PAIZI

SailWind Land Pattern Creator Guide and Reference

Document Revision : 1.1

派兹

Copyright and Disclaimer of SailWind Software

Copyright Information

All copyrights, patent rights, trademark rights, trade secrets, and other related intellectual property rights of the SailWind software (hereinafter referred to as the "Software"), including but not limited to its source code, object code, user interface design, graphics, images, audio, video, algorithms, data models, documentation, etc., belong to Chengdu Paizi Interconnect Electronics Technology Co. Ltd. (hereinafter referred to as the "Copyright Owner").

Installation and Use License

Users should clearly agree to all terms of this copyright and disclaimer before installing and using this software. By running this installation or software, the user indicates that they have read and agree to be bound by this copyright and disclaimer.

The copyright owner grants users a non exclusive, limited, and revocable installation license, allowing them to install the software on their designated computer devices and use its related features to complete their design tasks under the guidance of the software.

Users are not allowed to copy, distribute, modify, sell, rent, lend, transfer, reverse engineer, decompile, create derivative works, or otherwise use this software in any form, except with the explicit written permission of the copyright owner.

Disclaimer During Installation and Use

This software is provided as is, and the copyright owner does not guarantee that it is error free, defect free, and does not guarantee that the installation and use process will be successfully completed, nor does it make any commitment to the applicability, stability, security, or reliability of the installation and use process.

Users should bear the risk of using this software themselves. The copyright owner shall not be liable for any direct or indirect losses, data loss, business interruption, system damage, or other damages caused by the use or inability to use this software.

Limitations and Reservations of Rights

The use of this software is subject to the limitations and constraints of this copyright and disclaimer. The copyright owner reserves all rights not explicitly granted to users. Users are not allowed to perform any form of reverse engineering, decompilation, disassembly, decryption, modification, creation of derivative works, or use to create similar software on this software.

Other

The copyright owner has the right to modify the terms of this copyright and disclaimer at any time, and the modified terms will be notified to users through appropriate means. If the user continues to use this installation and software, it means that they have accepted the modified terms.

If any part of this copyright and disclaimer is deemed invalid or unenforceable for any reason, that part shall be deemed separate from the whole, but shall not affect the validity of other parts.

Based on the permanently authorized PADS® software of Siemens Industry Software Inc.

Contact Information

If you have any questions or suggestions about this installation or software, please contact:

Email: market@pzeda.com
Phone: 0755-86703052
Website: www.pzeda.com

Revision History

Revision	Changes	Date
1.0	Initial release	2024-04-25
1.1	Updated the CAD tools supported by SailWind Land Pattern Creator.	2024-06-25

Table of Contents

Chapter 1
Getting Started Guide

The SailWind Land Pattern Creator uses the concept of PLB files to store component dimensions and land pattern construction rules. Understanding the structure of these files and how they are applied during the land pattern creation process is necessary to achieving the desired output. This section will describe the process of creating and using the PLB files.

PLB File Explanation

A PLB file stores all component dimensions and IPC-7351B rules. The software comes with 17 PLB files for you to use as a starter library. There are over 10,000 different component packages stored in the PLB files.

The SMT, Through-hole parts and Connectors are intentionally kept in separate PLB files for faster search results, faster creation of each CAD library and to allow you to easily update rules for each component family and regenerate an entire new library. However, your personal user PLB file can contain all 3 component groups in a single file. The only time it's necessary to have more than one PLB file is if you work with different environment levels, then it's best to keep all the "Least" environment parts in a separate PLB file. The primary reason for PLB separation of environments is for making rule changes for a specific environment and mass creating a new library. The Multi-Part Wizard will take a PLB file through a single existing Preference rule file to create a CAD library and a new PLB file. So if you mix Least, Nominal and Most environments in a single PLB file and use Multi-Part Wizard to regenerate your CAD library, all of the PLB data will pass through a single set of rules and only output a single environment.

The following four PLB files do not have any component manufacturer's data. Their intended use is for the Multi-Part feature to mass create an entire CAD library.

- BGA7351B — contains all "Grid Array" parts for BGA, LGA and CGA component families
- SML7351B — Surface Mount Least environment
- SMN7351B — Surface Mount Nominal environment
- SMM7351B — Surface Mount Most environment

Since the PCBM-STARTER PLB file has component manufacturer part numbers it contains multiple manufacturer's data for the same land pattern. In some cases, a single land pattern will be listed 4 – 10 times with different manufacturer's part numbers and datasheet web-links. The primary purpose of the PCBM-STARTER PLB file is to search for manufacturer's data to quickly locate an existing land pattern to save time.

- PCBM-STARTER — contains all the BGA7351B and SMN7351B parts and includes manufacturer's data.

Levels A – C are IPC producibility levels for fabrication. Level P (Proportional) uses all 3 IPC levels starting with small holes Level C and medium hole sizes Level B and larger hole sizes use Level A and very large hole sizes have robust annular rings.

- CNA7251 — Connectors Level A (Most) environment
- CNB7251 — Connectors Level B (Nominal) environment
- CNC7251 — Connectors Level C (Least) environment
- CNP7251 — Connectors Level P (Proportional) environment
- THA7251 — Through-hole parts Level A (Most) environment
- THB7251 — Through-hole parts Level B (Nominal) environment
- THC7251 — Through-hole parts Level C (Least) environment
- THP7251 — Through-hole parts Level P (Proportional) environment

The following three PLB files contain non-standard parts that were manually created in a CAD tool using the component manufacturer's recommended land pattern. Since these parts were not created using SailWind Land Pattern Creator, they are unable to be viewed in the software. You can use them to recreate CAD library parts and view them in your CAD tool.

- CONNECTORS
- SM_MFR
- TH

The Wizard PLB file is primarily used to practice building library parts. It does not contain any parts, but it comes as the default PLB file prior to the creation of your personal PLB file that will replace the Wizard PLB file.

- Wizard

Tutorial

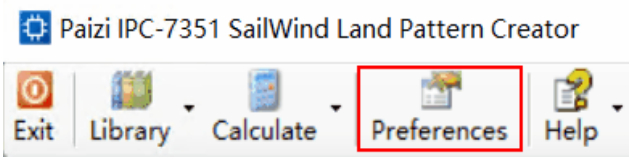
This section describes the steps required to configure and use the SailWind Land Pattern Creator. It explains how to set up your rule file, add personal attributes, create your PLB file, load it into the Library Manager, select your CAD tool, and finally, how to use the SailWind Land Pattern Calculator and Wizard.

Creating Your Personal Preference XML Rule File

It is important to create your corporate preference XML file to store all of your unique changes in the Preferences menu.

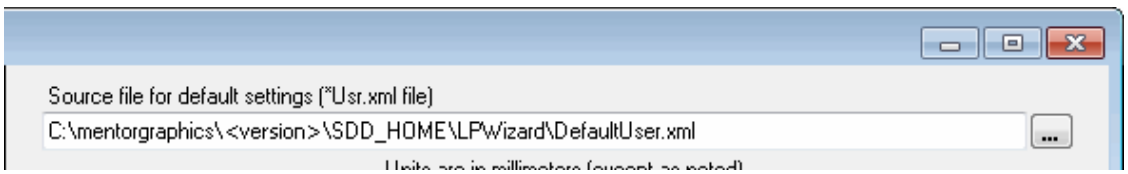
Procedure

1. To create your personal XML preference file, click the “Preferences” button in the main toolbar.

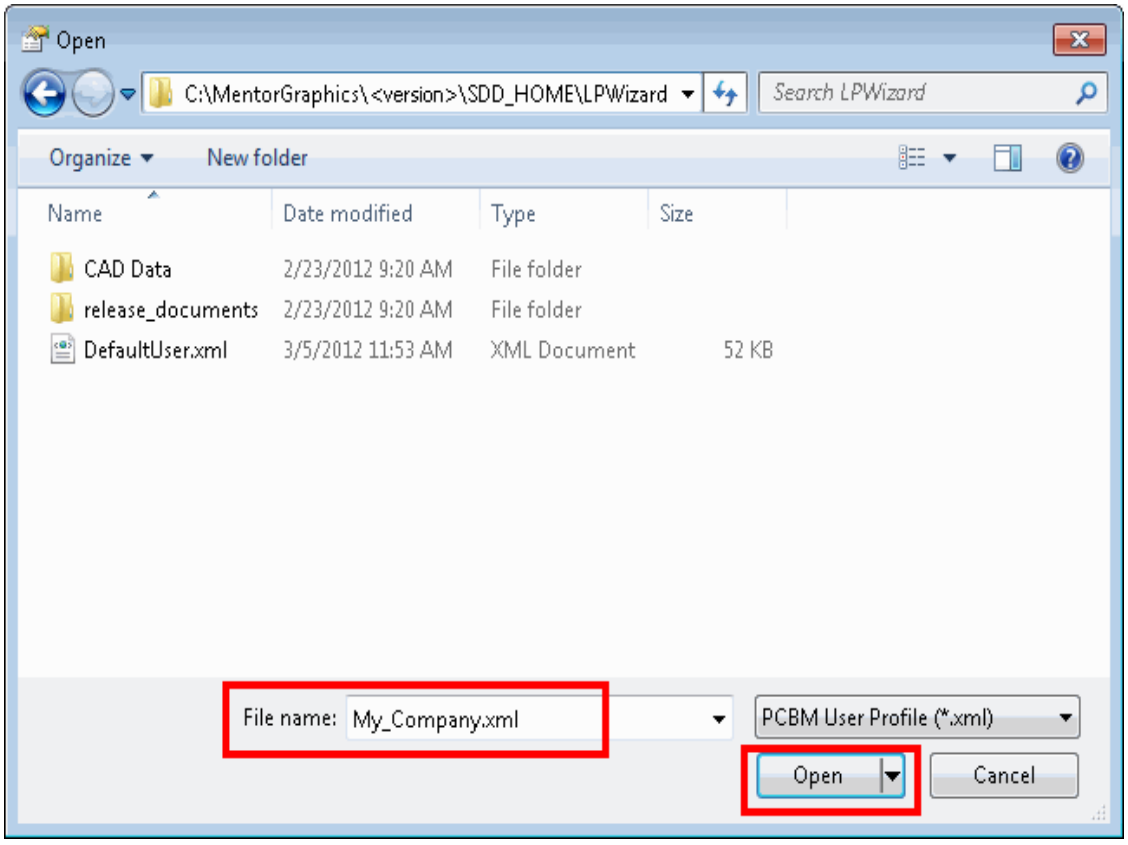


Note:
Pictures in this document are for reference only. In the case of interface difference due to such factors as version changes, the interface of SailWind Land Pattern Creator in practice shall prevail.

2. The default XML file is called – DefaultUser.xml. To make a copy of it, select the button on the right side.



3. The following dialog box appears and you need to change DefaultUser.xml to your company name or user name .xml.



4. Select the “Open” button and then click “Yes” in the pop-up dialog box to create your personal xml preference file.

Note:
You can, at any time, recreate DefaultUser.xml by deleting it in the main installation folder and when you run the software, it will automatically regenerate a brand new DefaultUser.xml file that matches the original default settings.

Adding Personal Attributes to Your PLB File

PLB Attributes are used for searching the library to quickly locate the correct land pattern. An example of a personal attribute would be “Corporate Part Number”. Most large companies assign a part number for every new component that is used in a PCB layout. When you add the Corporate Part Number in your PLB file, it becomes a powerful search feature for quickly locating the correct land pattern.

Procedure

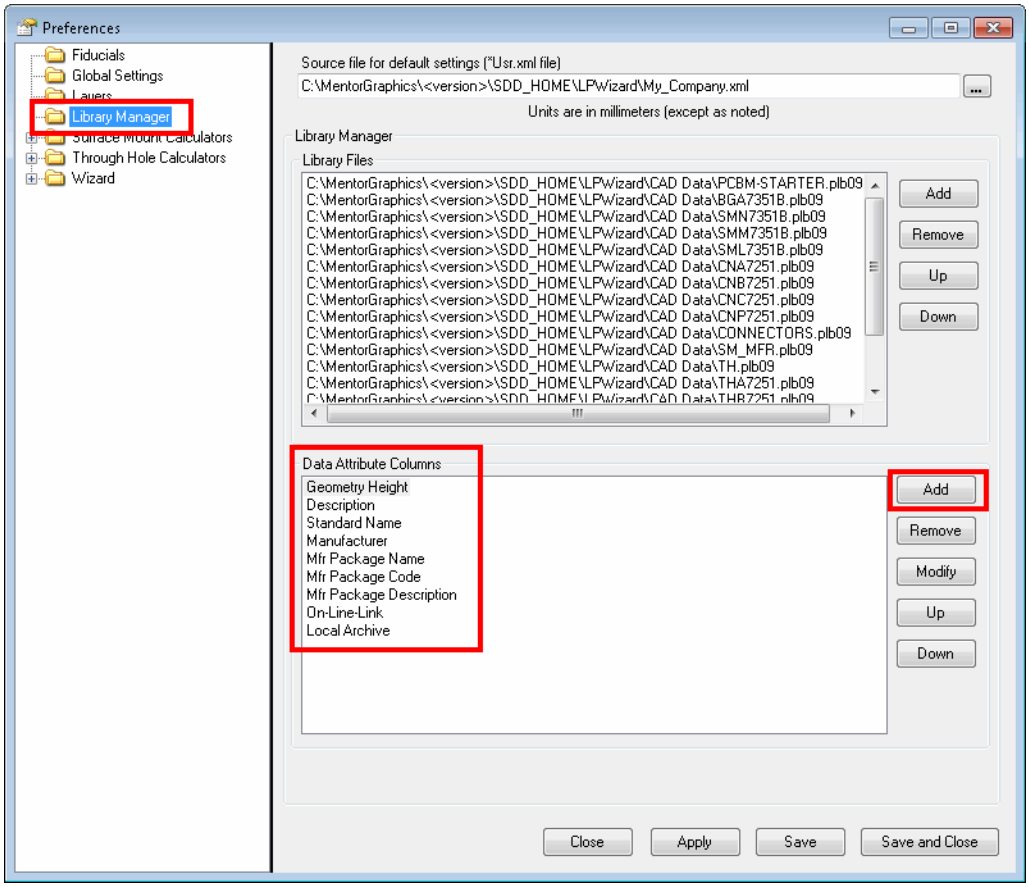
1. Next select the “Library Manager” folder in the upper left corner of the Preferences dialog box to view the Data Attribute Columns. The existing attributes are used in all of the PLB files that come with the software. You can remove any of these attributes except Geometry Height and Description. The “Geometry Height” attribute is used by the software to auto-generate the IDF 3D model data for every CAD library part. The “Description” attribute information is auto-generated by the component family and physical package dimensions.

If you remove or rename any existing attributes, because they appear in the SailWind Land Pattern Creator PLB files, when you transfer parts from the SailWind Land Pattern Creator PLB files to your PLB file, the existing attributes will transfer over into your PLB file.

Here is an explanation of the existing attributes:

- Geometry Height — used to auto-generate the IDF 3D model data
- Description — auto-generated by the component family and physical package dimensions
- Standard Name — used for JEDEC or industry standard part number
- Manufacturer — component manufacturer
- Mfr Package Name — the package name that appears on the component dimensional datasheet
- Mfr Package Code — the package code that appears on the component dimensional datasheet
- Mfr Package Description — the package description that appears on the component dimensional datasheet
- On-Line-Link — open datasheet on component manufacturer website
- Local Archive — open PDF file datasheet stored on your network

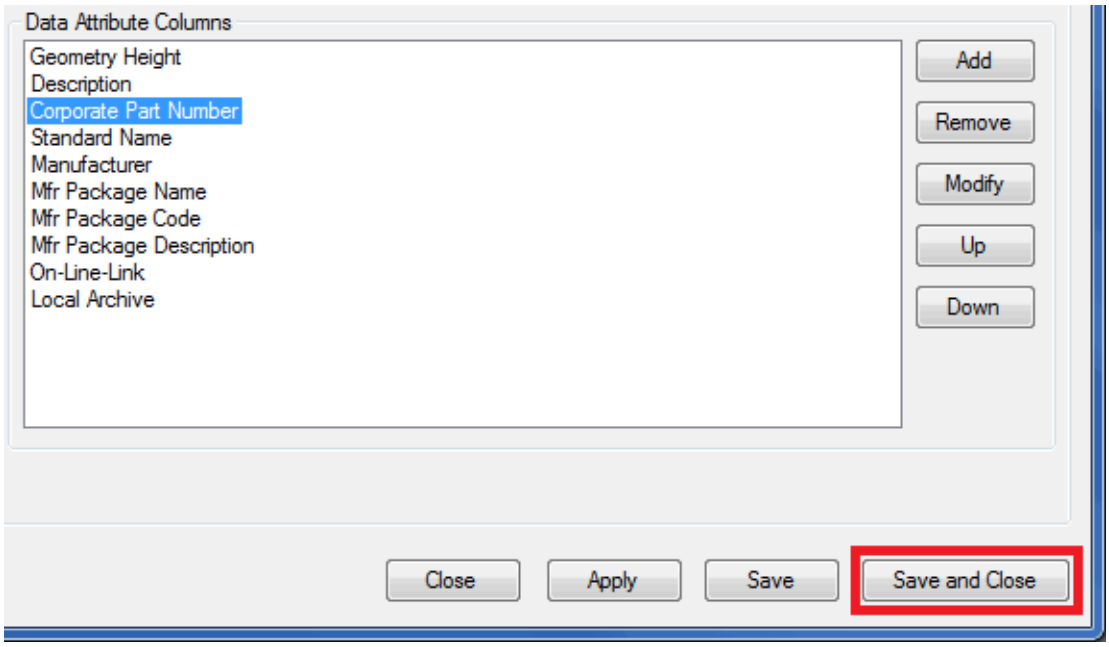
2. In the Library Manager preferences select the “Add” button in the “Data Attribute Column” section.



3. In this exercise you are going to add a new attribute called “Corporate Part Number”. After you select the “Add” button, a new attribute box will appear. Type the text – “Corporate Part Number” and then type “Enter” on your keyboard.

4. Next select the “Up” button to relocate the Corporate Part Number attribute to appear after the “Description” attribute.

5. Next select the “Save and Close” button in the lower right. This will close the Preferences window and save your PLB file.

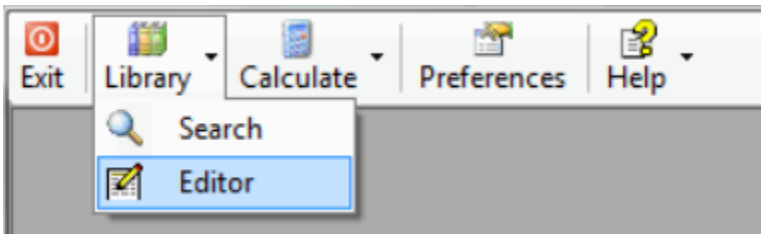


Creating Your Personal PLB File

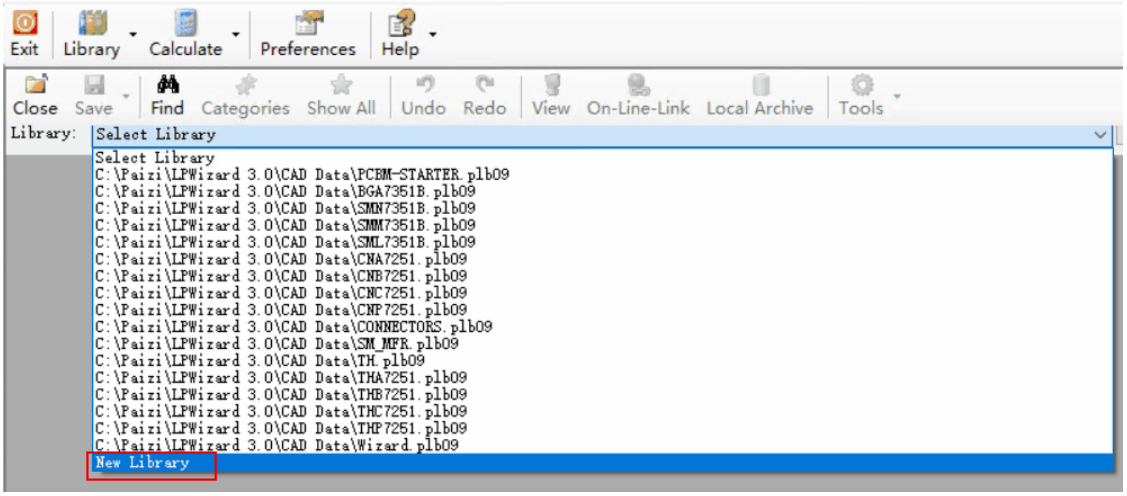
You can create your own .plb file to store all of your attribute preferences. Once created, this file is available for recall during future editing sessions. It can also be shared with other designers across your network.

Procedure

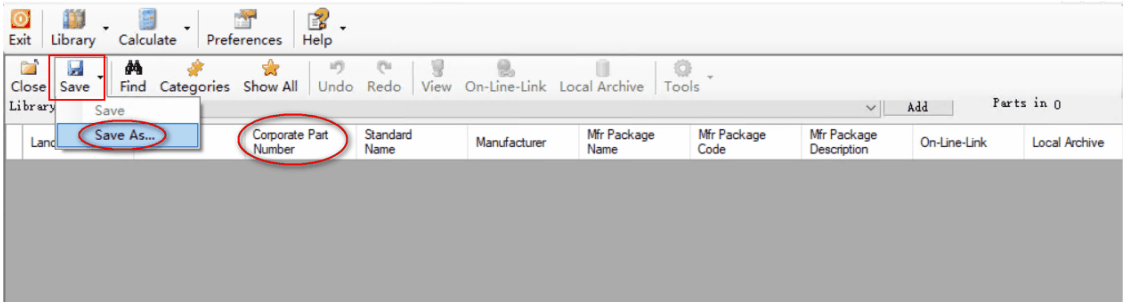
1. Select the drop-down button “Library” and click “Editor”.



2. In the Library Editor, select the “Library:” drop-down and select the last option “New Library”.

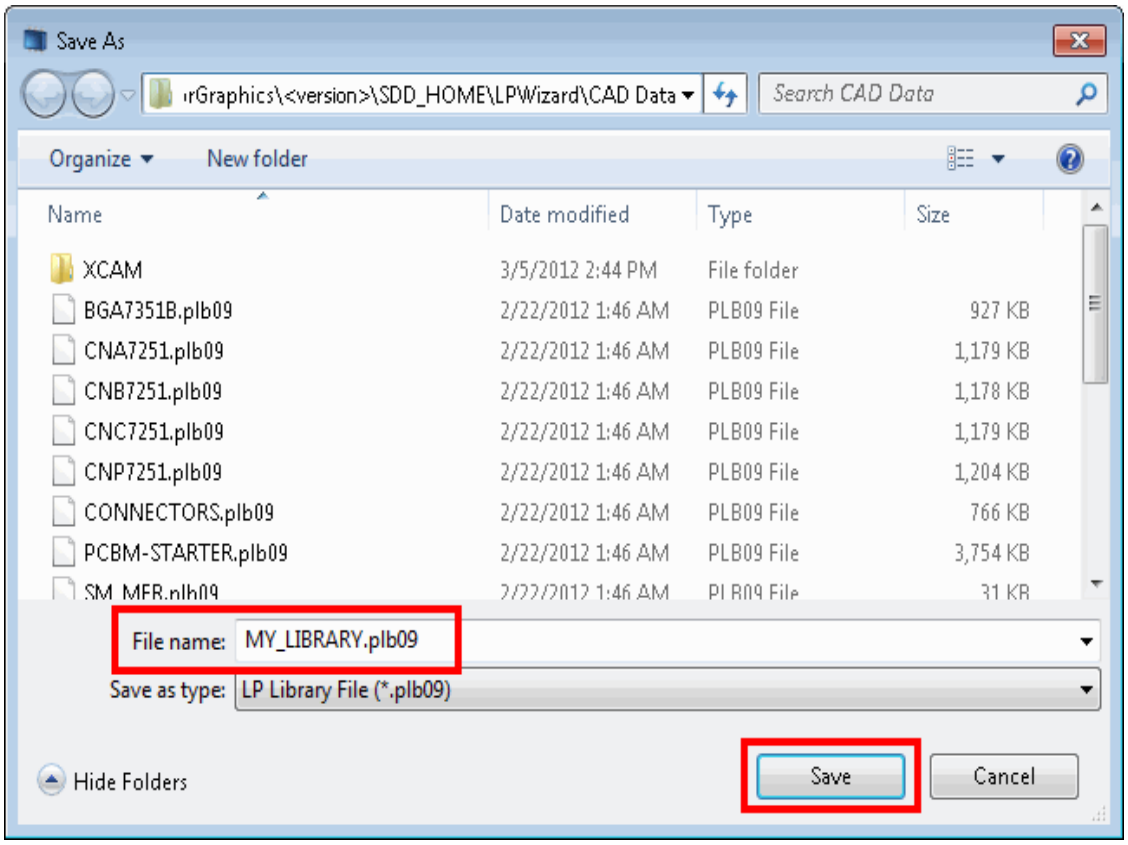


3. You have just created your personal PLB file with your personal attributes. Notice the “Corporate Part Number” is in the attribute header between “Description” and “Standard Name”. Now select the drop-down “Save” and click “Save As…”



4. Name your new personal PLB file after your company name. You can also save your PLB file in a folder on your network to allow other people to import it into the free Viewer or other SailWind Land

Pattern Creator users to access it to also add additional library parts to it. Click the “Save” button when you’re finished.

