Professional PCB Stackup Design System

SB200a

User Guide

Polar Instruments Ltd

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

Fax: +44 (0)1481 252476

email: mail@polarinstruments.com
http://www.polarinstruments.com

MAN 198–0601

SB200a User Guide

POLAR INSTRUMENTS LTD

COPYRIGHT

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

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

DISCLAIMER

1. Disclaimer of Warranties

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

2. Limitation of Liability.

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

TRADEMARKS

Microsoft, Windows, Windows 95, Windows 98, Windows NT, Windows 2000, Windows 2003, Windows XP and Excel are registered trademarks of Microsoft Corporation.

IBM is the registered trademark of International Business Machines Corporation.

All other trademarks acknowledged.

SB200a Specifications

Maximum layer count	64+
Via rules	Conventional, blind and buried
Materials library	Foils RCC foils Cores Pre-Preg Solder mask Ident Peelable mask Coverlay
Post press compensation	Yes (user defined)
Finished thickness compensation	Yes
Cu thickness calculation	Yes
Board thickness calculation	Yes
User library	Yes
Save builds	Yes

Personal Computer Requirements

Computer	IBM PC AT or compatible
Processor	Pentium 1GHz or better
Operating system	Microsoft Windows 98, Windows NT 4.0 SP6, Windows 2000 SP4, Windows 2003 SP1, Windows XP SP1 or later
Environment	Latest .NET framework installed
System memory required	128MB recommended
Hard disk space required	200MB (min.)
Video standard	XGA (1024 x 768) or higher Hi color 16 bit or higher
CDROM drive	
Mouse	Microsoft compatible
Licensing	Fixed: Parallel/USB key
	Floating FlexLM licence (Windows servers only)

Guide To The Manual

Introduction	Introduces the Polar Instruments SB200a.
Installation/set up	Instructions for installing/uninstalling and activating the SB200a software.
Using the SB200a	Discussion of the SB200a user interface; creating and editing stackups.
Adding controlled impedance structures	Working with the Si8000/Si9000 field solver to add controlled impedance structures to the stackup model. Using the goal seeking facilities of the field solver to obtain the correct impedance for a structure.
The SB200a Materials libraries	Using the SB200a materials libraries, creating new libraries, adding material to the library.
Design rule checking	Using the SB200a Design Rule Checker to correct stackup design errors.

Contents

Introduction to the SB200a

The SB200a Stackup Design System

The Polar SB200a PCB Stackup Design System is designed to reduce the amount of time consumed in PCB stackup documentation and control. The SB200a offers both interconnect designers (PCB layout engineers) and PCB front-end engineers a fast and professional solution to layer stackup creation. The SB200a is both graphical and easy to use while providing formal documentation for everyone involved in ensuring the correct materials are used in the build process. The SB200a allows the user to view stackup in 2d or 3d format. Layer and material annotation is clear and easy to read and each layer may be selected and queried to display the associated material type and properties, including the associated data file. Visible drill information ensures that designers instantly know which layers support conventional, blind and buried vias. The SB200a allows you rapidly to build and share stacks and verify via aspect ratios and track spacing rules. The stack contains base material information combined with layer description and a complete listing of transmission line structures deployed in the stack. Keeping all stack information in one file ensures that manufacturing data is accurately shared between original designer and fabricator.

Integration with the Si8000/Si9000

The SB200a is fully integrated with the Si8000 Controlled Impedance Quick Solver and the Si9000 PCB Transmission Line Field Solver so the user can quickly add controlled impedance structures to layers in the stackup. Using the Goal Seek facility of the Si8000/Si9000 the designer or board fabricator can rapidly arrive at the controlled impedance structure parameters to produce the target impedance.

Materials library

The SB200a supports a flexible materials library. This allows the designer to use standard materials data, and also provides the facility to create new material libraries. PCB fabricators can also build libraries of commonly stocked materials to give interconnect designers visibility of materials held in stock. Materials libraries may be downloaded from most popular base material suppliers in the Polar Material Partner program.

Preferred builds

PCB fabricators are able to create and share preferred builds and exchange this information with designers. Build information also includes blind and buried via information, simplifying sharing stackup and drilling information between board shops and the design community.

Dimensional information

Finished thickness is a critical dimension in many applications; the SB200a keeps track of finished thickness and tolerance, and allows fabricators the flexibility of adding in house post press thickness for prepreg layers; the SB200a takes into account plating thickness where appropriate.

High layer count boards

On boards with high layer counts changes can result in a non-symmetrical stack. The Design Rules Check checks for symmetry across the stack, and ensures material symmetry is maintained. Symmetrical build mode can be set to ensure that any changes are applied equally across the stack.

Documentation

In addition to saving the stack in efficient electronic format, the stack graphics can be output in a variety of formats, GERBER, DXF, bitmap or JPEG. In addition the stack data can be exported in comma-separated form for inclusion in other systems. Customisable printouts make it easy to discuss alternate builds and pricing impacts with fabricators.

Supplier management

When multiple sourcing PCBs or when moving from prototype to volume production, the stack and fabrication design rules checks ensure that you do not overlook the manufacturing capabilities of your chosen suppliers. In addition the professional documentation output from the SB9000 ensures that layer stack information is accurately conveyed from your team to your PCB suppliers.

Graphical interface

The SB200a offers an easy to interpret graphical interface. Clearly showing the layers supporting blind and buried vias, the SB200a also records the data file for each layer (including ident and peelable mask layers). The graphical interface is especially designed to simplify the process of communication between interconnect designer and fabricator. OEMs who need to manage boards sourced from multiple suppliers will also find this facility invaluable. In addition to physical layers the SB200a adds mask and notation for electrical layers.

Installing the SB200a

Activating the SB200a

Activation will depend on the license purchased. It will be necessary to activate the product license prior to performing calculations with the SB200a. The SB200a is licensed using the MacroVision FlexLM License Manager via a license file on a license server or the local machine or a software key connected to either the parallel port (typically LPT1) or a USB port. The FlexLM licensing files should be installed and running on a locally accessible server (the server must be running to manage licensing) or on the local machine for a node-locked license.

Licensing the SB200a using FlexLM

When the first stackup build is performed the SB200a FlexLM License finder asks for the name of the License Server or License File.

Ensure the **Specify the License File** option is selected and press **Next**.

Activation using a node-locked license

Enter the name of the license file or browse the local machine to locate the license file and press **Next**.

Activation using a license server

Enter the name of the license file or browse the network to locate the license file and press **Next**.

The name of the license server will normally be the Network Identification name of the computer acting as the Licensing Server.

The SB200a will perform stackup builds within the terms of the license purchased.

Installing the software key

If a software key is used insert the key into the parallel printer port or USB port; if required, plug the printer cable into the software key. Locate the license file as described above.

Note: If installing the software key on a Microsoft Windows NT or Windows 2000/XP system it will be necessary to be logged on as Administrator.

Uninstalling the software

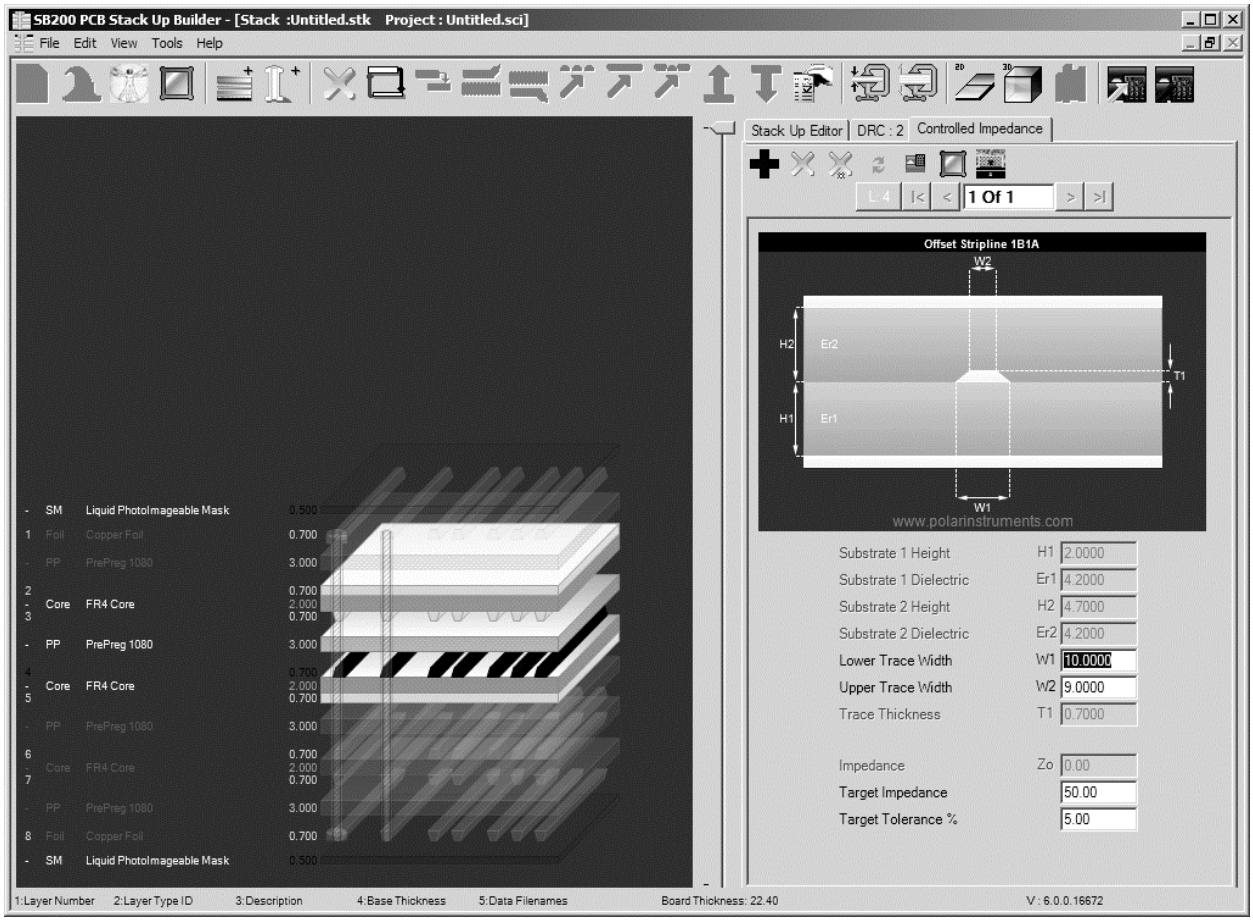
To uninstall the SB200a software click the Windows Start button and choose Settings and Control Panel.

Double-click Add/Remove Programs and choose SB200a from the list. Click Remove.

Using the SB200a Stackup Design System

The SB200a Stackup Design System

Double-click the SB200a icon to start the SB200a program. The main screen is displayed.



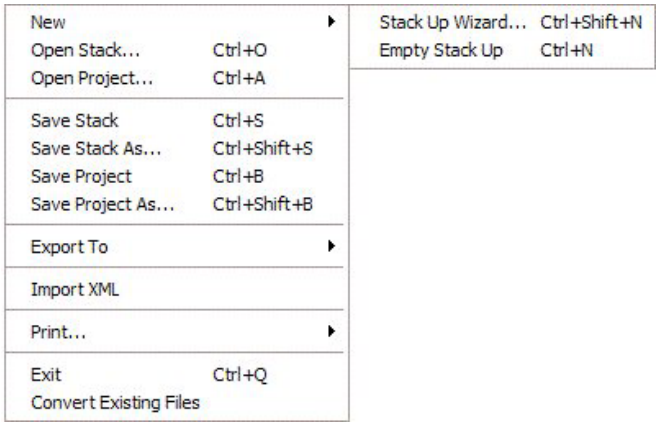
The SB200a screen comprises:

- The menu bar — drop-down menus containing all the SB200a commands
- The tool bar — incorporating short cut tool buttons to the most common menu commands
- The stackup build and construction window — where the board stack up is built and edited
- The Controlled Impedance window displaying the controlled impedance structures (if any) for the selected layer.

