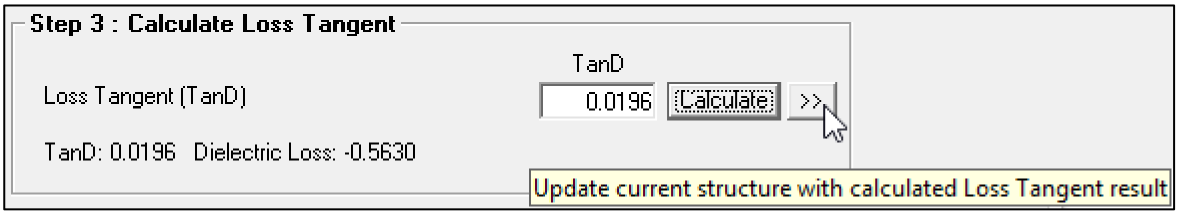
Under Step 2, Calculate the Conductor and Dielectric Loss. This will take the parameters for the current selected structure (Surface Microstrip 1B) and calculate the conductor loss at 8GHz and then take the calculated conductor loss from the total attenuation entered in Step 1 to calculate the remaining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

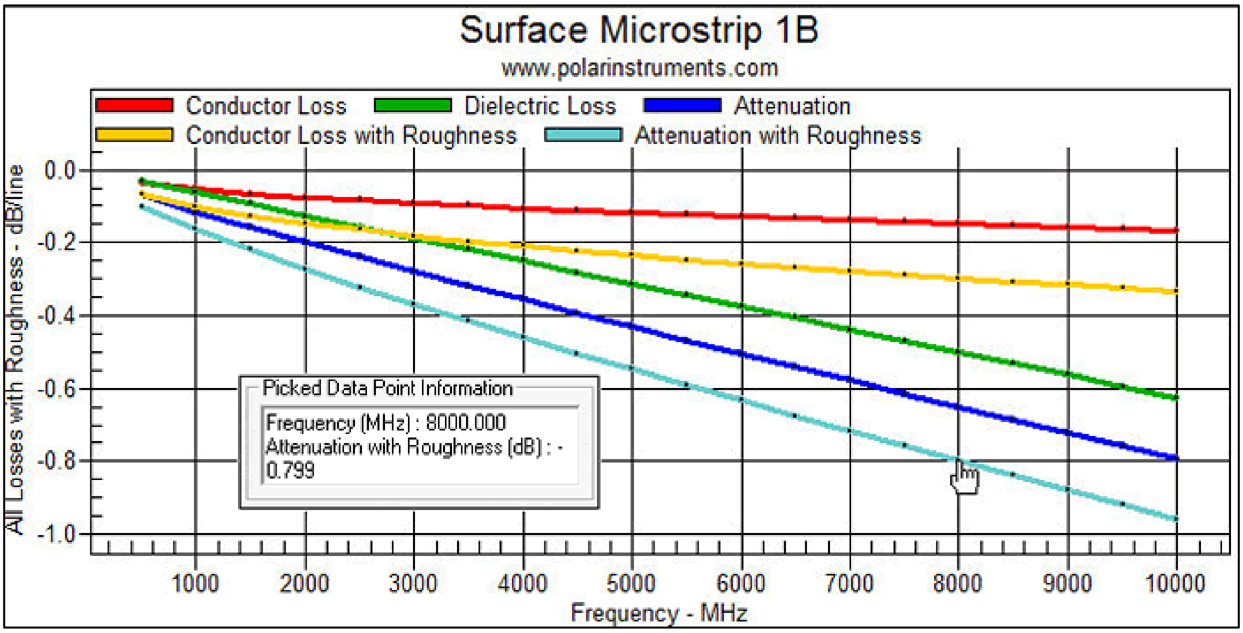


The result in this example is that the Loss Tangent (TanD) of 0.0196 is required to achieve a dielectric loss ~ –0.5019 dB/inch – the convergence value is used to give the target dielectric loss a tolerance.

Verifying results

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.

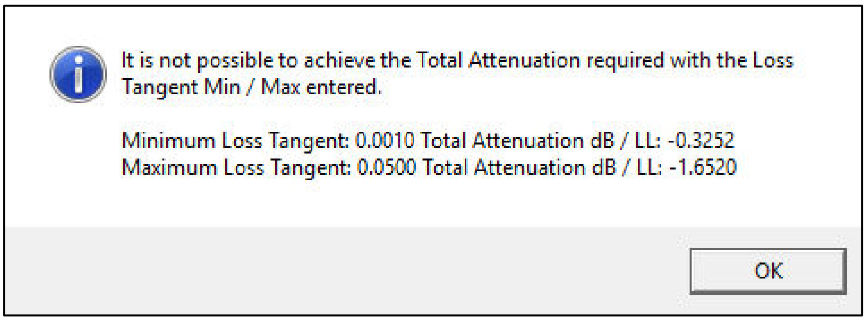
Close the dialog and click the frequency dependent Calculate button to update the Loss v Frequency graph.



Querying the 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.

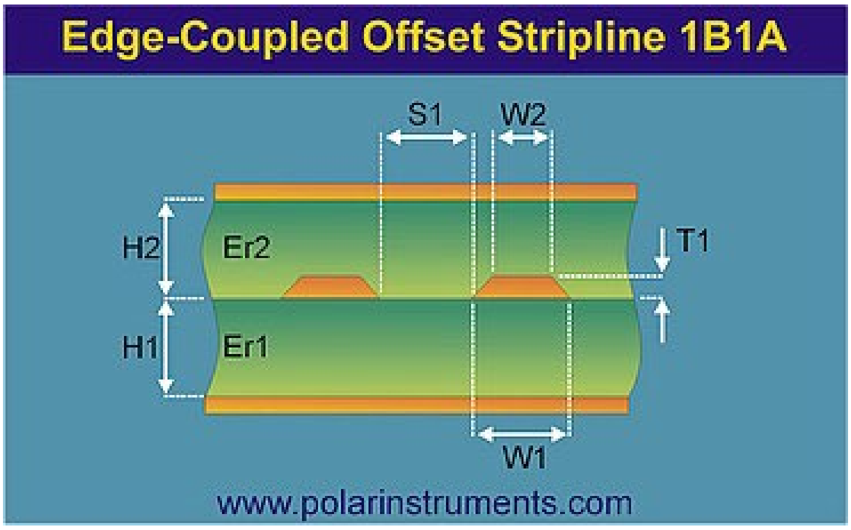
Multiple frequency loss tangent goal seek

Si9000e provides loss tangent goal seeking for multiple frequencies.

Up to five Loss Tangent values can be calculated in a single process; the calculated results can be exported to the Extended Substrate Data Library

Edge-Coupled Offset Stripline example

From the Structure bar select the Edge-Coupled Offset Stripline 1B1A structure



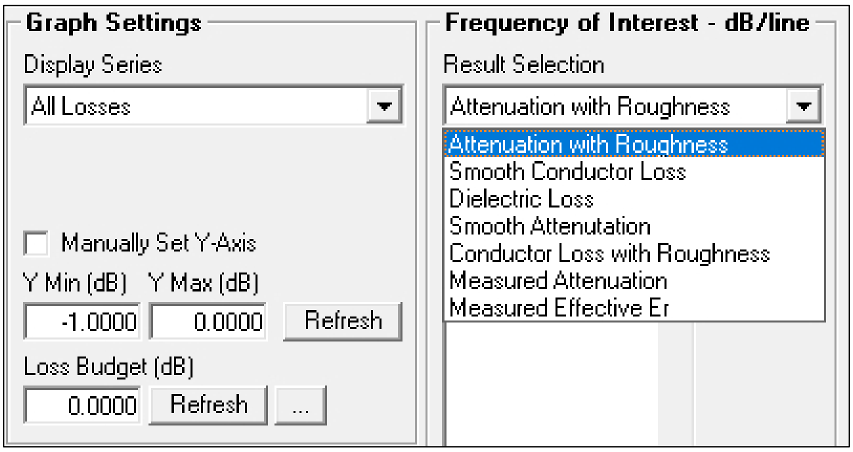
On the Lossless Calculation tab, supply the structure parameters and calculate the impedance.

Switch to the Frequency Dependent tab:

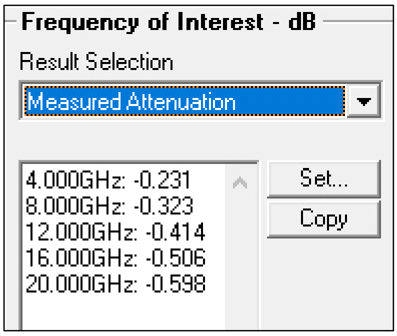
Result Selection

Designated frequencies can be specified directly or set from the Frequency of Interest table.

When using the Set from FOI (Frequency of Interest) option in the Loss Tangent Goal Seek dialog, the Total Attenuation data will depend on the Frequency of Interest Result Selection setting – below.



The readings in this example have been derived from measured data, so for this example it will be necessary to choose Measured Attenuation from the Result Selection dropdown in the Frequency of Interest pane.



Result Selection and Frequency of Interest table

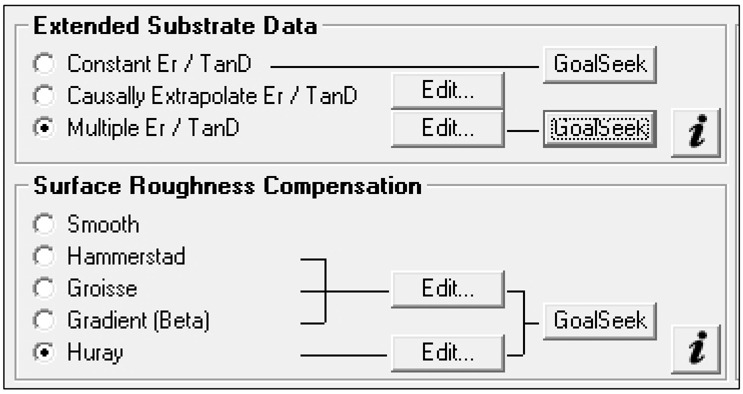
Surface Roughness Compensation

From the Surface Roughness Compensation option group:

- *Ensure the Smooth Surface Roughness Compensation option is not selected*
- Select the Surface Roughness Compensation option (Huray in the example below)

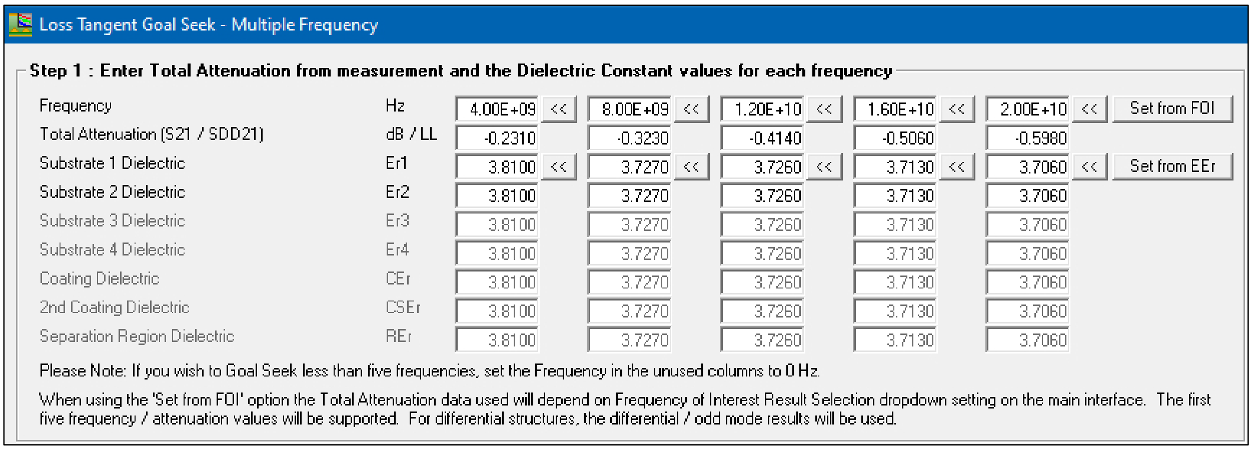
Extended Substrate Data

- Select the Multiple Er / TanD Extended Substrate Data option
- Click the Multiple Er / TanD Goal Seek button



The Loss Tangent Multiple Frequency Goal Seek dialog is displayed. The dialog allows for up to five Loss Tangent values to be calculated in a single process.

Input data and results are displayed simultaneously, with a separate column for each frequency. If frequencies of interest have been specified via the Frequency of Interest dialog, the first five frequencies of interest with their associated attenuation values can be added to the table via the Set from FOI (Frequency of Interest) button



Enter the Total Attenuation

Under Step 1, for each frequency, enter the Total Attenuation for the point of interest: the Frequency and the loss per length of line –dB / LL and the Dielectric Constant. The length of line will be the value entered into the Length of Line (LL) parameter on the main frequency dependent tab,

Using the frequencies of interest

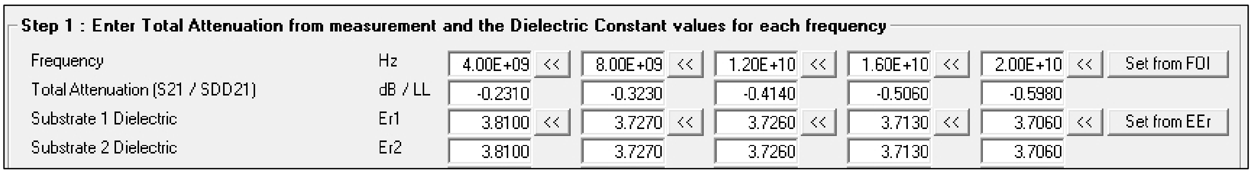
Set from Frequency of Interest

If frequencies of interest have been specified click the Set from FOI (Frequency of Interest) button to add up to the first five frequencies of interest with their associated attenuation values to the table. Frequencies of interest may be set directly or obtained from measured data.* See note below

** Note for Polar Atlas Users: Atlas users have the option of importing Polar Atlas measurement data into the Si9000e using Import Measurement Data from Atlas toolbar option. Once*

imported, you can use the Set from FOI' button to automatically set the Total Attenuation values from the imported Measured Attenuation. Similarly, the 'Set from EEr' button populates the Dielectric Constant values from the Measured Effective Er

The readings in the example below have been derived from measured data



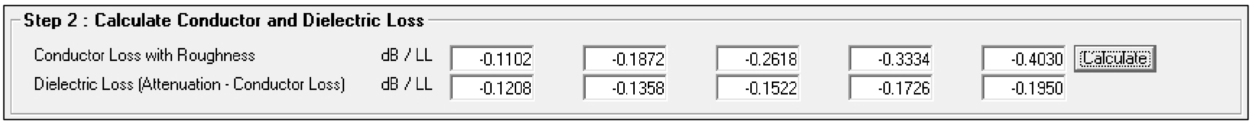
In the case of the Edge-coupled Offset Stripline 1B1A structure, the default value for LL is 10 mm – so Total Attenuation is displayed in dB/cm. The Total Attenuation is shown at each frequency.

Using the Measured Effective Er

The Dielectric Constant values may be entered directly – or you can use the Set from EEr button to populate the Dielectric Constant values from the Measured Effective Er

Calculate the Conductor and Dielectric Loss

Under Step 2, Calculate the Conductor and Dielectric Loss. Si9000e will take the parameters for the current selected structure (in this case, Edge-Coupled Stripline 1B1A) and calculate the conductor loss at each chosen frequency. It will then take the calculated conductor loss from the total attenuation entered in Step 1 to calculate the remaining dielectric loss at each frequency – see typical results below.

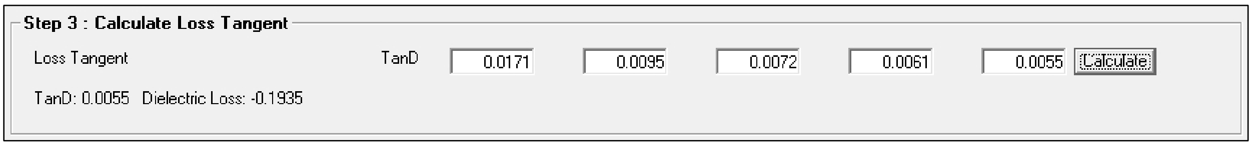


The conductor loss and remaining dielectric loss for each frequency are displayed in tabular form.

Calculate Loss Tangent

The Step 3 Calculate Loss Tangent option will allow the Si9000e to calculate the Loss Tangent (TanD) required to achieve the dielectric loss at each frequency as shown above.

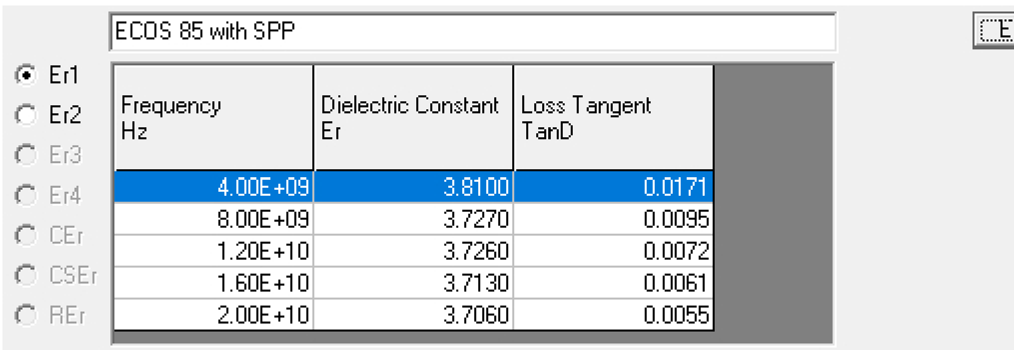
Using the Goal Seek Parameters to limit the min / max range of TanD the Si9000e will sweep the range of TanD values until a suitable TanD is calculated to achieve the displayed dielectric loss (i.e. Attenuation – Conductor loss) at each frequency. The progress of the calculation is updated on the dialog.



The result table above shows the Loss Tangent (TanD) required to achieve the dielectric loss at each frequency – the convergence value is used to give the target dielectric loss a tolerance.

Export results as an Extended Substrate Data table

The results can be exported in tabular form as an Extended Substrate Data table. Supply a descriptive name – in this example *ECOS 85 with SPP* for the table and click Export. The exported table can be specified in Extended Substrate Data calculations.