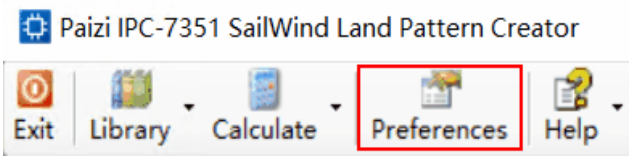


Loading Your Personal PLB File into the Library Manager

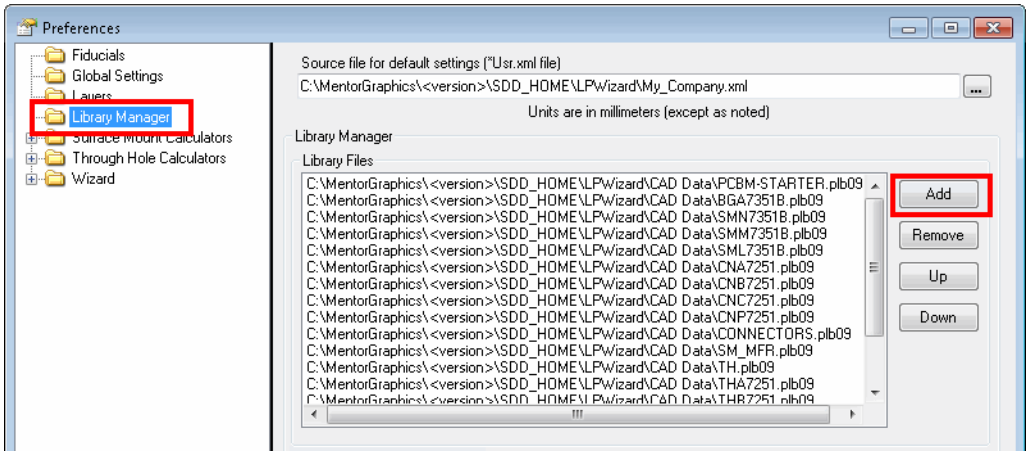
Your personal.plb file contains all of the attribute preferences that you have specified for your use in constructing land patterns. Once created, you must add it to the Library Manager so that it can be used during editing sessions. You can also use the Library Manager to remove unnecessary or obsolete .plb files.

Procedure

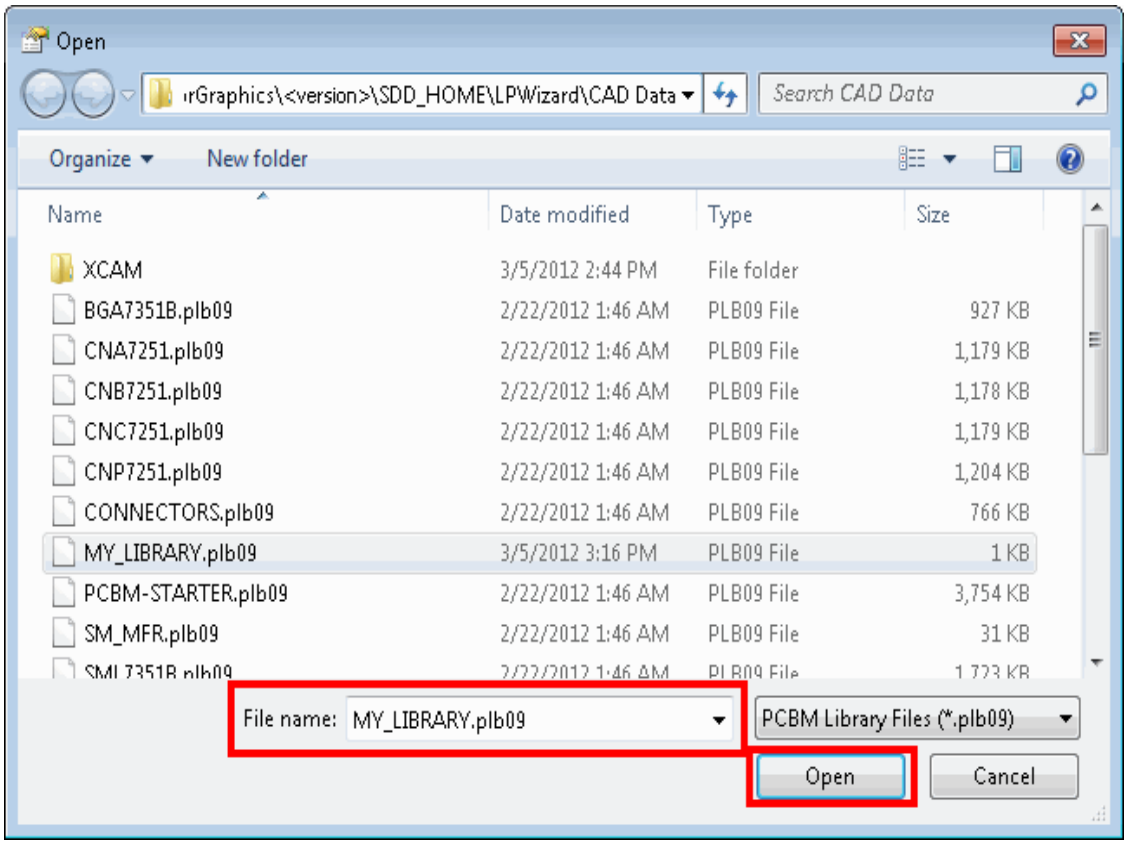
1. After you create your personal PLB file you need to load it in the Library Manager preferences. Select the “Preferences” button in the main toolbar.



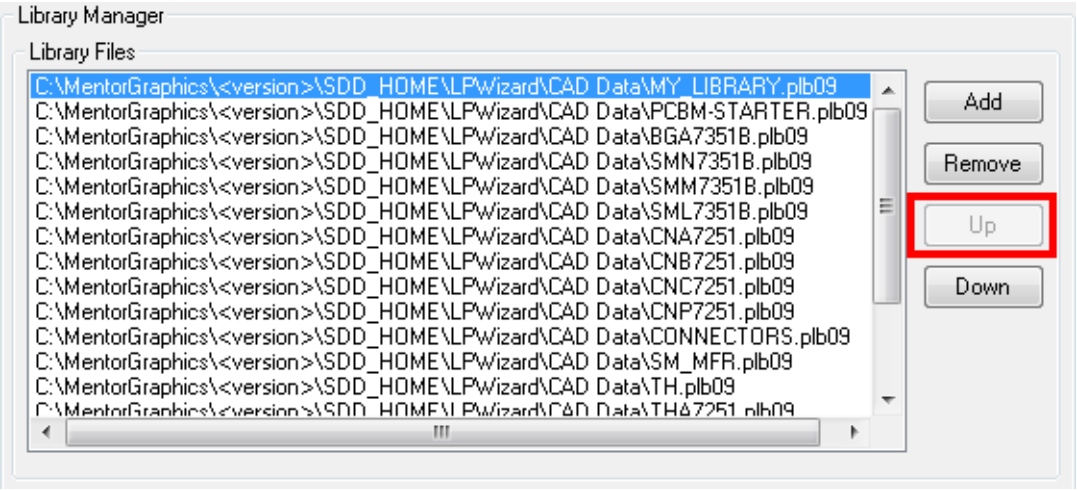
2. Then in the upper left, select the “Library Manager” folder, and in the Library Files section click the “Add” button.



3. When you click the **Add** button, the following dialog box appears to allow you to search for the location on your network where you placed your personal PLB file. After you select your personal PLB file, click “Open” in the lower right corner and your PLB file will be added to the bottom of the list.



4. Select your PLB file and use the “Up” button on the right side of the menu to relocate your personal PLB file to the top of the list.



5. While you're in the library manager, now is a good time to remove all the PLB files that you do not have any use for. For example, if you use the "Nominal" environment for your CAD library creation, you can remove the Least and Most PLB files from the list. You have to remove one PLB file at a time. When you remove PLB files from the list, they are still on your computer to reload any time you need them. Don't forget that the SMN7351B and the BGA7351B are merged together in the PCBM-STARTER PLB file.

   I would recommend that you remove these PLB files from your Library Manager:

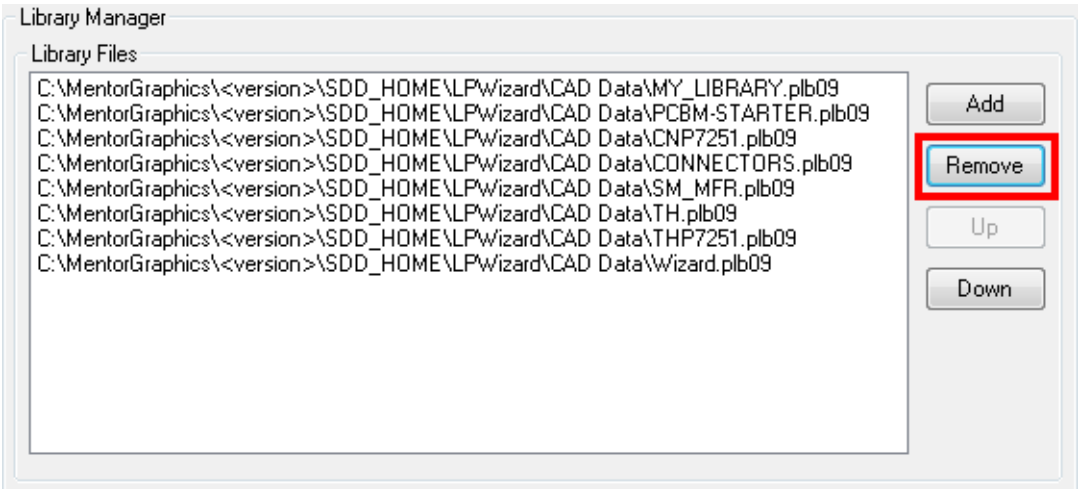
   - **BGA731B** — BGA's
   - **SML7351B** — Surface Mount Least environment
   - **SMN7351B** — Surface Mount Nominal environment
   - **SMM7351B** — Surface Mount Most environment
   - **CNA7251** — Connectors Level A (Most) environment
   - **CNB7251** — Connectors Level B (Nominal) environment
   - **CNC7251** — Connectors Level C (Least) environment
   - **THA7251** — Through-hole parts Level A (Most) environment
   - **THB7251** — Through-hole parts Level B (Nominal) environment
   - **THC7251** — Through-hole parts Level C (Least) environment

6. These are the PLB files that should be left in the Library Manager:

   - **MY_LIBRARY** — whatever name your personal PLB file is
   - **PCBM-STARTER** — BGA7351B and SMN7351B combined

- **CNP7251** — Connectors Level P (Proportional) environment
- **THP7251** — Through-hole parts Level P (Proportional) environment
- **CONNECTORS** — 5,000 manually built connectors
- **SM_MFR** — Unique manually built surface mount parts
- **TH** — Unique manually built Through-hole parts
- **Wizard** — Empty PLB file for testing purposes

This is what the Library Manager menu will look like when you are completed:

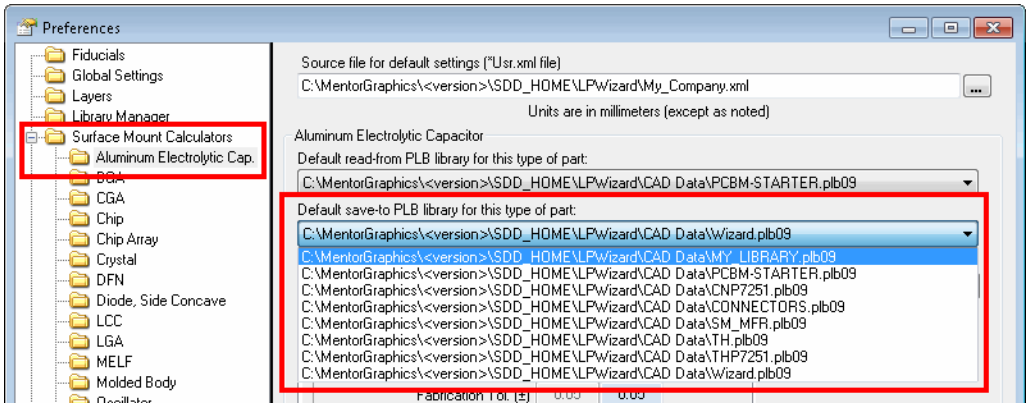


Loading Your Personal PLB File into the Default “Save-to” PLB Library

Load your personal PLB file into every component family template so that every time you build a CAD library part, the component dimensions and the rules will be saved to your personal PLB file.

Procedure

1. Expand the “Surface Mount Calculators” folder and then select the “Aluminum Electrolytic Capacitor” folder. Then select the drop-down for “Default save-to PLB library for this type of part:” and then search and select your personal PLB file.



2. A pop-up dialog box appears and asks you, “Would you like to make this the default library for all component families?” Click the “Yes” button.

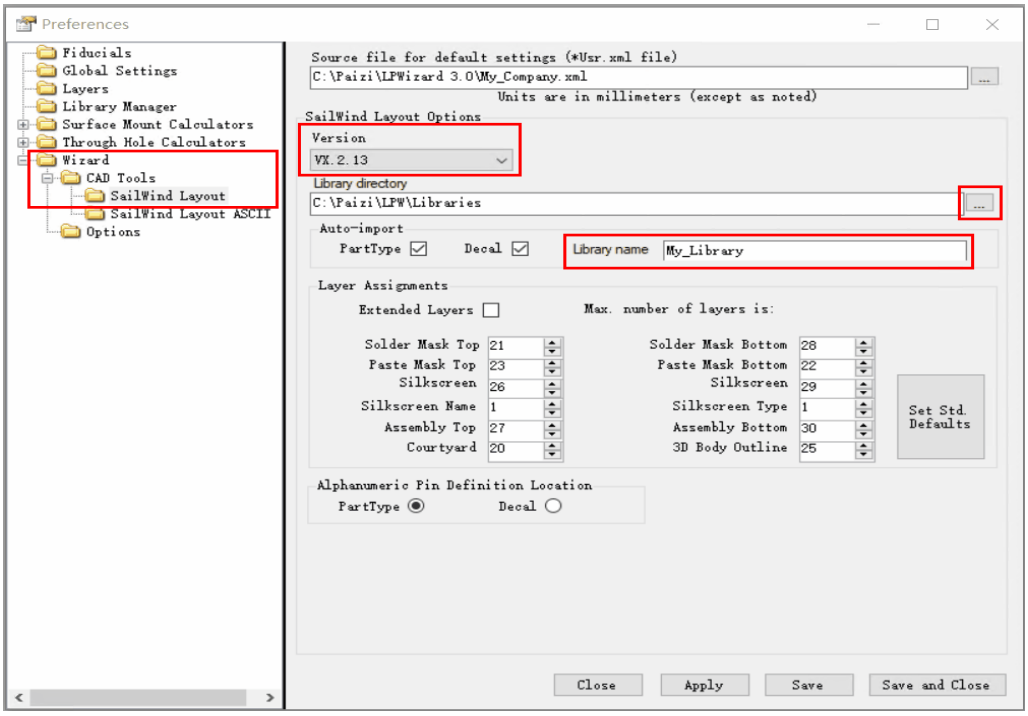
3. Now your personal PLB file is automatically loaded in every component family as the main “Save To” PLB file. You can verify this by selecting other component families to see if your personal PLB file is loaded.

Selecting Your Default CAD Tool

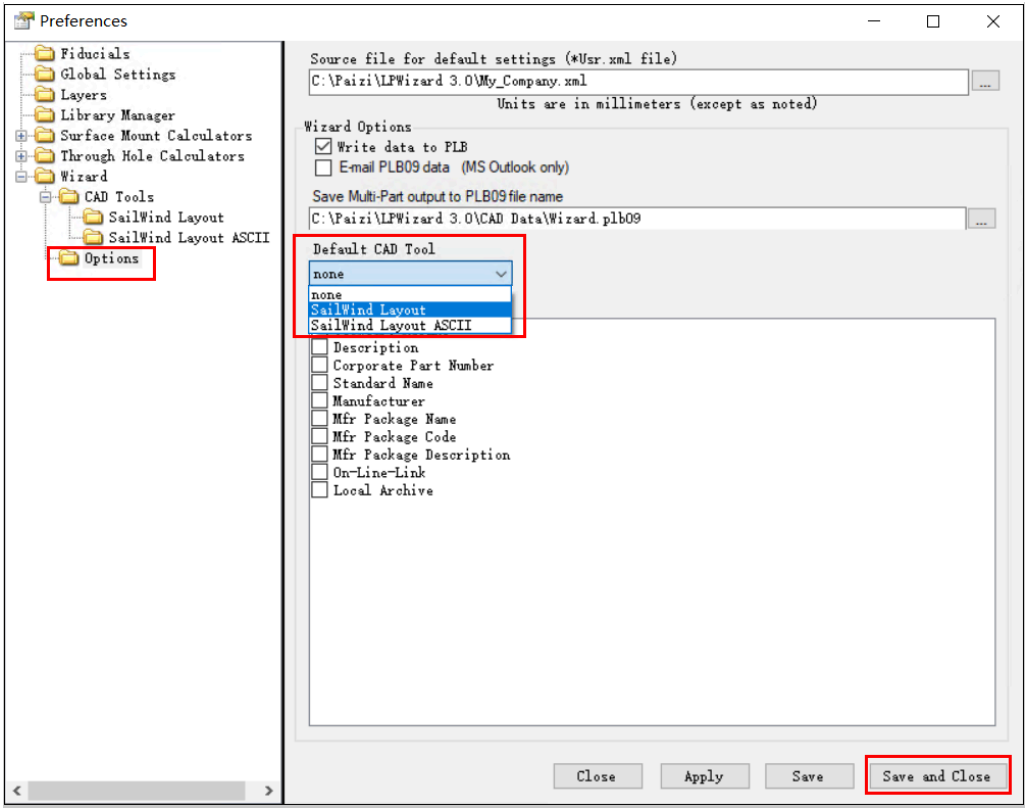
The final step in setting up your Preferences is selecting the CAD tool that you want to export to and the directory path to where you want to save your library data. If you export to multiple CAD tools, you will need to define all the folders that you want to save data to.

Procedure

1. Expand the “Wizard” folder, select the “CAD Tools” folder and then select your CAD tool. For this example I selected SailWind Layout. Select the “Library directory” browse button and search for the folder where your library is located. For the SailWind tool, you also need to set the “Version” in the drop-down and enter your CAD library name because the library data is moved from SailWind Land Pattern Creator directly into your SailWind library.



2. Next select the “Options” folder and select the “Default CAD Tool” drop down and pick your CAD tool. Next, select the “Save and Close” button in the lower right corner. You are now ready to start building CAD library parts in the land pattern calculator.

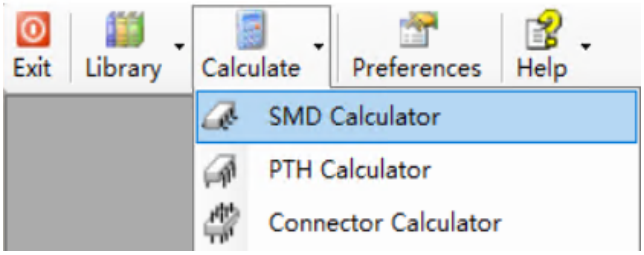


Using the Calculator and Wizard

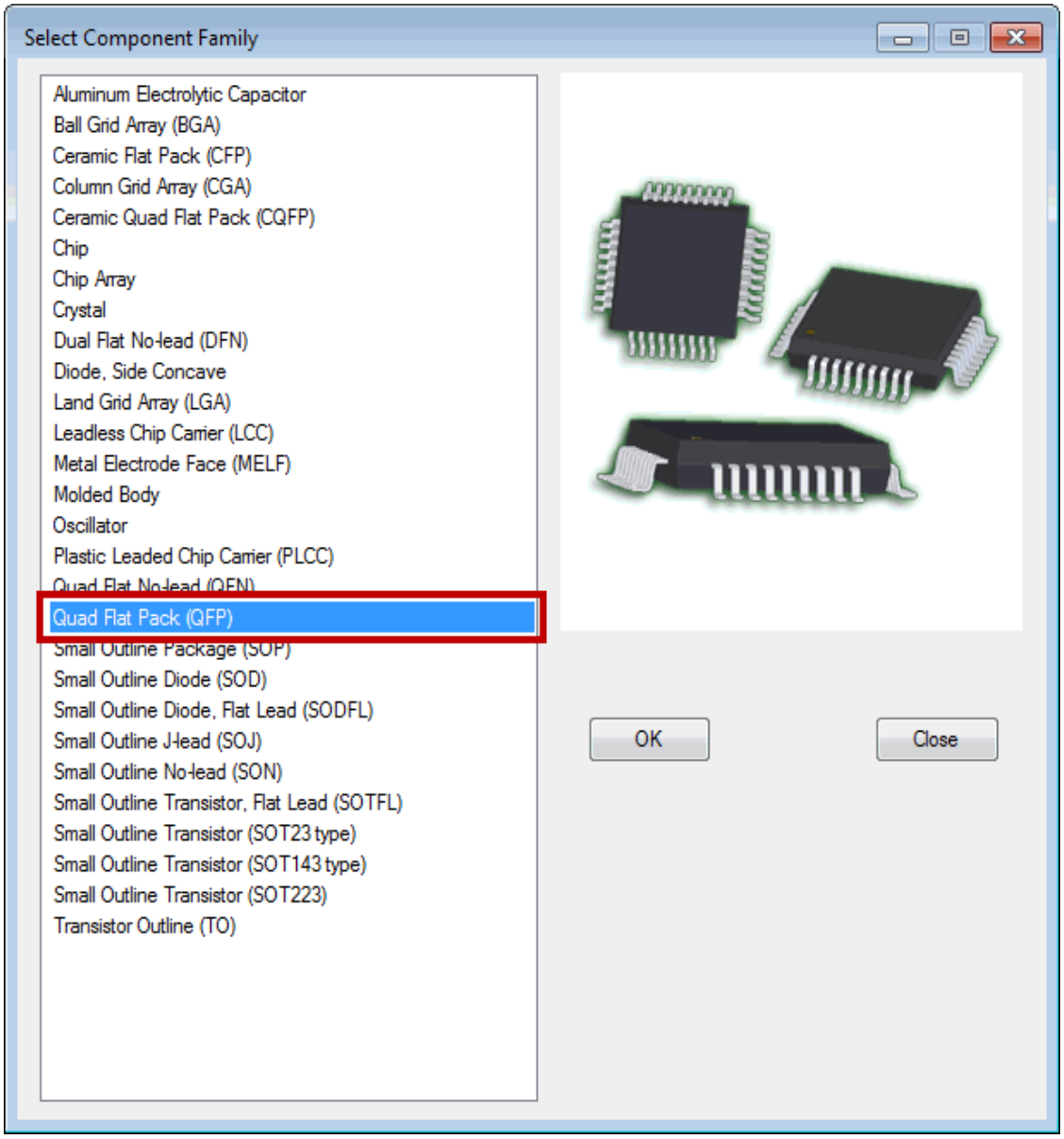
Once you have created your .plb file and made it available to your editing sessions with the Library Manager, perform a test run of the Calculator and Wizard to verify that everything works correctly and that your preferences are properly implemented.