- Stack Up Notes/DRC (design rules check) — a free form text area for explanatory or commentary notes/DRC
- Stack Up Information properties area — table containing information related to the whole stackup.
- Selected Item Information area — properties table containing the attributes of the layer currently selected in the stackup

The SB200a Menu System

The File menu



The **File** menu allows for creation of new stackups and projects and opening, saving, printing and converting existing stackups and projects.

Saving stackups

Click the Save button to save the stackup. Users are recommended to save the stackup frequently during the stackup creation process to avoid data loss; stackups are saved as .stk files

Saving projects

Stackups that incorporate controlled impedance structures are saved as *projects*. Use the Save Projects command to save a stackup and its controlled impedance structures; projects are saved as .sci files.

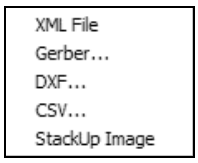
Printing stackup information

To print the stackup information, choose the Print… command from the File menu. Select to print either a summary report or a full output.

Exporting stackup information

The SB200 incorporates the facility to export the printed output in XML, Gerber, DXF, CSV as well as graphic image

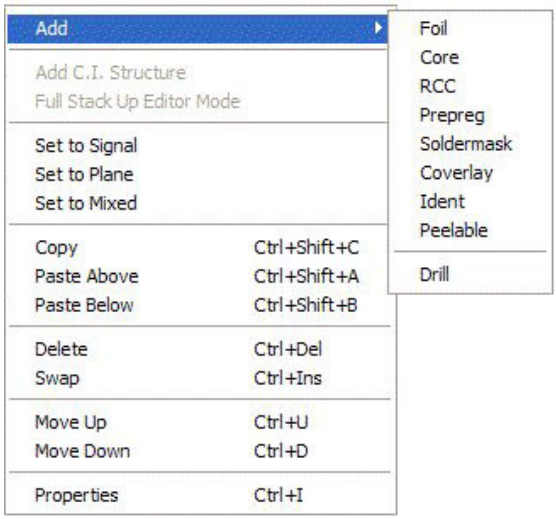
formats. From the File menu choose the Export To command and choose the format from the Export To sub-menu.



Backing up stackups and libraries

It is strongly recommended that stackup files (assigned the .stk extension), project files (assigned the .sci extension) and library files (assigned the .mlbx extension) be backed up to a secure location.

The Edit menu



The **Edit** menu contains the commands necessary to create and modify board stack ups. The designer or fabricator works within the free-form stackup build and construction window and adds layers of core, prepreg, foil, etc., from the materials library.

Layers can be changed to signal, plane or mixed, moved up or down or copied and pasted, or assigned properties as required.

The SB200a will apply design rules to the finished stackup to ensure compliance with good board design, (for example, the SB200a will check for symmetrical layers, misplaced layer types, adjacent copper layers, etc.)

The View menu

The **View** menu allows the SB200a to display the stackup in a 2-dimensional or 3-dimensional aspect.

The SB200a toolbar

The SB200a toolbar comprises shortcut links to the most popular commands.

Note: the toolbar buttons will

Pause the mouse over each tool button to display the tool's screen tip

File operations

 Create new stackup

 Stackup Wizard

Stack Building Operations

 Symmetrical Mode

 Mirroring Mode

 Add layer to the stackup

Click to select the layer type. The list of layer types is displayed in the associated sub-menu.

Layers available include:

Foil… Add foil layer to the stackup

Prepreg… Add prepreg layer

Core… Add core layer

Soldermask… Add solder mask

Coverlay… Add Coverlay layer

Ident… Add screened ident layer

Peelable… Add peelable mask

RCC… Add resin coated copper layer

 Add mechanical/laser drill between layers

Editing the stackup

 Delete selected stackup layer or drill

 Swap selected material

 Copy material of the selected layer

 Paste material above selected layer

 Paste material below selected layer

 Set the selected electrical layer as a signal layer

 Set the selected electrical layer as a mixed signal/plane layer

 Display properties dialog for the selected layer or drill

 Set the selected electrical layer as a plane

 Move selected layer up one layer

 Move selected layer down one layer

 Apply Finished Thickness

 Reset Finished Thickness

Changing the stackup view

 Display 2-dimensional view

 Display 3-dimensional view

Managing the materials library

 Display materials library

Exchanging data with the Si8000 or Si9000 Field solver

 Copy controlled impedance data to Field Solver

 Paste Controlled Impedance Data From Field Solver

Configuring the SB200

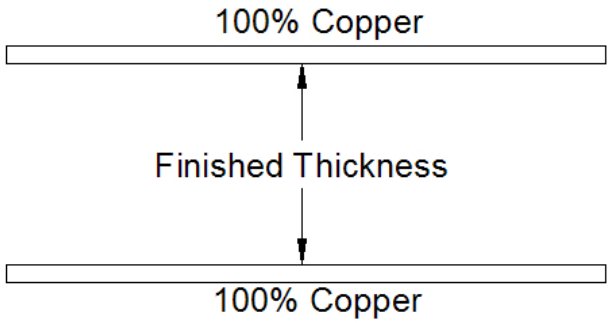
When first run, the SB200 environment is initialised to its factory settings. These may require adjustment before outputting a finished stackup and/or project. Select the Tools|Options menu to change the default settings.

Manufacturing Constraints

The Manufacturing Constraints options consist of a collection of manufacturing capabilities. They may refer to differing levels of technology offered by PCB manufacturers. The required information can normally be obtained from the manufacturer or the manufacturer's website.

Finished Thickness

Finished thickness is a critical dimension in many applications; the SB200a keeps track of finished thickness and tolerance, and allows fabricators the flexibility of adding in house post press thickness for prepreg layers. The SB200a also takes into account plating thickness where appropriate.



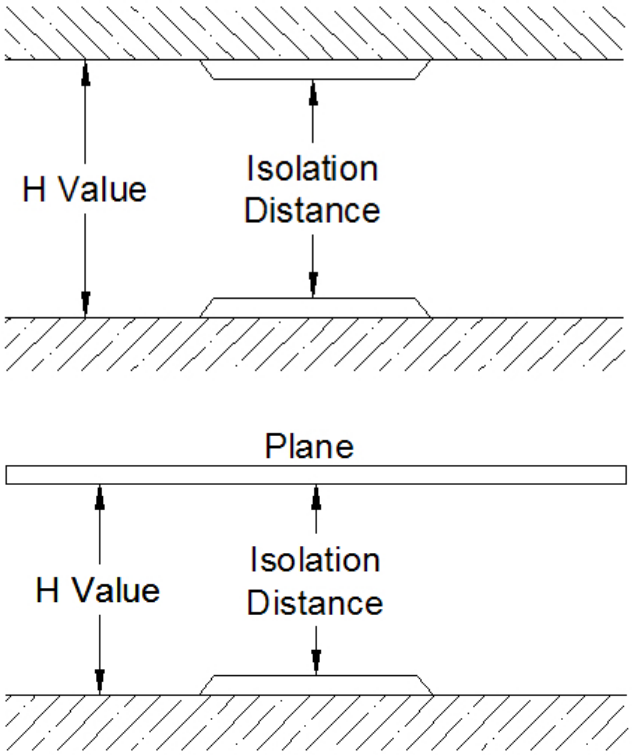
Finished thickness is the thickness of a prepreg after pressing between two solid sheets of copper, and should be defined in the materials library; it will be a lower value than the Base Thickness. This value will be dependant upon the board shop's process parameters.

SB200 H Value calculation

The SB200a is fully integrated with the Si8000 Controlled Impedance Quick Solver and the Si9000 PCB Transmission Line Field Solver so the user can quickly add controlled impedance structures to layers in the stackup. Using the Goal Seek facility of the Si8000/Si9000 the designer or board fabricator can rapidly arrive at the controlled impedance structure parameters to produce the target impedance. The H Value calculation produces the dielectric heights required for the Polar SI8000/9000 field solvers.

Isolation Distance

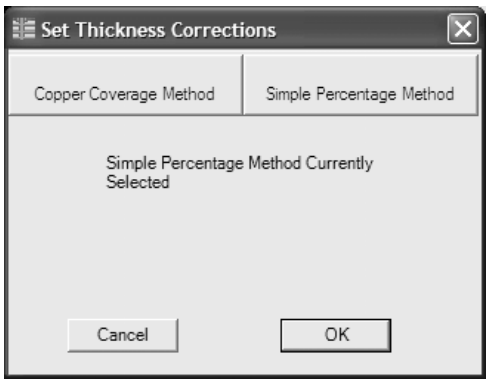
Copper may be embedded into the prepreg from either or both sides. Embedding copper in the prepreg reduces the thickness from the Finished Thickness. This reduced thickness is referred to as the *isolation distance*. Where more than one prepreg layer is involved the reduction is averaged across all the prepregs. A figure for isolation distance of less than zero implies an impossible build, i.e. the embedded copper totals more than the thickness of the prepreg.



The H Value is the sum of the isolation distance and the full thickness of the adjacent copper(s). However this does depend upon the controlled impedance structure and the layer type. Where a structure straddles one or more copper layers the full intervening copper thickness are added to the sum of the prepreg isolation distances. Power plane copper is not included in the H Value. Where a structure straddles a mixed layer, the copper of the mixed layer is included in the H Value, but if that mixed layer is the power plane for the structure the thickness of the copper is ignored. The H Value may be larger than the Finished Thickness because those areas of prepreg where there is no copper embedded are subjected to less pressure than they would be if all areas were covered with copper. An H Value larger than Base Thickness implies an impossible build. The H Value used in a structure can be calculated by summing the isolation distances, the intervening copper finished thickness and the signal trace thickness where applicable.

Thickness Options

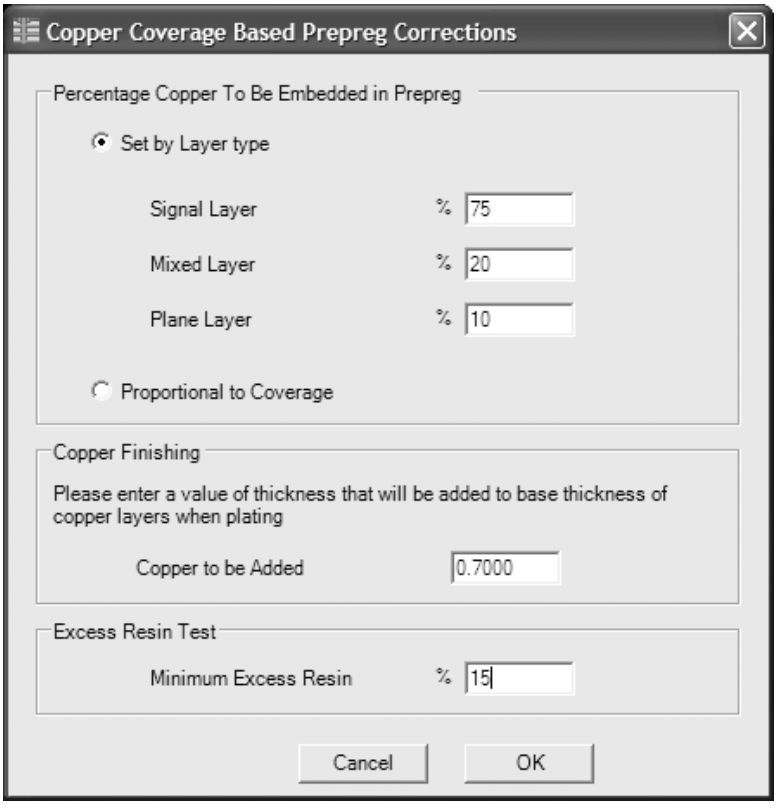
To specify the finishing method, choose Tools|Edit Thickness Options and click *Copper Coverage* or *Simple Percentage*.