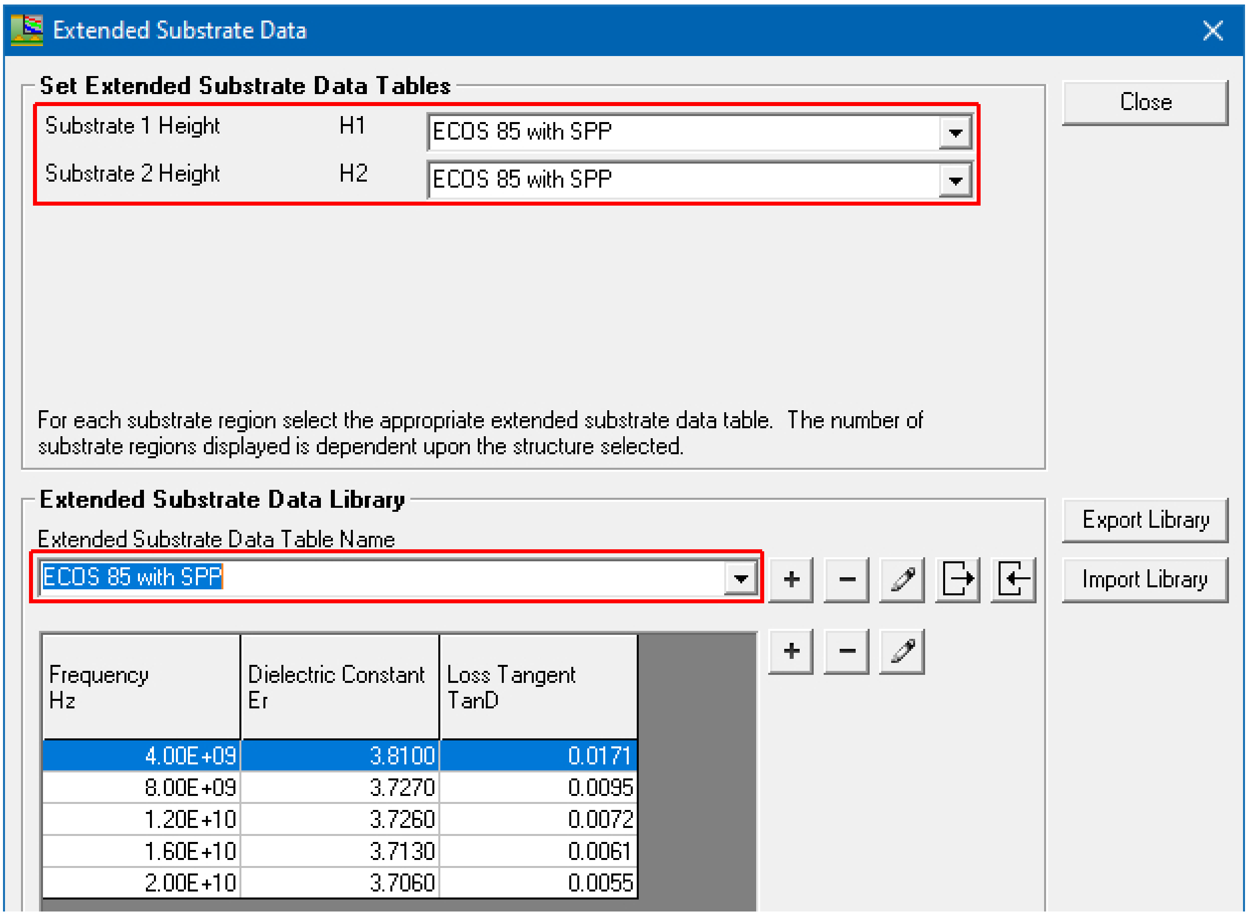
Using the exported table

From the Extended Substrate Data pane, choose Multiple Er / Tan D and click the Edit button.

For each substrate region:

- select the value from the exported table;
- specify the exported table from the Extended Substrate Data Library.

Close the dialog

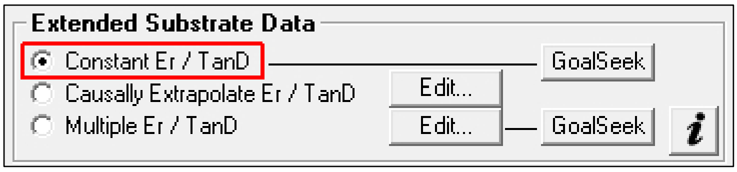


The table data may be used as described in *Using Extended Substrate Tables*. Multiple frequency goal seeking uses the data contained within the specified table.

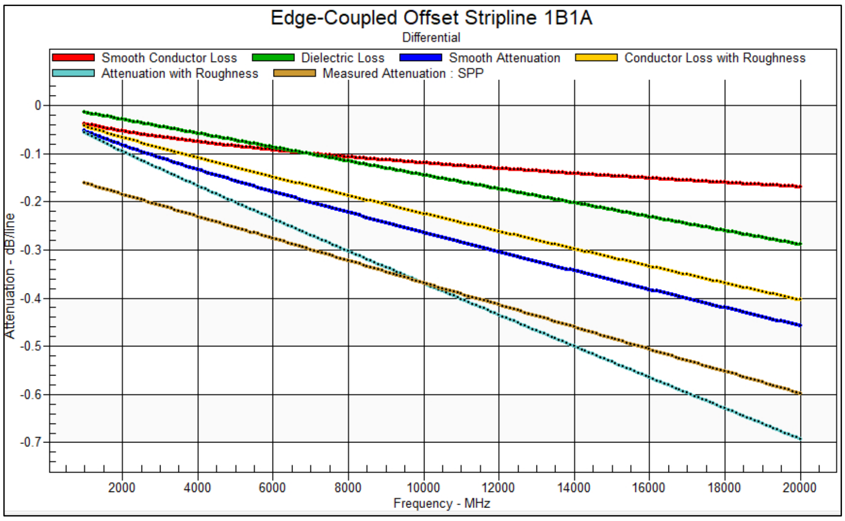
Comparing Single to Multiple Frequency Goal Seek

Using the results from the single and multiple frequency goal seek as an exported extended substrate data table illustrates the improvement in the correlation between the modelled Attenuation with Roughness (cyan) and Measured Attenuation (brown).

Single Frequency Goal Seek



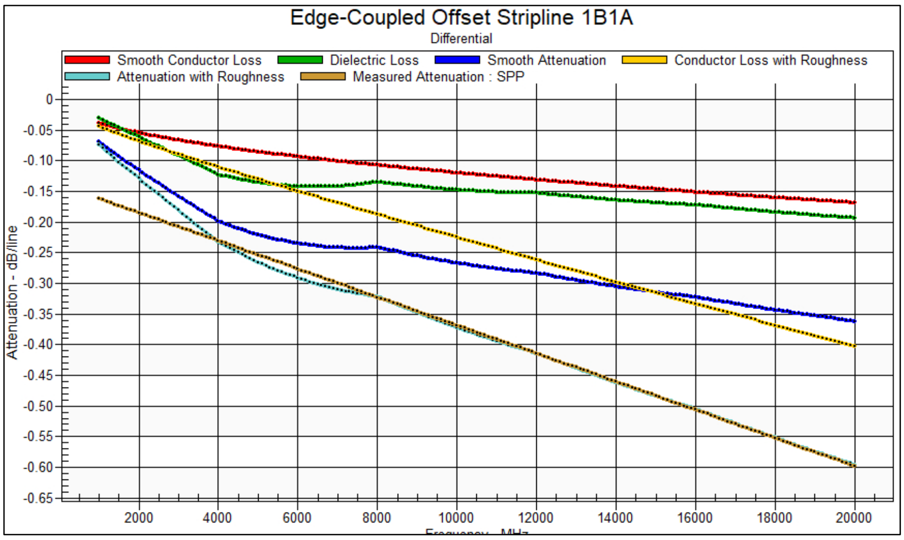
In the graph below, Attenuation with Roughness (cyan) and Measured Attenuation (brown) do not exhibit a high degree of correlation when using Constant Er / TanD mode.



Constant Er /TanD Goal Seeking

Multiple Frequency Goal Seek

Once the Multiple Er / TanD Loss Tangent Goal Seek is complete, the exported results show the improvement in the correlation between the calculated Attenuation with Roughness (cyan) and Measured Attenuation (brown).

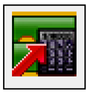


Speedstack Si to Si9000e data transfer (frequency dependent parameters)

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

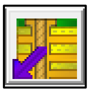


To Field Solver

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

From Field Solver

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



Paste Structure from 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

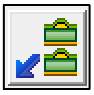
Copy Structure to Speedstack

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

Multiple structures

To Si Project

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



Paste from Speedstack into Si Project

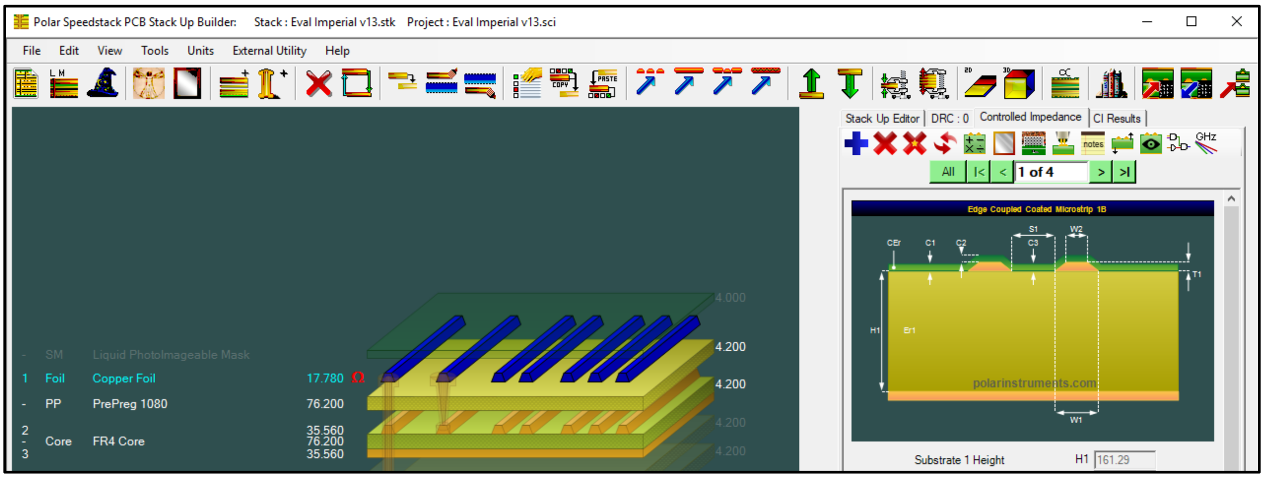
Use the Si9000e's Paste from Speedstack into Si Project to paste the set of structures into the Si9000e as a project.

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.

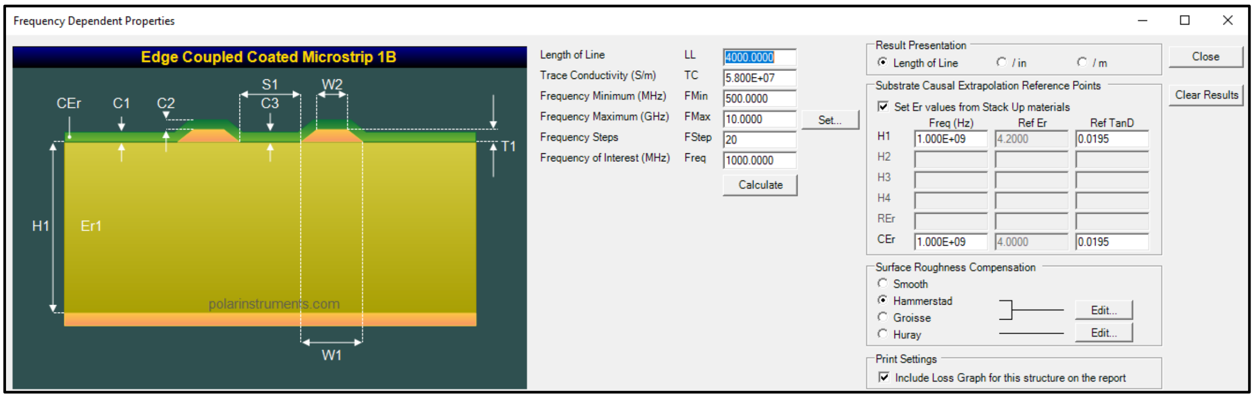
Using the data transfer icons within Speedstack allows a selected set of structure properties to be shared between other structures on the same electrical layer on the stackup.

To share parameters between structures, select the source structure (structure 1, Edge Coupled Coated Microstrip 1B.)

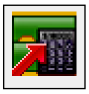


Frequency Dependent Properties

Select the Frequency Dependent Properties button to display the frequency dependent properties.



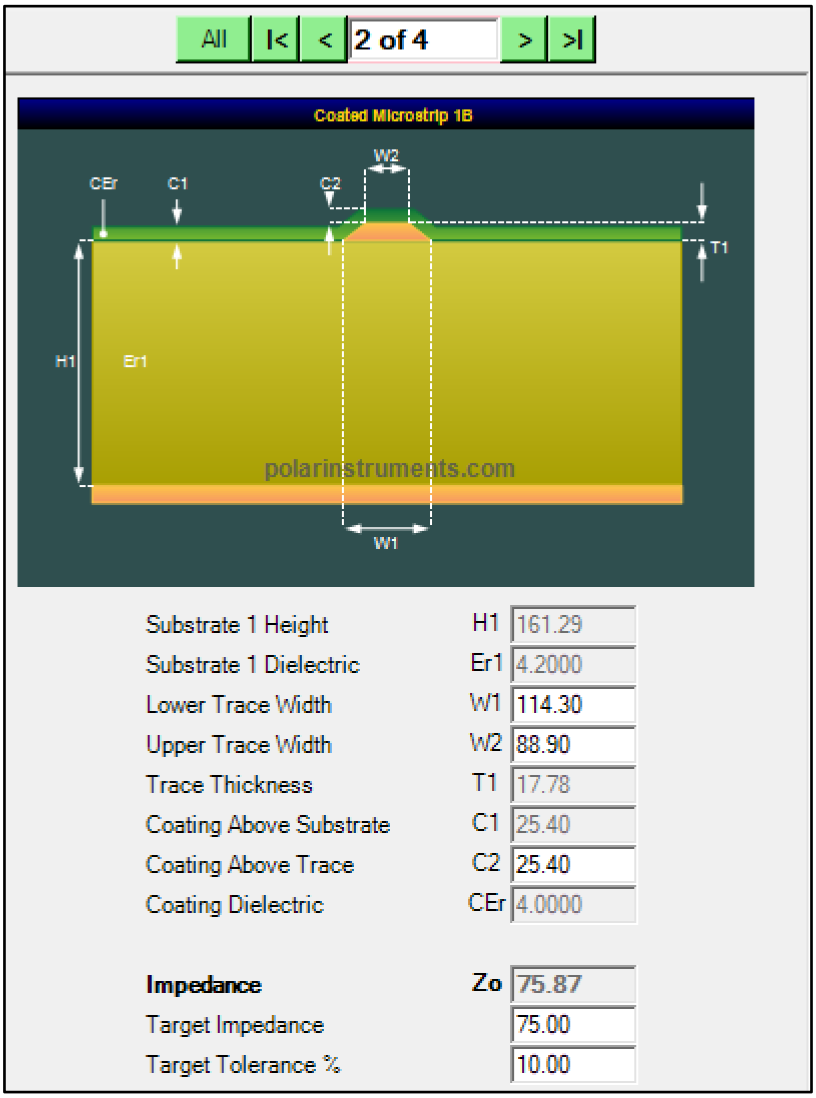
All the structure's properties, including all the frequency dependent parameters, will be available for sharing with the target structure.


To Field Solver

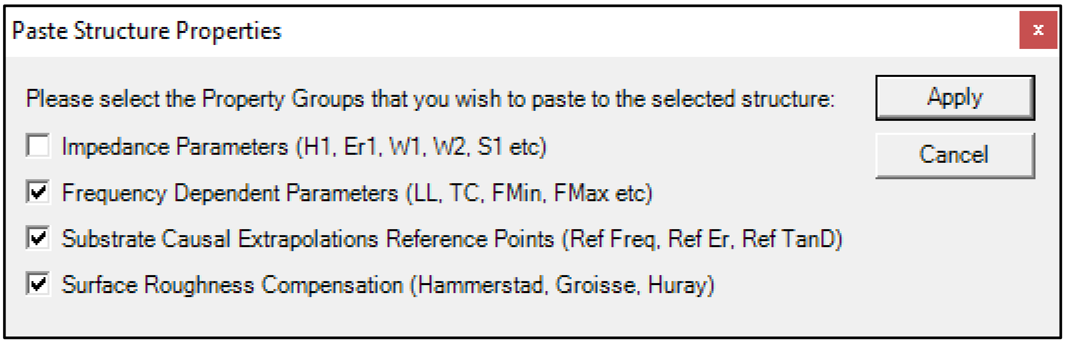
Close the dialog and click the To Field Solver button to copy the parameters to the clipboard.


From Field Solver

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.

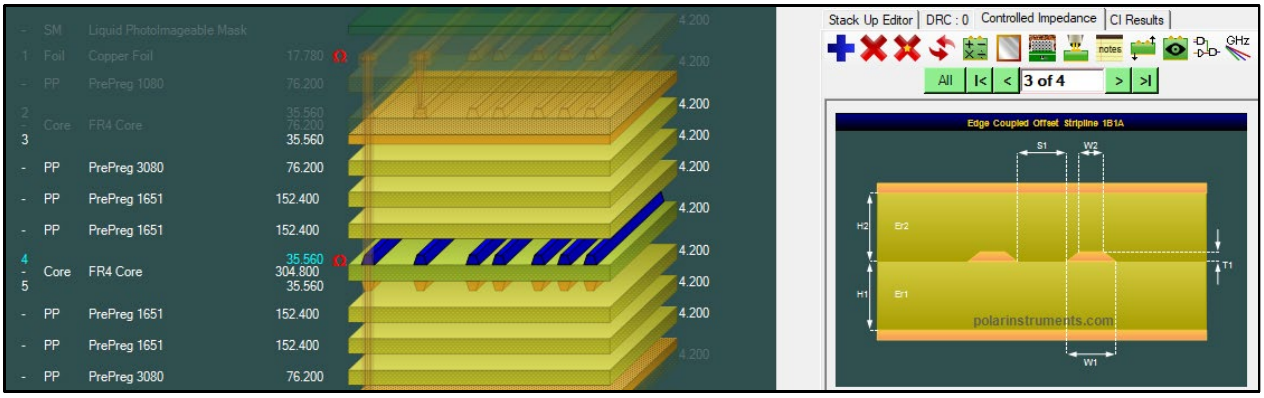
The frequency dependent parameters, along with the causal extrapolation reference points (frequency, Er and TanD) and surface roughness compensation method are applied to the target structure.