Procedure

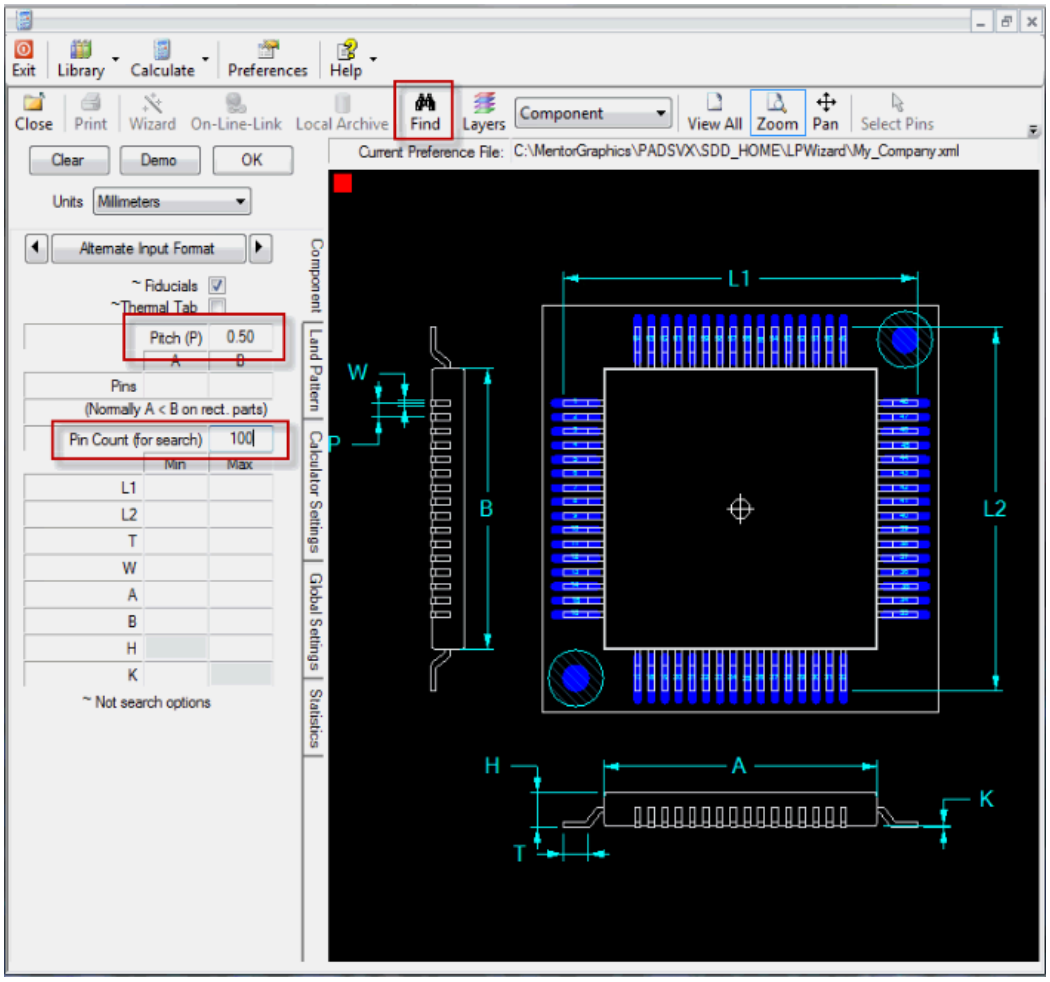
1. Select the “Calculate” drop-down in the main toolbar and select “SMD Calculator”.



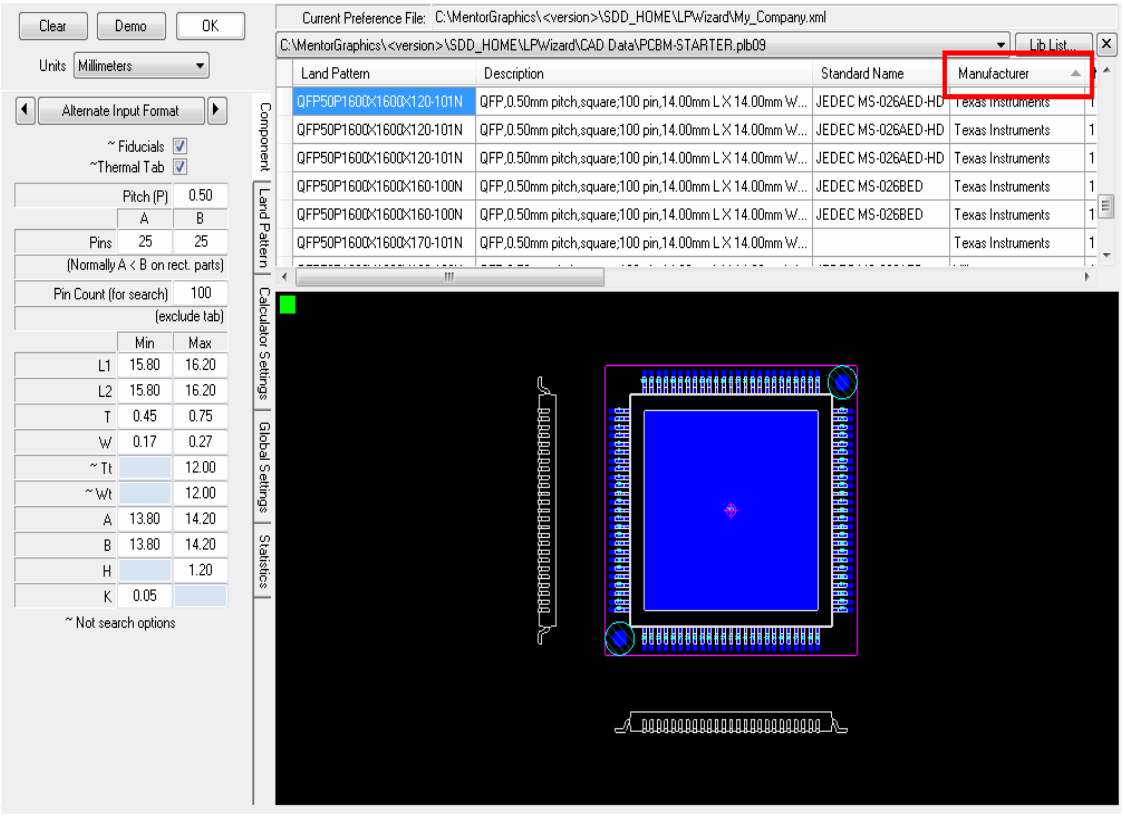
2. Select the “Quad Flat Plack (QFP)” component family in the list and click **OK** to open the Calculator.



3. For example, click **Clear** to void all parameters, then enter some component dimensions from the manufacturer's datasheet like Pitch (P) = 0.50 and Pin Count (for search) = 100 and then select the "Find" button in the toolbar. Only parts that contain that search criteria will appear in the list above the picture. The more dimensions that you enter into the component dimension column, the finer the search results will be. The goal is to quickly browse your existing library to see if the part you are looking for is already in the library.

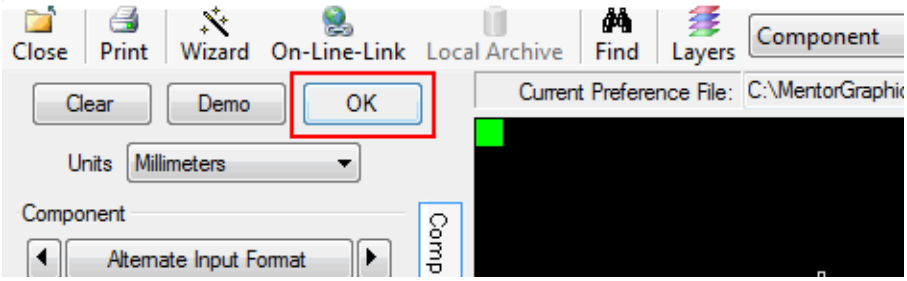


4. You can sort the attribute columns by “clicking” the header of each column. Example: If you are searching for a Texas Instruments land pattern you will be able to quickly scroll down to that manufacturer to see if the dimensional data in the PLB file matches the component dimensions on your datasheet. When you locate the correct part that you’re looking for, “double click” on that line item to open the land pattern in the Calculator.

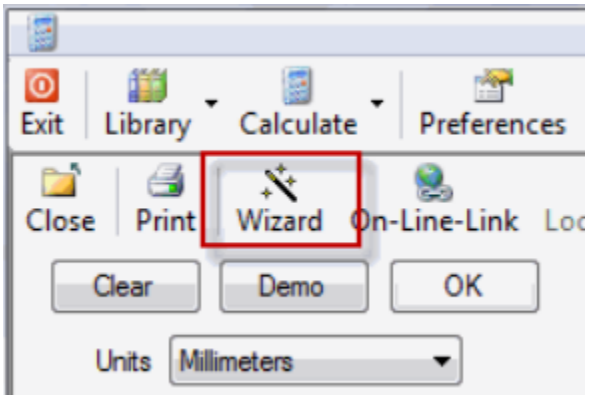


Note:
If you cannot locate an existing land pattern that matches your datasheet, open the land pattern with the most dimensional matches and then update the dimensions so that all of them match your component dimension datasheet.

5. Select the “OK” button when completed.



6. Next select the “Wizard” button in the toolbar to create the library part and write the component dimensions and your rule data to your personal PLB file.



When the SailWind Land Pattern Creator menu opens, your personal user PLB file will be loaded and ready to write to your CAD tool, library directory and library name are setup and ready to go.

Also notice that all the attribute data is carried over from the PCBM-STARTER file and is automatically loaded into the SailWind Land Pattern Creator menu for transfer into your PLB file.

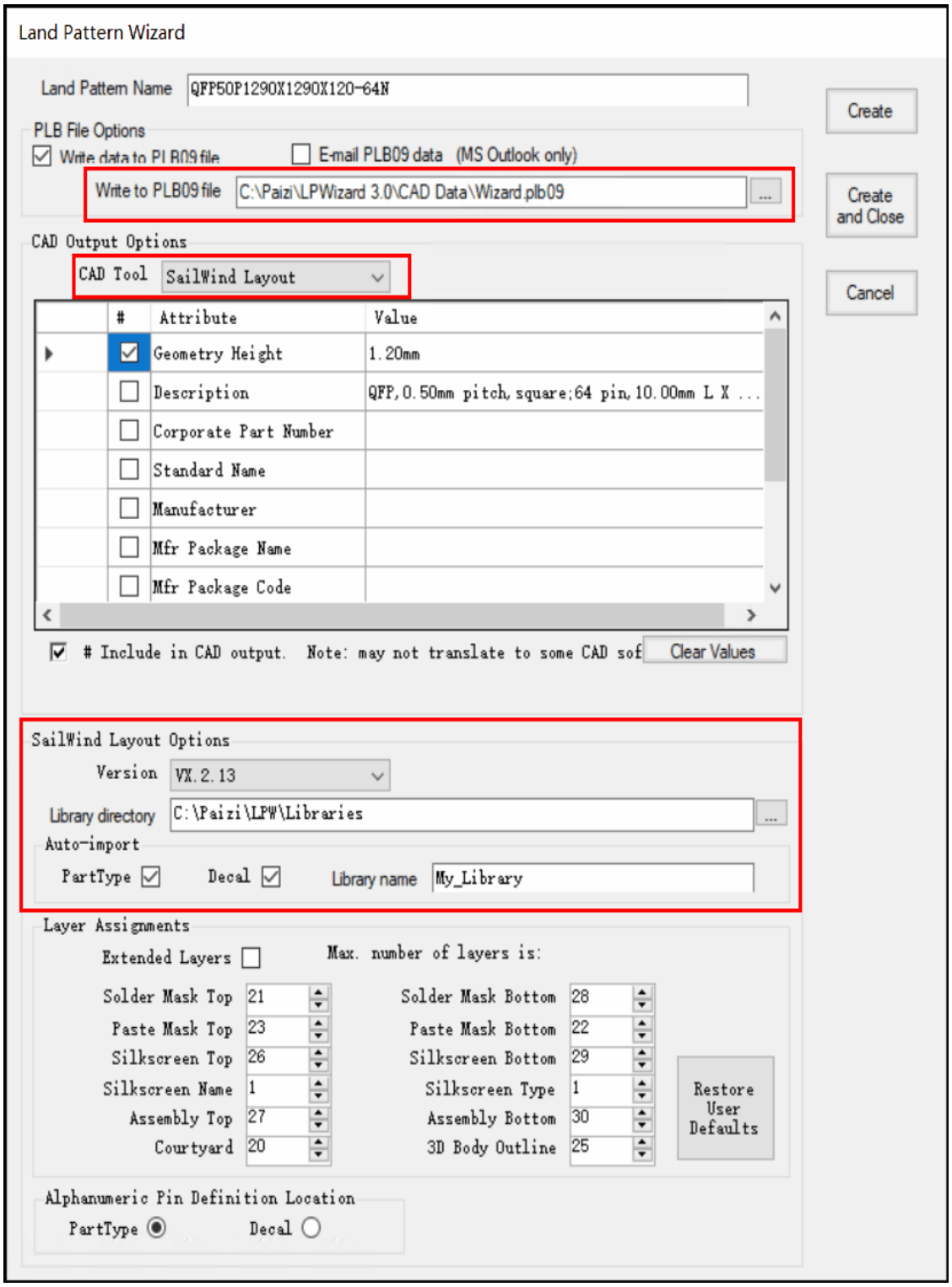
The software auto-generates the “Land Pattern Name” and the “Description” field data.

You can change any field that you want at this time.

7. When you are finished with all of your changes, select the “Create and Close” button in the upper right of the menu. This will do two things at the same time –

   - Write the component dimensions and rule data to your personal PLB file.
   - Automatically add the land pattern to your CAD library.

If you are using SailWind Layout for your CAD tool, you must open SailWind Layout before you select the “Create and Close” button.



Frequently Asked Questions

When using the Calculator and Wizard, you may encounter some unfamiliar situations. These frequently asked questions may be useful to help you get started.

What is a ‘Pin Package’?

The pin package is the number of pins that a component package can accommodate with no missing pins. For example, a three pin SOT would be a six pin package with three missing pins.

How can I create a 3-pin SOT23 ?

To create a 3 or 5 pin SOT, enter 6 for the pin package then fill in the rest of the component data and calculate a land pattern with 6 pins. Then click on the 'Select Pins' toolbar button, move the cursor to the pins you want to remove, click on each pin (pin will highlight), then select 'Delete' from the toolbar or right-mouse-button context menu.

How can I copy data from one PLB to another PLB ?

Open two sessions of the Library Manger in Edit mode. In one session open the destination library, in the other open the source library. In the source library click on a row header for each row you want to copy (or use shift-click to select multiple rows). Then right-mouse-button click to display the edit context menu. Select copy. In the destination library, place the cursor anywhere in the library table and right-mouse-button click to display the edit context menu and select paste. Any duplicate columns will be mapped and copied to the destination library while new columns will be added after existing columns. If column mapping is required this operation cannot be undone.

What do the Calculator Advisory message mean?

Advisory messages are displayed to alert to some condition that occurred either effecting a calculation or that resulted in an unexpected event.

Typical advisories will report that a calculation was modified to prevent a rule violation or that some aspect of the calculated geometry prevented the normal application of a land pattern element such as silkscreen or solder mask. Advisories will also be displayed if a land pattern that was selected from a library failed to verify correctly.

Example:

Advisory: 'Land pattern geometry is too small for silkscreen outline'

Meaning: Silkscreen can not be placed inside the courtyard without violating the silkscreen-to-land spacing rule.

Solution: change rules for silkscreen-to-land spacing or reduce the silkscreen line width.

How can I stop the Calculator from changing a land shape when it trims the land?

The calculator automatically trims some land patterns in conditions where a rule would be violated, such as land-to-land spacing or land extending under component body without sufficient solder clearance. Where automatic trimming is a factor there is a user option to select the shape of the trimmed land. This option can be found in the Calculator on the 'Calculator Settings' tab in the 'Settings' groupbox. The trim shape can be set either as a user preference which is applied for each calculation, or a calculator preference for individual calculations.

Why does the manufacturer’s recommended land pattern appear to be rotated 90 degrees from the calculator land pattern?

Typically the manufacturer hasn't taken the toe and heel goals into consideration (or they're too small). You can usually verify this by setting the Calculator Environment to 'User' and changing all goals to zero. The calculated land pattern will shrink considerably and probably rotate to the same orientation as that of the manufacturer.

Calculator Tips

When using the Calculator, there are a few useful tips that may assist you in your editing tasks, especially when you want to preserve attributes.

- When creating land patterns for chip components of less than 1.6mm in length, make the terminal (“T” dimension) the same for both Min and Max (ie. zero tolerance).
- To preserve attributes when editing a part selected from a library in the Calculator:

  a. After the land pattern is selected and in the calculator, briefly open the Wizard and double-click in any cell as if to edit it's contents.

  b. Press the 'Esc' key to abandon the edit and click 'Cancel' to close the Wizard.

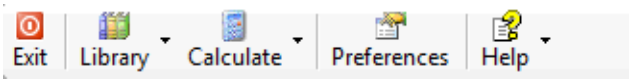
  c. When you re-enter the wizard you will be asked if you want to keep the last set of attributes - answer 'Yes'.

Main Toolbar Options

The Main Toolbar is the first toolbar to appear when the program is opened and the means by which all calculator functions are activated.

All functions are accessed using Tool Bar Buttons.

Figure 1. Main Toolbar



- **Exit**—closes the program.
- **Library**—is a pull-down menu that opens the Library Manager in either Search or Edit modes. In 'search' mode, libraries contents may be viewed as read only. In 'edit' mode library contents can searched, modified and saved.
- **Calculate**—is a pull-down menu that opens any of the Surface Mount, Through-Hole or Connectors calculators, Hole Size Calculator, Hole Pad Stack Calculator and Via Calculator. An additional Unit Conversion Calculator is provided here to conveniently convert back and forth between microns (micrometers), millimeters, mils and inches.

- **Preferences**—opens the user preference window. Here, program options may be selected and modified.
- **Help**—is a pull-down menu that can be used to access this document.

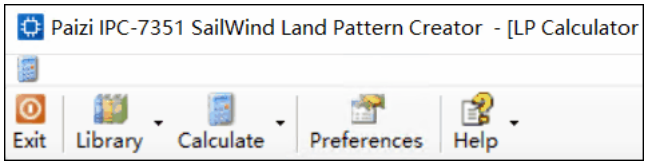
Main Toolbar Options Context Menu

- A Right-Mouse-Button click contextual menu is available with all the toolbar options (except Exit).

Operating Modes

The program may operate in any of four modes. The type of mode is displayed along with the revision level and build number in the header of the main menu window.

Figure 2. Operating Mode Example - Calculator



- **Viewer**—(basic shareware mode) Includes land pattern viewer and Library Manager Search function.
- **Calculator**—(advanced shareware mode) Viewer plus all land pattern calculators.
- **Librarian**—(licensed mode) Calculator, Library Manager Search and Edit functions, plus the ability to save calculator generated land patterns.
- **Wizard**—(licensed mode) Calculator, Librarian plus a pattern construction wizard capable of creating the land pattern in one or more CAD formats.

Chapter 2
User Preferences

The Calculator and Wizard are extremely flexible and can be configured to incorporate all of your specific design rules and standards. Set up preferences to customize the functionality of the software.

Creating a New Preferences User Profile

All user preferences are saved to a file. The main file used for this purpose is named DefaultUser.xml and is located in the program application directory. The DefaultUser file is sufficient to contain all the information for a single user configuration.

It may be advantageous in many cases to create and save additional configurations. One possibility being a setup where mils is the default unit of measurement and settings are modified to represent whole mils instead of their simple conversion from metric. In such a case new user configuration would be created as follows. Any number of files may be saved to reflect preferences for a particular type of calculation. For example, one set might be for printed-circuits, another for flex-circuits or another for a user-specific application.

Procedure

1. In the Preferences dialog box, at the top of any settings page, click the Browse button (...) associated with the text box Source file for default settings (*Usr.xml file).
2. Enter a name for the new User Profile and click the Open button. If the file exists it will be opened. If not, follow the prompts to create the new file.
3. Once the new file is created its name will appear in the Source file for default settings (*Usr.xml file) text box.
4. Make all desired changes to the User settings for each option.
5. Save and Close the Preference dialog box.

Results

The new defaults will be applied for all subsequent calculations in the current session and every subsequent session until the user profile is changed again.

Preferences Dialog Box Overview

Preferences allow you to set and save settings and controls that are used to perform land pattern calculations. This includes the whole range of general settings that control the display characteristics up to those defining the output to CAD.

- Preferences, Fiducials Page—Define the look of fiducials that are added to certain land patterns.
- Preferences, Global Settings—Make settings that apply to all calculations including Rules, Drafting, Unit selection and other related options.
- Preferences, Layers—Set the display characteristics for on-screen viewing and printing.
- Library Manager Page—Set up file management and library format options. Search and edit libraries.
- **Surface Mount (** "Grid Array" on page 47 **and** "Non Grid Array" on page 49 **) and** "Through Hole" on page 52 **Calculators—**Select 'default' or 'user' modes of calculation. Define user values to apply to calculations.
- **Wizard**
  - CAD Tool options are those which apply to specific cad tools.
  - "Options" on page 54 include those that apply to plb09 file operations and the selection of the default CAD tool, if any.

Preferences are accessed by clicking on their respective folders in the Preferences tree-view list (the left panel in the Preferences window).

Note:
There are 'Error Handlers' for each user entry that will indicate the presence of a missing or unacceptable entry. An error in any user-entered preference will prevent saving or changing to another preference page.

Once changed, User Preferences can be applied to the current session, saved or discarded.

- **Close** will simply close the window, however, if changes have been made a reminder will pop up asking if to save.
- **Apply** applies the preferences to the current session but lost after closing the program.