Each method requires that the amount of copper to be added where plating is required be set. In addition, where the Excess Resin design rule check is used the minimum acceptable value must be set.

Copper Coverage method

The Copper Coverage method allows the user to set the amount of copper that will be embedded into the prepreg. This can be set as a single value for each electrical layer type. Alternatively the amount of copper embedded will be calculated on an electrical layer by layer basis dependent upon the copper coverage for the layer set in the properties window. The greater the copper coverage the smaller the amount of copper embedded.



Set by Layer Type

Specify the percentage of copper for each layer type, signal, mixed or plane, that will be embedded in the adjacent prepreg.

Proportional to Coverage

Specify finishing dependant upon actual copper coverage to allow the values to be modified on an individual basis.

Copper coverage is inversely proportional to the degree of copper embedding. The amount of the copper embedded can be expressed as:

$$Ce = Cu(1-Cc/100)$$

where

Ce is embedded copper

Cu is trace thickness

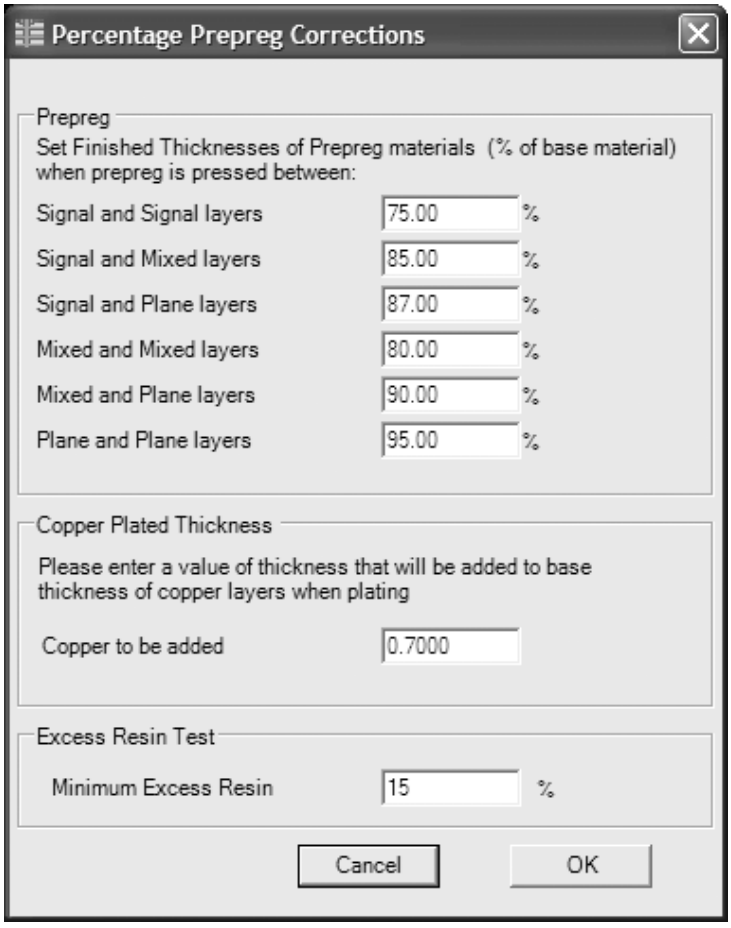
Cc is copper coverage as a percentage

The copper coverage for each layer can be set individually under Properties.

Simple Percentage Method

The Simple Percentage Method allows the user to set the percentage of Prepreg base height, which will be used to determine the Isolation Distance. The percentage is set for each electrical layer type pair.

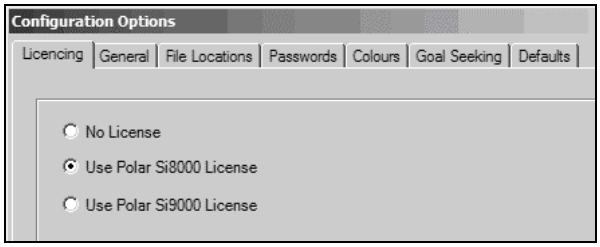
- Signal – Signal
- Signal – Mixed
- Signal – Plane
- Mixed – Mixed
- Mixed – Plane
- Plane – Plane



Environment Settings

From the Tools menu choose the Options command to display the Configurations Options dialog.

Controlled impedance calculations

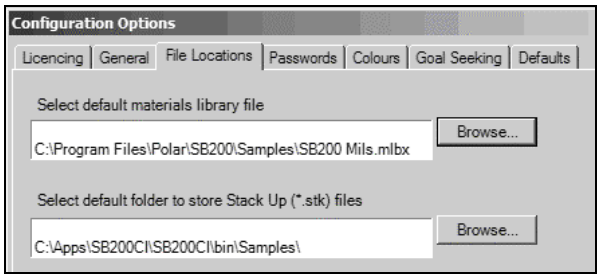


To activate the SB200 controlled impedance function, ensure that the Si8000 or Si9000 is installed and click on either **Use Polar Si8000 License** or **Use Polar Si9000 License** option as appropriate.

Choosing default file locations

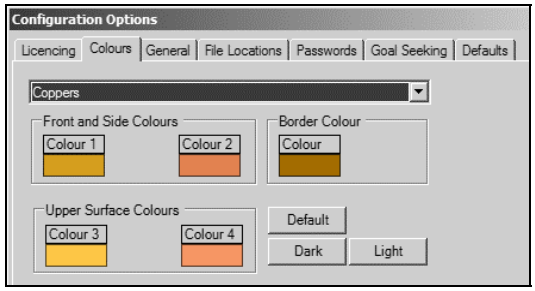
Users can choose which materials library the SB200 uses at start-up. Click the File Locations tab and use the Browse button to navigate to the library (.mlbx) file.

In addition a default location for stackup or project files may be specified. Browse to the folder and click OK ot confirm (create a new folder if necessary).



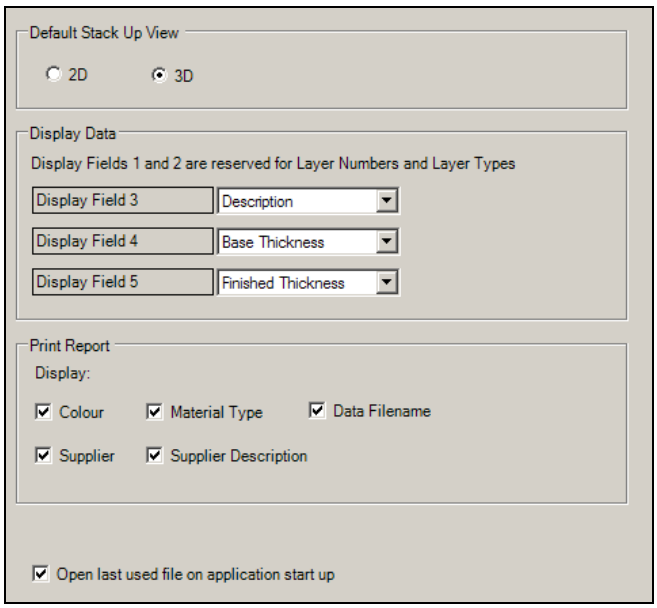
Choosing background and stackup layer colours

Choose the Colours tab to change stackup component colours from their factory defaults.



Specifying the default view and opening file

Select the General tab to specify the Default Stackup View (two or three-dimensional).

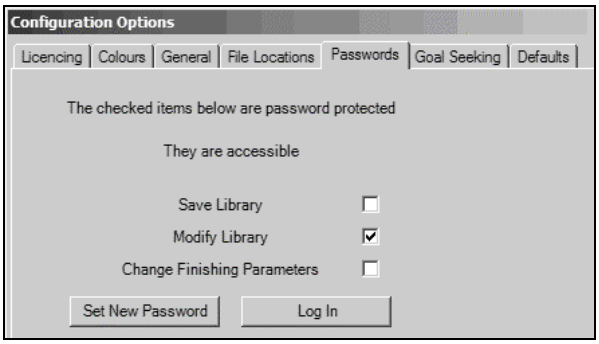


From the Display Data section select which data to display alongside the stackup; from the Print Report section select which data to display in the printout, click the **Open last used…** check box to specify that the SB200 should open the last used file on start-up.

Security — setting passwords

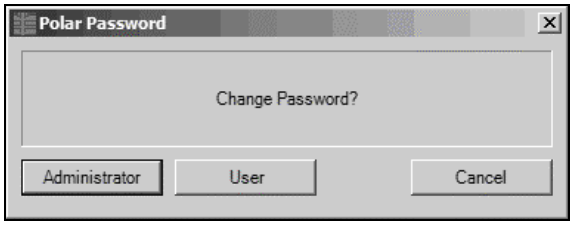
The SB200 security system provides for multi-level password protection. If a User password is defined, it will be requested each time the SB200 is started. Setting an Administrator password will prevent unauthorised changes to the SB200 configuration.

Select the Passwords Tab, and then Log In

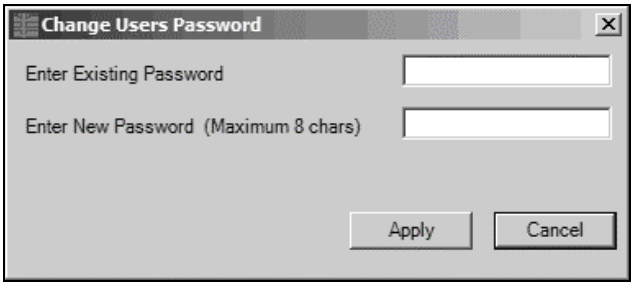


Default Passwords are blank for both the User and the Administrator. With blank passwords the password protection is transparent, and no protection is provided.

Click Set New Password



Chose whether it is the User password or the Administrators Password that is to be changed.



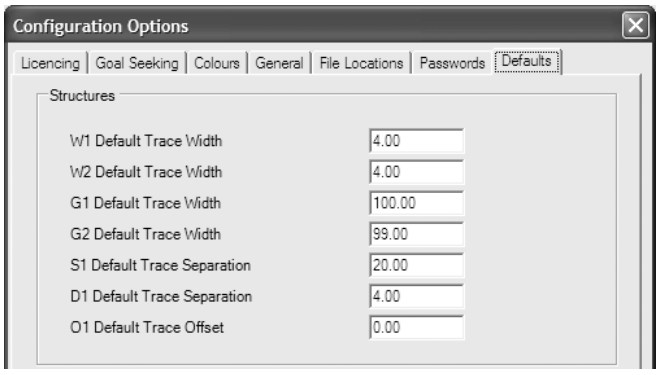
Enter the existing Password; this will be validated when moving to the Enter New Password text box. If the password is incorrect the focus will not leave the “Enter Password” text box.

Enter the new password, then retype it in the Confirmation text box which will appear. The two passwords will be compared and if different will enforce password re-entry.

Note: the new Users password will not be applicable until the program is re-started.

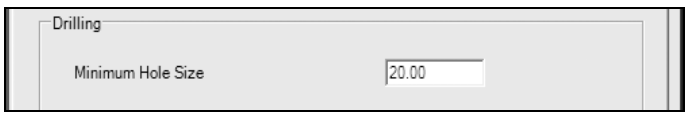
The Administrators password will be requested each time the Configuration Options are opened.