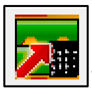
Transferring structures

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.





To Field Solver

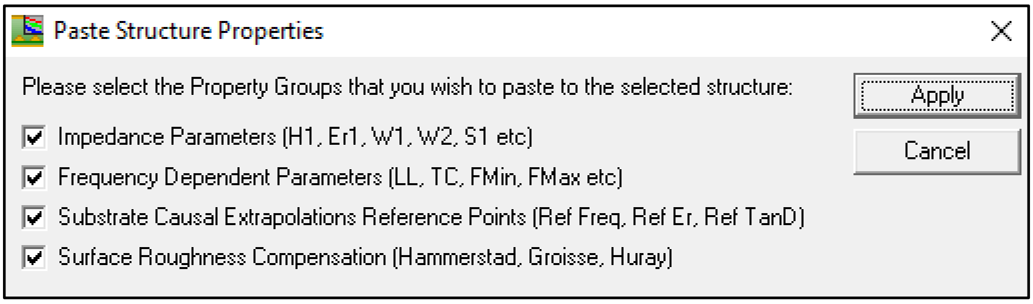
Click the To Field Solver button to transfer the structure and all parameters to the Si9000e.

Switch to the Si9000e.

Paste Structure from Speedstack

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

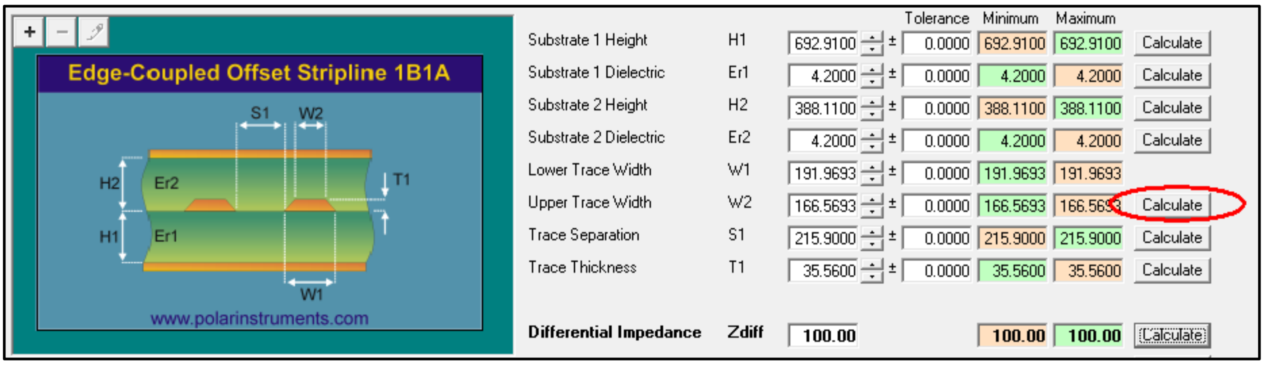
The Si9000e displays the Paste Structure Properties dialog.



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

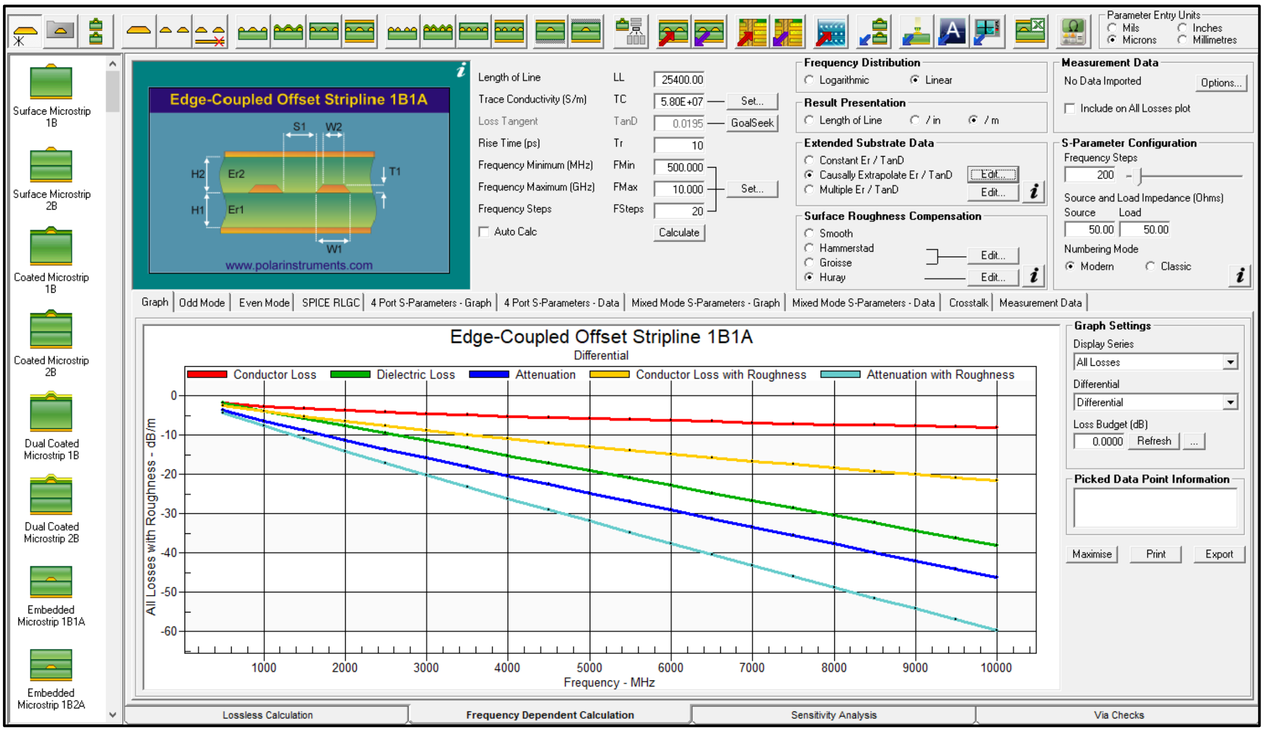
With the structure loaded into the Si9000e switch to the Lossless Calculation tab to display the structure graphic and lossless parameters.



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

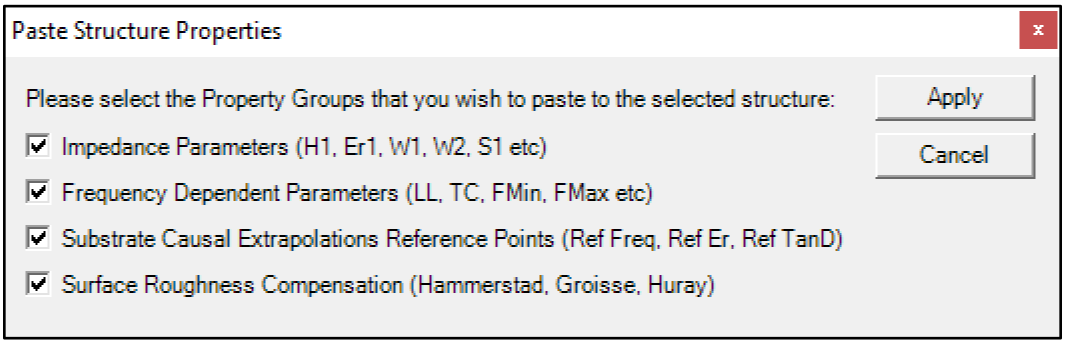
Enter the frequency dependent parameters, the extended substrate data settings, the surface roughness compensation method and values and click Calculate to refresh the results.



Copy Structure to Speedstack

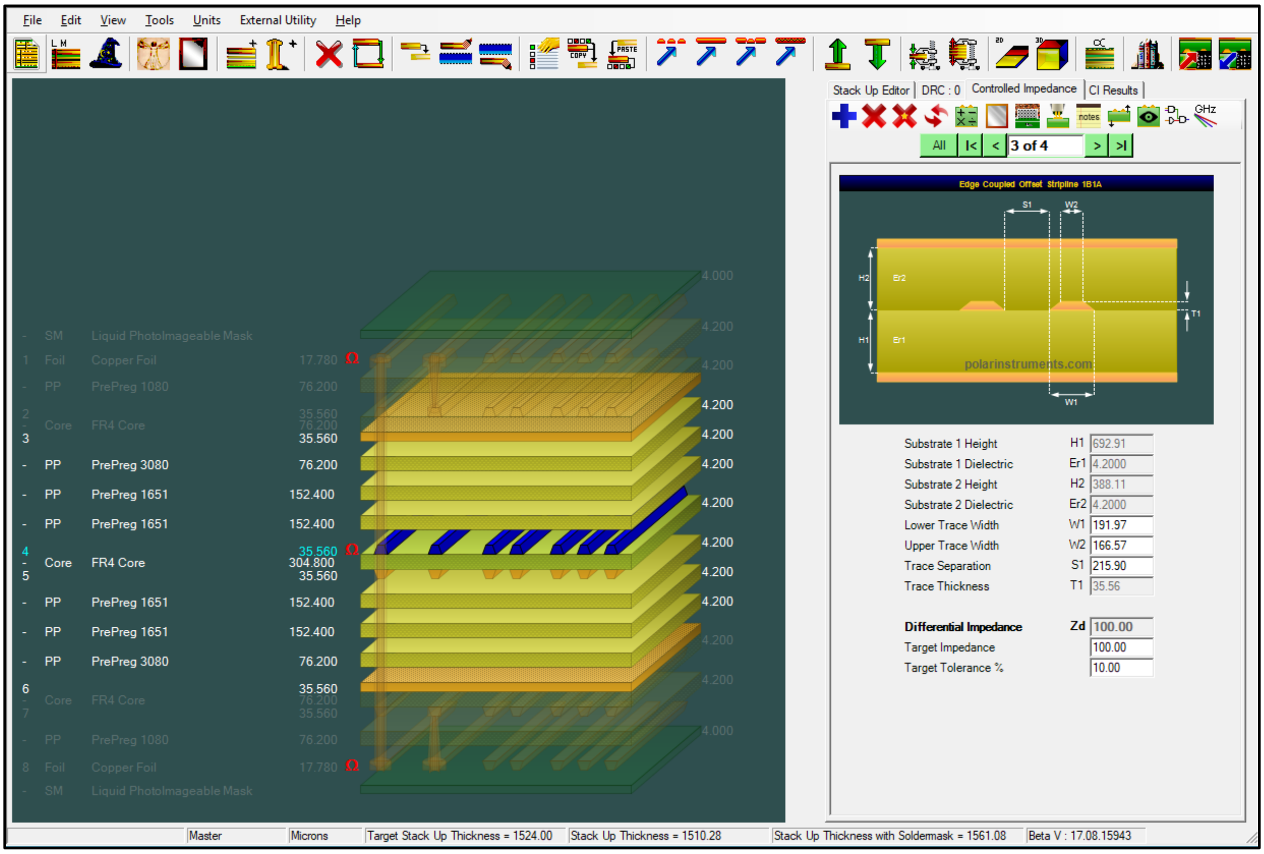
With all calculations complete click the Copy Structure to Speedstack to return the structure to the stackup in Speedstack.

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.



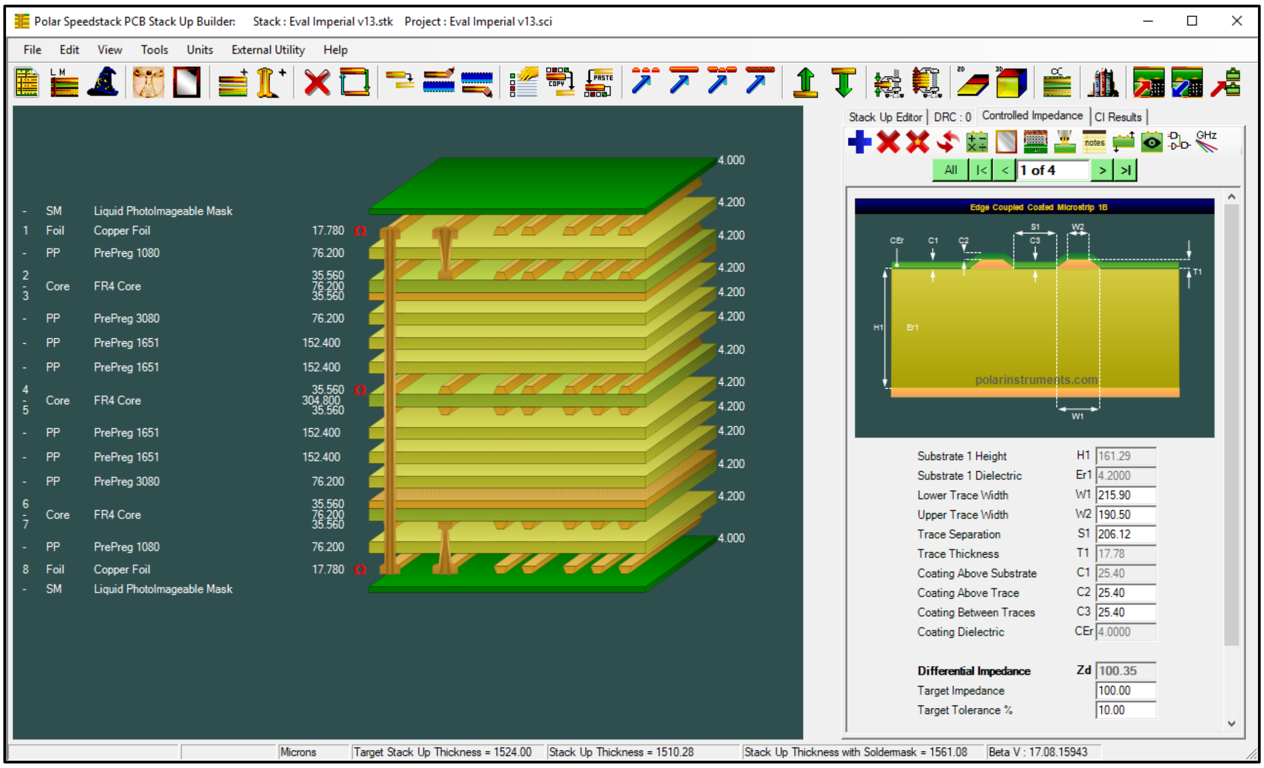
Transferring multiple structures via Si Projects

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.

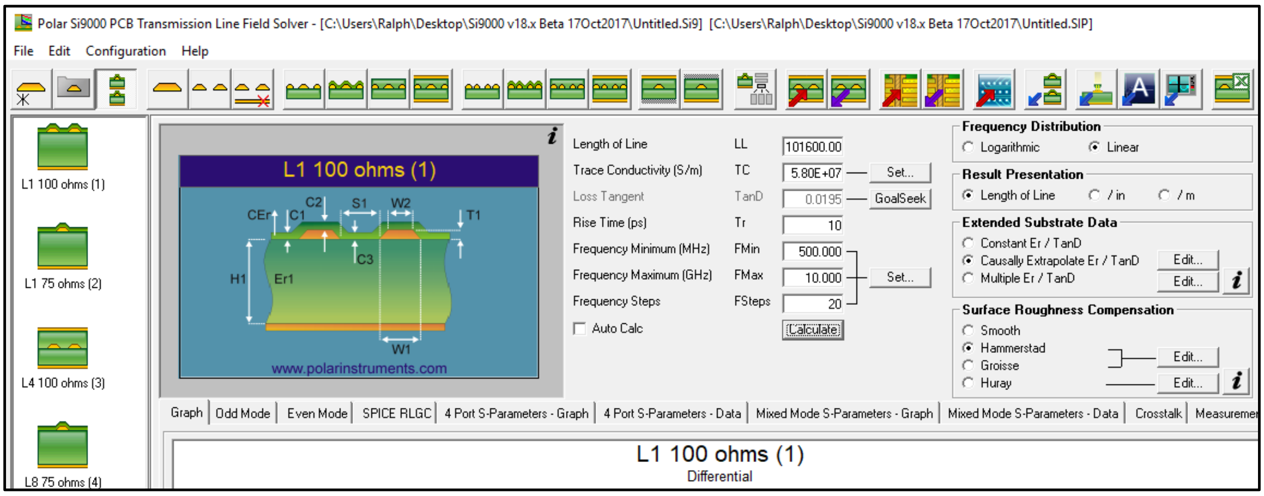


To Si Project

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.

Paste from Speedstack into Si 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.

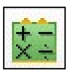
The Si Project window lists the transferred structures in Speedstack's display order, showing the order number and impedance value along with a thumb nail graphic indicating the structure configuration.

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.



Rebuild and Recalculate Displayed Structure

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

Rebuild and Recalculate All Structures

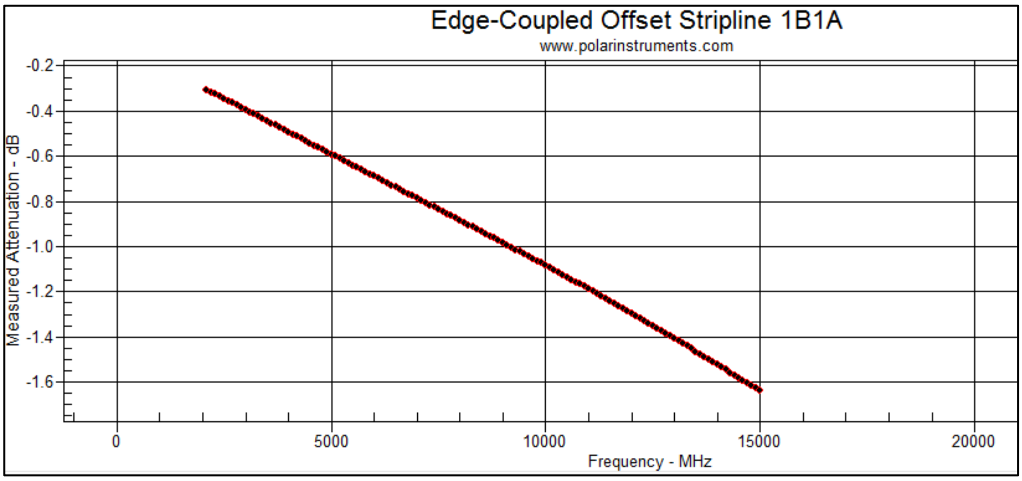
Click the Rebuild and Recalculate All Structures to update all structures in the stack.