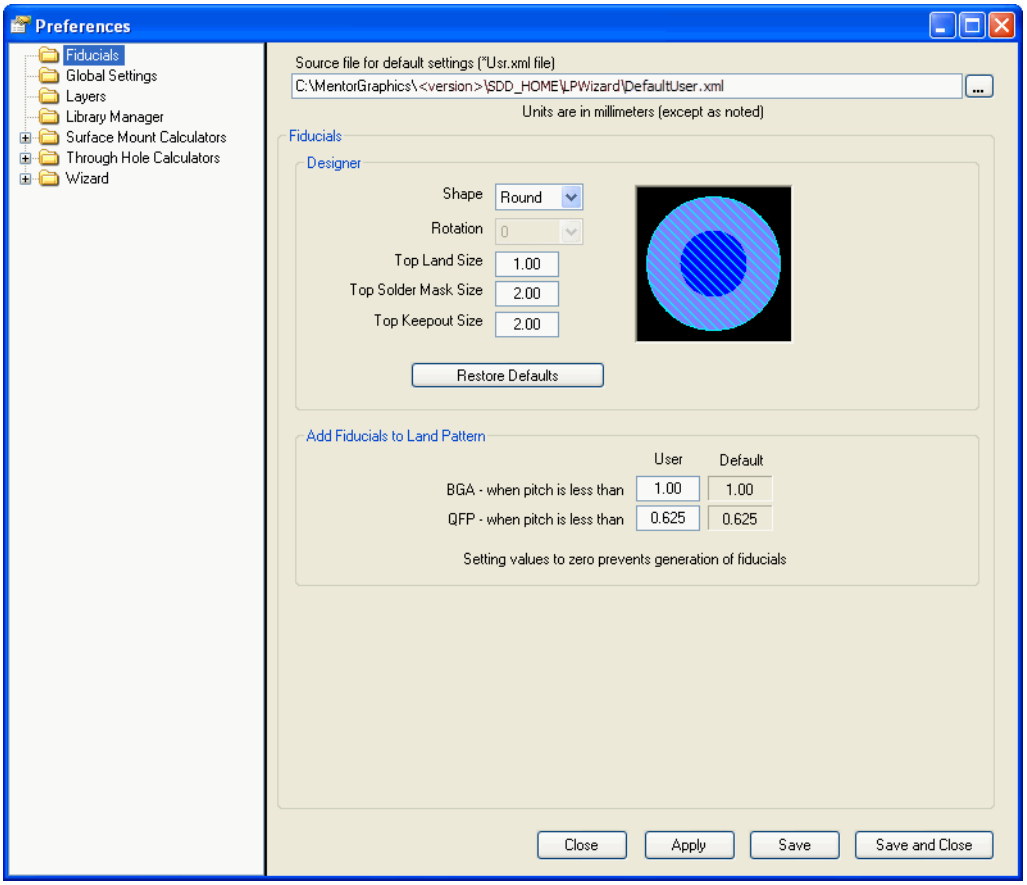
- **Save** and **Save and Close** updates the user preference to both apply the preference and save it for use in subsequent sessions.

Note:
Some preferences have a button or some other provision that will restore factory defaults to certain user settings. To restore all user settings to factory default, delete or rename the 'DefaultUser.xml' file, if that file is the source for user preferences (found in the program directory) or other user preference file.

Preferences, Fiducials Page

To access: **Preferences button > Fiducials folder**

The Fiducial Designer lets you create the shapes that will be added when fiducial alignment indicators are added to a land pattern.



Objects

Field	Description
Designer area	
Shape	There are three available shapes: • Round (most popular) • Square • Diamond
Rotation	Only available for Square shape
Top Land Size	Defines the overall pad size. The IPC default pad size is 1mm.

Field	Description
Top Solder Mask Size	IPC default solder mask size is 2:1 scale larger than the pad size
Top Keepout Size	IPC default keepout size is the same as the solder mask size
Restore Defaults	At any time, you can restore the default IPC values.

Add Fiducials to Land Pattern area—Fiducials may only be added to certain family types. When fiducials can be added there will be an option to add fiducials automatically or disabled completely.

To automatically add fiducials, fill in a value defining the condition under which to add the fiducials. This value typically represents a minimum pitch, below which fiducials will be added automatically at the time of calculation. Setting this value to zero will disable the auto-add feature.

Note:
Fiducials can always be added or removed in the calculator by selecting or clearing the fiducial option check box.

Preferences, Global Settings

To access: **Preferences** button > Global Settings folder

Global Preferences are applied universally to each land pattern, where applicable. These settings include Rules, Drafting, Environment and miscellaneous settings such as design units.

Description

The Global Settings screen has four tabs.

Figure 3. Global Settings Dialog Box

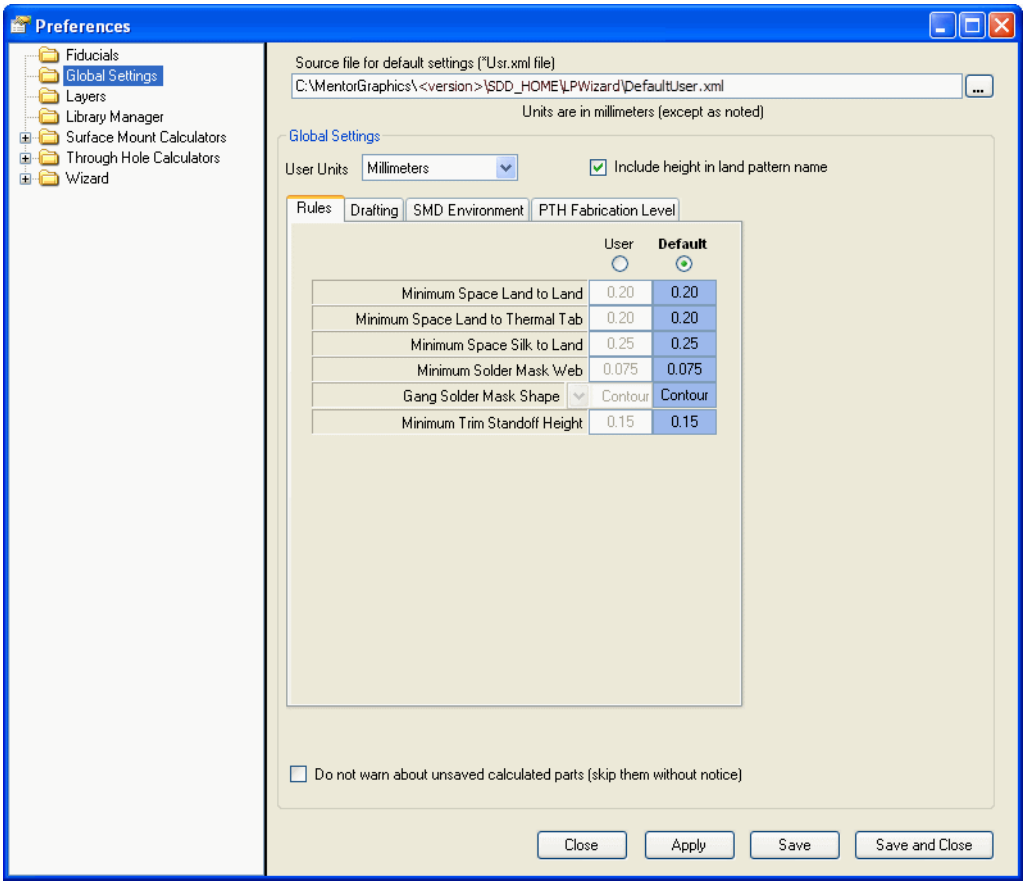


Figure 4. Rules Tab

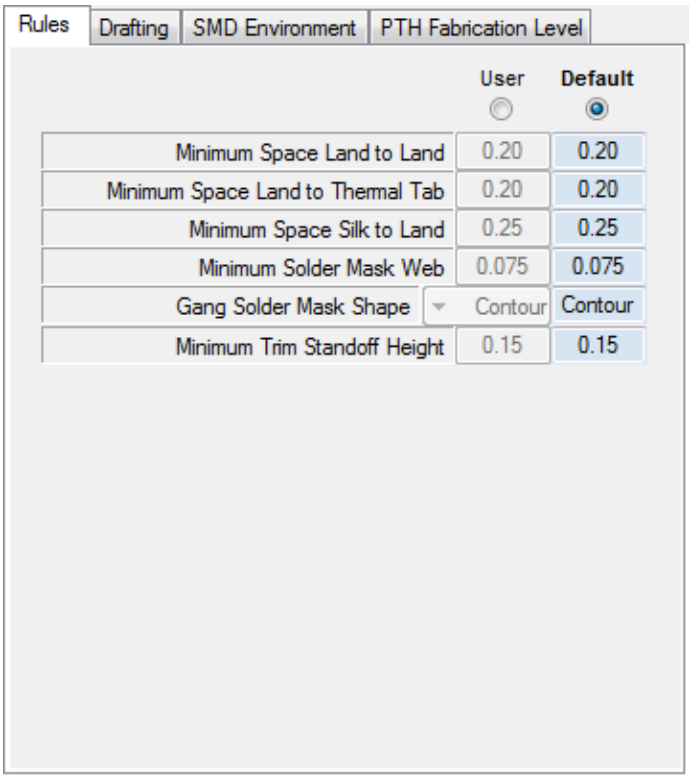


Figure 5. Drafting Tab

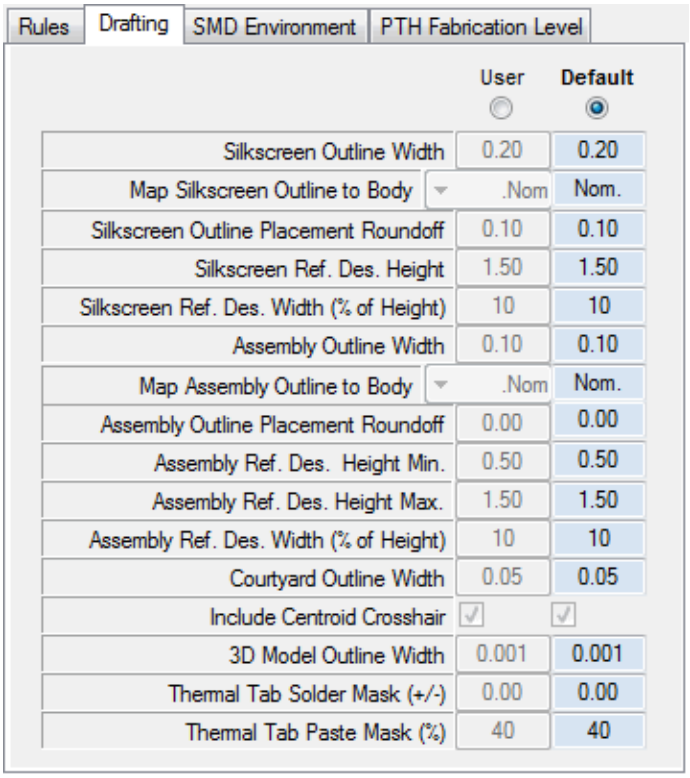


Figure 6. SMD Environment Tab

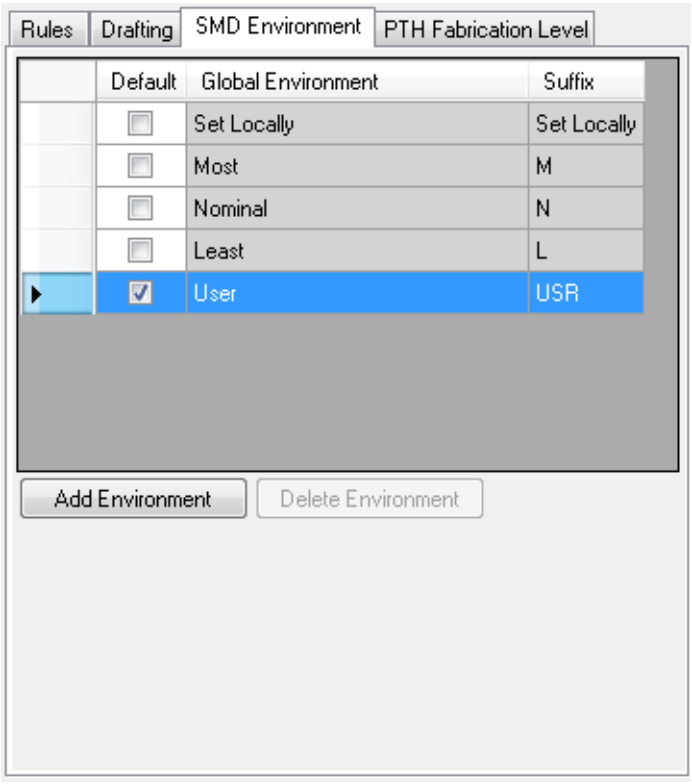


Figure 7. PTH Fabrication Level Tab

Objects

Table 1. Global Settings Dialog Box Contents

Fields	Description
User Units	You can set up Global Units so every component family will use them as the default. There are currently four different working units for the Calculator: • **Millimeters**—The default units. • **Microns (Micrometers)**—For the Xpedition CAD Output format, the land pattern name is appended with "_um" to prevent overwriting of the same land pattern in the default units (millimeters) • **Inches**—For the Xpedition CAD Output format, the land pattern name is appended with "_in" to prevent overwriting of the same land pattern in the default units (millimeters) • **Mils**—For the Xpedition CAD Output format, the land pattern name is appended with "_th" to prevent overwriting of the same land pattern in the default units (millimeters)
Include height in land pattern name	This feature turns on/off the Height dimension in the land pattern name. The IPC default is to have it turned on to discriminate the height of each component for 3D Modeling using SolidWorks or PRO-Engineer.

Table 1. Global Settings Dialog Box Contents (continued)

Fields	Description
Do not warn about unsaved calculated parts	This feature controls the visibility of a warning that appears when a calculated part has not been saved prior to either closing the calculator dialog box or changing the family type.

Table 2. Rules Tab Contents

Fields	Description
Minimum Space Land to Land	This value represents the minimum allowable space between the edge of any land and its neighbor. Trimming results when this space is smaller than this specified minimum value. Trimming can occur for: • Side-to-side spacing when the pitch minus the land width is less than the minimum. • Opposing lands when the land spacing minus the land length is less than the minimum • Corner-to-Corner when corner lands of a four sided pattern become too close. • Land-to-Thermal Tab when the signal lands become to close to a central thermal tab.
Minimum Space Land to Thermal Tab	This value represents the minimum allowable space between the edge of any land and the central thermal tab.
Minimum Space Silk to Land	This setting is for edge of silkscreen outline to edge of land. The IPC default is 0.25mm (10 mils) minimum spacing.
Minimum Solder Mask Web	This feature is primary used when you want to oversize the solder mask. It allows you to set a minimum web, or solder sliver, in between the edge to edge of the oversized solder mask. If this rule is violated, the Calculator automatically “Gang Mask” or "block" the entire row of lands to remove unmanufacturable solder mask webs.
Gang Solder Mask Shape	This option allows you to set the shape of the gang mask to conform to the pad shape or to appear as a rectangular block. **Tip** Setting Minimum Solder Mask Web value to zero disables solder mask blocking.
Minimum Trim Standoff Height	This feature defines your paste mask stencil thickness. When the minimum space between the bottom of the component and the printed circuit board surface is less than the paste mask stencil thickness, the J-STD-001 assembly standard recommends trimming the land out from under the component body. If the minimum standoff component height is less than the paste mask stencil thickness, the lands will be trimmed to avoid paste mask smearing under the component body.

Table 2. Rules Tab Contents (continued)

Fields	Description
	Tip Setting Minimum Trim Standoff Height value to zero disables automatic trimming.

Table 3. Drafting Tab Content

Field	Description
Silkscreen Outline Width	The aperture width that draws the silkscreen outline. The IPC default is 0.2mm (8 mils). **Tip** Setting this value to zero disables the generation of any silkscreen elements.
Map Silkscreen Outline to Body	This selection option permits the silkscreen size to adjust to the minimum, nominal or maximum body dimension.
Silkscreen Outline Placement Roundoff	This value sets the placement grid on which the silkscreen lines will be drawn after adjusting the for Map-to size.
Silkscreen Ref. Des. Height	The reference designator text character height for the silkscreen. The IPC default is 1.5mm (60 mils). **Tip** Setting this value to zero prevents text from being added.
Silkscreen Ref. Des. Width (% of Height)	This value represents the text character width expressed as a percentage of the height (typically 10%).
Assembly Outline Width	The aperture width that draws the assembly outline, The IPC default is 0.1mm (4 mils). **Tip** Setting this value to zero disables the generation of any assembly elements.
Map Assembly Outline to Body	This selection option permits the assembly layer size to adjust to the minimum, nominal or maximum body dimension.
Assembly Outline Placement Roundoff	This value sets the placement grid on which the assembly layer lines will be drawn after adjusting for the Map-to size.
Assembly Ref. Des. Height Min./Max.	The assembly reference designator is automatically scaled to fit in the closed polygon assembly drawing outline for all components. These settings define the minimum and maximum allowable assembly drawing reference designator height. The IPC defaults are: 0.5mm (20 mils) for minimum; 1.5mm (60 mils) for maximum. **Tips:** • Setting Min. to zero prevents text from being added. • Setting Min. and Max. to the same value disables automatic scaling.

Table 3. Drafting Tab Content (continued)

Field	Description
Assembly Ref. Des. Width (% of Height)	Width as a percent of height. With a width of 10 the width for a height of 2 would be 0.20. For CAD tools where text is drawn the width will be the aperture of the line. For CAD tools using fonts the width will be the font width.
Courtyard Outline Width	The aperture width that draws the courtyard outline, The IPC default is 0.05mm (2 mils). **Tip** Setting this value to zero disables the generation of any courtyard elements.
Include Centroid Crosshair	This feature allows you to turn “on” or “off” the 1mm X 1mm centric crosshair at the land pattern origin. The Centric Crosshair is placed on the same layer as the placement courtyard. The IPC default is set to “on”.
3D Model Outline Width	The aperture width that draws the 3D Model outline. The IPC default is 0.001mm (0.04 mils). The 3D Model outline is a closed polygon on an isolated layer primarily used for IDF export to SolidWorks or PRO-Engineer 3D Model tools. **Tip** Setting this value to zero disables the generation of this outline.
Thermal Tab Solder Mask (+/-)	The signal pin solder mask does not include the “Thermal Tab” solder mask setting. This is a separate global setting for all Thermal Tab solder mask. You can define a “solder mask defined” Thermal Land or a “non-solder mask defined” Thermal Land.
Thermal Tab Paste Mask (%)	This represents the percentage amount of Paste Mask that will be applied to the Thermal Land. The IPC default is 40%, but this value ranges widely from one assembly shop to another. It’s best to ask your assembly shop how much percentage of paste masks you should apply on your paste mask.

Table 4. SMD Environment Tab Contents

Field	Description
(For details on the meaning and application of environments see Glossary term Environment)	
Default column	Select the check box of the Global Environment setting to use by default.
Global Environment	The Environmentcan be set locally for each component family or globally. • **Set Locally** — each surface mount component family will default to the environment selected as the preference for that family. If set to a value by this control, the environment for all SMD families will default to this selection. • **Most - M**, **Nominal - N**, or **Least - L** — the displayed character is added to the end of the land pattern name.

Table 4. SMD Environment Tab Contents (continued)

Field	Description
	• **User** — specify one or more custom environments for use with "Surface Mount Calculators, Non-Grid Array Pages" on page 49 calculators and custom suffixes to add to the end of the land pattern name. New custom environments are automatically added to the Environment tabs of all non-grid array SMD calculator preference settings. Use the **Add Environment** and **Delete Environment** buttons to control the custom environments listed here.
User name suffix	Type the suffix to use when the Global Environment setting is set to User.

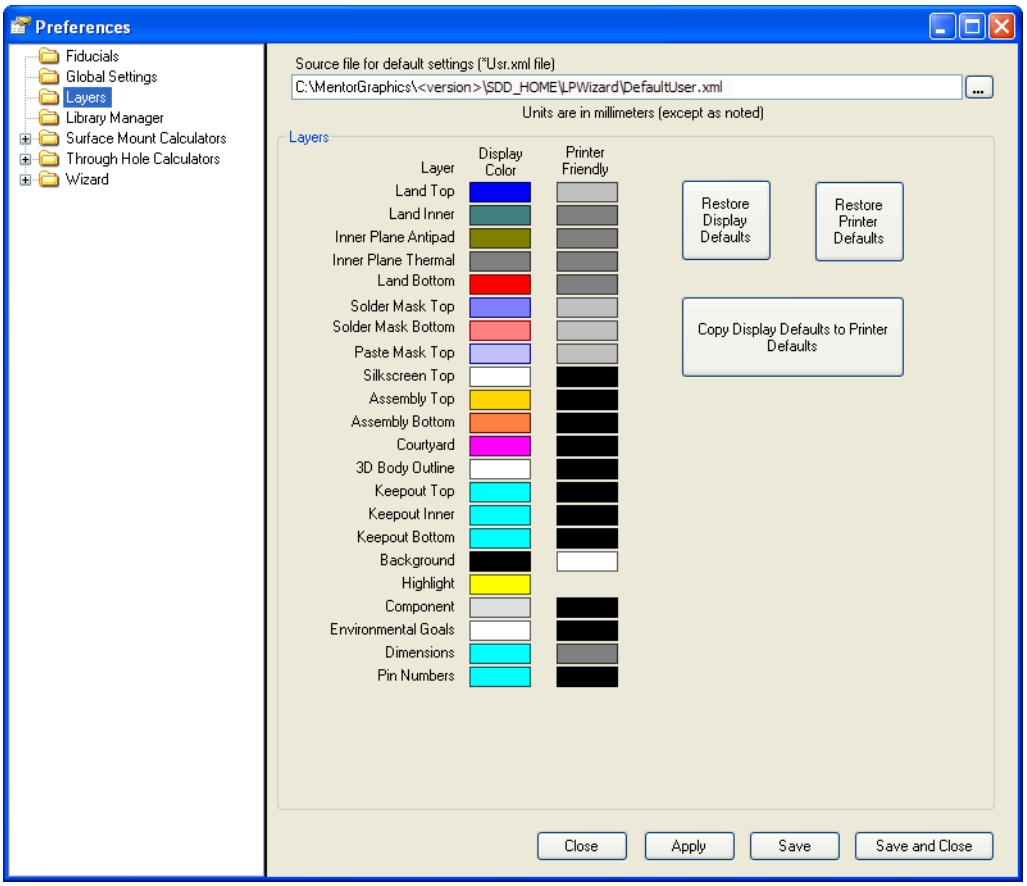
Table 5. PTH Environment Tab Contents

Field	Description
For details on the meaning and application of fabrication levels see Glossary term Fabrication Level	
Global Environment	The fabrication level can be set locally for each through-hole component family or globally. • **Set Locally** — each component family will default to the environment that selected by the preference for that family. If set to a value by this control, the fabrication level for all through-hole families will default to this selection. • **Most - A**, **Nominal - B**, or **Least - C** — the displayed character is added to the end of the land pattern name. • **User** — the characters specified by the User name suffix is added.
User name suffix	Type the suffix to use when the Global Environment setting is set to User.

Preferences, Layers

To access: **Preferences** button > Layers folder

You can reassign the colors for every layer for both the Display and a Printer.