Specifying default trace parameters



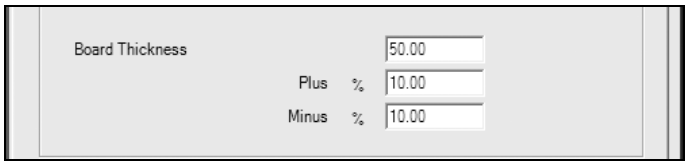
When adding a new structure default values are entered for the trace widths and separations. These values are set under the ‘Defaults’ tab

Specifying minimum drill hole size



In the Drilling section set the default value for the minimum drill hole size

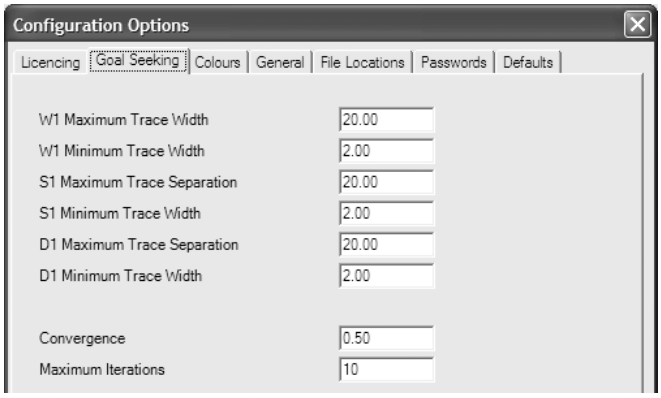
Specifying default board thickness



In the Board Thickness section set the default values for board thickness and tolerances.

Specifying Goal Seeking Parameters

Click the Goal Seeking tab to specify the defaults for trace widths and separations used during goal seeking.



During goal seeking the calculated value for impedance will progressively converge upon the target value. In the Convergence text box specify the difference between the target impedance and the actual impedance at which goal seeking will stop.

Use the Maximum Iterations text box to limit the number of iterations used during goal seeking.

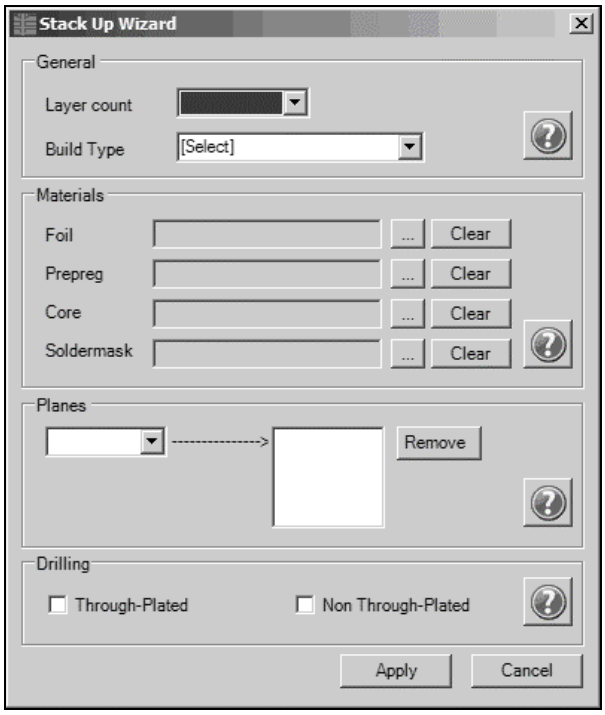
Creating a new stackup

Stackups may be created using the Stackup Wizard or manually using the editing window.

Using the Stackup wizard

Stackup wizard button

The Stackup Wizard guides the user through the process of creating complex stackups in only a few steps. Click the Stackup Wizard button or choose the Stackup Wizard command from the File|New sub menu. The stackup editing window is cleared and the Stackup Wizard is displayed.



Stackup Wizard

Using the Wizard the user can specify the layer count and build type, stackup materials, planes and drill types in a single operation.

Electrical layer count

Begin by specifying the electrical layer count — up to 64 electrical layers may be specified. Choose the number of layers from the drop down list box.

Build type

Choose the build type (Foil or Core) from the drop down list box. Core builds contain only core materials; most builds will be foil builds — containing internal layers of cores with two outer cores.

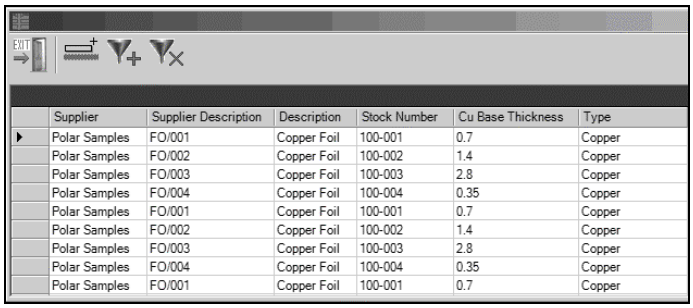
Choosing stackup materials

Note; if Core build type has been specified the Foil material control will be disabled.

Add material button

To include a foil layer click the Foil **Add Material** button; the library of foil materials is displayed.

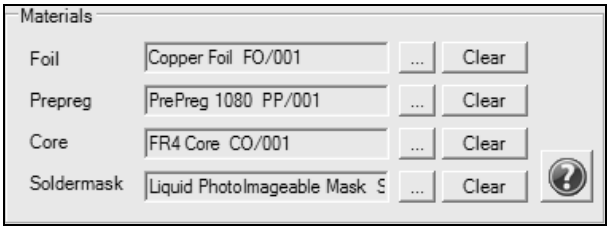
Choose the foil material from the list of materials available and click the **Select Highlighted Material** button; the material is added as a foil layer to the stackup.



Supplier	Supplier Description	Description	Stock Number	Cu Base Thickness	Type
Polar Samples	FO/001	Copper Foil	100-001	0.7	Copper
Polar Samples	FO/002	Copper Foil	100-002	1.4	Copper
Polar Samples	FO/003	Copper Foil	100-003	2.8	Copper
Polar Samples	FO/004	Copper Foil	100-004	0.35	Copper
Polar Samples	FO/001	Copper Foil	100-001	0.7	Copper
Polar Samples	FO/002	Copper Foil	100-002	1.4	Copper
Polar Samples	FO/003	Copper Foil	100-003	2.8	Copper
Polar Samples	FO/004	Copper Foil	100-004	0.35	Copper
Polar Samples	FO/001	Copper Foil	100-001	0.7	Copper

Repeat the procedure for prepreg and core materials and the (optional) solder mask layers.

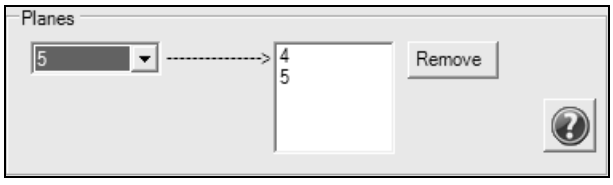
Use the **Clear** button to remove a layer from the stackup.



Stackup materials selected

Nominating power planes

Use the drop down list box to specify that a plane should be a power plane. Select all planes as required. To remove a power plane from the list select the plane number from the list and click Remove.



Layers 4 and 5 specified as power planes

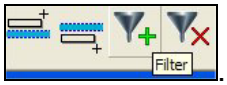
Adding drill information

To add a drill information between electrical layer 1 and the last layer click the Through-Plated and Non-Through-Plated check boxes as required.

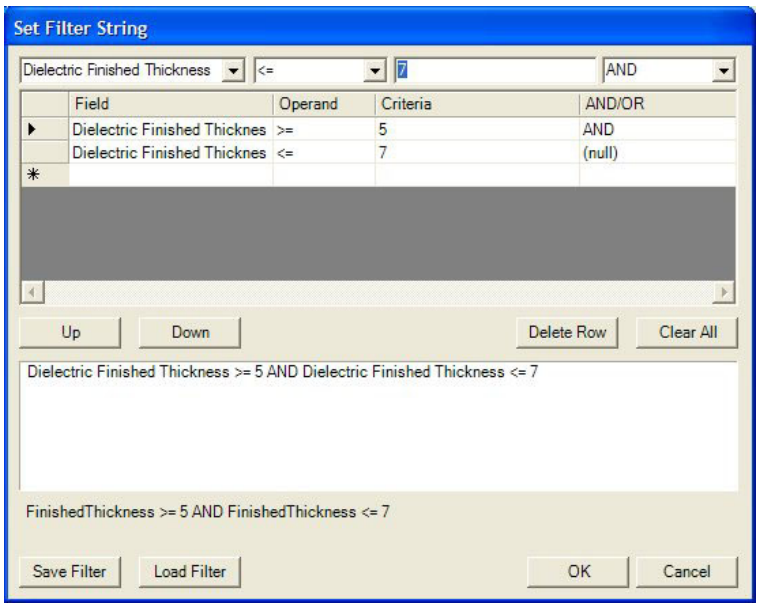
With all build options specified click **Apply** to complete the stackup. The finished stackup appears in the stackup window.

Filtering Materials

When using the Stack Wizard, adding or swapping materials, available materials (Foils, Prepregs, etc.) are listed in the associated material library dialog. Lists can be filtered for materials matching desired parameters (dielectric thickness, Er, etc.)



In the library window click the 'Filter' button to display the Set Filter String dialog.



Filter strings can be created and saved for future use. To recall an existing filter click the Load Filter button, choose the filter file and click OK.

Building the filter string

Build the filter string by selecting parameters, operands and criteria from the drop-down boxes. If the AND/OR box is selected another row is automatically added to the grid. The filter language is a sub-set of common database commands. Use the Up/Down buttons to select a row for deletion. The arrowhead at the side of the grid indicates the selected row.

Click OK to apply the filter immediately to the selected library. If desired, save the filter string for future use. (The SB200 provides for interaction between the library dialog and

filter form. This allows complex strings to be built line-by-line and tested without saving until the string is completed.)

When saving the filter choose a descriptive name for the file that reflects the purpose of the filter. The SB200 automatically names the files for the material type.

Click the Clear Filter button to display all materials of the selected type.

Creating and editing stackups

The SB200a allows the designer to add or edit stackup layers in any order, from top to bottom, bottom to top or from the centre layer outwards. In this example we create a four-layer stackup, starting at the centre core layer and adding layers above and below.

Adding layers to the stackup

Display materials library

Items added to the stackup are added from the currently open materials library. The SB200a opens *Program Files\Polar\SB200\default.mlb* if it exists; if a different library is required, open it via the Materials Library command.

Note: the SB200 does not ship with the default.mlb library. For this discussion open one of the two sample library files, *SB200 inch.mlb* or *SB200 mm.mlb* (stored in the *Program Files\Polar\SB200\Samples* folder at installation time for a default installation.)

Caution: Consistency of units

When defining dimensions for a stackup (for example, layer thicknesses) ensure that all measurements are defined using the same units (mils, mm, etc.) throughout the structure and its libraries.

Note: the libraries supplied for these examples are preloaded with sample data only.

Click the File|New command to clear the stackup screen and notes and information text areas.

Click the File|Save Stackup or Save Project command to save the stackup or project. Users are recommended to save stackups or projects frequently during the stackup creation process to avoid data loss. Stackup files (.stk), project files (.sci) and library files (.mlb) should be backed up to a secure location.

Adding a core layer

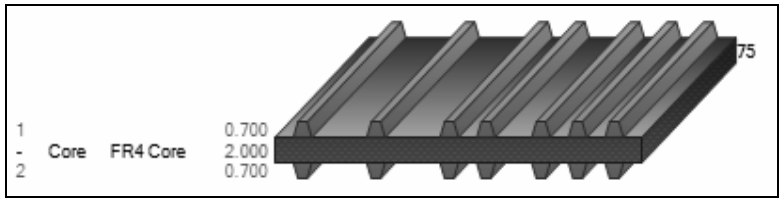
Click the **Add Layer Material** button and choose Core…the Core library is displayed

The Core library contains full details of the core material, including base and finished thicknesses, dielectric constant, and upper and lower copper thicknesses.

Base Thickness	Finished Thickness	Dielectric Constant	Upper Cu Thickness	Lower Cu Thickness
0.05	0.05	4.2	0.018	0.018
0.05	0.05	4.2	0.035	0.035
0.05	0.05	4.2	0.07	0.07

Click on the column button to sort the library list by the selected column. Choose a core type from the list of cores and click the Add Above button

Add core above selected layer. The core is added to the stackup screen.