Measured Attenuation and Measured Effective Er

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

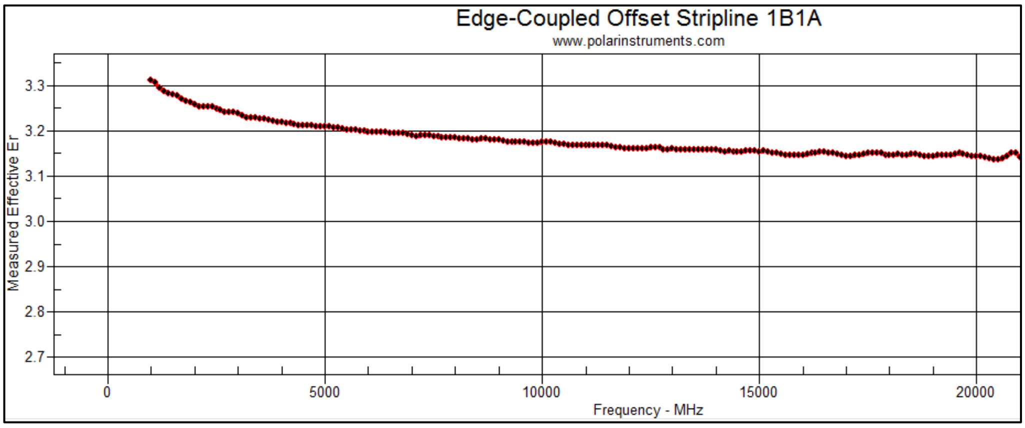
Selecting the Measured Attenuation display series allows the measurement data to be plotted without the modelling data.

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



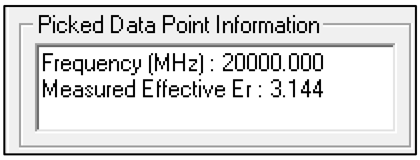
Measured Attenuation

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



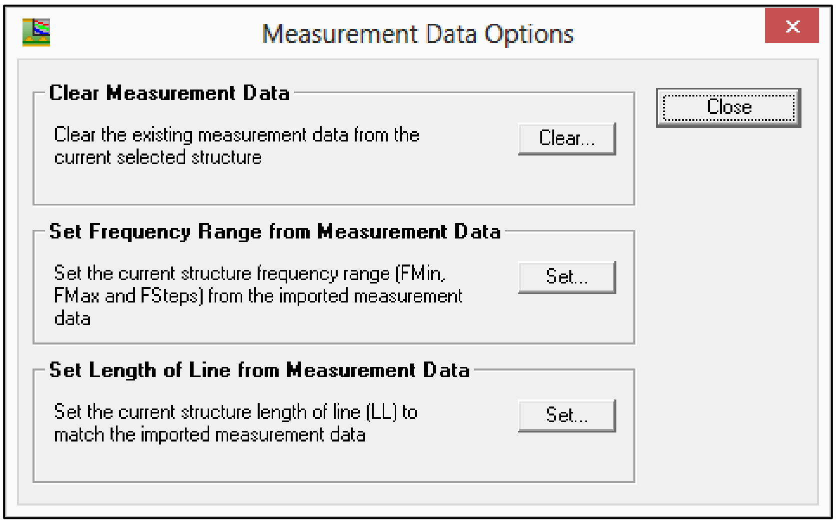
Measurement Data options

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.



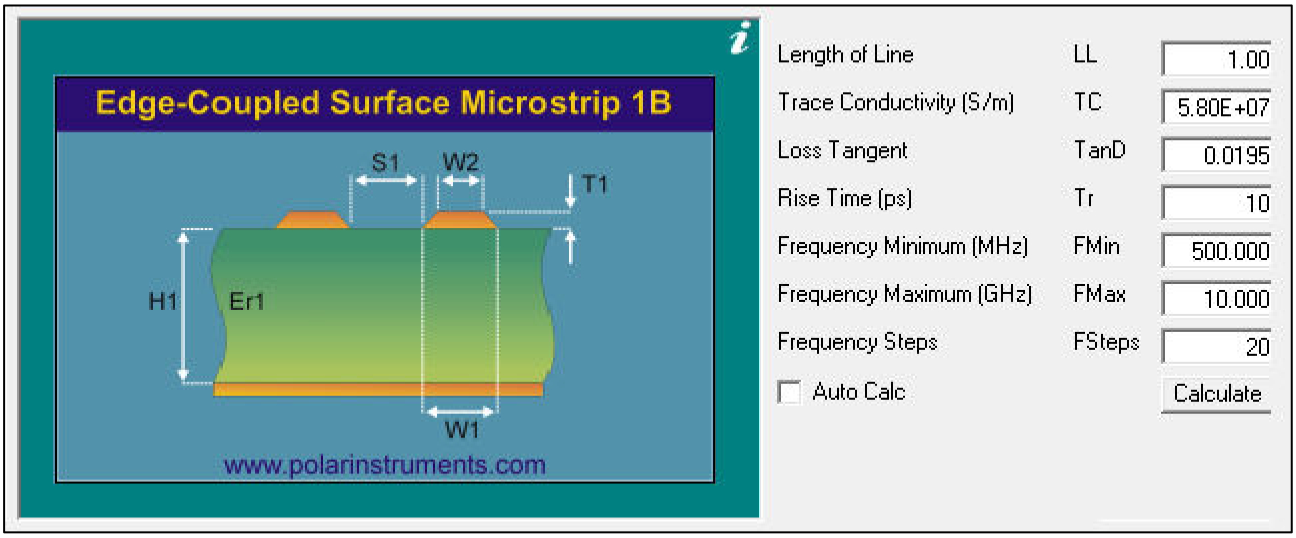
The new settings will be applied after the dialog box is closed. It will be necessary to click Calculate to update results.

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.)

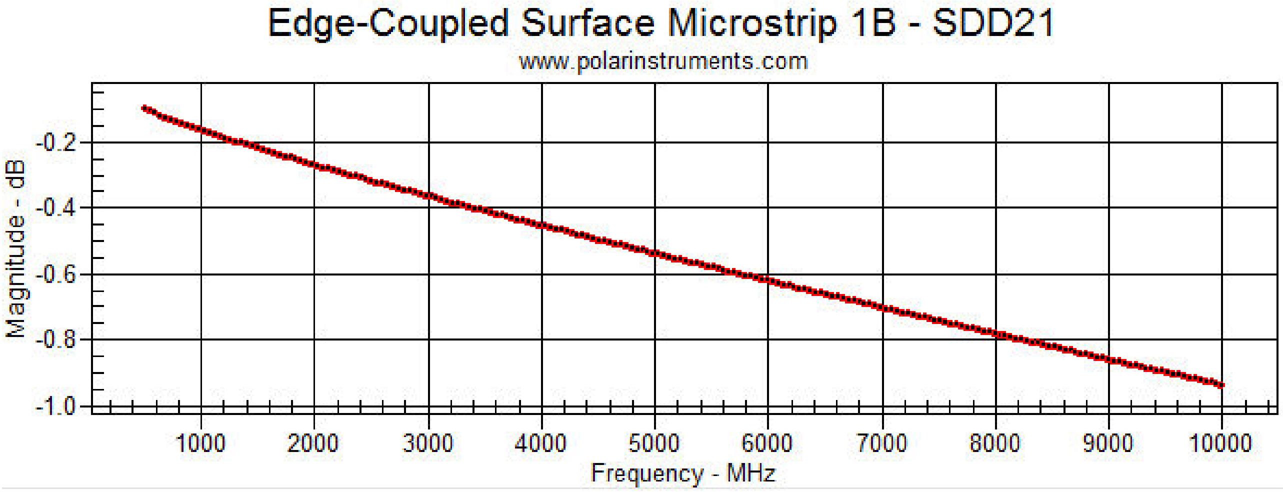
Using the final result from a Delta-L based measurement system and a correctly configured Si9000e you should be able to establish good correlation between Delta-L and Si9000e.

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.

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

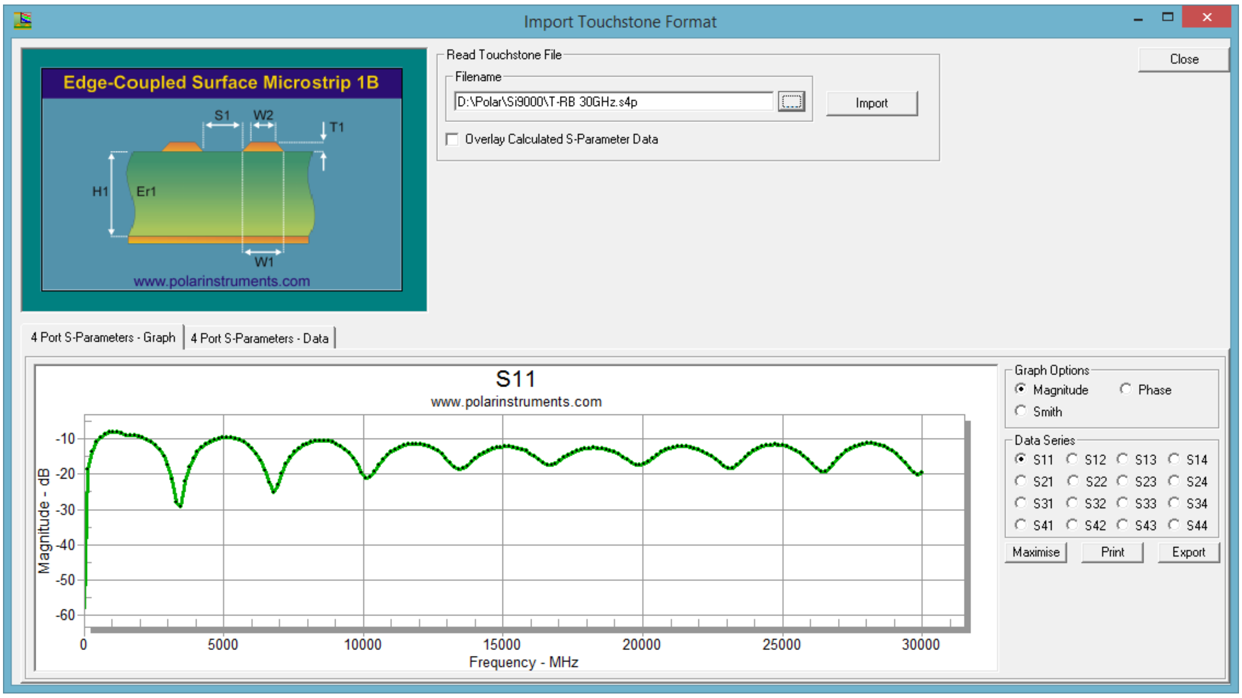
Import Touchstone files by using the File | Import Touchstone Format menu selection.

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 structure's S-parameter data.

Select the structure from the Si9000e structure bar

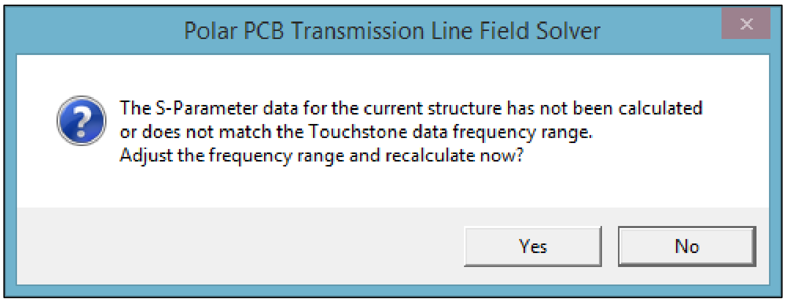
Select the File | Import Touchstone Format… command and select the .s2p or .s4p file and click Open.

The data will be displayed as a green dataset.

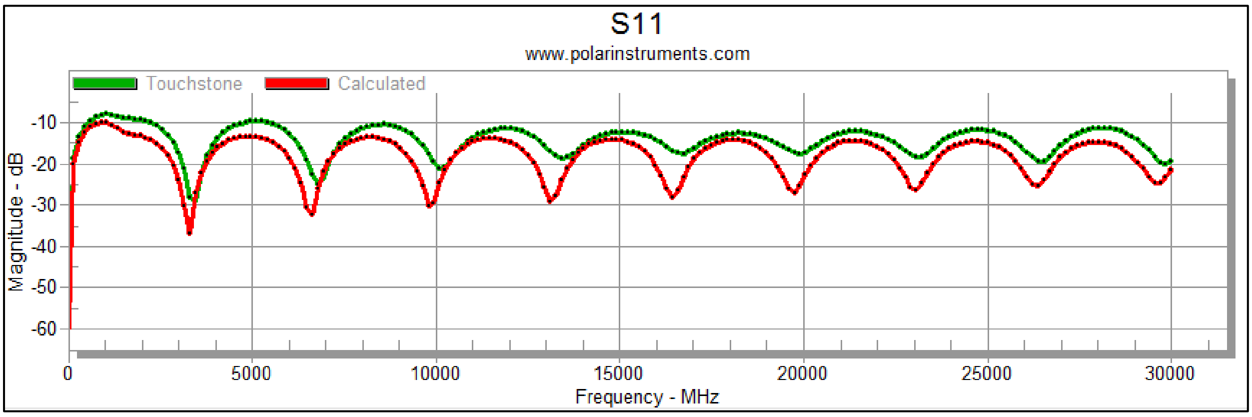


Select Overlay Calculated S-Parameter Data.

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.



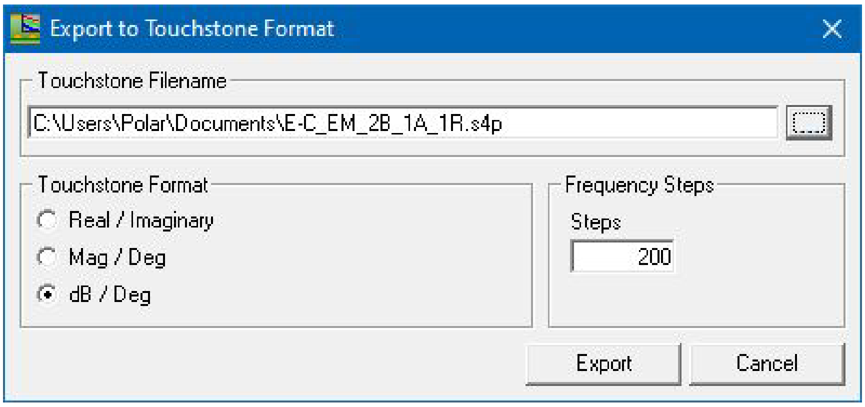
To change the structure parameters, it will be necessary to close the dialog, alter the parameters and calculate the structure and then return to the Import Touchstone option.

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.

From the File menu choose Export to Touchstone Format…



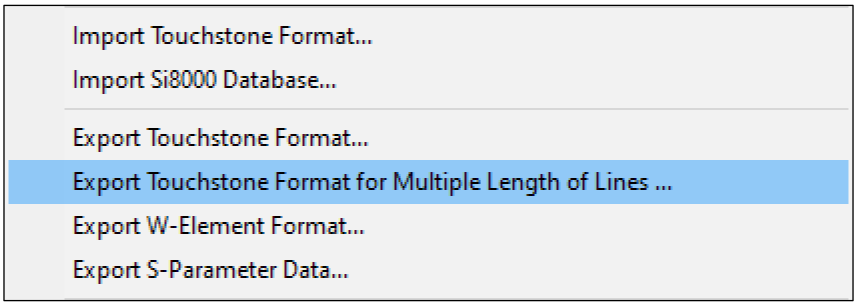
Specify the file name and location, choose from Real / Imaginary, Magnitude / Degrees or dB / Degrees

Touchstone formats, specify the number of frequency steps then click Export.

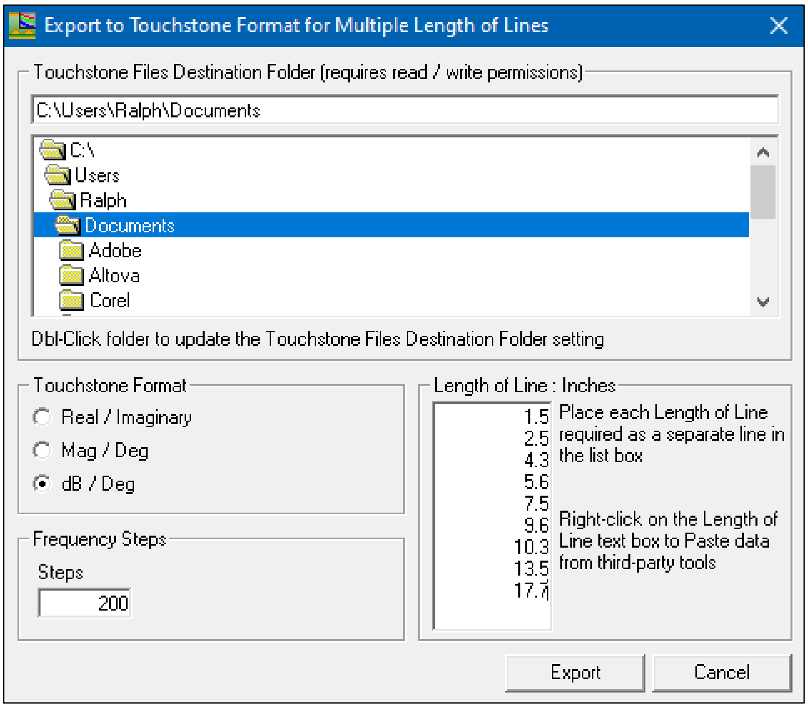
When overlaying the two sets of data the software will automatically check that the frequency range of the calculated data matches that of the Touchstone data; if this is not the case an option will be displayed offering an adjust and recalculate function – see above.