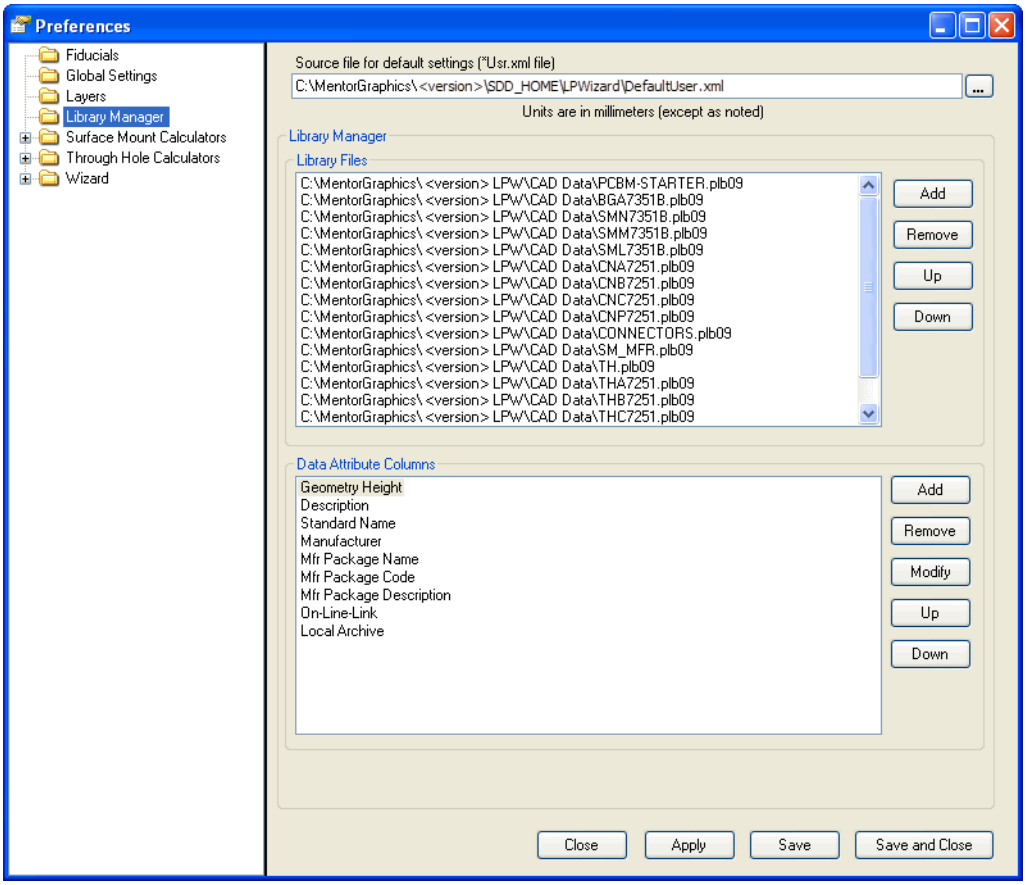
Objects

Field	Description
Color boxes	Click in any of the color boxes to display a color pallet from which the desired color may be selected.
Printer Friendly	For printing you can choose either color or gray-scale.
Restore buttons	Buttons are available to restore factory preset colors for both display and printer.
Copy button	Used to copy display colors to printer colors, if desired.

Library Manager Page

To access: **Preferences** button > Library Manager folder

The Library Manager Preferences allows you to set up and save the output file name and conditions for documentation creation.



Objects

Field	Description
Library Files	Land pattern libraries may be permanently added, removed or sorted by the Library Files preferences. PLB Files appearing in this list will also appear in the Lib List selection of the Library Manager and Calculator.
Data Attribute Columns	Libraries may contain any kind of information you want to document about a part or land pattern. These types of information, or 'Attributes', are organized in columns that are displayed when a plb09 library is opened in the Library Manager or Calculator. The names and organization of these columns may be formatted by this control.

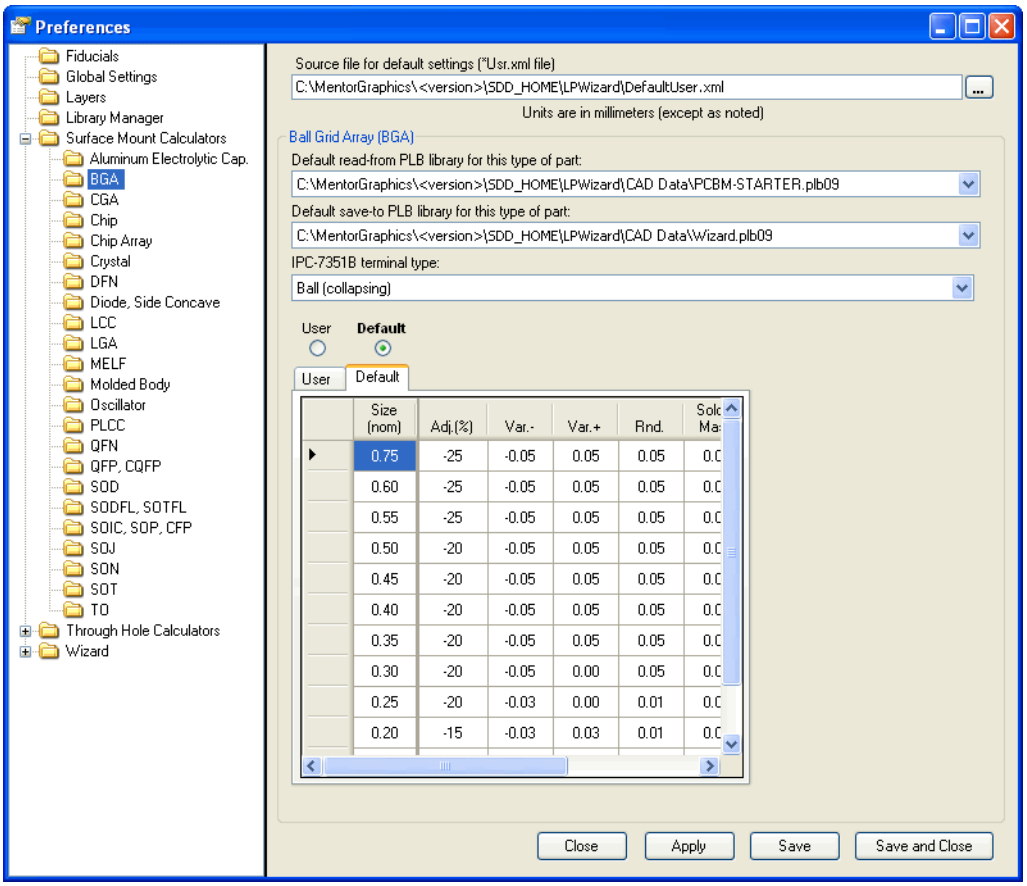
Field	Description
	Restriction: The Data Attribute Geometry Height can not be moved, modified or deleted.

Surface Mount Calculators, Grid Array Pages

To access: **Preferences** button > Surface Mount Calculators folder > grid array-type folder

The SMT Calculators, Grid Array pages allow you to configure and save your .plb file paths, terminal type settings and array numbering schemes for grid array components.

Figure 8. Surface Mount Calculators, Grid Array Page Example



Objects

Field	Description
Default read-from PLB library for this type of part	The default library that is searched first when using the calculator's Find function to locate land patterns in an existing library.
Default save-to PLB library for this type of part	The default library that is listed for saving to when saving this type of part.
IPC-7351B terminal type	Terminal type describes the terminal type to which settings will be applied. Some component families have only one terminal type. Others may have more depending on certain component variables. When more than one terminal type is available, setting

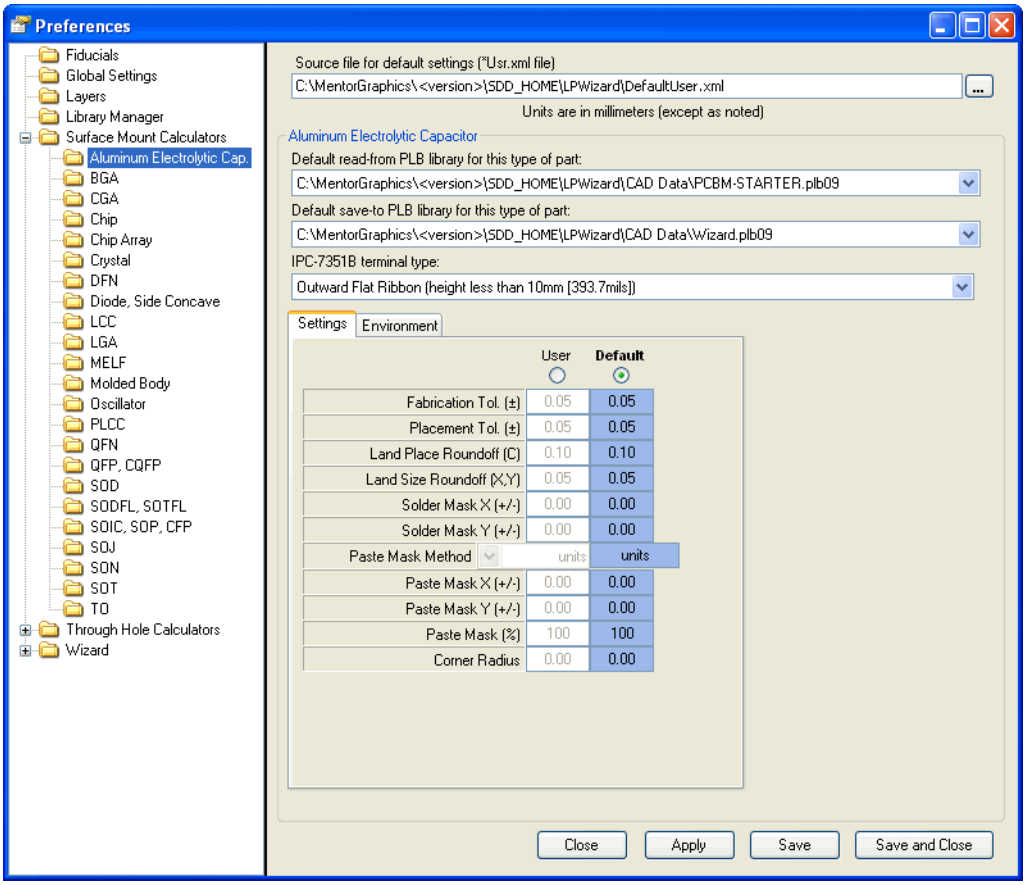
Field	Description
	and environment preferences will apply on to the terminal type shown.
Calculator Settings—Grid Arrays (BGA, LGA and CGA) differ from other SMD's in that they can have a relatively unlimited number of conditional settings based on their terminal size. For BGA's this will be the nominal ball; for LGA's and CGA's this will be the lead size. All settings for any particular size will be contained in one line of the table where columns define settings and rows define the size to which those settings apply. The setting values apply not only to the defined size but also to sizes that are greater but less than the next larger size in the table listings. If the size is listed first, it applies to all sizes that are equal to and greater than the listed size. If the size is listed last it will be applied to all sizes that are greater but less than the next larger size, plus all smaller sizes. • **Removing a Setting—**In the setting table, click on the row header then press the 'Delete' key. • **Modifying a Setting—**In the setting table, with the cursor in any cell, click twice (not a double-click) or, press the 'F2' key. Edit the contents, then press the 'Enter' key. • **Adding a Setting—**In the last row of the setting table, enter a value for 'Size (nom)'. If the size is larger than the one above, an 'Error Handler' will immediately appear indicating the value is out of order. To remove the error, click in the 'Size (nom)' column header. This will arrange the size values in numeric order. Error handlers will also appear for any missing or invalid table entries.	
User/Default	Only user settings can be modified. To enable user modifications, check the 'User' radio button and select the 'User' tab. • **Nominal Size Adjustment**—a percent value applied to the nominal size to produce the adjusted size. • **Land Variation Plus/Minus**—values added to the adjusted size to produce maximum and minimum material conditions. • **Round offs**—control the resolution of certain land pattern features. As features become smaller it may become advantageous to adjust the rounding factors in order to improve the solder joint while maintaining spacing clearances. **Note:** Setting roundoff values to zero disables rounding. • **Over/Undersize**—sets the size of the solder or paste mask relative to the land pattern size. • **Courtyard**—some families allow the you to select from a variety of possible shapes subject to certain constraints. Oblong shapes are only allowable when land length exceeds width.

Surface Mount Calculators, Non-Grid Array Pages

To access: **Preferences** button > Surface Mount Calculators folder > non-grid array type folder

The SMT Calculators, Non-Grid Array pages allow you to configure and save your .plb file paths, terminal type settings and tolerances for non- grid array SMT components.

Figure 9. Surface Mount Calculators, Non-Grid Array Page Example



Objects

Fields	Description
Default read-from PLB library for this type of part	The default library that is searched first when using the calculator's Find function to locate land patterns in an existing library.
Default save-to PLB library for this type of part	The default library that is listed for saving to when saving this type of part.
IPC-7351B terminal type	Terminal type describes the terminal type to which settings will be applied. Some component families have only one terminal type. Others may have more depending on certain component

Fields	Description
	variables. When more than one terminal type is available, setting and environment preferences will apply on to the terminal type shown.
Calculator Settings	
Settings	Setting preferences are applied first time any calculation takes place. After they may be modified for recalculations. If preferences are changed the changes will not appear until the calculator is reset (by clicking the "Clear" button on the component tab). • **Fabrication Tolerance**—this figure describes how accurately the fabrication process can produce a land pattern feature of a given dimension. Within plus or minus .05 mm is typical for most applications but may want to increase for high volume applications. • **Placement Tolerance**—this figure describes how accurately the placement process can locate a component on any given land pattern. Within plus or minus .05 mm is typical for most applications but may want to increase for high volume applications. • **Round offs**—control the resolution of certain land pattern features. As features become smaller it may become advantageous to adjust the rounding factors in order to improve the solder joint while maintaining spacing clearances. **Tip** Setting roundoff values to zero disables rounding. • **Over/Undersize**—sets the size of the solder or paste mask relative to the land pattern size. Paste mask can be adjusted in either units or as a percent of coverage. If 'units' are selected the mask can be adjusted in both the x and y dimensions relative to the land. If 'percent' is selected the mask is adjusted equally in x and y. • **Land pattern Shape**—some families allow you to select from a variety of possible shapes subject to certain constraints. Oblong shapes are only allowable when land length exceeds width. • **Corner Radius**—rectangular lands may have either right angle corners or rounded corners. If rounded corners are desired, enter a value for the corner radius. A value of zero will result squared-off corners. If twice the corner radius exceeds either the length or width of the land (whichever is smaller) the radius will be reduced to half of that value. **Note:** Rounded corners are not supported by every CAD tool. • **Lands on Assembly**—some families allow selection of the appearance of lands on the assembly drawing layer. Options are none, filled shape or outline shape. • **Append Thermal Tab Name**—some families support thermal tabs/pads. Enable this setting to append the land pattern name with thermal pad information.
Environment tab	Environment preferences are applied first time any calculation takes place. After they may be modified for recalculations. If preferences are changed the changes will not appear until the

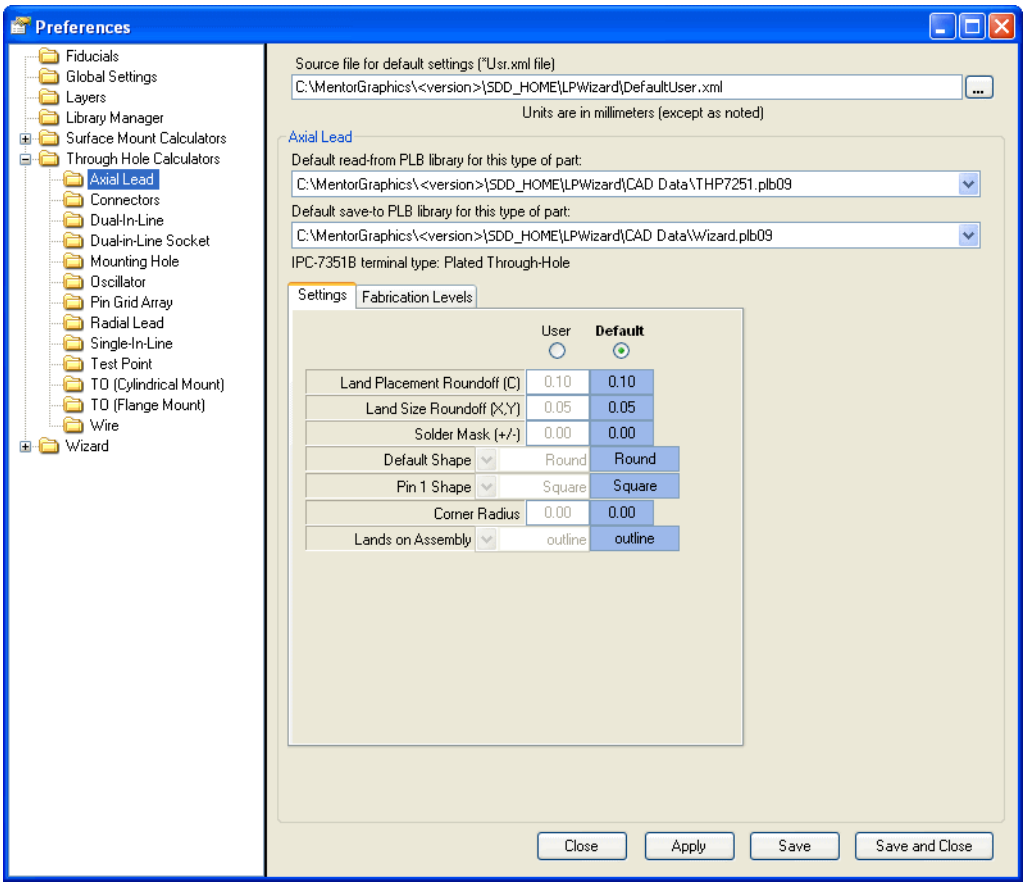
Fields	Description
	calculator is reset (by clicking the "Clear" button on the component tab). This is the control for the Local environment. If the Global environment is set (See Help topic Preferences, Global Settings) it overrides the local setting. The environment name suffix is also set by a Global preference. User specified environment columns are generated by the creation of additional user specified environments created on the SMD Environment tab of the Preferences, Global Settings.

Through-Hole Calculators, Device Pages

To access: **Preferences** button > Through Hole Calculators folder

The Through-Hole Calculators, Device pages allow you to configure and save your .plb file paths, terminal type settings and tolerances for through-hole components.

Figure 10. Through-Hole Calculators, Device Page Example



Objects

Field	Description
Default read-from PLB library for this type of part	The default library that is searched first when using the calculator's Find function to locate land patterns in an existing library.
Default save-to PLB library for this type of part	The default library that is listed for saving to when saving this type of part.
IPC-7351B terminal type	Displays the terminal type to which settings will be applied. Some component families have only one terminal type. Others may have more depending on certain component variables. When

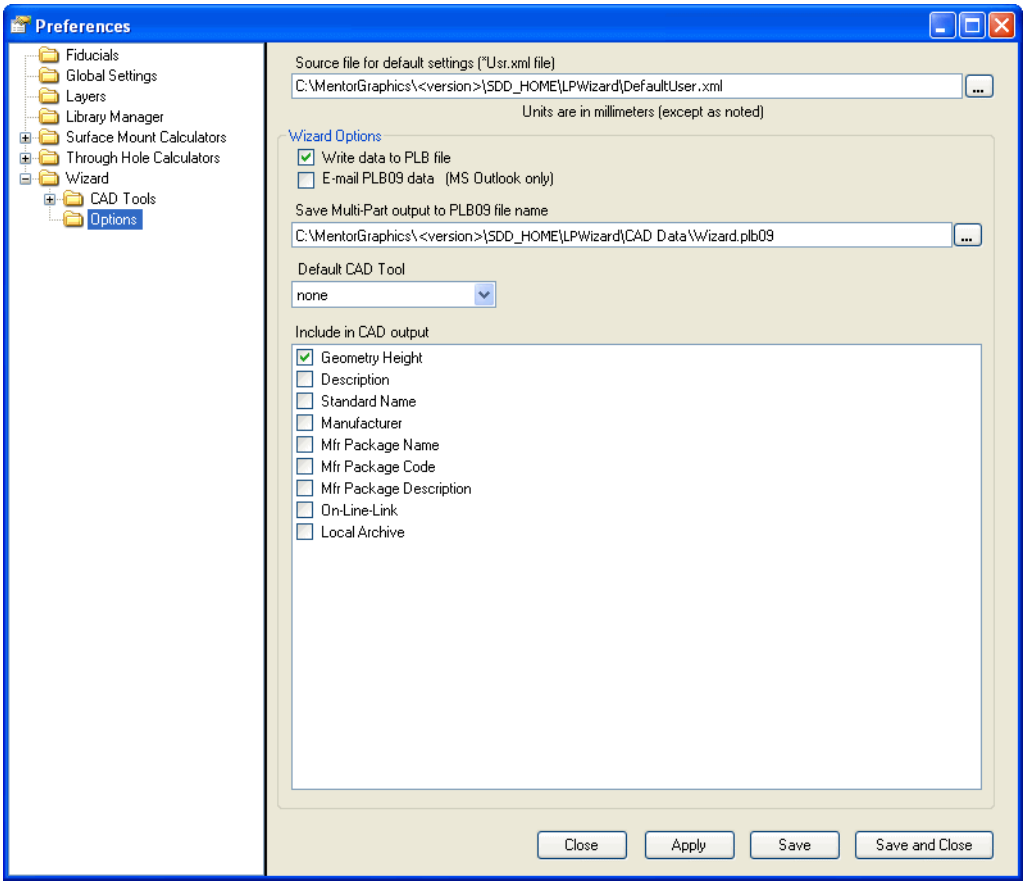
Field	Description
	more than one terminal type is available, setting and environment preferences will apply to the terminal type shown.
Calculator Settings	
Settings tab	Setting preferences are applied first time any calculation takes place. After they may be modified for recalculations. If preferences are changed the changes will not appear until the calculator is reset (by clicking the 'Clear' button on the Component tab). • **Round offs**—control the resolution of certain land pattern features. As features become smaller it may become advantageous to adjust the rounding factors in order to improve the solder joint while maintaining spacing clearances. **Tip** Setting roundoff values to zero disables rounding. • **Over/Undersize**—sets the size of the solder mask relative to the land pattern size. • **Land pattern Shape**—some families allow you to select from a variety of possible shapes subject to certain constraints. • **Corner Radius**—rectangular lands may have either right angle corners or rounded corners. If rounded corners are desired, enter a value for the corner radius. A value of zero will result squared-off corners. If twice the corner radius exceeds either the length or width of the land (whichever is smaller) the radius will be reduced to half of that value. **Note:** Rounded corners are not supported by every CAD tool. • **Land on Assembly**—some families allow selection of the appearance of lands on the assembly drawing layer. Options are none, filled shape or outline shape.
Fabrication Levels tab	Fabrication Level preferences are applied first time any calculation takes place. After they may be modified for recalculations. If preferences are changed the changes will not appear until the calculator is reset (by clicking the 'Clear' button). This is the control for the Local fabrication level. If the Global fabrication level is set (See Help topic Preferences, Global Settings) it overrides the local setting. The fabrication level name suffix is also set by a Global preference.

Wizard, Options Page

To access: **Preferences** button > Wizard folder > Options folder

Preferences marked “Librarian” are those that apply to the saving of a land pattern calculation to library, or *.plb*(09) file and the selection of a default CAD tool. Preferences marked “Wizard” are those that may be applied specifically to a CAD output.

Figure 11. Wizard, Options Page



Objects

Field	Description
Write data to PLB file checkbox (Librarian)	The calculated land pattern will be saved to a *.plb* file.
E-mail PLB Data checkbox (Librarian)	The calculated land pattern will be mailed, as a *.plb* file, named for the land pattern and containing the single calculated land pattern, will be attached to an e-mail that may be sent to another party for review.
Save Multi-Part output to PLB09 file name (Librarian)	The file name indicated in the Wizard Preference is the file name that will be saved when new land patterns are generated using the Multi-Part Wizard feature (for details on the Multi-part Wizard see the Help topic

Field	Description
	"Running Multi-Part Wizard" on page 74). Clicking the **Browse** button (button displaying 3 dots) allows selection of an existing file. If no preference exists, one may be chosen at the time the multi-part options is implemented. For single land patterns, the name of the file is determined by another user preference, dependent on the family type (for details on PLB save preferences see the Help topics: • "Surface Mount Calculators, Grid Array Pages" on page 47 • "Surface Mount Calculators, Non-Grid Array Pages" on page 49 • "Through-Hole Calculators, Device Pages" on page 52
Default CAD Tool (Wizard)	This pull-down menu may be used to select a default CAD format in which to construct the land pattern. Only CAD formats that are licensed for the installation will be displayed. If 'none' is displayed, no CAD output will occur. Wizard preferences are unique for each CAD tool and displayed according to the selected Librarian CAD tool. Generally, these preferences will include (where required or supported by the individual tool): • An output directory. • An output software version. • Available user defined options. • Layer definitions.
Include in CAD output (Librarian)	All library attributes are displayed in a list. Each of these attributes, checked or not, will be included in the output to the PLB file. If checked, attributes will be included in the CAD output format permitting (not all CAD formats support user defined attributes). In normal calculations, attributes listed here are derived from the Library Manager data columns (see the Help topic "Library Manager Page" on page 45). If the Wizard is entered immediately after selecting a land pattern from a library, the attributes from that library are displayed.