Stackup core layer

Layers may also be added below the selected layer.

Add core below selected layer

As each layer is added the stackup information table is updated to reflect the current status of the stackup.

Stack Up Information

Field	Value
Electrical Layer Count	2
Stack Up Thickness	1.0700
Dielectric Thickness	1.0000
Copper Thickness	0.0700

Stackup information table

With the core selected, the Selected Item table displays the properties of the core.

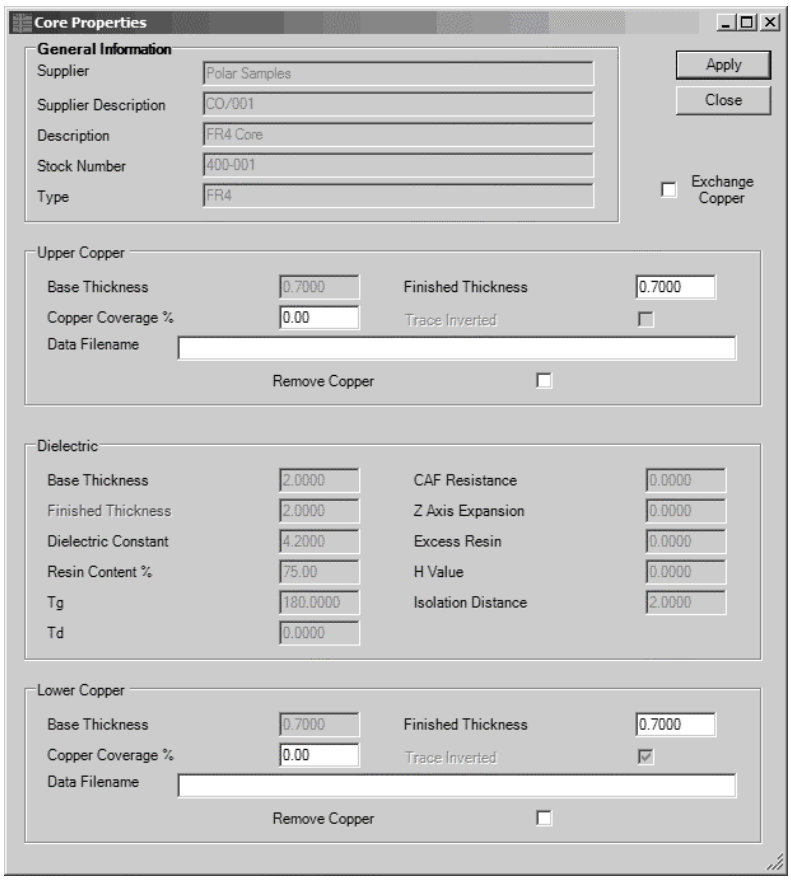
Selected Item Information : Core

Field	Value
Supplier	SuperLaminates Inc
Supplier Description	CO/026
Description	FR4 Core
Stock Number	400-026
Upper Cu Base Thickness	0.035
Upper Cu Finished Thickness	0.035
Data Filenames	
Dielectric Base Thickness	1
Dielectric Finished Thickness	1
Dielectric Constant	4.2
Lower Cu Base Thickness	0.035
Lower Cu Finished Thickness	0.035
Data Filenames	

Core layer information

Editing the selected layer properties

To change the properties of the selected object (for example, to modify the value for the finished thickness of the dielectric), right click the object in the stackup and choose Properties from the shortcut menu; in this example the Core Properties dialog is displayed.



Change the Dielectric finished thickness to the corrected value and click Apply.

Adding data file names

If available, add the data file name(s) to the upper and lower copper layers and click Apply.

Close the dialog when all changes are completed.

Changes will be reflected in the Stackup Information table

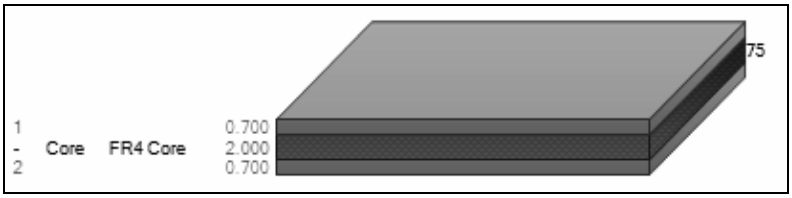
Changing a layer function

In this example we change both signal layers above and below the core dielectric to planes.

Set Layer Plane

Click the lower signal layer and click the Set Layer Plane button. Repeat for the upper signal layer.

The changes are reflected in the stackup window



Exchanging layers

Library Swap button

To change just the core dielectric (leaving the copper layers unaffected), select the core material (for example the FR4 in the graphic above) and click the Swap Selected Material button. Choose the new core type from the library and click the Swap button.

The layer properties will change to reflect the new material and changes appear in the Stackup Information table.

Adding Prepreg layers

Add Layer Material button

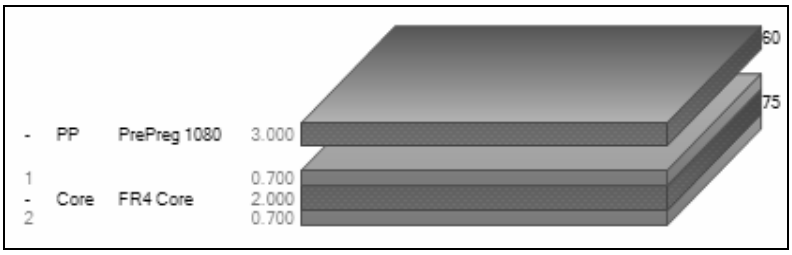
With the core selected, click the **Add Layer Material** button and choose Prepreg…; the Add Prepreg library is displayed.

Description	Stock Number	Base Thickness	Finished Thickness	Dielectric Constant
PrePreg 1080	300-001	0.075	0.075	4.2
PrePreg 3080	300-002	0.077	0.077	4.2
PrePreg 3113	300-003	0.102	0.102	4.2
PrePreg 1651	300-004	0.152	0.152	4.2
PrePreg 7628	300-005	0.2	0.2	4.2

The Prepreg library contains details of the Prepreg material, including the Prepreg's base and finished thickness and dielectric constant.

Add Prepreg Above

Choose the Prepreg material from the database and click the Add Prepreg Above button.



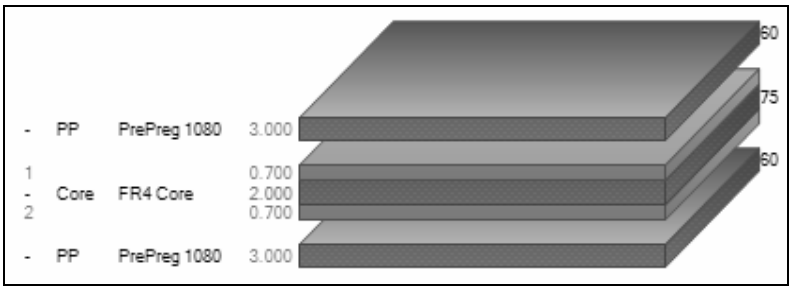
The Prepreg layer is added above the core.

To change the properties of the Prepreg material right- click the layer and choose Properties from the short cut menu. For example, the value for Finished Thickness can be modified to reflect the effects of the pressing process.

Add Prepreg Below

Click the Add Prepreg button to display the Add Prepreg library and click the Add Below button

The layer of Prepreg is added below the core.



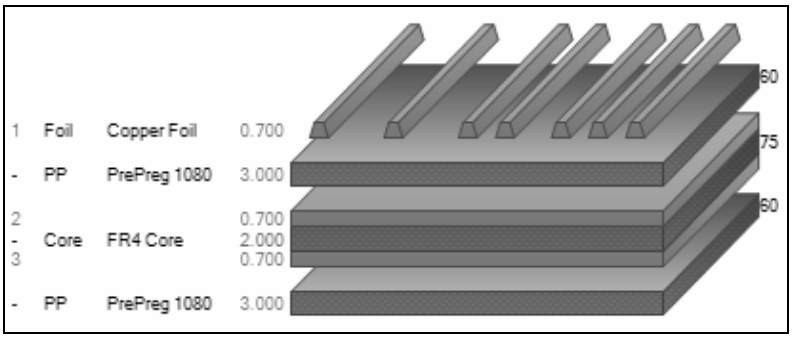
Modify the properties as necessary.

Adding a layer of copper foil

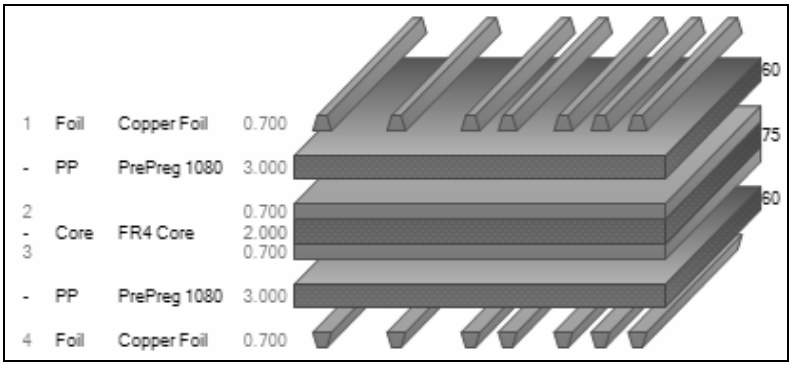
Select the upper layer of Prepreg and click the **Add Layer Material** button and choose Foil to display the copper foil library.

	Supplier	Supplier Description	Description	Stock Number	Cu Thickness
▶	SuperLaminates Inc	FO/001	Copper Foil	100-001	0.018
	SuperLaminates Inc	FO/002	Copper Foil	100-002	0.035
	SuperLaminates Inc	FO/003	Copper Foil	100-003	0.07

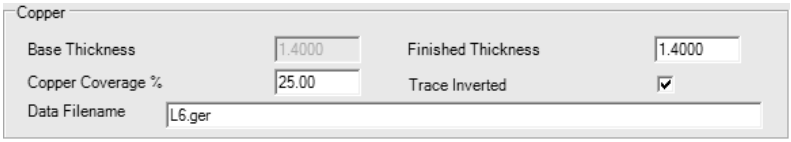
Choose the foil type and click Add Above, the copper foil layer is added above the selected Prepreg layer.



Repeat the procedure for the lower Prepreg layer: select the lower Prepreg layer and add a layer of copper foil below the layer.



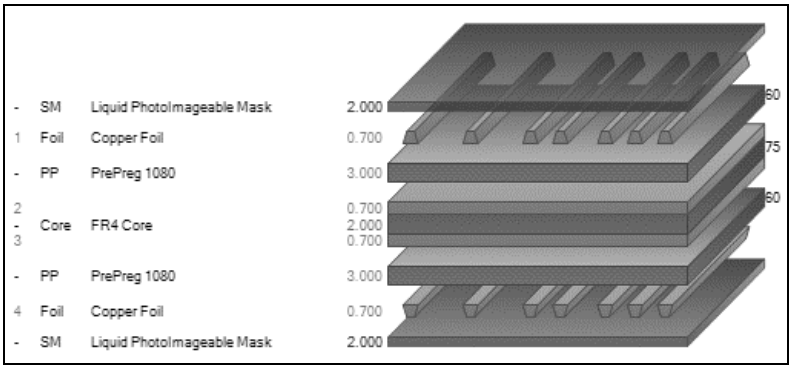
To alter the foil properties, right-click the foil layer and choose Properties. Using the Properties dialog the user can, for example, specify that the trace is shown inverted.



Note the stackup is built symmetrically about the centre layer.

Adding solder mask layers

With the upper layer of foil selected, click the **Add Layer Material** button and choose Soldermask to add a layer of LPI solder mask above the foil. Repeat the process for the lower foil layer.