Exporting Touchstone format for multiple line lengths

Si9000e can export Touchstone format files for multiple line lengths in a single step. From the File menu choose Export Touchstone Format for Multiple Length of Lines…

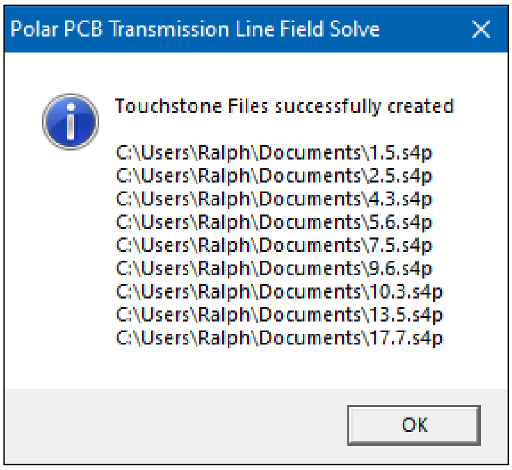


In the example dialog below, nine line lengths have been specified with the line lengths specified in inches.



Choose the Touchstone Format, specify the number of frequency steps and click Export

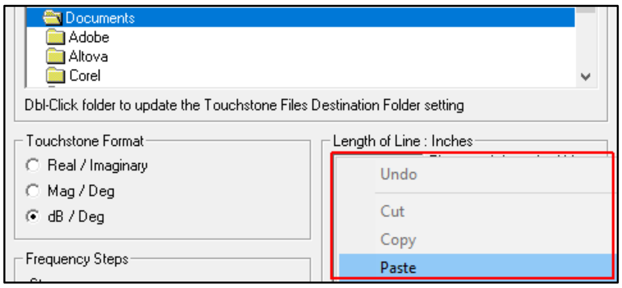
The Si9000e calculates and then saves the Touchstone files to the designated folder – as reported below.



Line lengths may also be pasted in from third party products (for example, from a spreadsheet, as illustrated below.)

Copy the line lengths from the spreadsheet, right click the Length of Line panel and paste them into the panel.

Length of Line (mils)
1000
2000
3000
4000
5000
10000
15000
20000
25000



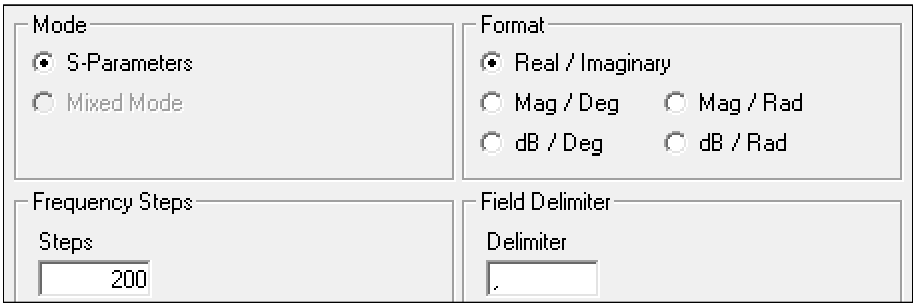
Click Export – the Touchstone files are saved as above.

Exporting to W-Element format

To export (RLGC) data in W-Element (HSPICE) format choose the Export to W-Element format then choose the file name and location.

Exporting S-parameter data

To export s-parameter, choose Export S-Parameter Data and specify the Mode and format, number of frequency steps and text field delimiter (comma, pipe, etc.)



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.

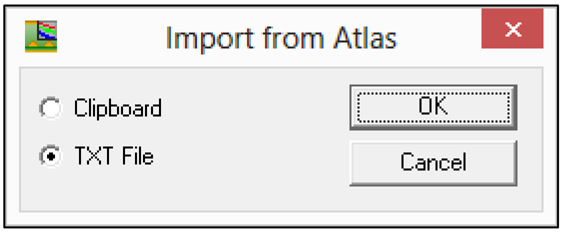
Data may be imported via the Windows clipboard where Atlas and Si9000e coexist on the same machine or via text files where data are transferred from a separate Atlas system.

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.

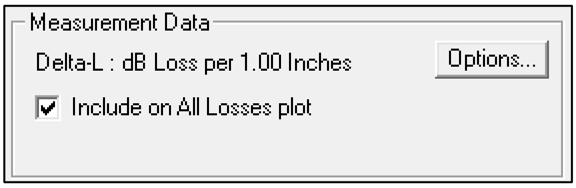
Importing Atlas data via a text file

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



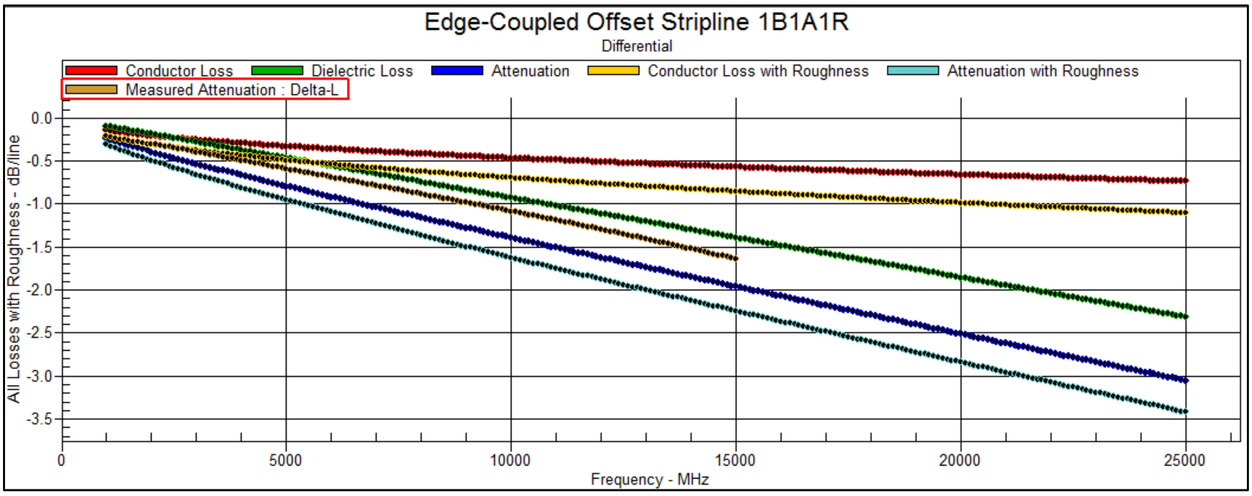
Once imported, the Measurement Data frame on the Frequency Dependent interface updates, summarising key information about the data imported.

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.

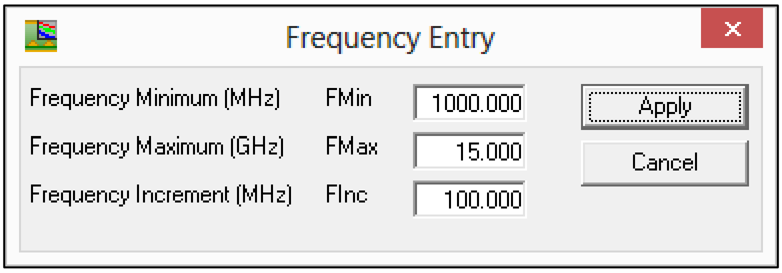
The imported measurement data may be overlaid onto modelled data for analysis. In the example below, the set of measured attenuation data for the Delta-L test is shown as an additional Measured Attenuation Delta-L data set on the All Losses plot.



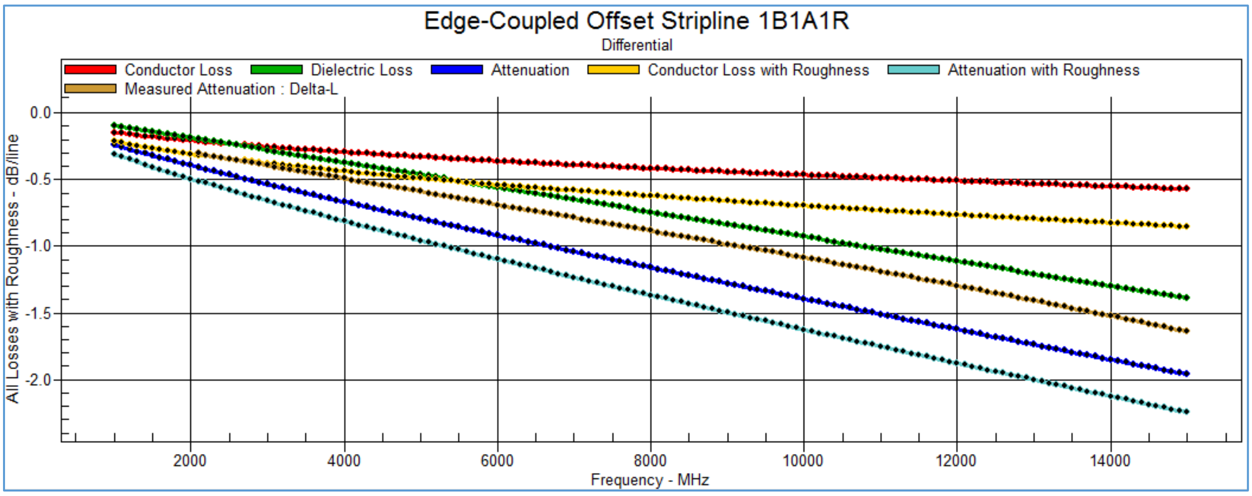
Comparing the Attenuation with Roughness curve with the Measured Attenuation indicates the degree of correlation.

Set frequency range

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.

Frequency Hz	Measured Attenuation : dB Loss per 1.00 Inches	Effective Er
1.000E+09	0.000	3.312
1.100E+09	0.000	3.306
1.200E+09	0.000	3.295
1.300E+09	0.000	3.286
1.400E+09	0.000	3.281
1.500E+09	0.000	3.279
1.600E+09	0.000	3.277
1.700E+09	0.000	3.271
1.800E+09	0.000	3.264
1.900E+09	0.000	3.262
2.000E+09	0.000	3.259
2.100E+09	-0.306	3.254
2.200E+09	-0.316	3.253
2.300E+09	-0.326	3.253
2.400E+09	-0.335	3.252
2.500E+09	-0.345	3.248

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

Using the Si Excel Interface

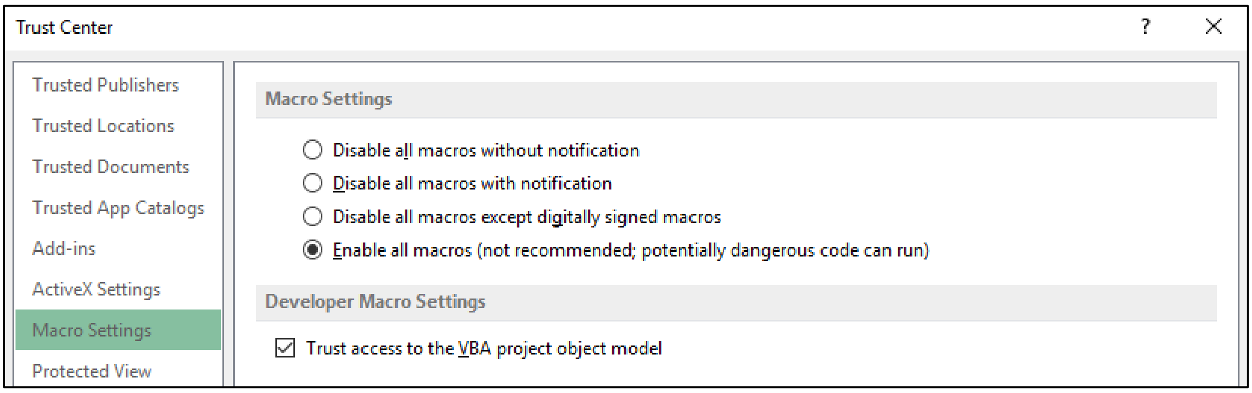
Note: Si8000m/Si9000e is compatible with Microsoft™ Excel™ 2003 or later ***(32-bit only)***

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.

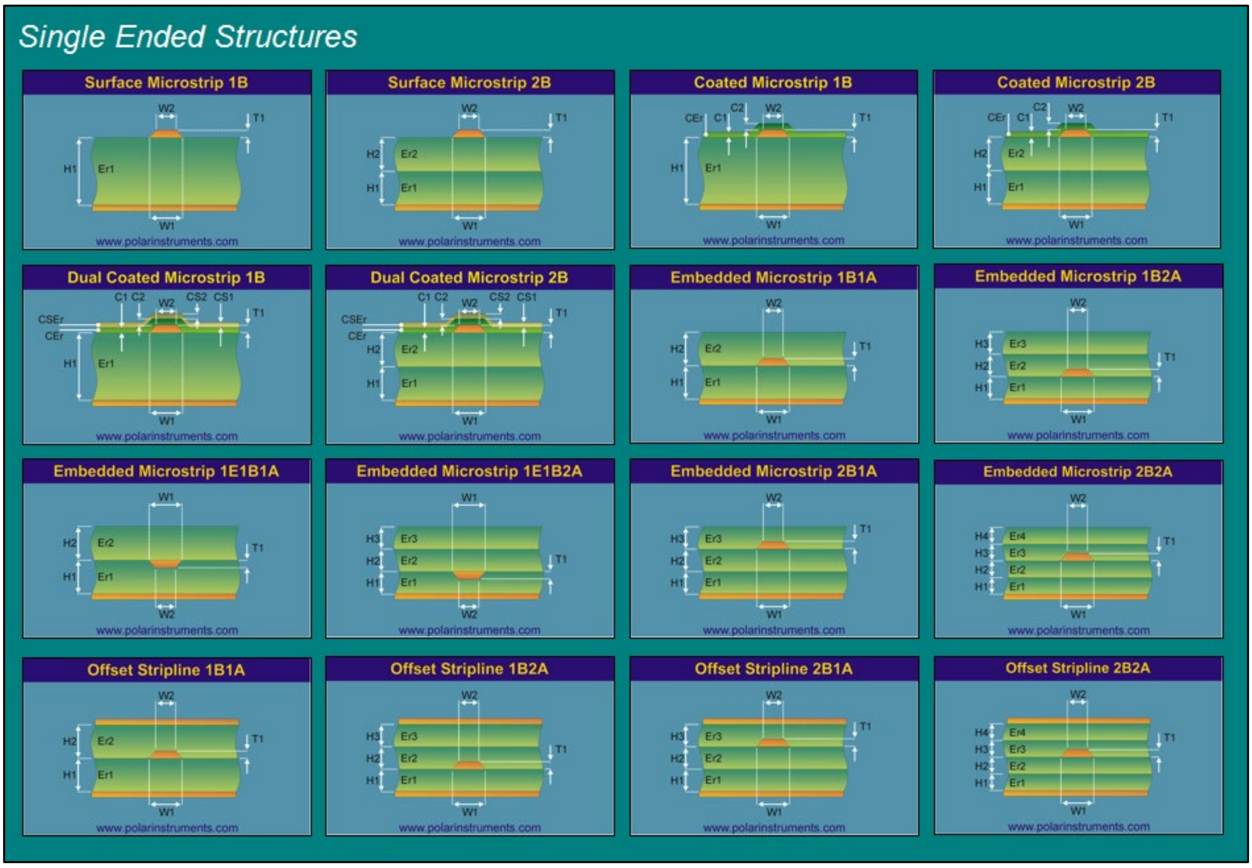


(See the discussion on security levels in Excel’s help.)

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.



Structure index sheet — Single ended structures

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

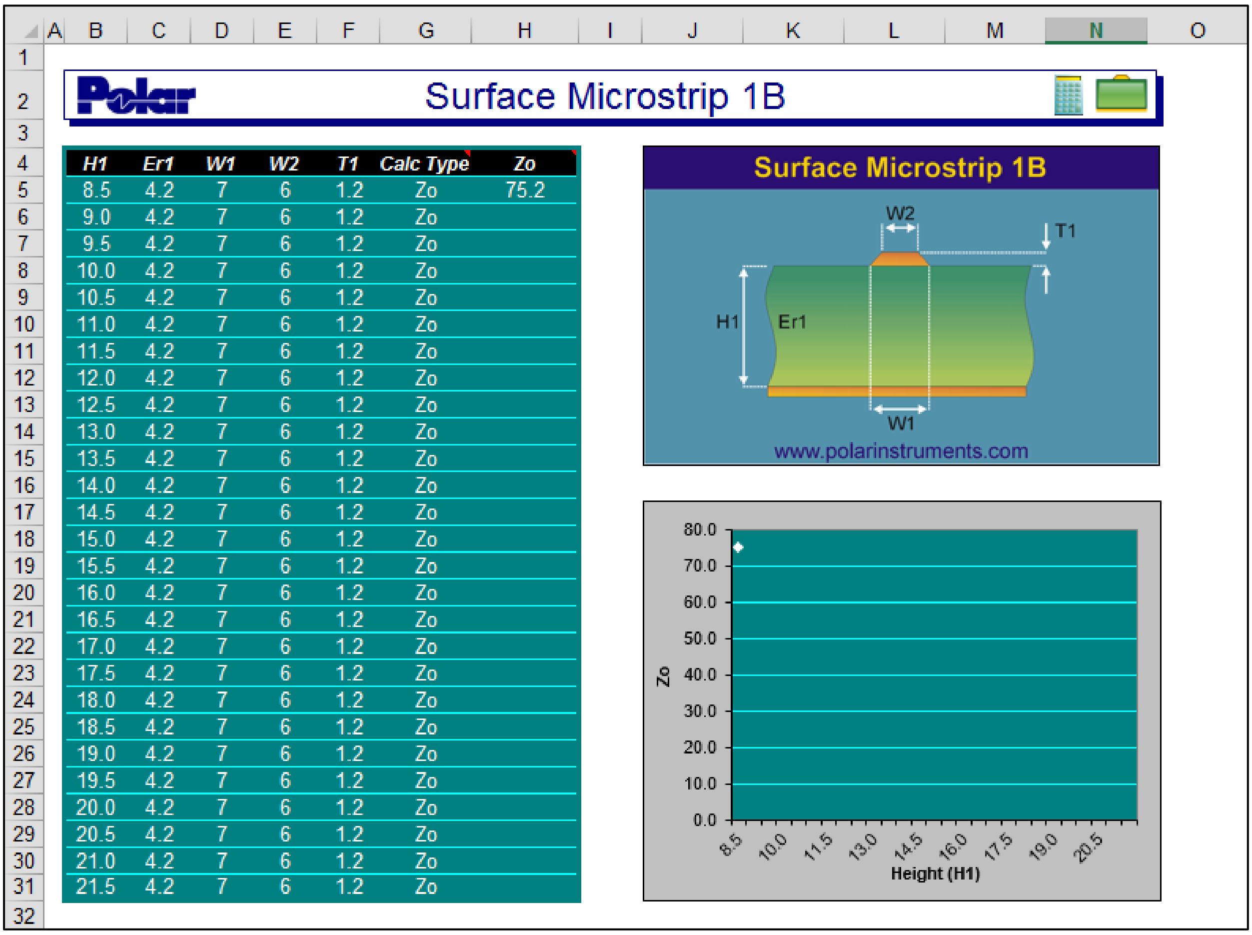
Differential offset coplanar

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.