Chapter 3
Land Pattern Calculator

The Land Pattern Calculator allows you to specify dimensions of a particular component and then calculates the proper land pattern dimension based upon internal formulas and user-customized .plb file settings.

The calculator can be opened two ways: from the Calculator tool bar button (in the main window) or from the Library Manager.

When opened from the Library Manager the calculator displays the selected land pattern from the library currently in the Library Manager. Otherwise, a sample part is displayed with prompts for your input.

Like the Library Manager, the calculator can open and search libraries. Calculator searches are different to the extent that only components of the same type as the calculator will be displayed. Searching can be filtered according to component data that you enter.

Each calculator stores a set of default settings and environmental values which are automatically applied based on component data. The defaults are recommendations based on studies by IPC regarding surface mount solder joint reliability. If desired, you may substitute your own settings in order to customize the land pattern design.

Calculator inputs are divided in to four categories: Tool Bar Controls, Component, Calculator Settings, and Global Settings. Generated calculations are divided into two categories: Land Pattern and Statistics.

Calculator Toolbar Controls

The Calculator toolbar contains a broad range of buttons that provide access to all of the Calculator functions.

Figure 12. Calculator Toolbar

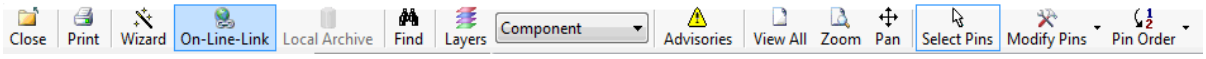


Table 6. Calculator Toolbar Controls

Name	Description
Close	Closes the calculator window and returns to another open window, if any.
Print	Prints a two page 'data sheet' for the calculated land pattern.
Wizard	Opens the Land Pattern Wizard Dialog Box and allows you to save the land pattern to a plb09 library file or create an output to selected CAD tool.

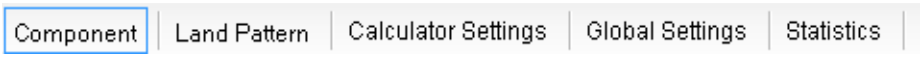
Table 6. Calculator Toolbar Controls (continued)

Name	Description
On-Line-Link	If the displayed component was loaded from a library it may have a link to a manufactures web site with an on-line data sheet. If such a link exists this toolbar button will be enabled and clicking it will go to the link.
Local Archive	If the displayed component was loaded from a library it may have a link to a local directory with component documentation. If such a link exists this toolbar button will be enabled and clicking it will open the file.
Find	Opens the default library associated with calculator family type and loads the library for viewing and searching. Search results can be 'filtered' by providing additional information about characteristics of the search object. For example, if searching for a part with a particular pin configuration, enter the pitch and the total number of pins to restrict the search to land patterns with that pitch and pin count. Some characteristics can not be used to filter the search. Such characteristics are usually indicated by a '~' preceding the value label.
Layers	Opens a layer control panel that allows you to turn on and off layers, dimensions, pin numbers and a display grid locked to the component origin.
Top-Most View Layer	This pull down menu allows you to select the layer that will be displayed 'on top' in display.
Advisories	Additional advisories may result from loading a library land pattern when a verification discrepancy encountered. For information on verification, see Help topic Verifying Land Patterns.
View All	Adjusts the display so that all elements are visible within the viewing area.
Zoom	Sets the viewer left-mouse function to Zoom Window.
Pan	Sets the viewer left-mouse function to Pan.
Select Pins	Pin modification is possible in several component families (BGA, QFP, SO and others). Clicking this button changes the cursor to an arrow and pin may be selected for modification by moving the cursor over the lands and clicking the left mouse button. Pins may also be selected by dragging the cursor to form a box surrounding the pins to be selected. Selected pins are highlighted and subject to modification.
Modify Pins	Pin modifications include: setting the location of pin one, setting the location of the cathode (on diodes), hiding, deleting, restoring, renumbering and reordering.
Pin Order	Pin order clockwise or counter-clockwise may be selected.

Generic Tab Controls

The tab controls are displayed vertically. The tabs allow access to groups of settings for the components, the land patterns and the calculator, as well as global settings and statistics.

Figure 13. (Vertical) Calculator Tabs



Component Tab

This tab contains controls for the entry of component data. Each calculator has a "Demo" button which will load and calculate a sample part for you.

Transfer the component dimensional data from the manufacturer's data sheet in to the appropriate fields of the calculator's Component tab. A provision for Alternate Input Format is provided to allow easy entry of component data whether it is given in the form of minimum-maximum or nominal-with-plus-or-minus tolerances. In some cases Alternate Input Format also accommodates different combinations of available component data that can result from variations in manufacturer supplied data sheet specifications.

Note:
While the Alternate inputs are generally transferable from one format to another, min-max is easily derived from nominal-with-tolerance but nominal-with-tolerance can not be accurately derived from min-max.

Optional Entries

When there is an optional entry the data field will be invisible or disabled until a box or button is checked indicating that the option is to be included.

- Thermal Tabs/Pads are optional for SOIC, SON, QFP and QFN.

  Tip
  When you enable the Thermal Tab, you have the option to append the thermal tab information to the land pattern name. See the "Append Thermal Tab Name" check box on the “Land Pattern Tab” on page 60.

- Fiducial's are optional for BGA and QFP. Optional fiducial's are added automatically for: BGA's when the pitch is equal to or less then 0.80mm; and in QFP's when the pitch is equal to or less then 0.50mm. You always have the option of adding or removing fiducial's but the locations are always fixed as follows: BGA's in upper right and lower left corners at the courtyard intersections; QFP's in upper right and lower left corners centered with pad rows.

- Pull-Back Leads are optional on SON and QFN when terminals are pulled back from the edge of the body.

Click OK once all entries have been made. If any errors are detected a red indicator will flash in the field. Moving the cursor over the indicator will display a 'tool tip' with an explanation of the violation.

At this point a complete calculation will be performed using the default preferences for the Settings, Environment, Rules and Drafting variables. Entries for these variables are disabled unless the User radio button is checked. All land pattern dimensional fields will populate and the Viewer window will display the component and calculated land pattern.

Note:
Parts selected from a library search can be modified to produce new land patterns. If you can find a component close to the one you are looking for, all you have to do is modify the numbers that are different and click the **OK** button.

Land Pattern Tab

This tab is for viewing land pattern calculated dimensions. It also allows you to modify automatically generated names.

Land Pattern and Pad Stack names are generated automatically but you may change them prior to saving or creating the CAD land pattern (using the Land Pattern Wizard).

The Append Thermal Tab Name check box appears if you select the Thermal Tab check box on the “Component Tab” on page 59. The thermal tab is optional for SOIC, SON, QFP and QFN families. After you select the check box, you must click **OK** for the regenerated name to appear.

The View Pin and Padstack Data button opens the Pins, and Padstacks tabs to list the associated data. The button is only available when the Dimensions area is populated.

Note:
The automatic naming convention is not foolproof. It is possible for different components to produce identical names. This can happen for several reasons. For instance, if height is not a factor in the land pattern name so parts with identical footprints might have a different height component unless you intervene in the naming process. Also, the automatically generated name is typically based on certain component Nominal dimensions. Nominal dimensions are derived from averaging the min and max values so that two dimensions with the same nominal value might have vastly different min and max values which will likely result in different land patterns. When you save your part to the plb09 library a check will be made for duplicate names. If a duplicate name is encountered you will have the option to add the duplicate, overwrite the original or cancel the operation. For hints on how to deal with duplicate names see Help topic “Land Pattern Name Syntax” on page 128.

Default settings and environment are always applied to the first calculation of a new land pattern. After that, changed entries will remain until the “Clear” button (on the component tab) is used to clear the component data fields and return the calculator to its default state.

Calculator Settings Tab

This tab contains controls for entering design constraints that will effect how the land pattern will calculated. Entries on this tab apply specifically to the kind of land pattern being created, BGA, chip, QFP, etc. All of the values on this tab can be assigned default values by the user Preferences (See the “Help topics for User Preferences, Surface Mount and Through Hole Devices” on page 31). Any setting may be modified if the User radio button is checked.

The “(Global)” list of environments can contain multiple user specified environments and suffixes set up on the “Preferences, Global Settings” on page 36.

Global Settings Tab

This tab contains controls for entering design constraints that will effect how all land patterns will calculated. All of the values on this tab can be assigned default values by the user Preferences (See Help topic “Preferences, Global Settings” on page 36). Any setting may modified if the User radio button is checked.

Statistics Tab

This tab displays information about the how the calculation satisfied the environmental goals. See Help topic “Analyzing Results” on page 65.

Surface Mount Grid Array Specific Tab Controls

The BGA calculator is set up slightly different from the other SMD calculators owing to the variety of design variation possibilities. The SMD Grid Array calculators have a number of variations that can be accessed from these tabs. These include groups of settings for the components, the land patterns and the calculator, as well as global settings and statistics.

Component Tab for SM Grid Array Devices

Transfer the component dimensional data from the manufacturer's data sheet as described in Help topic “Surface Mount Device Non-Grid Array Specific Tab Controls” on page 62. The following is an enumeration of significant differences that set BGA's apart from other families of component types.

Fiducials—are optional. Fiducial's are added automatically for: BGA's when the pitch is equal to or less then 0.80mm. After a calculation, Fiducial's may be unconditionally added or removed by checking the Fiducial Checkbox. Locations are always fixed as follows: in upper right and lower left corners at the courtyard intersections.

The value for either Rows or Columns may be numeric or alphanumeric. The selection for which is which is set by a pull down menu control. Alphanumeric values may be entered either as an alphanumeric or as a number. For example, a device with 23 rows may be entered either as '23' or its alphanumeric equivalent 'AC'.

Ball Type - Collapsing and Non-collapsing balls—At some point balls get small enough that the weight of the part and the surface tension of the solder in the ball prevent it from collapsing. This effects the size of the pad to the extent that pads for collapsing balls are typically reduced by a percentage of the ball size while non-collapsing balls are typically increased by a percentage of the ball size. Non collapsing balls usually occur only on very small BGA's.

Click OK once all entries have been made. If any errors are detected a red indicator will flash in the field. Moving the cursor over the indicator will display a 'tool tip' with an explanation of the violation.

At this point a complete calculation will be performed using the default preferences for the Settings, Environment, Rules and Drafting variables. Entries for these variables are disabled unless the User radio button is checked. All land pattern dimensional fields will populate and the Viewer window will display the component and calculated land pattern.

Notes:

- Settings are applied first time any calculation takes place. After they may be modified for recalculations. If preferences are changed the changes will not appear until the calculator is reset (by clicking the 'Clear' button).

- Parts selected from a library search can be modified to produce new land patterns. If you can find a component close to the one you are looking for, all you have to do is modify the numbers that are different and select the "OK Button".

Calculator Settings Tab for SM Grid Array Devices

The surface mount grid array family can have a number of settings that can be applied over a range of components based on Ball (or Lead) type and size. Settings are available to control both the land pattern, solder mask and paste mask sizes. All of the values on this tab can be assigned default values by the user Preferences (See Help topic “Surface Mount Calculators, Grid Array Pages” on page 47). Any setting may modified if the User radio button is checked. The definition of the rule being applied to any particular calculation is displayed above the collection of text boxes on this tab.

Settings are applied first time any calculation takes place. After they may be modified for recalculations. If preferences are changed the changes will not appear until the calculator is reset (by clicking the 'Clear' button).

Statistics Tab for SM Grid Array Devices

This tab displays the calculated potential Material Conditions. Maximum Material Condition defines the land pattern dimension.

Surface Mount Device Non-Grid Array Specific Tab Controls

The non-grid array SMD calculators have a number of variations that can be accessed from these tabs. These include groups of settings for the components, the land patterns and the calculator, as well as global settings and statistics.

Component Tab for SM Non-Grid Array Devices

This tab contains controls for the entry of component data. Each calculator has a "Demo" button which will load and calculate a sample part for you.

Transfer the component dimensional data from the manufacturer's data sheet in to the appropriate fields of the calculator's Component tab.

Optional Entries

When there is an optional entry the data field will be invisible or disabled until a box or button is checked indicating that the option is to be included.

- Thermal Pads are optional for SOP, SON, QFP and QFN.
- Fiducial's are optional for BGA and QFP. Optional fiducial's are added automatically for: BGA's when the pitch is equal to or less then 0.80mm; and in QFP's when the pitch is equal to or less then 0.50mm. You always have the option of adding or removing fiducial's but the locations are always fixed as follows: BGA's in upper right and lower left corners at the courtyard intersections; QFP's in upper right and lower left corners centered with pad rows.
- Pull-Back Leads are optional on SON and QFN when terminals are pulled back from the edge of the body.

Click OK once all entries have been made. If any errors are detected a red indicator will flash in the field. Moving the cursor over the indicator will display a 'tool tip' with an explanation of the violation.

At this point a complete calculation will be performed using the default preferences for the Settings, Environment, Rules and Drafting variables. Entries for these variables are disabled unless the User radio button is checked. All land pattern dimensional fields will populate and the Viewer window will display the component and calculated land pattern.

Notes:

- Settings are applied first time any calculation takes place. After they may be modified for recalculations. If preferences are changed the changes will not appear until the calculator is reset (by clicking the "Clear" button on the component tab).

- Parts selected from a library search can be modified to produce new land patterns. If you can find a component close to the one you are looking for, all you have to do is modify the numbers that are different and select the "OK Button".

Land Pattern Tab for SM Non-Grid Array Devices

This tab is for viewing land pattern calculated dimensions. It also allows you to modify automatically generated names.

Land Pattern and Pad Stack names are generated automatically but you may change them prior to saving or creating the CAD land pattern (using the Land Pattern Wizard).

Note:
The automatic naming convention is not foolproof. It is possible for different components to produce identical names. This can happen for several reasons. For instance, if height is not a factor in the land pattern name so parts with identical footprints might have a different height component unless you intervene in the naming process. Also, the automatically generated name is typically based on certain component Nominal dimensions. Nominal dimensions are derived from averaging the min and max values so that two dimensions with the same nominal value might have vastly different min and max values which will likely result in different land patterns. When you save your part to the plb09 library a check will be made for duplicate names. If a duplicate name is encountered you will have the option to add the duplicate, overwrite the original or cancel the operation. For hints on how to deal with duplicate names see Help topic “Land Pattern Name Syntax” on page 128.

Default settings and environment are always applied to the first calculation of a new land pattern. After that, changed entries will remain until the “Reset” button is used to clear the component data fields and return the calculator to its default state.

For details on Preferences see the Help topic “Surface Mount Calculators, Non-Grid Array Pages” on page 49.

Calculator Settings Tab for SM Non-Grid Array Devices

This tab contains controls for entering design constraints that will effect how the land pattern will calculated. Entries on this tab apply specifically to the kind of land pattern being created, BGA, chip, QFP, etc. All of the values on this tab can be assigned default values by the user Preferences (See Help topic “Surface Mount Calculators, Non-Grid Array Pages” on page 49). Any setting may modified if the User radio button is checked.

The “(Global)” list of environments can contain multiple user specified environments and suffixes set up on the “Preferences, Global Settings” on page 36.

Global Settings Tab for SM Non-Grid Array Devices

This tab contains controls for entering design constraints that will effect how all land patterns will calculated. All of the values on this tab can be assigned default values by the user Preferences (See Help topic “Preferences, Global Settings” on page 36). Any setting may modified if the User radio button is checked.

Statistics Tab for SM Non-Grid Array Devices

This tab displays information about the how the calculation satisfied the environmental goals. See Help topic “Analyzing Results” on page 65.

Through-Hole Device Specific Tab Controls

The through-hole calculators have a number of variations that can be accessed from these tabs. These include groups of settings for the components, the land patterns and the calculator, as well as global settings.

Component Tab for TH Devices

This tab contains controls for the entry of component data. Each calculator has a "Demo" button which will load and calculate a sample part for you.

Transfer the component dimensional data from the manufacturer's data sheet in to the appropriate fields of the calculator's Component tab.

Optional Entries

When there is an optional entry the data field will be invisible or disabled until a box or button is checked indicating that the option is to be included. Optional entries, when left blank, will be automatically calculated and optimized to produce a value based on other component and settings data.

Note:
Optional entries may not occur in some component families. Optional Entries, when present, will be annotated with an asterisk (*). If left blank, their values will be calculated and filled in automatically.

- Lead Spacing and Mounted Height are optional for the Axial component family. Both values may be automatically calculated to compensate for lead bending, based on lead diameter, or overridden by user entered values so long as possible.
- Standoff is optional for the Radial component family. This value may be automatically calculated to compensate for lead bending, based on lead diameter, or overridden by a user entered value so long as possible.
- Gauge is optional for the Wire component family. Selecting a wire gauge will automatically produce a wire diameter. If a simple diameter is desired, it can be entered directly.

Click OK once all entries have been made. If any errors are detected a red indicator will flash in the field. Moving the cursor over the indicator will display a 'tool tip' with an explanation of the violation.

At this point a complete calculation will be performed using the default preferences for the Settings, Environment, Rules and Drafting variables. Entries for these variables are disabled unless the User radio button is checked. All land pattern dimensional fields will populate and the Viewer window will display the component and calculated land pattern.

Notes:

- Settings are applied first time any calculation takes place. After they may be modified for recalculations. If preferences are changed the changes will not appear until the calculator is reset (by clicking the 'Clear' button on the component tab).
- Parts selected from a library search can be modified to produce new land patterns. If you can find a component close to the one you are looking for, all you have to do is modify the numbers that are different and select the "OK Button".

Calculator Settings Tab for TH Devices

This tab contains controls for entering design constraints that will effect how the land pattern will calculated. Entries on this tab apply specifically to the kind of land pattern being created, axial, radial, DIP, etc. All of the values on this tab can be assigned default values by the user Preferences (See Help topic “Through-Hole Calculators, Device Pages” on page 52). Any setting may modified if the User radio button is checked.

Other controls are the same as described in Help topic “Surface Mount Device Non-Grid Array Specific Tab Controls” on page 62.

Note:
Default settings and fabrication level are always applied to the first calculation of a new land pattern. After that, changed entries will remain until the “Clear” button is used to clear the component data fields and return the calculator to its default state.

For details on Preferences see the Help topic “Through-Hole Calculators, Device Pages” on page 52.

Analyzing Results

Solder Joint Analysis is the process of evaluating the relationship between the terminal, which is the contact area of the lead, and the land pattern. Understanding these relationships is important to determining the proper setting for the calculators and obtaining the desired results from the land pattern generation processes.

The table of Solder Joint Analysis gives you important feedback about the relationship between land and terminal. The Calculator takes all component, process and environmental variables into account then reports the minimum and maximum Protrusions of the land for the Toe Fillet, Heel Fillet and Side Fillet or Periphery for terminals with no vertical dimension.

For any given land pattern protrusion or periphery three statistical values will be displayed: the Goal; the Minimum and the Maximum.

The Goal, of course, is the target you specify as the minimum acceptable value (this is the value assigned to the calculation when the Most, Nominal or Least environment is selected, or the manually entered values in the User environment).

Its recommended, if you are not familiar with Solder Joint integrity, to accept the defaults for Settings, Tolerances, and Environment Goals.

The Minimum is a calculated value that will result in a worst case scenario if the component is in it's extreme condition, the processes (fabrication and assembly) are at their maximum tolerance, including the specified rounding condition and any trimming required to spacing minimums. Note that positive or negative values have no significance. Its the relationship that's important. The closer this value is to the goal, the better. If its larger than the goal then excessive board space is being used. If its smaller than the goal then its reliability is lessened.

The Maximum is a calculated value that will result in the most favorable scenario if the component is in its nominal condition, the processes are held to minimum tolerance, including the specified rounding condition and any trimming required to spacing minimums.

The ability to calculate and display these values is the true value of the land pattern calculator for those who require the best reliability in their PCB applications.

Chapter 4
Library Manager

The Library Manager may be opened in either (or both) of two modes. Search only or Search and Edit. Edit mode is only available in licensed installations.

Use Model and Disclosure for Supplied Libraries

The PCBM-STARTER.plb09 library supplied with the software is for "Reference Only" and can contain Land Patterns with the same name as expected, with different manufacturers. This library was not intended for use with the Multi-Part Wizard and will create import errors/warnings when duplicate land pattern names are encountered.

The SM_MFR.plb09, CONNECTORS.plb09 and TH.plb09.plb09 contain parts that were manually built in a CAD tool. You cannot view these parts in the SailWind Land Pattern Creator, but you can build them using the Multi-Part Wizard one part at a time or batch create the entire library.

CAUTION:
Since these land patterns were manually built, Paizi claims no responsibility of their accuracy or completeness. Please use extra caution to check these library parts prior to adding them to your CAD library.

All PLB library files are supplied free as sample land patterns and to provide you with a large selection of Land Patterns to "harvest" as applicable and are for "Reference only". Component Links, descriptions and availability, etc, as with all components, is subject to change without notice. Please see our new Link-Checker to aid you in determining broken links in both supplied sample libraries and your created library file with On-Line-Links.

Mode Comparison

The Library Manager can be utilized in Search Mode or in Search and Edit Mode. Each mode has its own set of features. Understanding the differences in these features helps you to determine which mode you should use.

Search Mode

- Libraries can be loaded and it's contents browsed or searched. The contents of any given library may vary and may be structured to suit your needs.
- Land patterns can be opened and viewed in the calculator.
- Links to on-line or local archive data can be opened.

Search and Edit Mode (licensed installations only)

- All functions of Search are available.
- Find and Replace option is available.
- Libraries can be modified and saved - see Help topic “Edit Options” on page 72.
- Libraries can be verified - see Help topic “Verifying Land Patterns” on page 73.
- Libraries can be constructed or reconstructed - see Help topic “Running Multi-Part Wizard” on page 74.
- Columns may be dragged-and-dropped to create a new order. Click on any column header with the left mouse button and, while holding the button down drag the column to a new location. The library must be saved to preserve any column move changes.
- Column headers may be edited. Double-click on any column header with the left mouse button to enable column header editing.

Library Manager Toolbar Controls

The Library Manager toolbar contains a broad range of buttons that provide access to all of the Library Manager functions.

Figure 14. Library Manager Toolbar

Table 7. Library Manager Toolbar Controls

Name	Description
Close	Closes the Library Manager window.
Save (not available in search mode)	Pull down options include: • **Save**—the current library is saved by it's current name. • **Save As**—the current library is saved by a user defined name.
Find	Opens the 'Find/Replace' dialog box that can be used to locate and, if desired, replace string values in the current library.
Categories	Displays a list of general collections of land patterns that exist in the current library. This list is not normally visible unless specifically activated by this button. Selecting an item from the list of categories will display, in the library table, a complete list of land patterns with the characteristics of that category.
Show All	Displays the entire contents of a library. This is usefully for displaying a library after invoking Categories or Find > List All.
Undo	Returns the library to the state prior to the last edit.
Redo	Returns the library to the state after to the last undo.
View (on screen)	Recalculates and displays a selected land pattern using the calculator/viewer. **Note:** A PLB library can contain both land patterns that were created with the software and those that were manually created and added to a library file for the sole purpose of documentation. Only land patterns created by the calculator can be viewed on screen.
On-Line-Link	If a link to on-line data is available, this button will be enabled and may be used to launch an Internet browser to the link provided for the selected land pattern.
Local Archive	If a link to local access data is available, this button will be enabled and may be used to open the data file provided for the selected land pattern.
Tools (not available in search mode)	Pull down menu options include: • **Verify**—the current library is recalculated using the current user preferences and compared to reference values contained in the current library file (see Help topic “Verifying Land Patterns” on page 73). • **Wizard**—the current library is recalculated subject to certain user options (see Help topic “Running Multi-Part Wizard” on page 74). • **Link Checker**—this tool may be periodically to verify the validity of On-Line-Link data web sites (see Help topic “Running the Link Checker” on page 79).