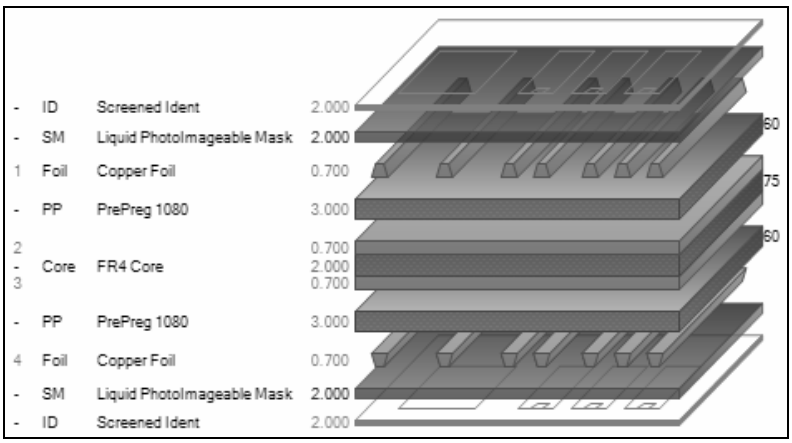


Adding the Ident layers

Select the lower LPI Soldermask layer and click the **Add Layer Material** button and choose Ident to add a layer of Screened Ident below the layer. The sample Ident library includes ink thickness and colour

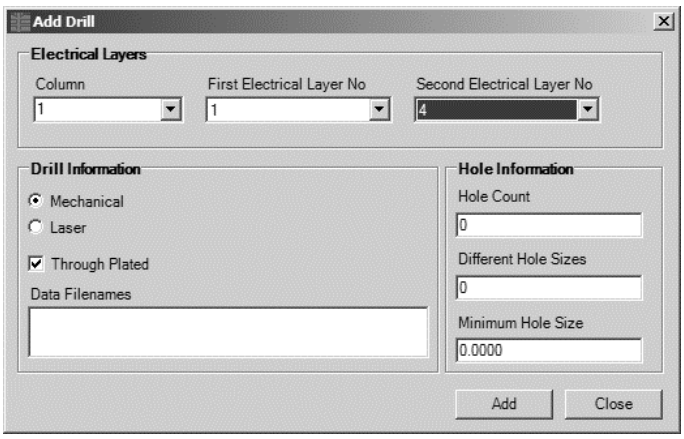
	Supplier	Supplier Description	Description	Stock Number	Ink Thickness	Ink Colour
▶	SuperInks Inc	ID/001	Screened Ident	600-001	0.05	White
	SuperInks Inc	ID/002	Screened Ident	600-002	0.05	Yellow
	SuperInks Inc	ID/003	Screened Ident	600-003	0.05	Black

Repeat for the upper layer.



Adding drill information

To add drill information between layers click the Add Drill button; the Add Drill dialog is displayed.



Select the column in which to place the drill.

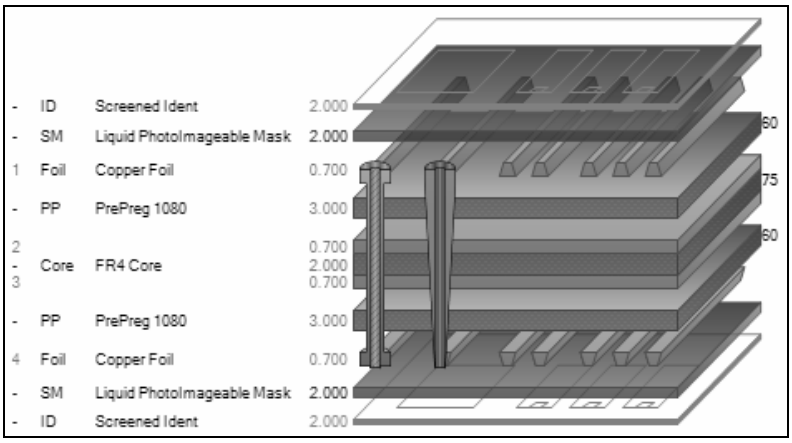
Choose the first and second electrical layer numbers (layers 1 and 4 in the example).

Specify the drill type, mechanical or laser and whether through plated.

Optionally, add the NC drill data filenames.

Optionally, add the hole count, number of different hole sizes and the minimum hole size. Click Add and close the dialog. The drill information is added to the stackup. In the example we have added through plated and laser drill information.

This completes the stackup; the finished stackup is shown below



Completed stackup

Deleting a layer

Delete layer

To remove a layer from the stackup select the layer and click the Delete button.

Copying layers

Copy Material

With layers defined it will often be found more convenient to copy an existing layer and paste it into the stackup than to create a new layer “from scratch”. Select the layer to be copied and click the Copy Material button.

Click the layer nearest the destination location and choose Paste Above or Paste Below as appropriate

Note: when modifying the stackup it may be necessary to redefine the drill information to reflect the changes.

Moving layers

Move selected layer up one layer

To move layers within the stackup use the Move Up and Move down buttons.

Move selected layer down one layer

When a layer is moved it is exchanged with the layer above or below, respectively.

Applying finished thicknesses

Apply Finished Thickness

To apply the finished thickness factor throughout the board, click this button with no layers selected, otherwise the finishing will only be applied to the selected layer

Reset Finished Thickness

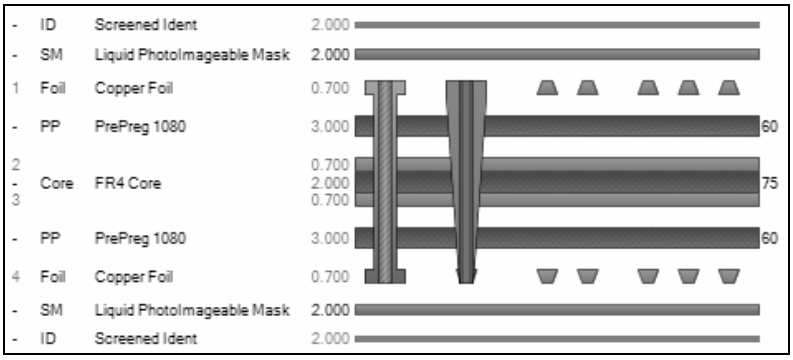
To reset the finished thickness back to the original base thickness of the materials throughout the board, click this button with no layers selected, otherwise the reset will only be applied to the selected layer

Displaying the stackup in 2-dimensional view

View 2D button

To change the view of the stackup from its default 3-dimensional aspect, click the View 2D button.

The stackup is displayed in 2-dimensional view.



Display 3D button

Click the View 3D button to restore the 3 dimensional view.

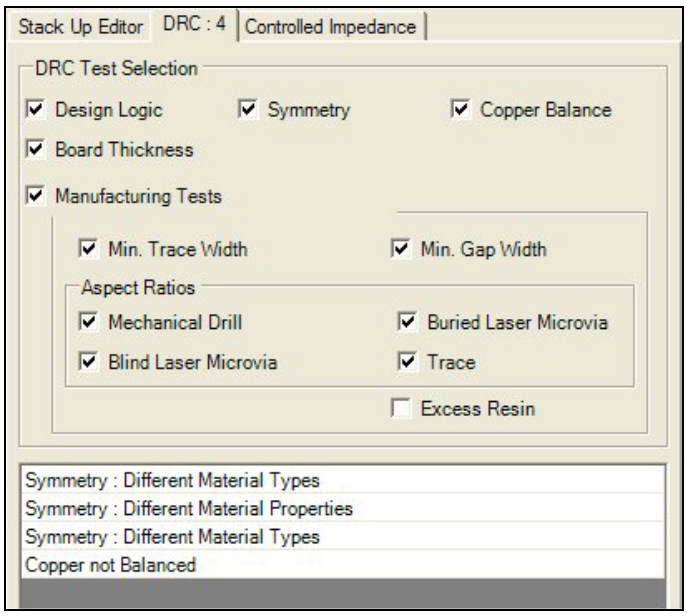
Design Rule Checking

The SB200a includes facilities to check for errors in stackup design, such as layers placed in invalid order or asymmetrical structures.

The Design Rule Checker (DRC) displays results in the DRC dialog. As each design rule is broken the SB200a increments the error count on the DRC tab.

Viewing design rule errors

Click the DRC tab to view errors.



The Design Rule Checker checks include checking for:

Two Adjacent Copper Layers

Resin Coated Copper on Internal Layer

External Prepreg

Internal Solder Mask

Internal Ident

Internal Peelable Mask

Symmetry – Different Material Types

Copper not Balanced

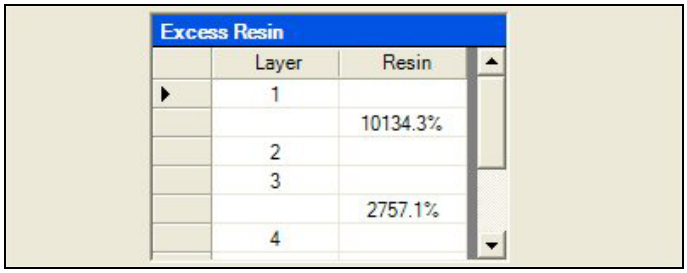
Board Thickness (If the board is outside tolerance the Stack Information in the Stack editor is displayed in red)

Manufacturing tests

- Minimum Trace width (Test carried out when calculating Controlled Impedance)
- Minimum Trace Separation (Test carried out when calculating Controlled Impedance)

- Drill Aspect Ratios for Plated holes
- Track Aspect ratio
- Excess Resin test

If the Excess Resin check box is ticked values are shown as below; scroll through the layers as required



Users can choose to display all errors or to select from a combination of Design Errors, Symmetry errors and Copper balance errors, etc.; check the boxes as required.

Click on the errors shown in the list to highlight the errors in the stackup screen. Errors are highlighted in red.

Correcting design rule errors

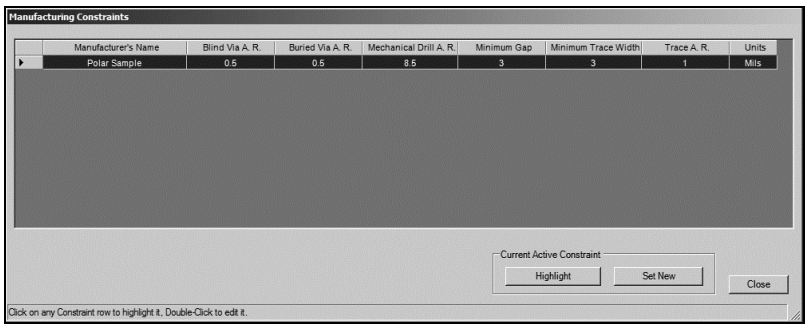
Users are strongly recommended to work through and correct errors in the order in which the errors are listed. Note that clearing each error may clear other errors in the process.

Manufacturing tests should be fixed before sending the PCB for manufacture. Hole sizes should be adjusted to comply. Failures with track and gap should be corrected, possibly by changing prepreg thickness and/or dielectric constants.

A collection of manufacturing constraints can be defined and the required one selected.

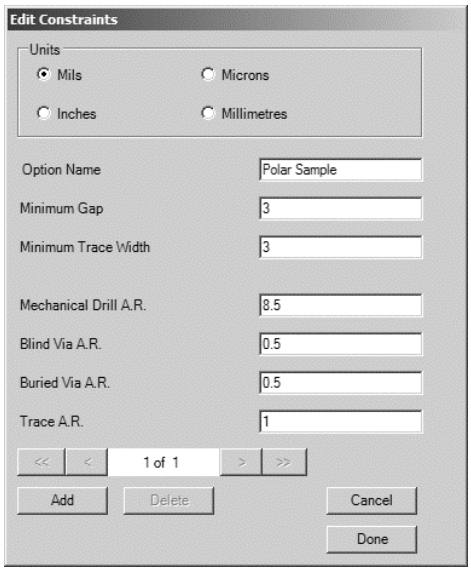
Creating and using manufacturing constraints

From the Tools menu, select Manufacturing Constraints: the Manufacturing Constraints window opens, displaying any manufacturing constraints added.