H1	Er1	W1	W2	T1	Calc Type	Zo
8.5	4.2	7	6	1.2	Zo	75.2
9.0	4.2	7	6	1.2	Zo	
9.5	4.2	7	6	1.2	Zo	
10.0	4.2	7	6	1.2	Zo	
10.5	4.2	7	6	1.2	Zo	
11.0	4.2	7	6	1.2	Zo	
11.5	4.2	7	6	1.2	Zo	
12.0	4.2	7	6	1.2	Zo	
12.5	4.2	7	6	1.2	Zo	
13.0	4.2	7	6	1.2	Zo	
13.5	4.2	7	6	1.2	Zo	
14.0	4.2	7	6	1.2	Zo	
14.5	4.2	7	6	1.2	Zo	
15.0	4.2	7	6	1.2	Zo	
15.5	4.2	7	6	1.2	Zo	
16.0	4.2	7	6	1.2	Zo	
16.5	4.2	7	6	1.2	Zo	
17.0	4.2	7	6	1.2	Zo	
17.5	4.2	7	6	1.2	Zo	
18.0	4.2	7	6	1.2	Zo	
18.5	4.2	7	6	1.2	Zo	
19.0	4.2	7	6	1.2	Zo	
19.5	4.2	7	6	1.2	Zo	
20.0	4.2	7	6	1.2	Zo	
20.5	4.2	7	6	1.2	Zo	
21.0	4.2	7	6	1.2	Zo	
21.5	4.2	7	6	1.2	Zo	

Surface Microstrip sample worksheet

Moving through the structure sheets

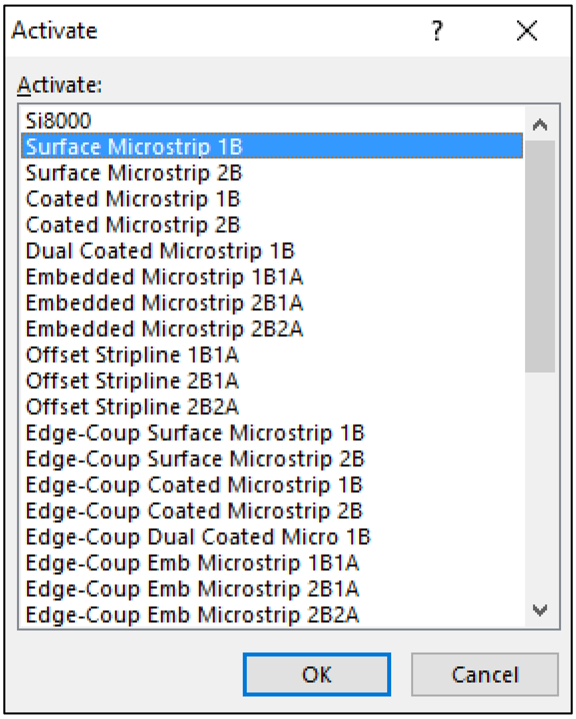
Structure sheets may also be selected via the Tab Scrolling Buttons,


.

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 (H1) with other parameters fixed).

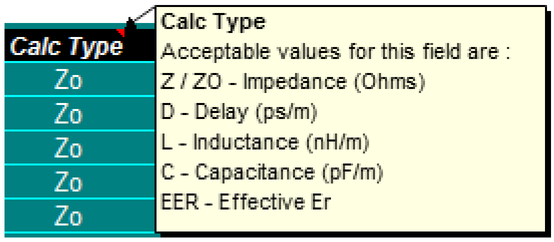
H1	*Er1*	*W1*	*W2*	*T1*	*Calc Type*	*Zo*
8.5	4.2	7	6	1.2	Zo	75.2
9.0	4.2	7	6	1.2	Zo	
9.5	4.2	7	6	1.2	Zo	
10.0	4.2	7	6	1.2	Zo	
10.5	4.2	7	6	1.2	Zo	
11.0	4.2	7	6	1.2	Zo	
11.5	4.2	7	6	1.2	Zo	
12.0	4.2	7	6	1.2	Zo	
12.5	4.2	7	6	1.2	Zo	
13.0	4.2	7	6	1.2	Zo	
13.5	4.2	7	6	1.2	Zo	
14.0	4.2	7	6	1.2	Zo	

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.

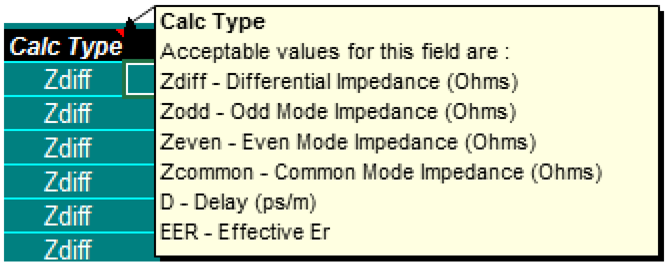
Choosing the calculation type

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.

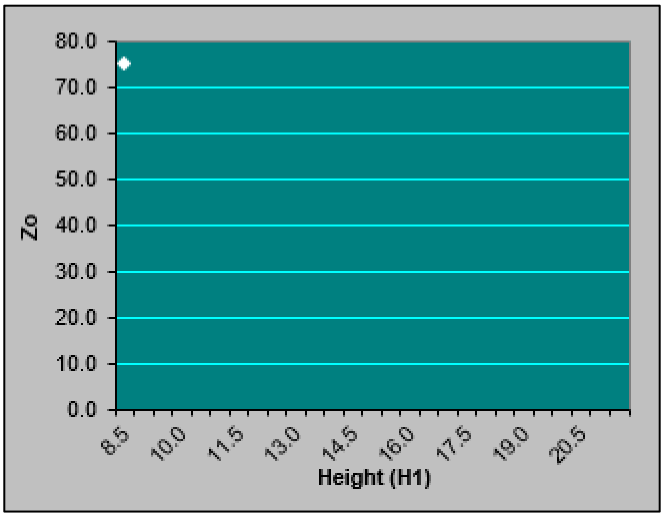


Differential calculation types

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 Z_0 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, H1, 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.)

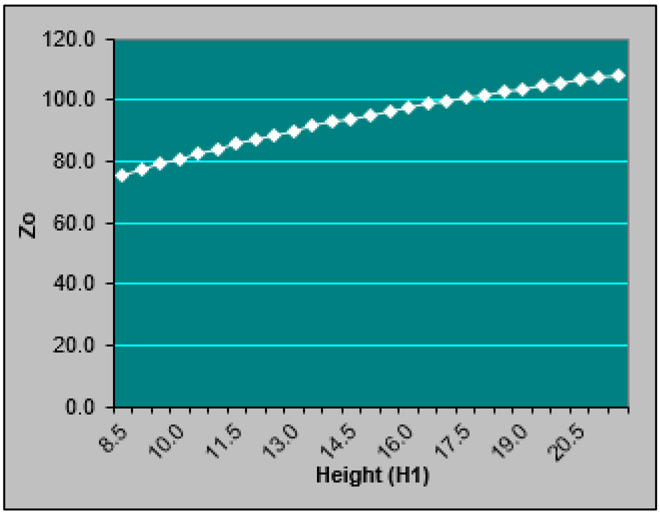
H1	Er1	W1	W2	T1	Calc Type	Zo
8.5	4.2	7	6	1.2	Zo	75.2
9.0	4.2	7	6	1.2	Zo	
9.5	4.2	7	6	1.2	Zo	
10.0	4.2	7	6	1.2	Zo	
10.5	4.2	7	6	1.2	Zo	
11.0	4.2	7	6	1.2	Zo	
11.5	4.2	7	6	1.2	Zo	
12.0	4.2	7	6	1.2	Zo	
12.5	4.2	7	6	1.2	Zo	
13.0	4.2	7	6	1.2	Zo	
13.5	4.2	7	6	1.2	Zo	
14.0	4.2	7	6	1.2	Zo	

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.

H1	Er1	W1	W2	T1	Calc Type	Zo
8.5	4.2	7	6	1.2	Zo	75.2
9.0	4.2	7	6	1.2	Zo	77.1
9.5	4.2	7	6	1.2	Zo	79.0
10.0	4.2	7	6	1.2	Zo	80.8
10.5	4.2	7	6	1.2	Zo	82.5
11.0	4.2	7	6	1.2	Zo	84.1
11.5	4.2	7	6	1.2	Zo	85.7
12.0	4.2	7	6	1.2	Zo	87.1
12.5	4.2	7	6	1.2	Zo	88.6
13.0	4.2	7	6	1.2	Zo	90.0
13.5	4.2	7	6	1.2	Zo	91.3
14.0	4.2	7	6	1.2	Zo	92.6
14.5	4.2	7	6	1.2	Zo	93.9
15.0	4.2	7	6	1.2	Zo	95.1

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



Plot of Z_0 as Height (H1) varies

Choosing other parameters

Z_0, D, L, C and Er can be plotted against any of the function parameters.

For example, to display Z_0 as Er1 varies, in the example reset H1 to a single value, e.g. 8.5, and plot Z_0 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 E_r 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.

Click the **Calculate** icon to refresh the Z_0 column.

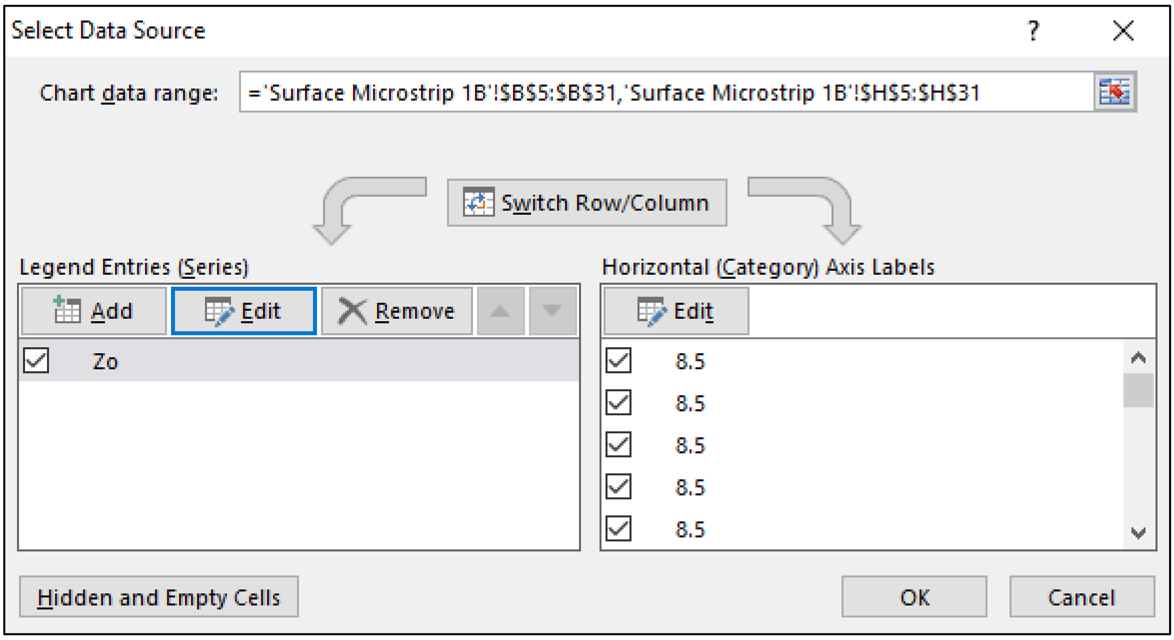
H1	Er1	W1	W2	T1	Calc Type	Zo
8.5	3.8	7	6	1.2	Zo	78.4
8.5	3.85	7	6	1.2	Zo	78.0
8.5	3.9	7	6	1.2	Zo	77.6
8.5	3.95	7	6	1.2	Zo	77.1
8.5	4	7	6	1.2	Zo	76.7
8.5	4.05	7	6	1.2	Zo	76.3
8.5	4.1	7	6	1.2	Zo	75.9
8.5	4.15	7	6	1.2	Zo	75.6
8.5	4.2	7	6	1.2	Zo	75.2
8.5	4.25	7	6	1.2	Zo	74.8
8.5	4.3	7	6	1.2	Zo	74.4
8.5	4.35	7	6	1.2	Zo	74.1

Z0 against Er1 with other parameters fixed

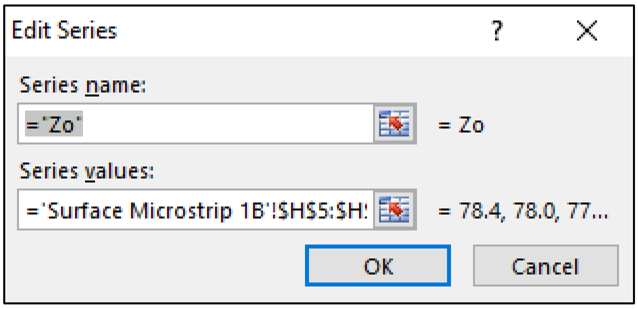
Modifying the chart

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…

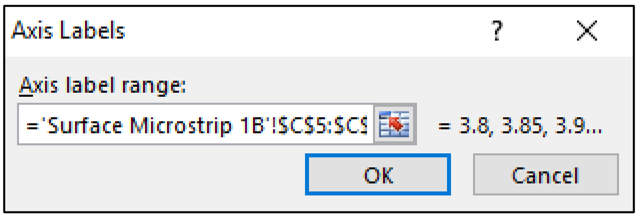


From the Select Data Source dialog box, click the Z_0 Series and choose Edit; the Series page shows the source data cell ranges for the chart.



Click the Collapse Dialog button,  , and select the new range of values of Z_0.

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

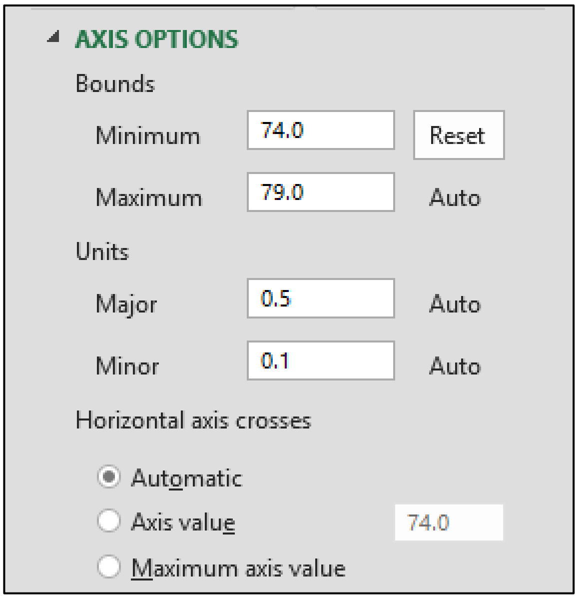


Click the button again to restore the dialog box and press **OK**.

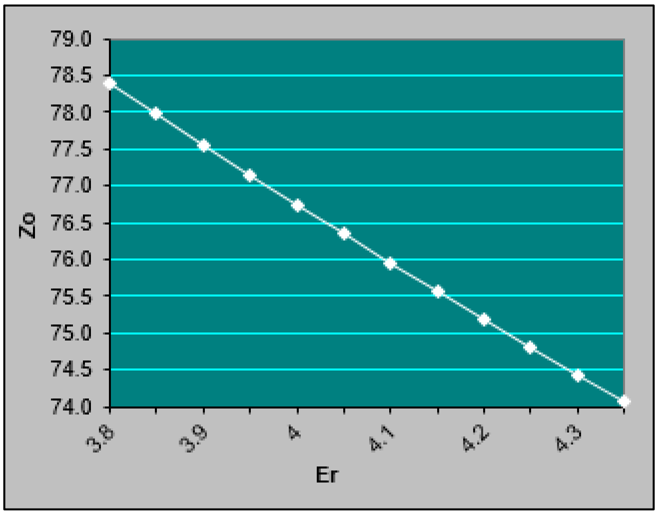
Click the Category Axis Title label and replace the H with Er. Right click the horizontal (Er) axis and format as required.

Right click the value (Z_0) 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.

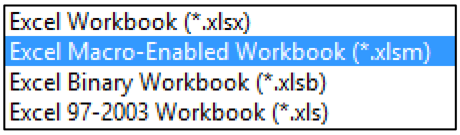
Using the controlled impedance functions in other workbooks

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.

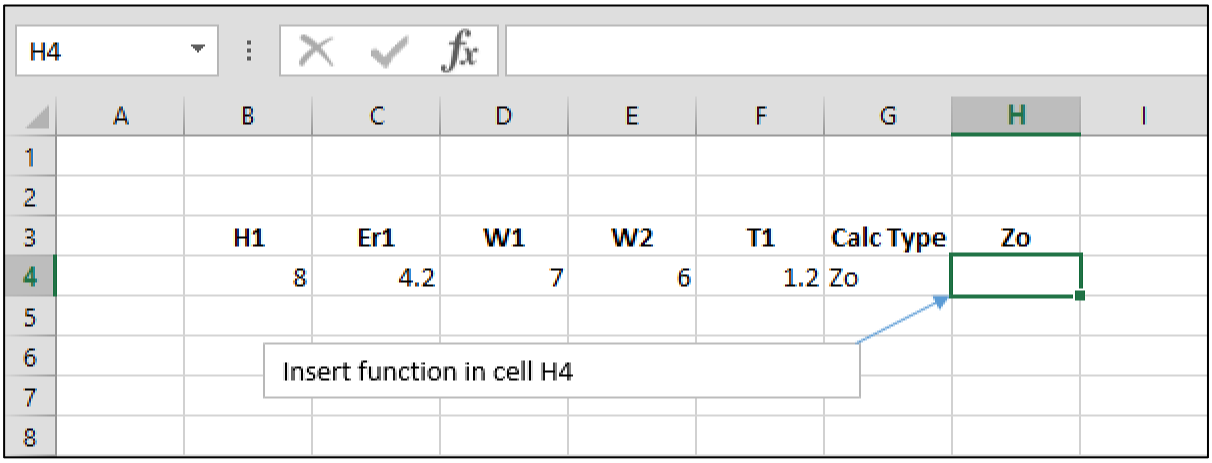
The functions use the board parameters, H1, W1, Er1, etc. as arguments. Parameter values can be derived from existing data in worksheet cells or inserted into the Function Arguments dialog directly.