Context Menu

Click the right-mouse button while in the library data table to display a list of options that are, generally, associated with editing functions.

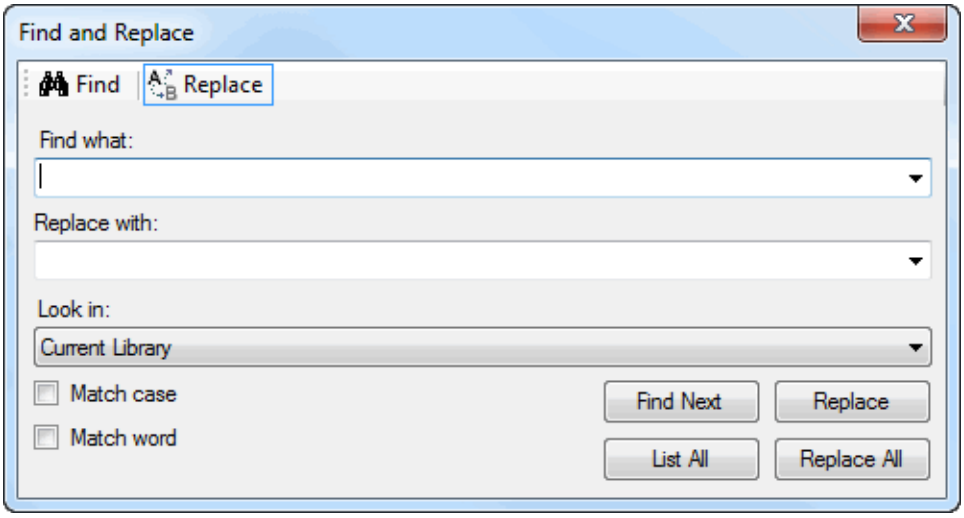
Table 8. Library Data Table Context Menu Items

Name	Description
View (Same as toolbar functions)	• On Screen • On-Line-Link • Local Archive
New	• Row — adds a new row to the library data table. • Column — adds a new column to the library data table.
Copy	Copies the contents of a cell (if a cell is selected and its contents are highlighted) or row (if the row if the entire row is selected by clicking the row header) to the Windows clipboard. Multiple rows may be selected and copied.
Paste	Pastes the contents of the Windows clipboard into a cell (if a cell is selected) or the library data table (if no cell is selected). Single or multiple rows may be copied.
Autosize Columns	Adjusts the widths of all columns to display the entire contents the largest cell in that column.
Delete	• Row — removes all highlighted rows. • Column — removes the column containing the selected cell.
Tools (same as toolbar functions)	• Verify • Wizard • Link Checker

Find/Replace Dialog Box

To access: Library Manager toolbar > **Find** button

This dialog opens in Find-only mode when in the Search mode and both Find and Replace when in the Editor mode.



Objects

Name	Description
Find/Replace (mode buttons)	Determines the operating mode. In 'Find' there are no replace options visible.
Find what box	Defines the string to find (see following Match Case and Match Word).
Replace with box	Defines the replacement string for any found strings. Not available in Find mode.
Look in list	Defines the area or areas in which to perform the Find operation.
Match Case check box	If checked, the string to find case must exactly match the search string.
Match word check box	If checked, the string to find must exactly match the search string (case withstanding). If unchecked, the string to find need only contain an instance of the search string.
Find Next	When the button is clicked cursor will advance to the next found string.
List All	When the button is clicked only lines containing a found string will be visible. In those lines, cells containing the found string will be highlighted.
Replace	When the button is clicked selected found strings will be replaced by the replace string. Not available in Find mode.

Name	Description
Replace All	When the button is clicked all found strings will be replaced by the replace string. Not available in Find mode.

Edit Options

The Library Editor allows you to edit libraries while maintaining the file format required for the program. Library files may be edited to manually modify existing content, add or remove content, or create new libraries.

Opening two Editor sessions allows the contents of any library to be copied to another.

To open an existing library, select it from the Library pull down list. If the library doesn't appear in the list click the Add Lib Button to browse for the library. Documentation for all library components is, by default, contained in files with a ".plb09" file extension in a program sub directory called CADData. Upon selection, the library will load into the library display table.

To create a new library select New Library from the Library pull down list.

All editing options are available in a right-mouse-button (RMB) pop up menu.

Individual Cells may be edited by either clicking in empty cells or highlighting the contents of a cell and entering new values.

Find and Replace can be performed to edit the contents of any cell or collection of cells.

The contents of an individual cell may be copied to another cell by highlighting the contents of a cell followed by RMB copy. Move the cursor to the destination cell and RMB paste.

Individual or multiple rows can be copied by clicking on the row header (or shift-click for headers) followed by RMB copy. RMB paste will add the copied rows to the destination library.

Columns may be added, deleted or dragged and dropped to change the left-to-right order.

Note:
Generally, the names of any added columns are at your discretion, however there are some reserved column headings:

- **Land Pattern**

  This column contains the names for all land patterns in the library and cannot be deleted.

- **Description**

  This column contains a general description of the land pattern. Descriptions are automatically generated by the calculator.

- **On-Line-Link**

  This column is reserved for Internet links to land pattern data.

- **Local Archive**

  This column is reserved for links to land pattern data that may be stored on a local storage device such a hard drive.

Undo and Redo is available for up to 32 of the most recent edits.

Modifications that can't be undone include copy-paste operations where the source and destination have different column content. When this condition occurs, the destination columns are re mapped to include any non-existent source columns. A message will allow cancellation of the operation when this condition exists.

When finished with changes click on the **Save** toolbar button. You will then be prompted for a file name. If you keep the same name as the original file, a backup file of the original will be created with a 'bak' file extension.

CAUTION:
You should not simultaneously edit parts in a particular library while using a wizard to create parts in the same library. Doing so runs the risk of overwriting the work of one with the work of the other. Additionally, any parts added by the wizard will not be observed in the editor until the library is reloaded.

Verifying Land Patterns

The Verify feature is used to check land patterns that were created with a previous version of software against those same land patterns as created with the current version. This is useful in identifying changes to land pattern calculations resulting from modifications to standard calculation formulae, user preference changes or possible anomalies in the software.

Verify is performed on every land pattern that is opened in the Calculator. If a notice is generated, the 'Notice' toolbar button will flash. Clicking the button will open a message box displaying information about the notice. Not all notices indicate problems. Many are simply advisory.

A Verification Report can be generated by the Library Manager Editor and may be performed on land patterns or complete libraries that have been created using the Calculator.

Procedure

1. Load the library on which to perform Verify. For more information on selecting and loading libraries see the Help Contents section for “Edit Options” on page 72.
2. Select the row or rows containing the land patterns to verify by clicking on the row headers to select the full row.
3. Click on the **Tools** toolbar button and select the pull down option **Verify**.
4. Verify is performed and a report is generated containing a list of discrepancies, if any.

Results

Verify performs a calculation using all the same component data and settings that were used to create the original land pattern. It then compares the new calculation to the original calculation using information saved from the original calculation in the plb09 library file. The types of discrepancies that can be found with Verify are as follows:

- Differences that might result from a change to the user selected preferences or default settings.
- Differences in a that might result from a change to a program calculation method or formula.

Verify checks basic land pattern dimensions such as land lengths, widths, spacings, pad stack names, and the fundamental settings that were used in performing the calculation for those dimensions.

CAUTION:
Verify does not check everything!

The following are some examples of potential land pattern differences that may or may not be detected by verify:

Verify can tell you that a land pattern was created with 'Default' or 'User' Drafting or Rule preferences but that currently those same preferences are set otherwise. Other than that it will not alert you to specific differences in the actual settings. For example, it does not detect differences between original silkscreen line width and current line width but will alert you if the original silkscreen line width was set to the Default line width but the current line width is set to the User line width.

Verify will alert you to differences between settings that directly effect the land pattern functional characteristics. For example, if the current default solder mask oversize doesn't match the original a notice will be generated.

Verify does not compare a library's environment to the currently selected environment. For example, if the current default preference is to create land patterns in a nominal environment but the library being verified is in the least environment, the library will be verified against the current default least environment regardless of the selected default environment.

Running Multi-Part Wizard

The Multi-Part feature is used to recreate a group of land patterns or entire libraries. This is useful for quickly converting libraries from one environment to another or rebuilding a group of land patterns to correct or modify one or more user settings. Multi-Part may be used to update the plb library data or with the Land Pattern Wizard to create land patterns for the various available CAD Tools.

CAUTION:
Before continuing, please read “Use Model and Disclosure for Supplied Libraries” on page 67.

Restrictions and Limitations

This may only be performed on land patterns that have been created using the Calculator.

Procedure

1. Load the library on which to perform the Multi-Part feature. For more information on selecting and loading libraries see the Help Contents section for “Edit Options” on page 72.
2. Select the row or rows containing the land patterns to create using Multi-Part.

   CAUTION:
   The CAD Data (plb09) file you select will be completely overwritten!

3. On the Library Editor toolbar click **Tools** and then click **Wizard**.

   Alternative: In the Land Pattern list, right-click, point to Tools and click Wizard.

4. Make your required settings in the “Multi-Part Wizard Dialog Box” on page 76.

5. Once the Wizard setup has been completed, CAD output options may be set in the “Land Pattern Wizard Dialog Box” on page 82.

Results

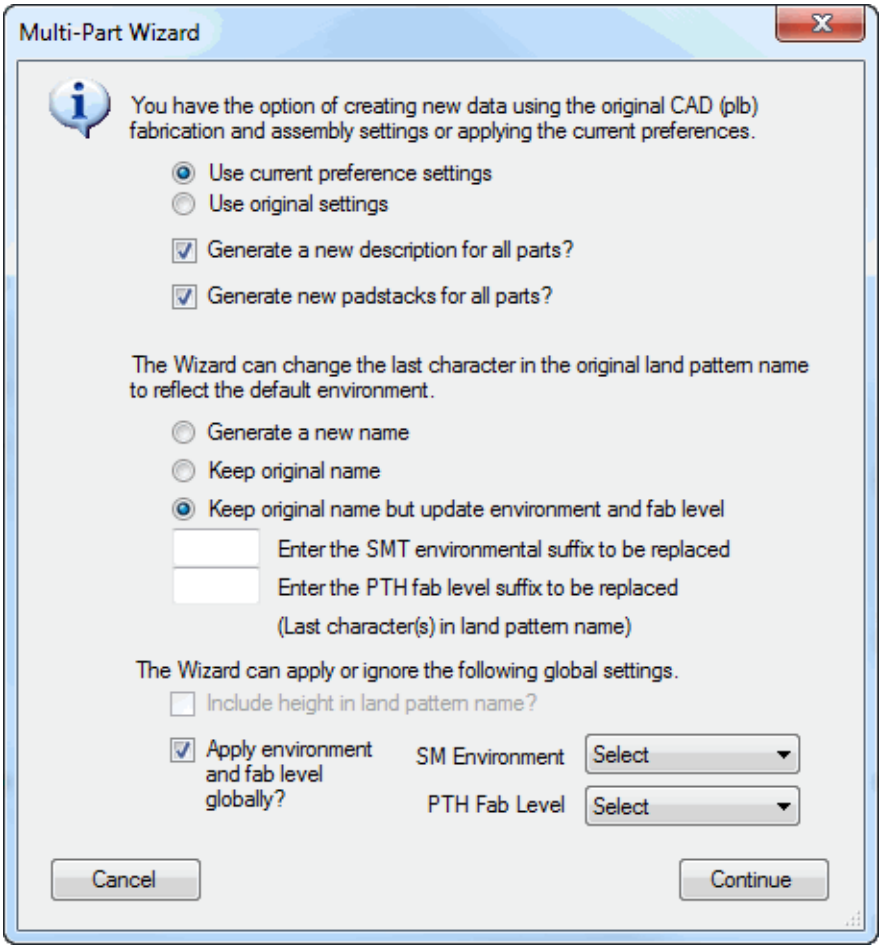
The “Multi-Part Wizard Report.txt Log File” on page 78 is automatically generated and displayed in your text editor.

Multi-Part Wizard Dialog Box

To access:

- On the Library Editor toolbar click **Tools** and then click **Wizard**.
- In the Land Pattern list, right-click, point to Tools and click Wizard.

The Multi-Part Wizard dialog box is used to set up the Multi-Part Wizard. Carefully review the content in this topic to assure that you get the results you expect! Settings include fabrication and assembly tolerances, round offs, spacings, and environment.



Objects

Table 9. Multi-Part Wizard Dialog Box Objects

Name	Description
Use current preference settings	Use the current settings as set in user preferences. When you select this option you will also be able to modify environment variables.

Table 9. Multi-Part Wizard Dialog Box Objects (continued)

Name	Description
	Restriction: User settings include all preference settings except those for SMD Environment and PTH Fabrication Level. See the "Apply environment and fab level globally" description.
Use original settings	Recreate land patterns using the settings that were originally applied when the library was last created. When you select this option you will not be able to modify any environment variables.
Generate a new description for all parts	Select to generate a new description; when unchecked, the description from the original library file will saved. **CAUTION:** Do not check this box if you manually edited or created descriptions for your land patterns and you do not want to loose those descriptions!
Generate new padstacks for all parts	Select to generate new pad stack names for selected land patterns; when unchecked, the pad stack names from the original library will be saved. **CAUTION:** Do not check this box if you manually edited or created pad stack names for your land patterns and you do not want to loose those names!
Land Pattern Name Options **Restriction:** All sections below are only available if you select the “Use current preference settings” option.	
Generate a new name	Select to create a new land pattern name based on the IPC standard naming convention. For details on land pattern names see the Help topic IPC-7351B Naming Convention for Standard SMT Land Patterns. **CAUTION:** Do not check this box if you manually edited or created names for your land patterns and you do not want to loose those names!
Keep original name	Select to apply the land pattern name from the original library. **Note:** If you manually edited or created names for your land patterns and you do not want to loose those names check this box unless your are recreating the library in a different environment - see next item!
Keep original name but update environment and fab level	Select to apply the land pattern name from the original library but replace the original environment suffix with default suffix as defined by the user preferences. In the space provided, enter the suffix that will be replaced. For example, if converting a nominal environment library to a least environment, you would enter the character 'N' indicating you wish the environmental suffix 'N' to be replaced with default suffix for the least environment (SOIC127P1032X264-16N would become SOIC127P1032X264-16L).
Global Settings	

Table 9. Multi-Part Wizard Dialog Box Objects (continued)

Name	Description
Include height in land pattern name	Select to add the height attribute to automatically generated land pattern names. **Restriction:** This check box is not available if you select “Keep original name.”
Apply environment and fab level globally	Select to apply the chosen environment to all selected land patterns where applicable. **Restriction:** This check box is only available if you select “Generate a new name.”

Multi-Part Wizard Report.txt Log File

The *Multi-Part Wizard Report.txt* log file is automatically generated and displayed in Notepad each time the Multi-Part Wizard is executed.

Log File Format

```
LP Suite Multi-Part Wizard Report

LP Program Version: <program version>
<time stamp>

<number> Land patterns were added to plb file:
<.plb file path>

<number> Land patterns were added to library:
<folder path>

<error messages>
```

Error Messages

The <error messages> (if any) can be one from the list below:

- Environmental suffix not found in land pattern <land pattern name>
- Unable to Build <land pattern name> - calculation failed
- Unable to Build <land pattern name> - bad build data
- Unable to Build <land pattern name> - bad component data

Sample Log File

```
LP Suite Multi-Part Wizard Report

LP Program Version: 10.4
9/5/2011 11:53:53 AM

4 Land patterns were added to plb file:
D:\MyLPW_Libs\LPWizard\CAD Data\Wizard.plb09.

4 Land patterns were added to library:
C:\Users\gmueller\Desktop\NewParts\NewParts.

Unable to Build BGA100C100P10X10_1100X1100X200 - bad build data
```

Running the Link Checker

Internet links frequently change or become obsolete. The Link Check feature may be used to verify the integrity of links to manufacturer's on-line data sheets.

A Link Integrity Check may be performed by the Library Manager Editor on land patterns or complete libraries that have been created using the Calculator and that contain the On-Line-Link attribute.

Restrictions and Limitations

The Link Check routine is necessarily slow and no other SailWind Land Pattern Creator Functions may be used when it is running. The rate at which links are checked is dependent on a number of variables but, primarily, on the speed of the Internet connection. Expect to wait about 1-2 seconds for each link you wish to check. The operation may be canceled at any time.

Procedure

1. Load the library on which to perform Link-Check. For more information on selecting and loading libraries see the Help Contents section for “Edit Options” on page 72.
2. Select the row or rows containing the Internet links to verify by clicking on the row headers to select the full row.
3. Click on the **Tools** toolbar button and select the pull down option “Check-On-Line Links.”
4. Checks are performed. Selected rows with valid links are highlighted in green, rows with invalid links, if any, are high-lighted in red. Highlighting is removed when the currently selected or a new library file is loaded.

Chapter 5
Land Pattern Wizard

The land pattern wizard is used to generate the land patterns for your CAD tool. Choose a particular component type, enter component dimensions, specify the rules file you would like to use and a customized land pattern is generated that is ready to use in your CAD tool.

Preferences are unique for each CAD tool and displayed according to the selected Librarian CAD tool. Generally, these preferences will include, where required or supported by the individual tool:

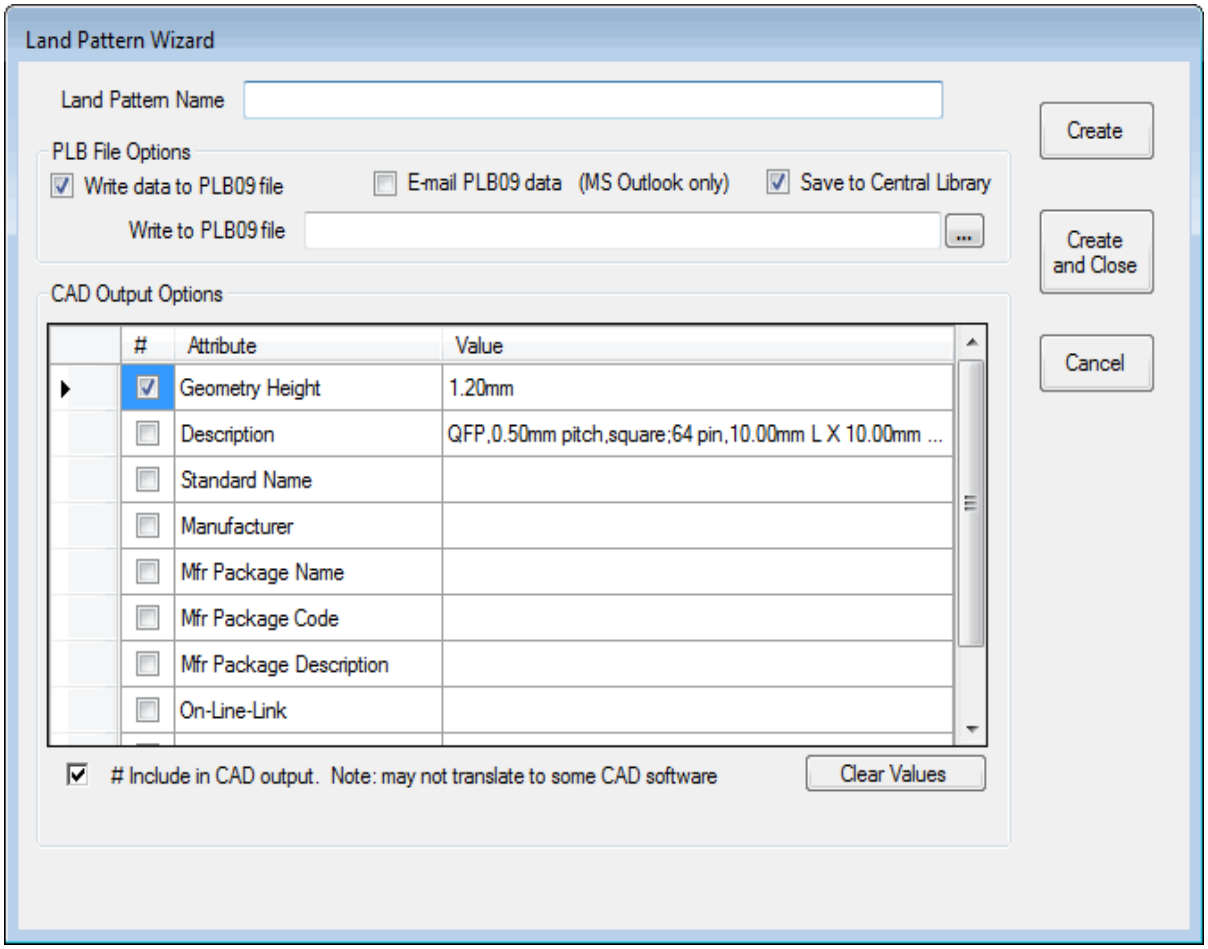
- An output directory
- An output software version
- Available user defined options
- Layer definitions

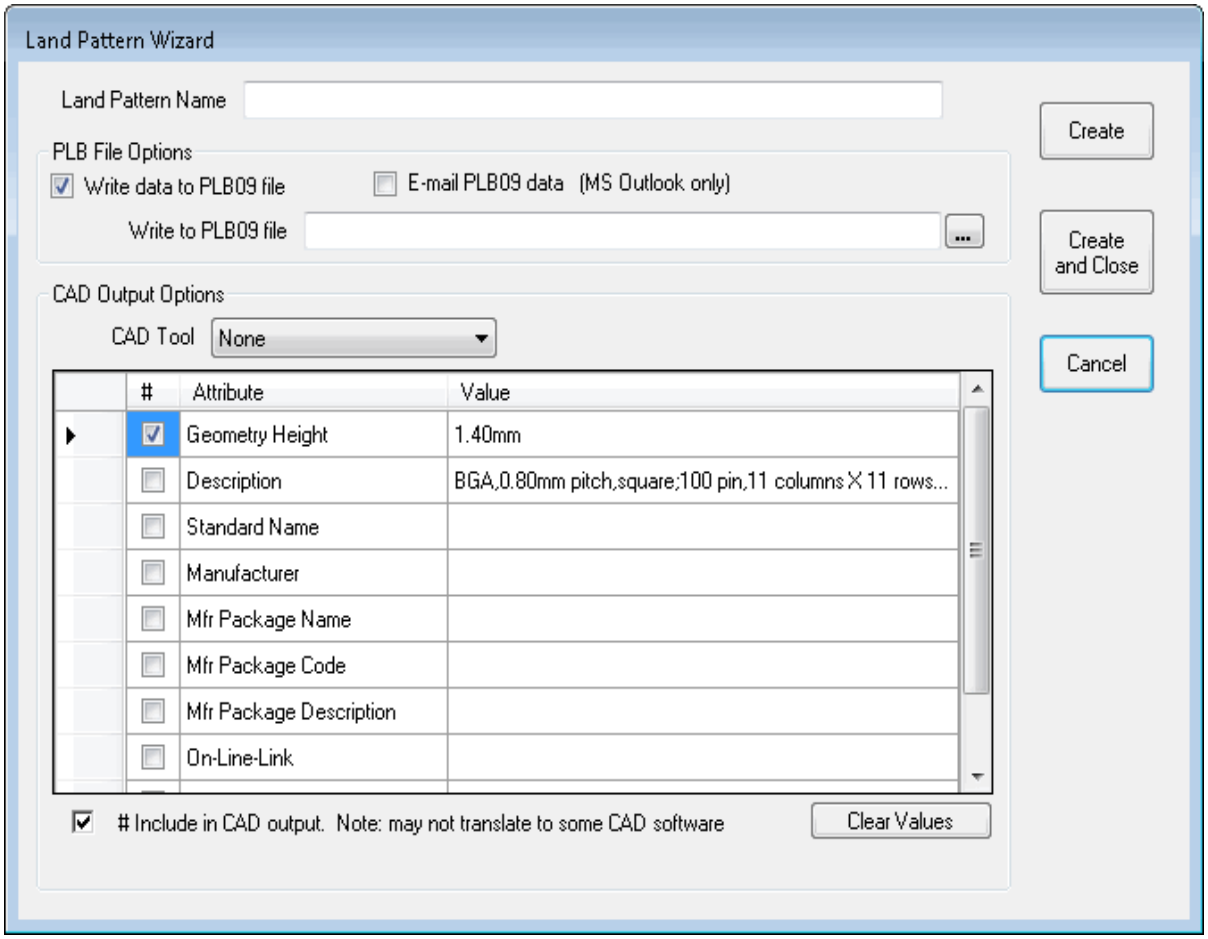
Land Pattern Wizard Dialog Box

To access: Calculator view > **Wizard** button

The Land Pattern Wizard dialog box allows you to specify the land pattern name, PLB file options, CAD Output options and to initiate the creation process. The Wizard can be entered from both the Calculator and Library Manager.