By default there will always be at least one. It is important to always have one constraint set active.

Double-click on a constraint row will bring up the Edit Constraints dialog; use the dialog to add, delete or edit constraints.



To Edit a constraint set, use the navigation buttons to select the set to modified, change the values as required and then press **Done**.

To Delete a constraint set, use the navigation buttons to select the set, then press **Delete**.

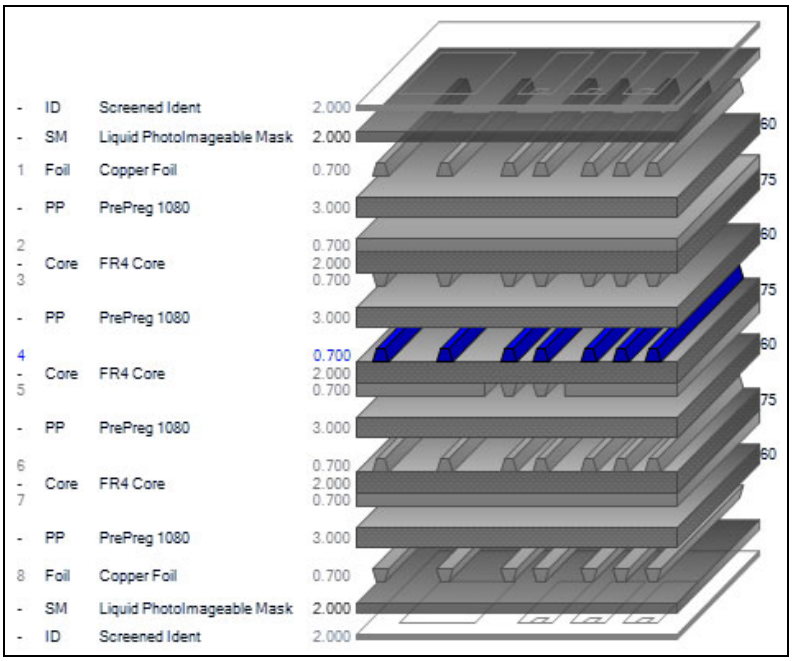
To Add a new constraint set, press the **Add** button, this will add a new (empty) constraint row, enter the name and constraint values and press **Done**.

Adding controlled impedance structures

The SB200 incorporates the facility to add controlled impedance structures to a layer in the stackup. The SB200 is integrated with the Polar Instruments Si8000/9000 Controlled Impedance Field Solver so impedance values for a structure may be calculated at the click of a button.

Structure parameters may be copied to the Field Solver for processing (for example by the Si8000/9000 Goal Seeking function) and calculated values pasted back to the SB200 for insertion into the stackup.

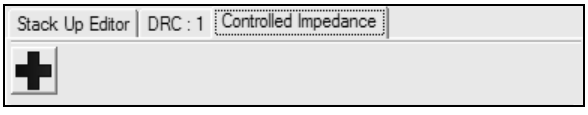
In this example we add a controlled impedance structure to signal layer 4 of the sample stackup below.



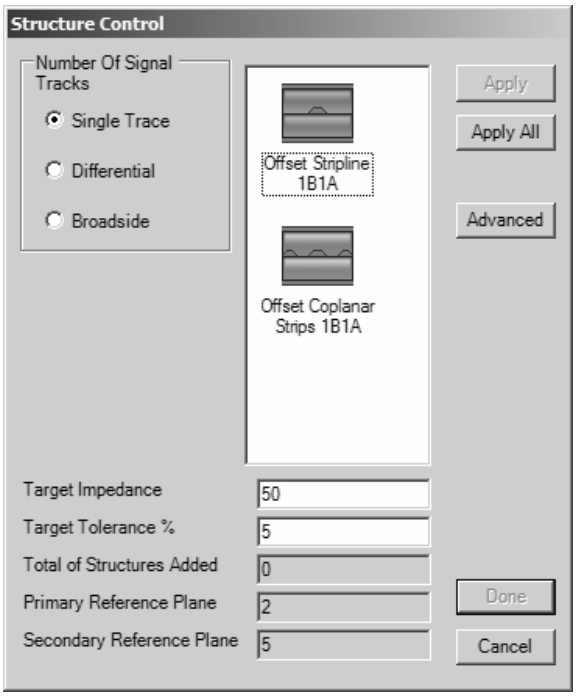
Sample stackup (showing signal layer 4 selected)

Note that in this example layer 5 is a mixed signal/plane layer. Potential reference planes for signal layer 4 are therefore plane layer 2, mixed signal/plane layer 5 and plane layer 7.

With layer 4 selected, click the Controlled Impedance tab. The **Add Structure** button is displayed.



Click the **Add Structure** button; the Structure Control dialog is displayed containing the controlled impedance structures applicable to the selected layer in the stack. Choose values for the target impedance and tolerance. If necessary, resize the Structure Control dialog to view all structures.



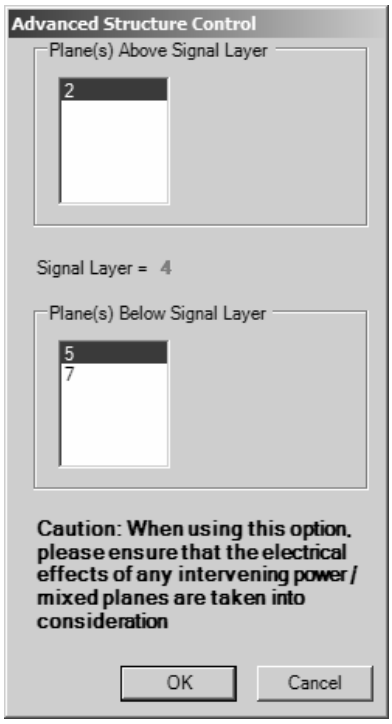
Click the Single Trace, Differential or Broadside option button as appropriate (in this case, we choose Single Trace|Offset Stripline 1B1A).

Note: Broadside only appears as an option where the signal trace is between two reference planes. When selecting Broadside structures it will be necessary to choose the second trace.

Specify the values for Target Impedance and Tolerance.

Choosing reference planes

As there are multiple reference planes available (layers 2, 5 and 7, it will be necessary to specify which planes to use for this structure. Click the **Advanced** button.

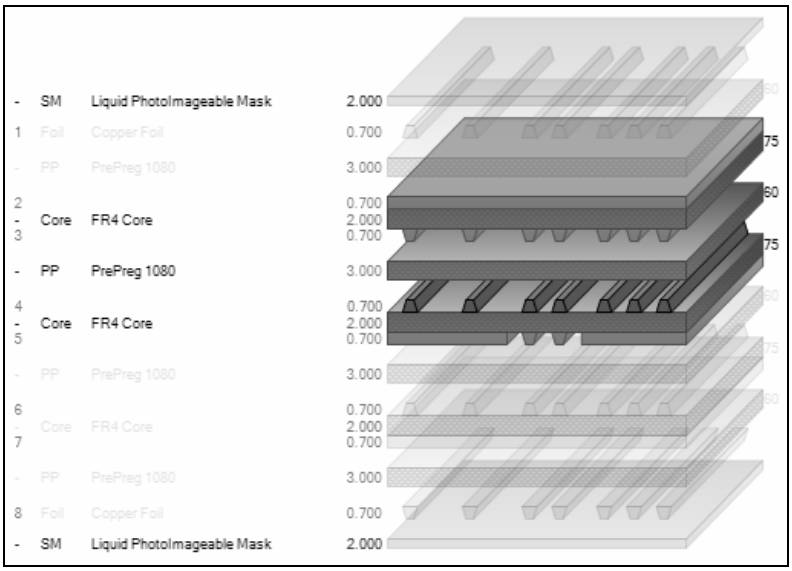


Choose a reference plane from the list of available planes. In the example structure plane layer 2, mixed plane 5 and plane layer 7 are available for reference.

Note: if plane layer 7 is chosen as reference, it will be necessary to take into account the electrical effects of mixed signal/layer plane 5.

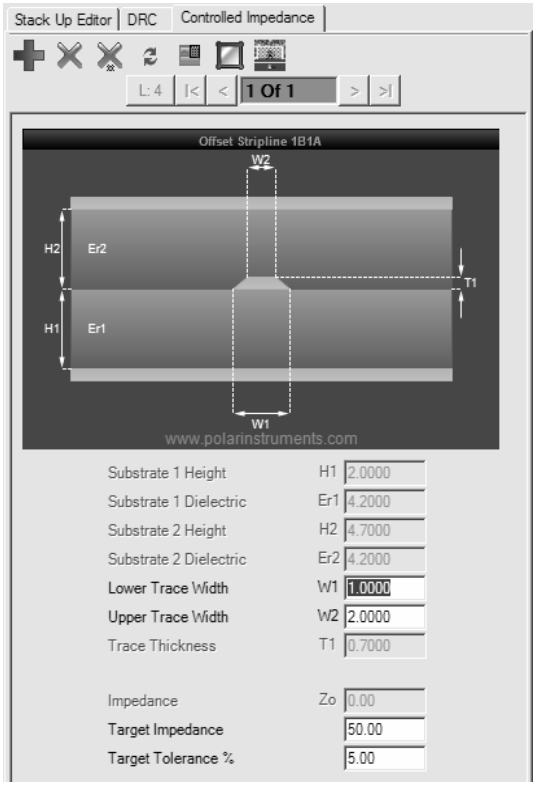
In this example we choose mixed signal/plane layer 5. Press **OK** to confirm.

Repeat for all structures to be added. Click **Apply** for each structure then click **Done** to finish. In this example we choose a single structure.



The stackup window changes to reflect the selected signal layer and its associated reference planes.

The applied structure is displayed in the Controlled Impedance pane.



The window displays the parameters of the controlled impedance structure. Fields shown "greyed out" are values derived from the choice of materials in the stackup. For this structure, enter appropriate values for lower and upper trace widths.

Calculate button

Click the **Calculate** button to display the impedance value of the structure with the current parameters. The parameters may then be varied to alter the value of the final impedance.

In the example above the user can vary the trace width in order to approach the value of the target impedance; other parameters are changed by modifying the stackup dimensions (for example, core thickness H1).

Clicking **Apply All** in the Structure Control dialog adds a single instance of all structures matching the stackup layer and the chosen criteria; the user can then choose the structure producing the value nearest the target impedance and delete the structures that are not needed. Controlled impedance operations are performed via the Controlled Impedance toolbar.

Controlled Impedance toolbar

 Add controlled impedance structure to current layer

 Delete structure from current layer

 Clear all structures from current layer

 Refresh and calculate impedance

 Calculate structure

 Mirror Structures

 Goal Seek

Display layer and Navigate through structures

Changing parameter values

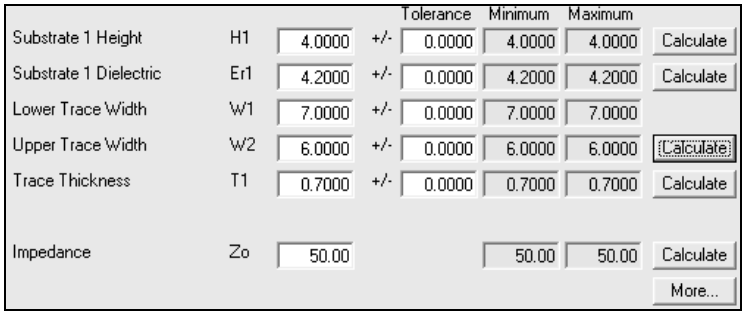
Clicking the Calculate function yields a value for impedance. The user can now vary parameters (for example, the dielectric height) to yield a value for impedance closer to the target impedance. For this example, select the dielectric layer; click the Swap Selected Material button and choose a different core (ensure the same dimensional units are used throughout the structure) and click the Refresh and Calculate Impedance button. The impedance is recalculated to its new value. To arrive at a solution acceptably close to the target impedance, use the goal seeking function of the Si8000 to alter other parameters (in this case, upper and lower trace widths).