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.

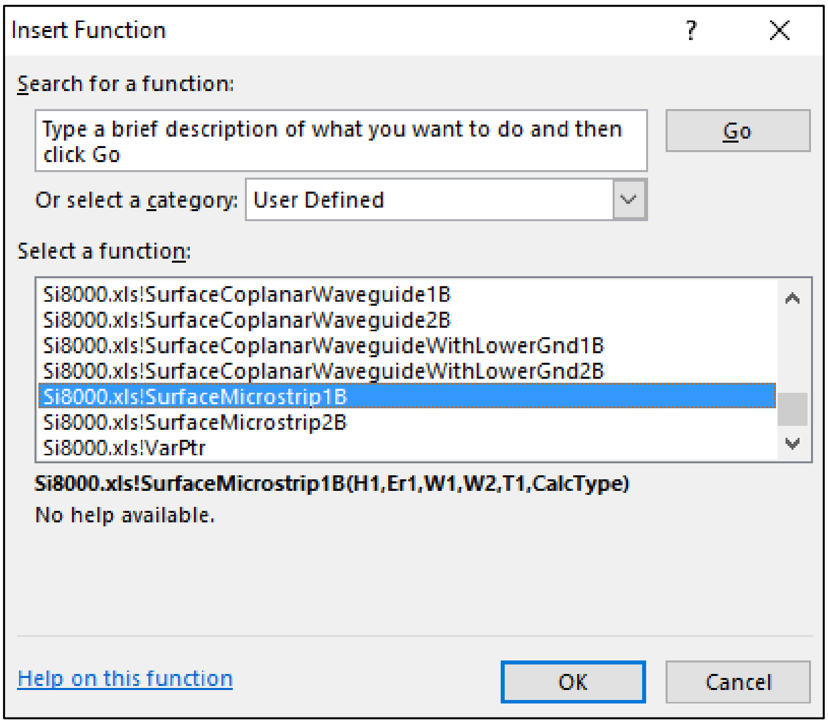
In the example below cells B3 – H3 contain the labels for a Surface Microstrip 1B structure. The parameter values for the Surface Microstrip structure are contained in cells B4 to G4. The Surface Microstrip 1B function will be inserted into cell H4 and reference cells B4 – G4.



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.

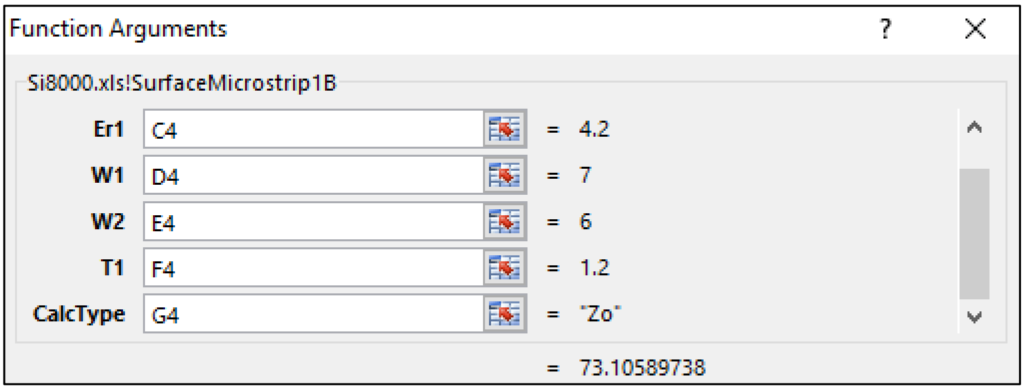


If necessary, scroll to the controlled impedance structure functions; click the function associated with the surface microstrip structure (Si8000.xls!SurfaceMicrostrip1B in this example) and click OK: the Function Arguments dialog is displayed.

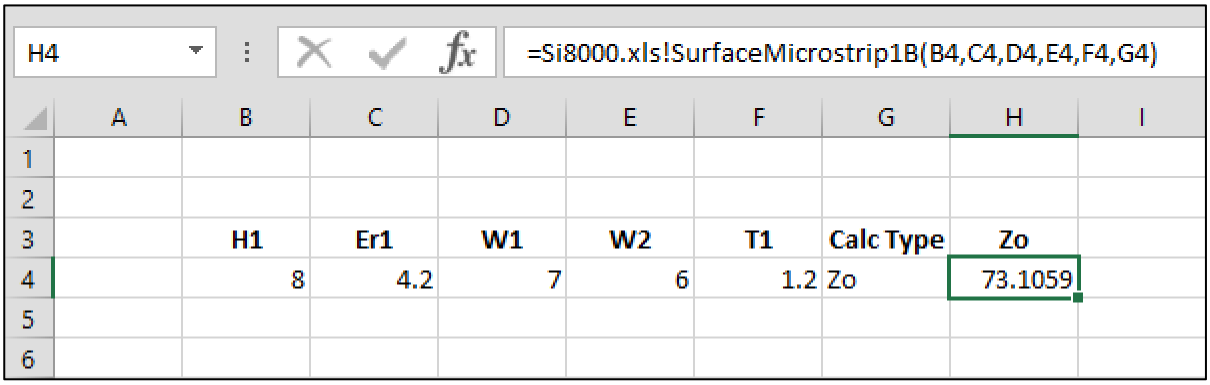
Using the Function Arguments dialog to enter formulas

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

to select the required number of decimal places.

Format as required.

H1	Er1	W1	W2	T1	Calc Type	Zo
8	3.80	7	6	1.2	Zo	76.24228
8	3.85	7	6	1.2	Zo	75.82773
8	3.90	7	6	1.2	Zo	75.41997
8	3.95	7	6	1.2	Zo	75.01880
8	4.00	7	6	1.2	Zo	74.62406
8	4.05	7	6	1.2	Zo	74.23555
8	4.10	7	6	1.2	Zo	73.85313
8	4.15	7	6	1.2	Zo	73.47663
8	4.20	7	6	1.2	Zo	73.10590
8	4.25	7	6	1.2	Zo	72.74079
8	4.30	7	6	1.2	Zo	72.38116
8	4.35	7	6	1.2	Zo	72.02687

Z_0 calculated for changing E_r

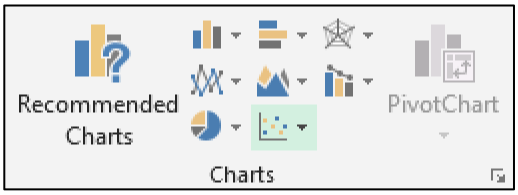
Charting results

Use the Excel Chart Wizard to chart the results.

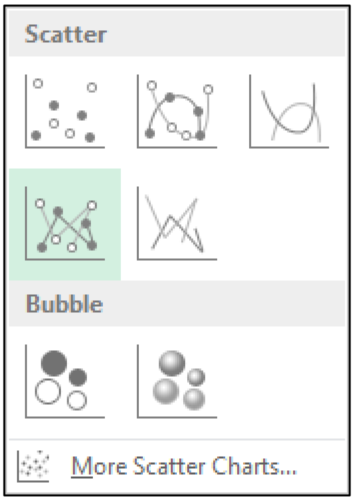
Select the area to be charted: in this example the Er1 and Z_0 ranges (to select non-adjacent ranges, press Ctrl while dragging the mouse over each range). If necessary, decrease decimal to the appropriate resolution.

H1	Er1	W1	W2	T1	Calc Type	Zo
8	3.80	7	6	1.2	Zo	76.24228
8	3.85	7	6	1.2	Zo	75.82773
8	3.90	7	6	1.2	Zo	75.41997
8	3.95	7	6	1.2	Zo	75.01880
8	4.00	7	6	1.2	Zo	74.62406
8	4.05	7	6	1.2	Zo	74.23555
8	4.10	7	6	1.2	Zo	73.85313
8	4.15	7	6	1.2	Zo	73.47663
8	4.20	7	6	1.2	Zo	73.10590
8	4.25	7	6	1.2	Zo	72.74079
8	4.30	7	6	1.2	Zo	72.38116
8	4.35	7	6	1.2	Zo	72.02687

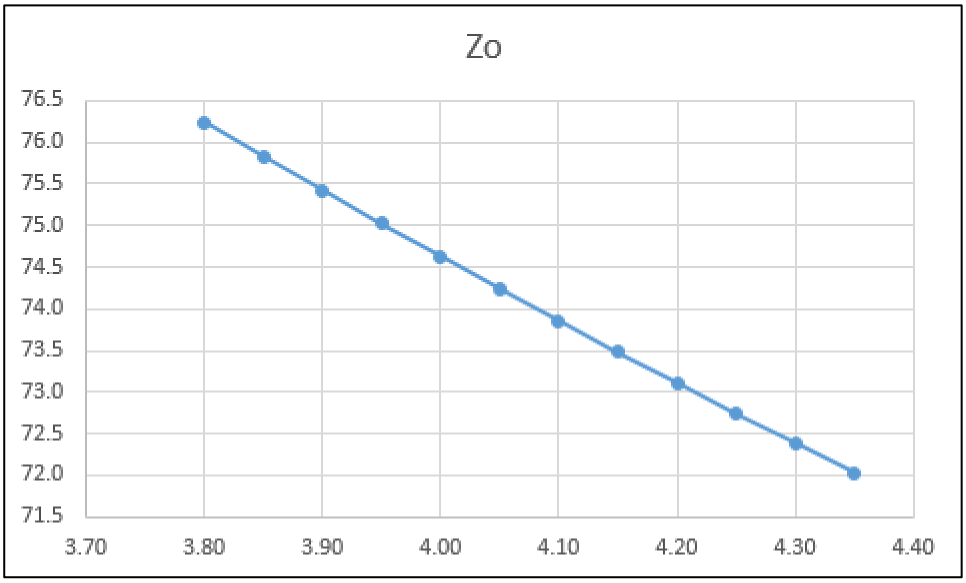
From the Insert tab on the ribbon Click the **Insert Scatter (X,Y)** button



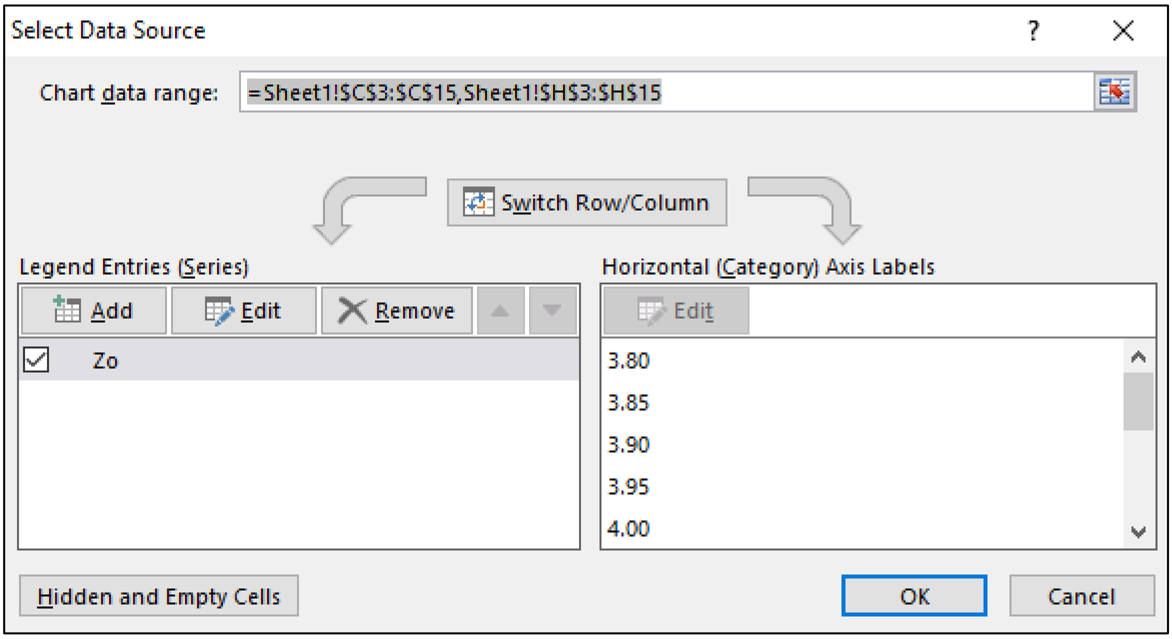
From the scatter type choose Scatter with Straight Lines and Markers



Excel charts the selected data.

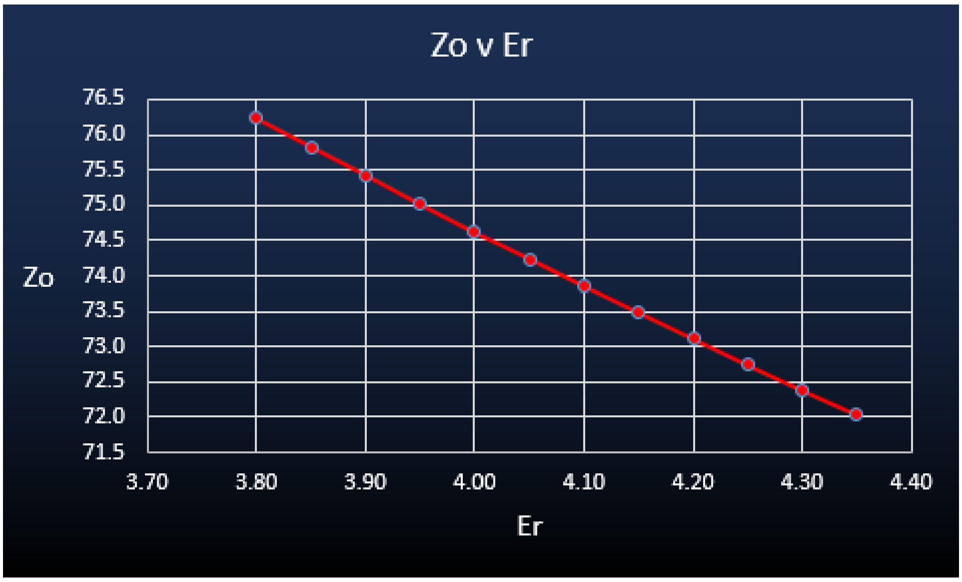


Right click the chart and choose Select Data and check the Data Source.

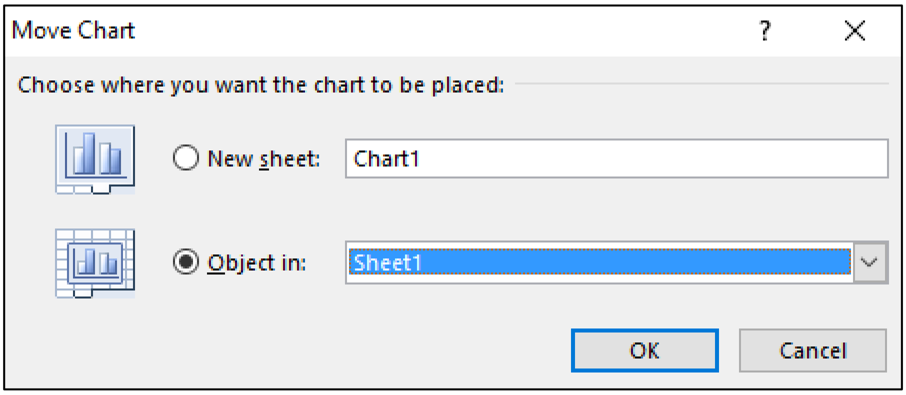


Check the Chart Data Range: in this case the cell references are correct.

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 Z_0 for surface and coated microstrip

Inserting the first data series

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

H1	Er1	W1	W2	T1	Calc Type	Zo
8	3.80	7	6	1.2	Zo	76.2
8	3.85	7	6	1.2	Zo	75.8
8	3.90	7	6	1.2	Zo	75.4
8	3.95	7	6	1.2	Zo	75.0
8	4.00	7	6	1.2	Zo	74.6
8	4.05	7	6	1.2	Zo	74.2
8	4.10	7	6	1.2	Zo	73.9
8	4.15	7	6	1.2	Zo	73.5
8	4.20	7	6	1.2	Zo	73.1
8	4.25	7	6	1.2	Zo	72.7
8	4.30	7	6	1.2	Zo	72.4
8	4.35	7	6	1.2	Zo	72.0
8	4.40	7	6	1.2	Zo	71.7

In this example, plot Z_0 against E_r.

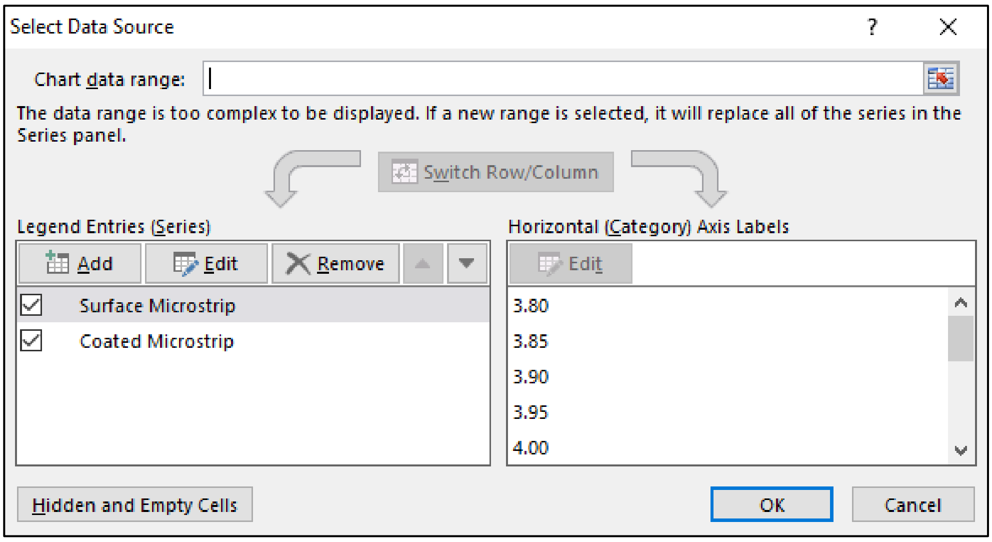
To format any chart item, right click the item and change its properties via the short cut menu.

Adding the second data series

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

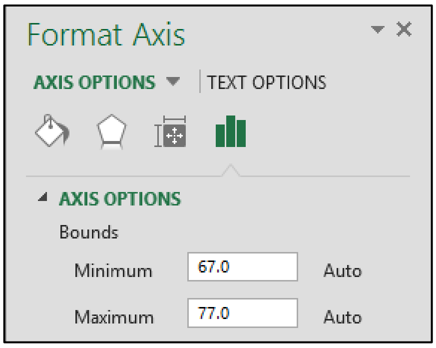
H1	Er1	W1	W2	T1	C1	C2	Cer	Calc Type	Zo
8	3.80	7	6	1.2	1	1	3.80	Zo	72.2
8	3.85	7	6	1.2	1	1	3.85	Zo	71.7
8	3.90	7	6	1.2	1	1	3.90	Zo	71.3
8	3.95	7	6	1.2	1	1	3.95	Zo	70.9
8	4.00	7	6	1.2	1	1	4.00	Zo	70.5
8	4.05	7	6	1.2	1	1	4.05	Zo	70.2
8	4.10	7	6	1.2	1	1	4.10	Zo	69.8
8	4.15	7	6	1.2	1	1	4.15	Zo	69.4
8	4.20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

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.

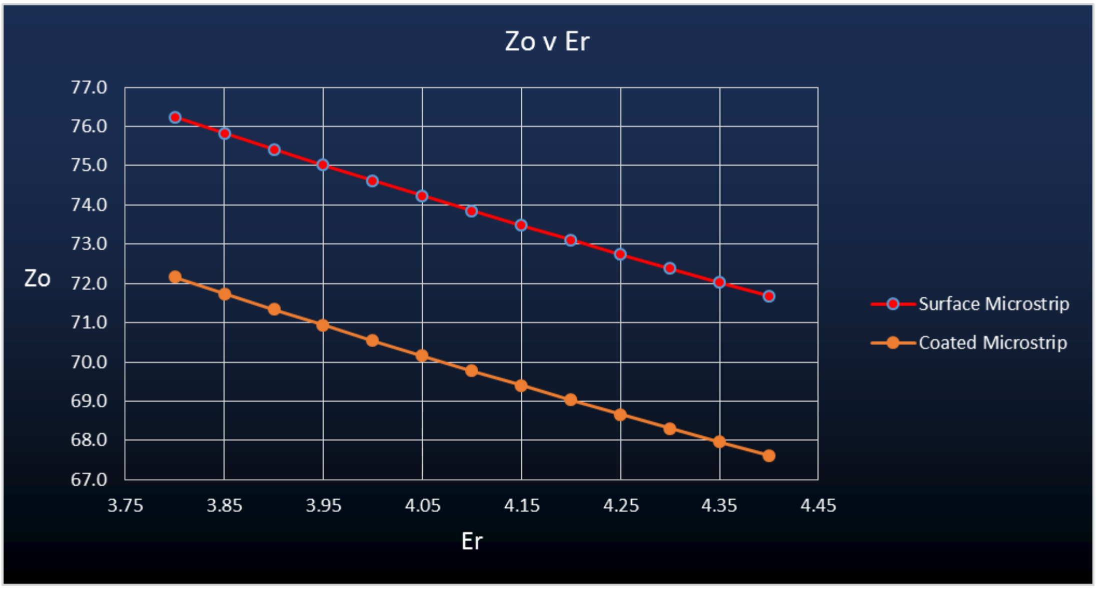


Add the name *Coated Microstrip* to the Name text box, click the Z_0 series from the Series list and add the name *Surface Microstrip*; press **OK**.

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_{even} and Z_{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_{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 Z_{odd} label.

Paste the Z_{odd} values into the column.

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

Press the **Calculate** button. Partial results are shown below.

H1	Er1	H2	Er2	W1	W2	S1	T1	Calc Type	Zeven	Zodd
3.00	4.2	3.00	4.2	7	6	10	1.2	Zeven	21.4	21.3
3.00	4.2	3.00	4.2	7	6	9.75	1.2	Zeven	21.4	21.3
3.00	4.2	3.00	4.2	7	6	9.5	1.2	Zeven	21.4	21.3
3.00	4.2	3.00	4.2	7	6	9.25	1.2	Zeven	21.4	21.3
3.00	4.2	3.00	4.2	7	6	9	1.2	Zeven	21.4	21.3
3.00	4.2	3.00	4.2	7	6	8.75	1.2	Zeven	21.4	21.3
3.00	4.2	3.00	4.2	7	6	8.5	1.2	Zeven	21.4	21.3
3.00	4.2	3.00	4.2	7	6	8.25	1.2	Zeven	21.4	21.3
3.00	4.2	3.00	4.2	7	6	8	1.2	Zeven	21.4	21.3
3.00	4.2	3.00	4.2	7	6	7.75	1.2	Zeven	21.4	21.2
3.00	4.2	3.00	4.2	7	6	7.5	1.2	Zeven	21.4	21.2
3.00	4.2	3.00	4.2	7	6	7.25	1.2	Zeven	21.4	21.2

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

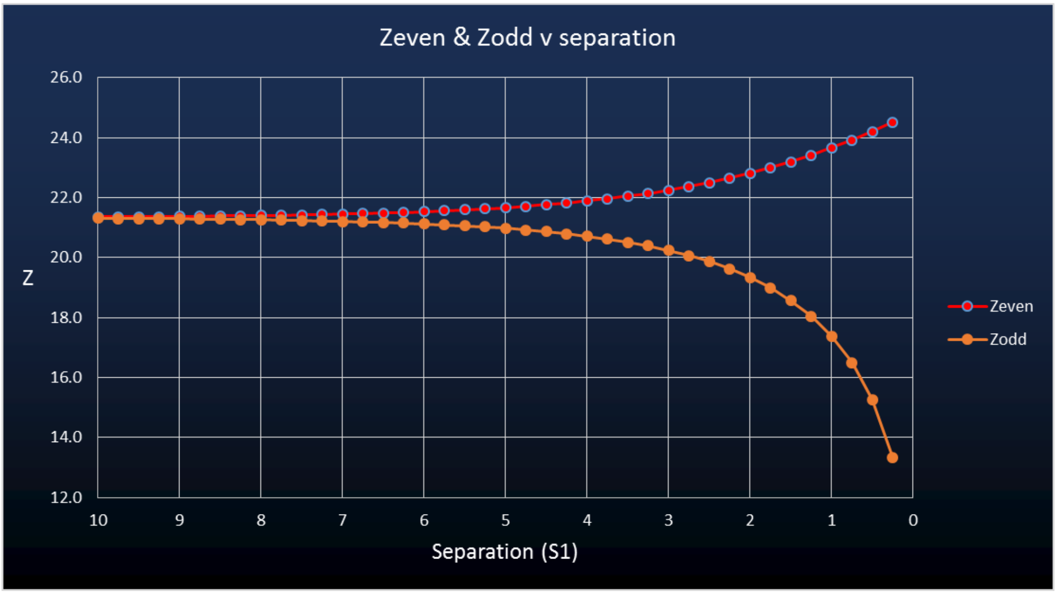
Drag the column of Z_{odd} values onto the chart.

Modify the chart so the Source Data Category (X) axis labels, and Series refer to the S1, Z_{even} and Z_{odd} 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.

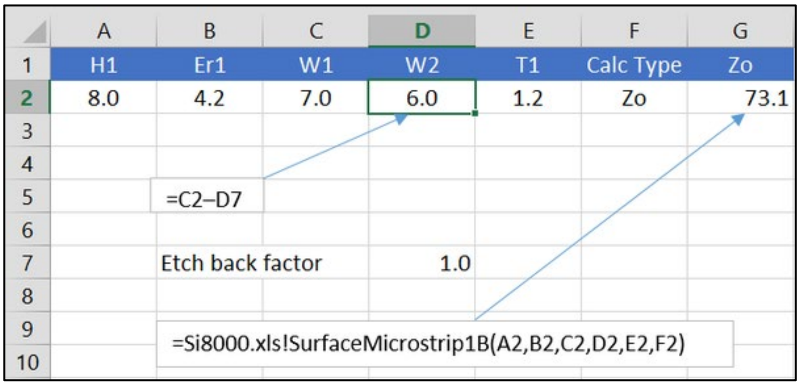


Using more complex models

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.



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:

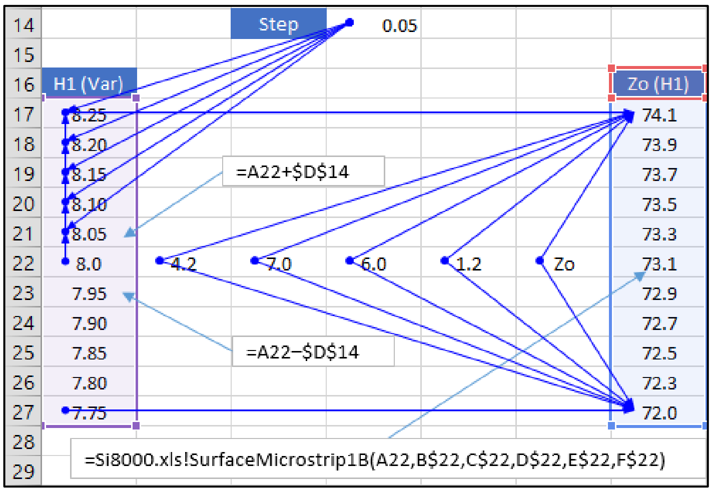
=Si8000.xls!SurfaceMicrostrip(A2,B2,C2,D2,E2,F2)

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)



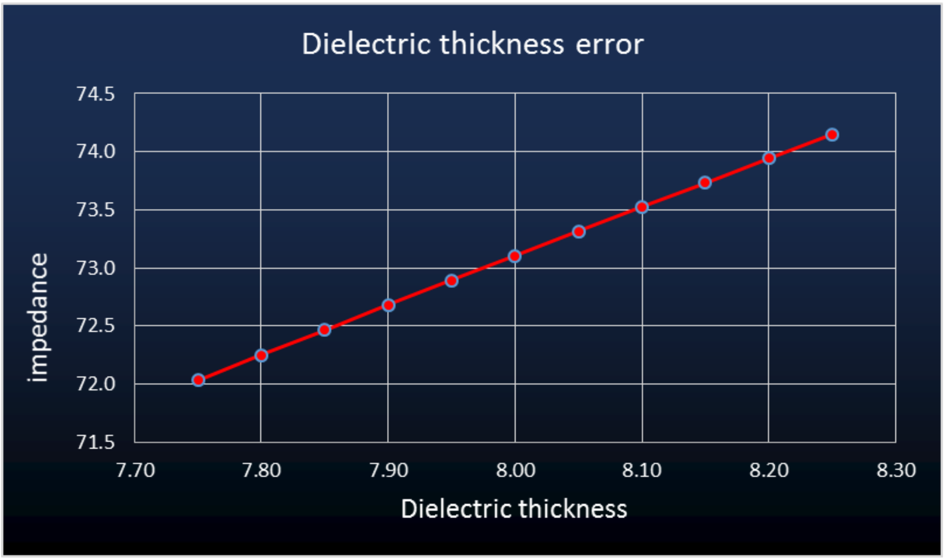
Copy the formula in G22 up to cell G17 and down to G27 as shown above

Create a step value of **0.05** in D14, enter the equation **=A22+$D$14** in cell A21 and fill it up to A17.

Enter the equation **=A22-$D$14** in cell A23 and fill it down to A27.

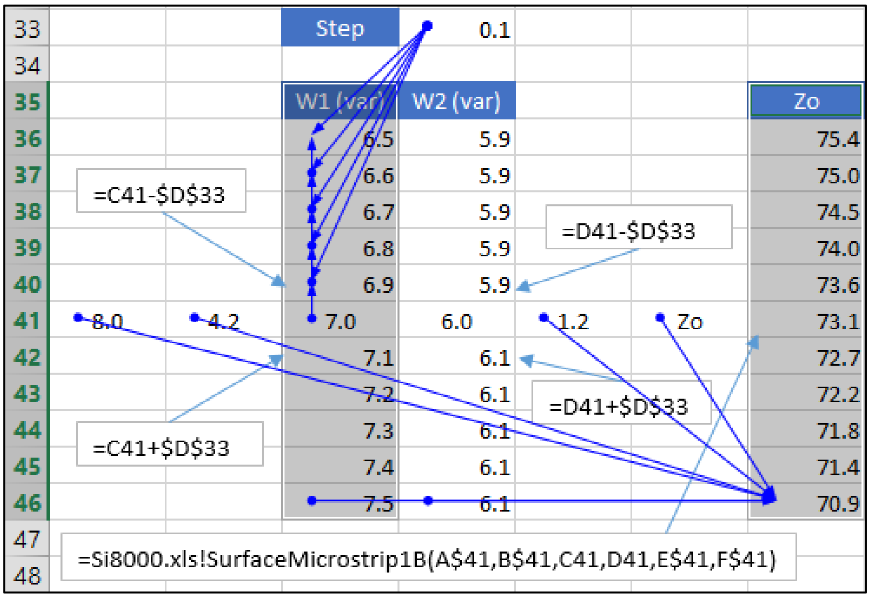
Use the Auditing Toolbar Trace Precedent and Dependent arrows to check references are as shown above. Press Shift-F9 to recalculate.

Select ranges H1(var) A17:A27 and Z_0(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

Finally, chart the effect of etch back error.

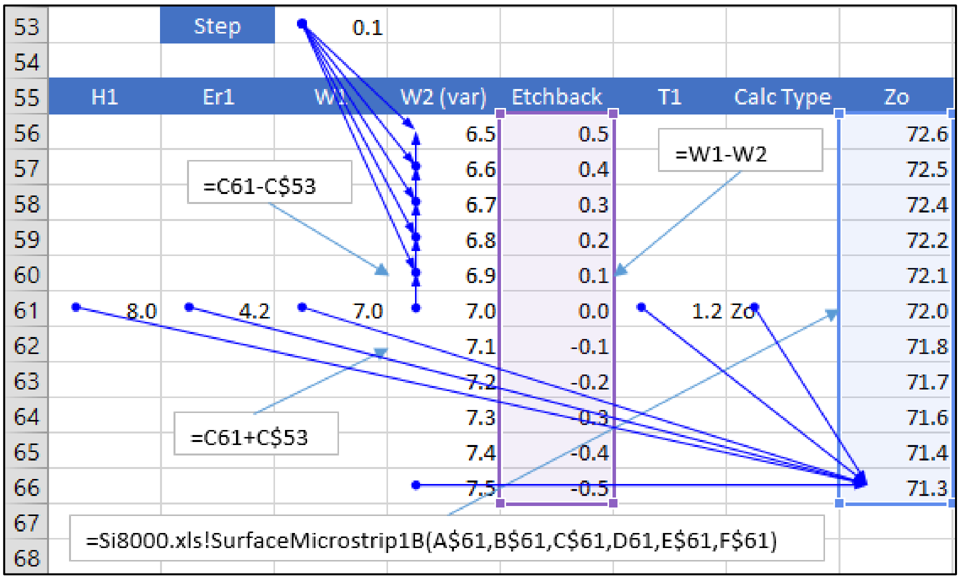
Create references to cells A2:F2 in cells A61:F61.

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-C$53** in cell D60 and fill up to D56.

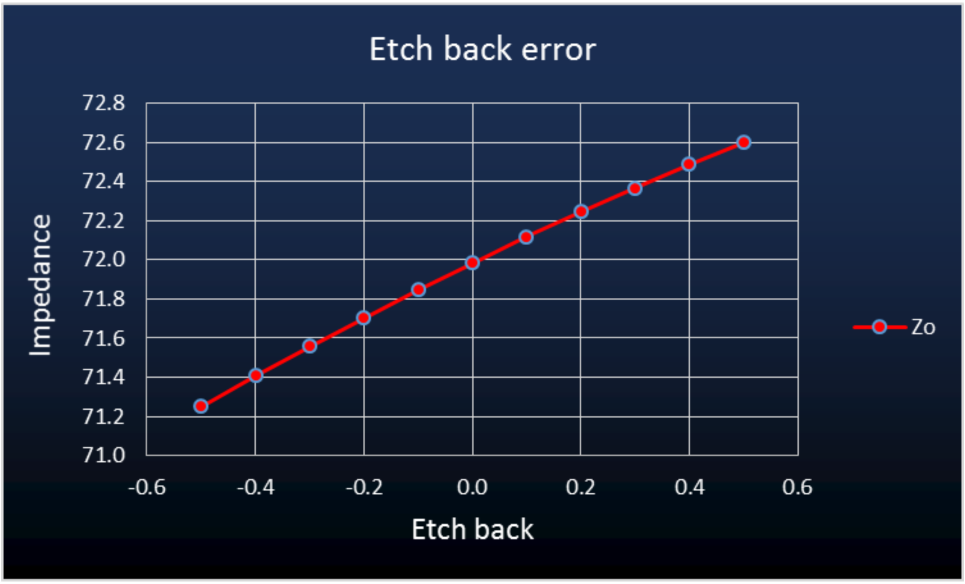
Enter the formula **=D61+C$53** in cell D62 and fill down to D56.

Insert cells E55:E66 and add label

Audit the precedents and dependencies.

Select cell ranges Etchback (E56:E66) and Z_0 (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

AC	Alternating Current
CMOS	Complementary Metal Oxide Silicon
DC	Direct Current
ECL	Emitter Coupled Logic
EMI	Electromagnetic Interference
FR-4	Epoxy Glass Dielectric Material
TDR	Time domain Reflectometry
TTL	Transistor-Transistor Logic
Z_o	Characteristic Line Impedance
Z_o'	Characteristic Line Impedance (Loaded)
E_r	Relative Permittivity (homogeneous dielectric materials)
E'_r	Effective Relative Permittivity (non-homogeneous dielectric materials)

References

Wadell, Brian C – Transmission Line Design Handbook, Artech House 1991

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

Hart, Bryan – Digital Signal Transmission, Pub: Chapman and Hall 1988

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

Sadiku, Matthew N O – Numerical Techniques in Electromagnetics, Pub: CRC Press 1992

Silvester P P – Microwave Properties of Microstrip Transmission Lines. IEE Proc vol 115 No 1 January 1969 pp43–48

Silvester P P & Ferrari R L – Finite Element for Electrical Engineers Pub, Cambridge University Press 1983

Brebbia, C A – The Boundary Element Method for Engineers, Pub: Pentech Press 1980

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
IEEE Trans MTT-26 No 2 February 1979 pp119–127

Bogatin, Eric; Justice, Mike; DeRego, 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