Goal seeking with the Si8000/9000

The SB200 Stackup Design System is fully integrated with the Si8000/9000 Controlled Impedance Field Solvers. The user is able to transfer stackup layer dimensions to the Field Solver, solve for stackup parameters to produce the target impedance (or calculate that the target impedance is unachievable with the current values) then transfer the solved dimensions back to the SB200.

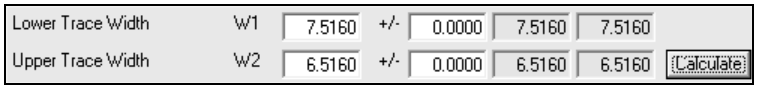
Ensure the Field Solver is running and that its units match the SB200 units. With the stackup parameters displayed in the Controlled Impedance window, click the Copy to Field Solver button to transfer the current SB200 parameters to the Field Solver. Switch to the Field Solver and click the Paste from SB200 button to load the parameters into the associated Field Solver fields.

For the example data below we seek a final value for impedance of 50 Ohms; H1, Er1 and T1 are fixed.



Field Solver parameter fields with SB200 data loaded

Click the Upper Trace Width Calculate button to goal seek on trace width. The Field Solver returns values for trace width to produce 50 Ohms final impedance.



Solved values for impedance

Click the Copy to SB200 button, switch to the SB200 and click the Paste from Field Solver button to display the solved parameters for the target impedance. It may be necessary to round some dimensions (e.g. dielectric heights) to the nearest practical values and recalculate the impedance.

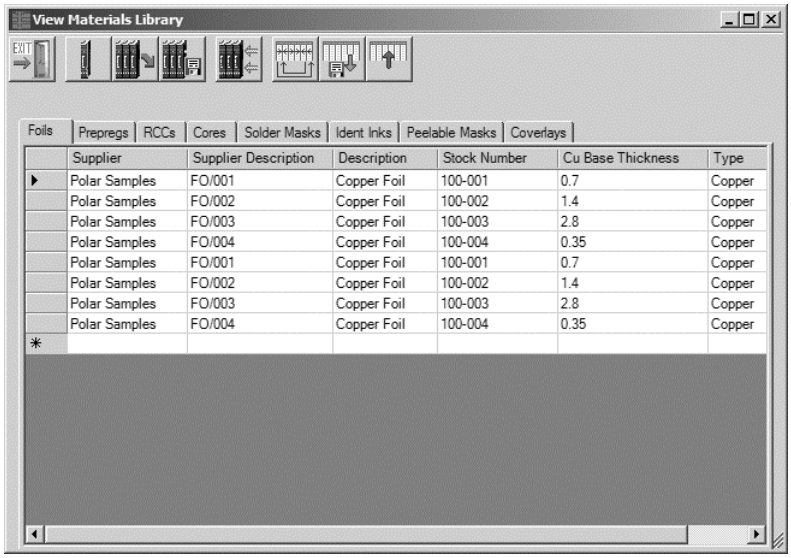
Using the SB200 materials libraries

Materials Library

The materials libraries allow designers to manage their own libraries of board materials.

Click the Materials Library button to display the Materials Library window.

When the SB200 is started the material library specified as the default materials library file (via Configuration Options|File Locations) is opened.



Each library component type is accessible via its associated tab. Click on the Tab to view or edit the component type.

Materials library toolbar

The Materials Library is managed via its toolbar

 Clear Materials Library

 Open Materials Library

 Save Materials Library

 Import Material into the Library

 Select & arrange Column fields

 Save column arrangement

 Load column arrangement

Creating a new library

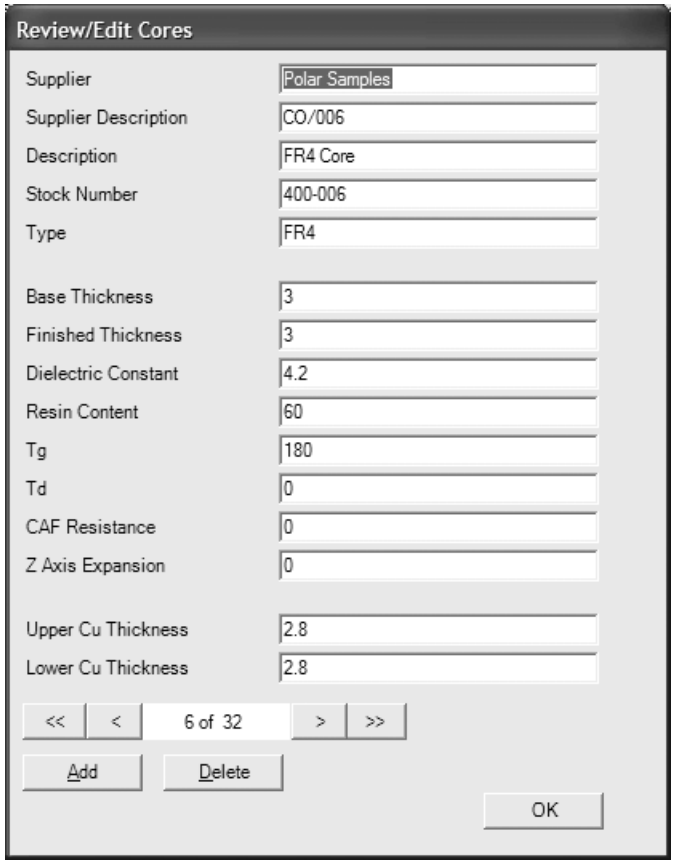
To create a new library, click the Clear Materials Library button; the library is removed from the library manager. Click the Save Materials Library button to create the new library. To have the library load as the SB200 starts, specify it as the default materials library file via Configuration Options|File Locations.

Adding material to the library

Caution: consistency of units

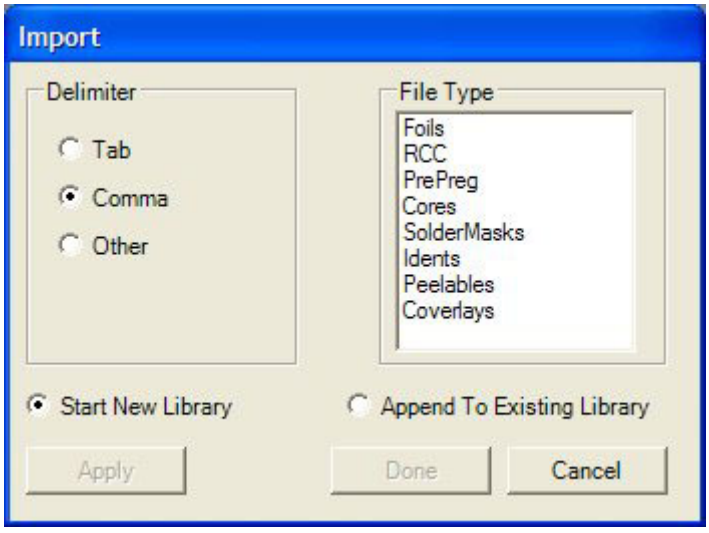
When defining dimensions, e.g. layer thicknesses, for a stackup ensure that all measurements are defined using the same units (mils, mm, etc) throughout the structure and its libraries.

Open the library to be modified. To add materials to a library click the associated component type tab; click onto a material, or empty line. An editing box will open which will contain the material clicked on, or the last material in that type library. The material can be edited or deleted, or a new material can be added. To speed up the process of adding families of materials, when a material is added the properties of the last material are copied to the new material. The details can then be edited. Clicking OK will add any new materials to the end of the list.



Importing material to the library

The SB200 allows users to add existing material lists to its library; material data must be arranged in the format and order used by the SB200 library. Sample files in comma separated value format and Microsoft Excel spreadsheet and template formats suitable for importing to the SB200 are included in the SB200|Material Library Import folder. Click the Materials Library button to open the Library, and then click the Import button to open the Import dialog.



Library Import dialog

Creating a new materials library

Choose the Start New Library option and choose the field delimiter type. The library import function can accept files in a variety of formats, tab delimited, comma separated and Excel worksheet and template formats. Sections of the sample files included with the SB200 are shown below.

	1	2	3	4	5
1	* Cores				
2	*				
3	*Type	Supplier	Supplier Description	Description	Stock Number
4	FR4	Polar Samples	CO/001	FR4 Core	400-001
5	FR4	Polar Samples	CO/002	FR4 Core	400-002

Sample library file in Microsoft Excel format

```
* Cores,,,,,,,,,,
*,,,,,,,Dielectric,Dielectric,Dielectric,Dielectric,Dielectric
*Type,Supplier,Supplier Description,Description,Stock Number,Upp
Thickness,Dielectric Constant,Resin Content,Tg
FR4,Polar Samples,CO/001,FR4 Core,400-001,0.7,0.7,2,2,4.2,75,180
FR4,Polar Samples,CO/002,FR4 Core,400-002,1.4,1.4,2,2,4.2,75,180
FR4,Polar Samples,CO/003,FR4 Core,400-003,2.8,2.8,2,2,4.2,75,180
FR4,Polar Samples,CO/004,FR4 Core,400-004,0.7,0.7,3,3,4.2,60,180
FR4,Polar Samples,CO/005,FR4 Core,400-005,1.4,1.4,3,3,4.2,60,180
```

Sample library file in comma separated format

Files for importing into the library must be in the above format, with columns in the correct order. Examine the sample files supplied in the Material Library Import subfolder for more information. (It may be helpful to export data files from the current system into Microsoft Excel and save in Microsoft Excel workbook or text file format.)

Specify the delimiter if necessary and choose the file type (Foil, RCC, Prepreg, etc.) and click Apply. Choose the file from the list displayed in the Open dialog and click Open. Repeat the procedure for every file type; Click Done when all file types have been imported. Save the file as an .mlbx library file.

Adding material data to an existing library

To add material data to an existing library, open the library and click the Import Material button

Selecting Materials from the Library

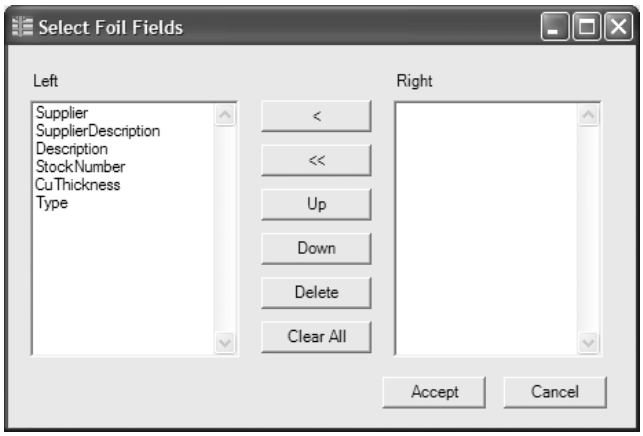
Column Order (Materials Library)

The default setting displays all columns. The columns displayed and the order they are displayed can be set in the materials Library form.

Arranging Columns in Library Forms

The Library windows can be customised in respect of which columns to display and in which order.

Click the Go to Materials Library button and select Arrange Columns; the dialog associated with the selected material tab (Foils, Prepregs, etc.) is opened.



The Left box of the dialog shows the columns that will be displayed and the order top to bottom is the order they will be displayed left to right in the library window.

Click Accept to return to the Materials Library, which will show the columns as set.

Until the column order is saved the column order is only available during the current session. Click Save Column Order to define the selected column order as the default order whenever the program is run.