Figure 15. Land Pattern Wizard Dialog Box Land Pattern Wizard Dialog Box





Objects

Table 10. Land Pattern Wizard Dialog Box Objects

Field	Description
Write data to PLB file checkbox	Select to save the calculated land pattern to a plb file.
E-mail PLB Data checkbox	Select to mail the calculated land pattern as a plb file. The file is named for the land pattern and contains the single calculated land pattern, attached to an e-mail that may be sent to another party for review. **Restriction:** This feature is only functional when Microsoft Outlook is installed.
Save to Central Library	Saves the exported file directly into the Central Library.
Save output to PLB09 file name	This is the file name that will be saved when a land pattern (or land patterns in the case of the Multi-Part wizard) are generated.
CAD Tool	This pull-down menu may be used to select a default CAD format in which to construct the land pattern. Only CAD formats that are

Table 10. Land Pattern Wizard Dialog Box Objects (continued)

Field	Description
	licensed for the installation will be displayed. If 'none' is displayed, no CAD output will occur. **Tip** This feature is irrelevant if no CAD tools are licensed.
Include in CAD output	All library attributes are displayed in a list. Each of these attributes, checked or not, will be included in the output to the PLB file. If checked, attributes will be included in the CAD output format permitting (not all CAD formats support user defined attributes). In normal calculations, attributes listed here are derived from the user preferences for Library Manager data columns. If the Wizard is entered immediately after selecting a land pattern from a library, the attributes from that library are displayed. **Note:** This feature is irrelevant if no CAD tool is selected.

SailWind Layout Preference

In the Preferences dialog box, you can specify the details of how SailWind-compatible land patterns will be generated.

Figure 16. Preferences Dialog Box, SailWind Layout Output Defaults

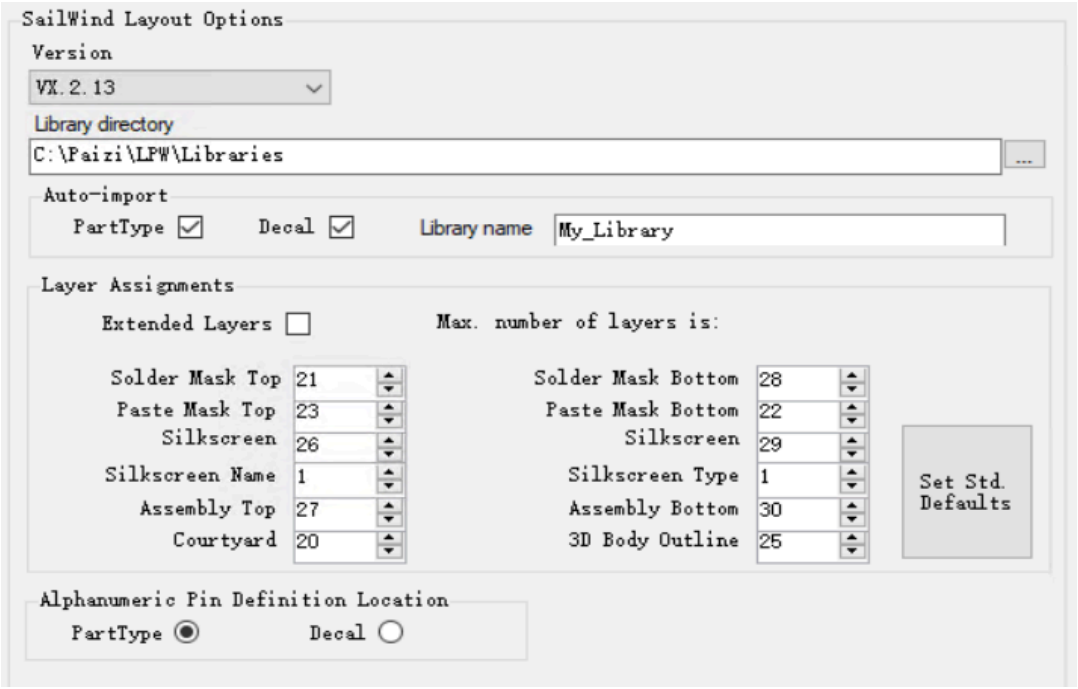


Table 11. Preferences Dialog Box, SailWind Layout Output Content

Name	Description
Version	Set the version of SailWind software you want for output compatibility.
Library Directory	Browse for a directory where you want to place the files it creates.
Auto Import area	Check either PartType, Decal or both if you want to automatically import the output. Enter the name of the SailWind Layout library where you want to import the new parts. **Restriction:** This area is not available when using the SailWind Layout ASCII output.
Layer Assignments area	Assign the layer numbers you want associated with the various elements of the land pattern (silkscreen, solder mask, etc.)
Alphanumeric Pin Definition Location	Part Type — A numeric decal is created, and the alphanumeric pin numbers in the part type are mapped to the numeric decal numbers on the Pin Mapping tab in the part type. Decal — An alphanumeric decal is created to match the alphanumeric pin numbers in the part type.

SailWind Layout Part Type and Decal Names

Normally, Part Type and Decal names assume the same name as the generated land pattern. User defined names may be substituted.

Use the following method: Create two new SailWind library attributes, "PartType" and "Decal", and Include them in the CAD output (See Help topic “Wizard, Options Page” on page 54). Before creating the land pattern from the Wizard, fill in the names to use for the Part Type and Decal and select the 'Include in CAD output' check box. When this method is used, if neither the user-defined Part Type name or Decal name is the same as the land pattern name, a new Part Type attribute for 'Land Pattern Name' is automatically added as a reference to the original name.

Once the options have been set, click the **Create** or **Create and Close** button to begin the translation process.

Tip
SailWind Layout output is generated in Millimeters and Mils units only.

Importing Files into SailWind Layout

There are 3 methods to import files into a SailWind Layout library. The software creates Part Type (.p) and Decal (.d) files that can be imported from the SailWind Layout design space, imported using the SailWind Library Manager, or automatically using a direct interface to SailWind Layout.

When translation is complete, the SailWind library is created if it doesn't exist, and the new parts added.

Choose one of the following:

- Importing into a SailWind Layout Library Automatically — If you have both SailWind Land Pattern Creator and SailWind Layout on your computer, this method is automatic and the easiest.
- Importing Manually Using the SailWind Library Manager — If you don't have both SailWind Land Pattern Creator and SailWind Layout on your computer, this method can be used to create SailWind *.p* & *.d* files for import into a library using the SailWind Library Manager.
- Importing to a SailWind Library Using an ASCII File into a Design — If you don't have both SailWind Land Pattern Creator and SailWind Layout on your computer, this method can be used to create a SailWind ASCII file for import into a blank design where you can save the part types and decals from the design area into the library.

Importing into a SailWind Layout Library Automatically

Transfer Calculator land pattern data automatically into a SailWind library name of your choice.

Restrictions and Limitations

- The included extended library files; *PCBM-STARTER.plb09*, *SM_MFR.plb09*, *CONNECTORS.plb09* and *TH.plb09* include special user defined attributes. In order to include all these attributes in your SailWind Land Patterns they must be defined in SailWind using the Attribute Dictionary or you will receive expected Warnings. If you do not wish to import these attributes you can simply ignore the Warnings. You should review the supplied *plb09 file attributes included to determine their desired use.

Prerequisites

- A session of SailWind needs to be open in order for the data to be automatically passed into the SailWind library.
- Ensure that neither the Part Type or Decal editors are open.
- Do not attempt automatic import when multiple sessions of the SailWind software are running.

Procedure

1. Choose one of the following:
   - Use the single part wizard within the Calculator:
     i. On the toolbar, click **Wizard**.
   - Use the multi-part wizard within the Library Editor:

i. Open SailWind Land Pattern Creator and use the **Library** > **Editor** command to choose the library (*.*plb09*) file that contains the Land Pattern you want to create.

ii. Use the gray box to the left of the Land Pattern name to select Land Patterns.

iii. Next either right-click and click the **Tool > Wizard** popup menu item or click from the menu bar to launch the "Multi-Part Wizard Dialog Box" on page 76.

iv. Provide the desired settings and click **Continue** to launch the Wizard.

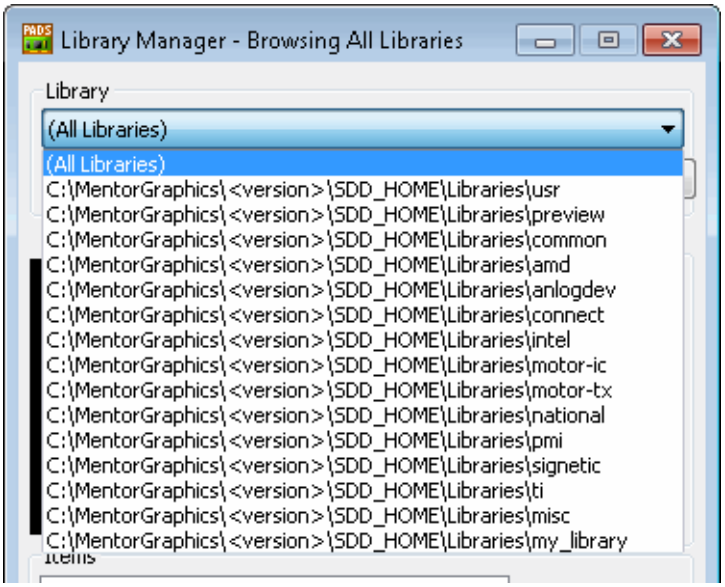
2. In the CAD Tool list, select **SailWind Layout**.
3. Match the SailWind output Version to your opened SailWind Layout version.

CAUTION:
Importing incompatible versions can result in library corruption. A warning message appears if the files are not compatible with the running SailWind software version.

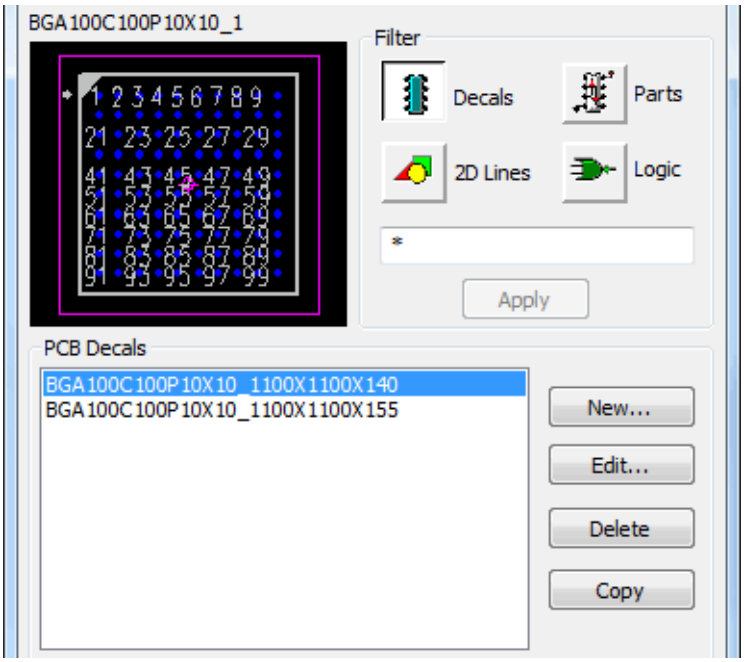
4. In the Auto-import area, select both Part Type and Decal check boxes.
5. Provide the library directory and library name as required.
6. Click either **Create** or **Create and Close**.

   When complete, a prompt will display saved-to information:

7. In SailWind Layout, open the Library Manager and select the created library in the Library list - (added to end of the list by default).



8. The Decal(s) and Part Type(s) will appear in Library Manager appear, like below, and are available for use, editing, exporting, etc.



Importing Manually Using the SailWind Library Manager

Create SailWind .p & .d files and import them into a library using the SailWind Library Manager. This option does not require a session of SailWind to generate the SailWind CAD Output.

Restrictions and Limitations

- The included extended library files; PCBM-STARTER.plb09, SM_MFR.plb09, CONNECTORS.plb09 and TH.plb09 include special user defined attributes. In order to include all these attributes in your SailWind Land Patterns they must be defined in SailWind using the Attribute Dictionary or you will receive expected Warnings. If you do not wish to import these attributes you can simply ignore the Warnings. You should review the supplied *plb09 file attributes included to determine their desired use.

Procedure

1. Choose one of the following:

   - Use the single part wizard within the Calculator:

     i. On the toolbar, click **Wizard**.

   - Use the multi-part wizard within the Library Editor:

i. Open SailWind Land Pattern Creator and use the **Library > Editor** command to choose the library (*.plb09) file that contains the Land Pattern you want to create.

ii. Use the gray box to the left of the Land Pattern name to select Land Patterns.

iii. Next, either right-click and click the **Tools > Wizard** popup menu item or click from the menu bar to launch the "Multi-Part Wizard Dialog Box" on page 76.

iv. Provide the desired settings and click **Continue** to launch the Wizard.

2. In the CAD Tool list, select **SailWind Layout**.
3. Match the SailWind output Version to your opened SailWind Layout version.

CAUTION:
Importing incompatible versions can result in library corruption. A warning message appears if the files are not compatible with the running SailWind software version.

4. In the Auto-import area, clear both the Part Type and Decal check boxes.
5. Provide the desired library directory location for saving the .p & .d files.
6. Click either **Create** or **Create and Close**.
7. Read the Caution prompt and continue as desired.
8. You can do either of the following:

- Navigate to the supplied Library directory to retrieve the .p & .d. files
- Import .p & .d files into SailWind by opening the SailWind Library Manager, selecting Decals and importing the .d file and selecting Parts and importing the .p file.

Importing to a SailWind Library Using an ASCII File into a Design

Create a SailWind ASCII file, import the ASCII data into a blank design and save the part types and decals from the design area into the library.

You can use SailWind ASCII for Single Part or Multi-Part Cad Output. When using SailWind ASCII Cad Output, a **.asc* (ASCII) file is always created that contains all of the part type and decal definitions. The default file name when using the Multi-Part Wizard is "NewParts", but you can rename that file to whatever name you chose. After the **.asc* file is Imported, you save the part types and decals to a library. Once in the SailWind Library, they are ready for use in a new design.

Restrictions and Limitations

- The included extended library files; *PCBM-STARTER.plb09*, *SM_MFR.plb09*, *CONNECTORS.plb09* and *TH.plb09* include special user defined attributes. In order to include all these attributes in your SailWind Land Patterns they must be defined in SailWind using

the Attribute Dictionary or you will receive expected Warnings. If you do not wish to import these attributes you can simply ignore the Warnings. You should review the supplied *plb09 file attributes included to determine their desired use.

Procedure

1. Choose one of the following:

   - Use the single part wizard within the Calculator:

     i. On the toolbar, click **Wizard**.

   - Use the multi-part wizard within the Library Editor:

     i. Open SailWind Land Pattern Creator and use the **Library** > **Editor** command to choose the library (*.plb09) file that contains the Land Pattern you want to create.

     ii. Use the gray box to the left of the Land Pattern name to select Land Patterns.

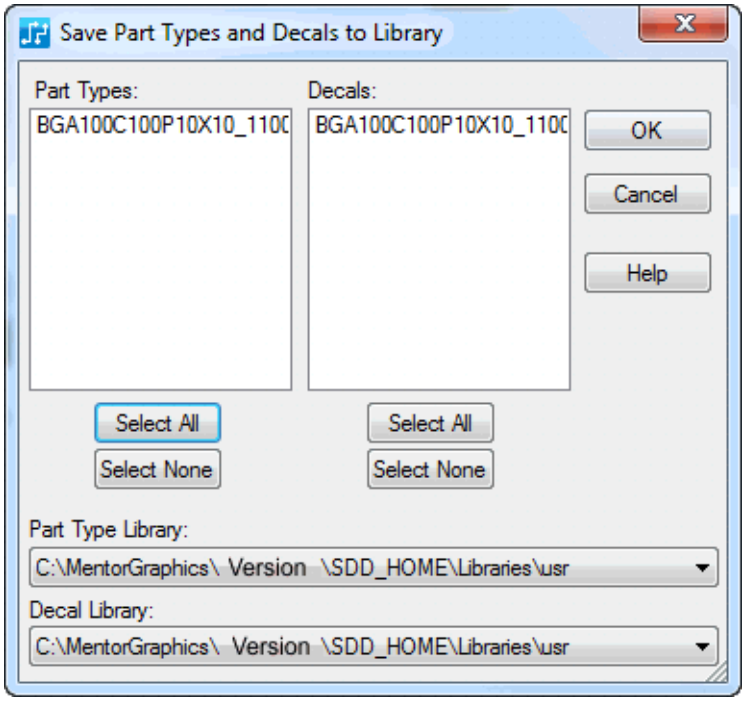
     iii. Next, either right-click and click the**Tools > Wizard** popup menu item or click **Tools > Wizard** from the menu bar to launch the “Multi-Part Wizard Dialog Box” on page 76.

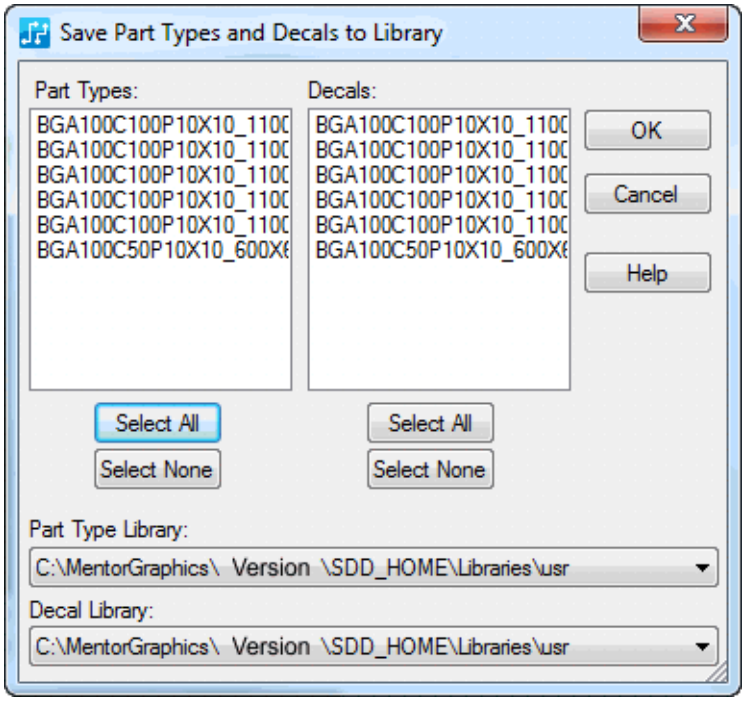
     iv. Provide the desired settings and click **Continue** to launch the Wizard.

2. In the CAD Tool list, select “SailWind Layout ASCII.”
3. Match the SailWind output Version to your opened SailWind Layout version.

CAUTION:
Importing incompatible versions can result in library corruption. A warning message appears if the files are not compatible with the running SailWind software version.

4. Provide the desired directory location for saving the *.asc* file.
5. Click either **Create** or **Create and Close** as desired.
6. In SailWind Layout, in a new blank design, click the **File > Import > ASCII** menu item and import the *.asc* file into the design.
7. Once the library parts are imported into a the blank design, right-click and click **Select Components**. The parts are imported, stacked on the origin, like when you import a netlist.
8. Select the component or right-click and click **Select All**.
9. Right-click and click **Save to Library**.
10. In the Save Part Types and Decals to Library dialog box, click **Select All** for both Part Types and Decals.


Save Part Types and Decals to Library
Part Types:
Decals:
BGA100C100P10X10_110(
BGA100C100P10X10_110(
OK
Cancel
Help
Select All
Select None
Select All
Select None
Part Type Library:
C:\MentorGraphics\ Version \SDD_HOME\Libraries\usr
Decal Library:
C:\MentorGraphics\ Version \SDD_HOME\Libraries\usr


Save Part Types and Decals to Library
Part Types:
Decals:
BGA100C100P10X10_110(
BGA100C100P10X10_110(
BGA100C100P10X10_110(
BGA100C100P10X10_110(
BGA100C100P10X10_110(
BGA100C50P10X10_600X(
BGA100C100P10X10_110(
BGA100C100P10X10_110(
BGA100C100P10X10_110(
BGA100C100P10X10_110(
BGA100C100P10X10_110(
BGA100C50P10X10_600X(
OK
Cancel
Help
Select All
Select None
Select All
Select None
Part Type Library:
C:\MentorGraphics\ Version \SDD_HOME\Libraries\usr
Decal Library:
C:\MentorGraphics\ Version \SDD_HOME\Libraries\usr

11. **Select** the desired save-to library locations and click **OK**.
12. The land patterns are now available in the chosen library and ready for use.

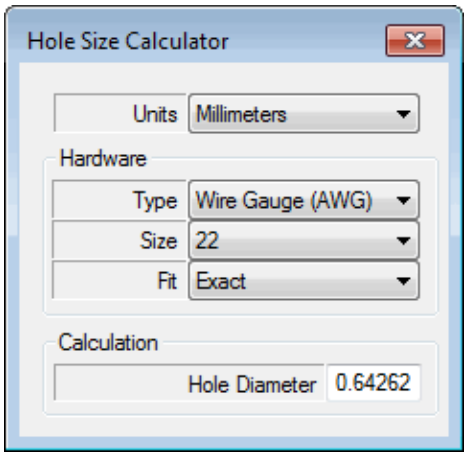
Chapter 6
Hole and Via Calculators

Calculators are also available for non-part objects that you use in your design.These include calculators for vias and generic holes such as mounting and tooling holes.

Hole Size Calculator

The Hole Size Calculator provides a quick and easy way to determine the size of a hole based on what goes into it.

Figure 17. Hole Size Calculator



Type options include wire, screw or user defined.

- **Wire Gauge** — the **Size** list is populated with a variety of standard AWG wire sizes.
- **ANSI Screw** — the **Size** list is populated with a variety of standard ANSI screw sizes.
- **Metric Screw** — the **Size** list is populated with a variety of standard Metric screw sizes.
- **User** — a diameter size can be entered by typing in a value for a single size. A square or rectangular size can be specified by typing in two values (the first for the length, the second for the width) separated by a comma.

The hole size is calculated using one of six Fit options.

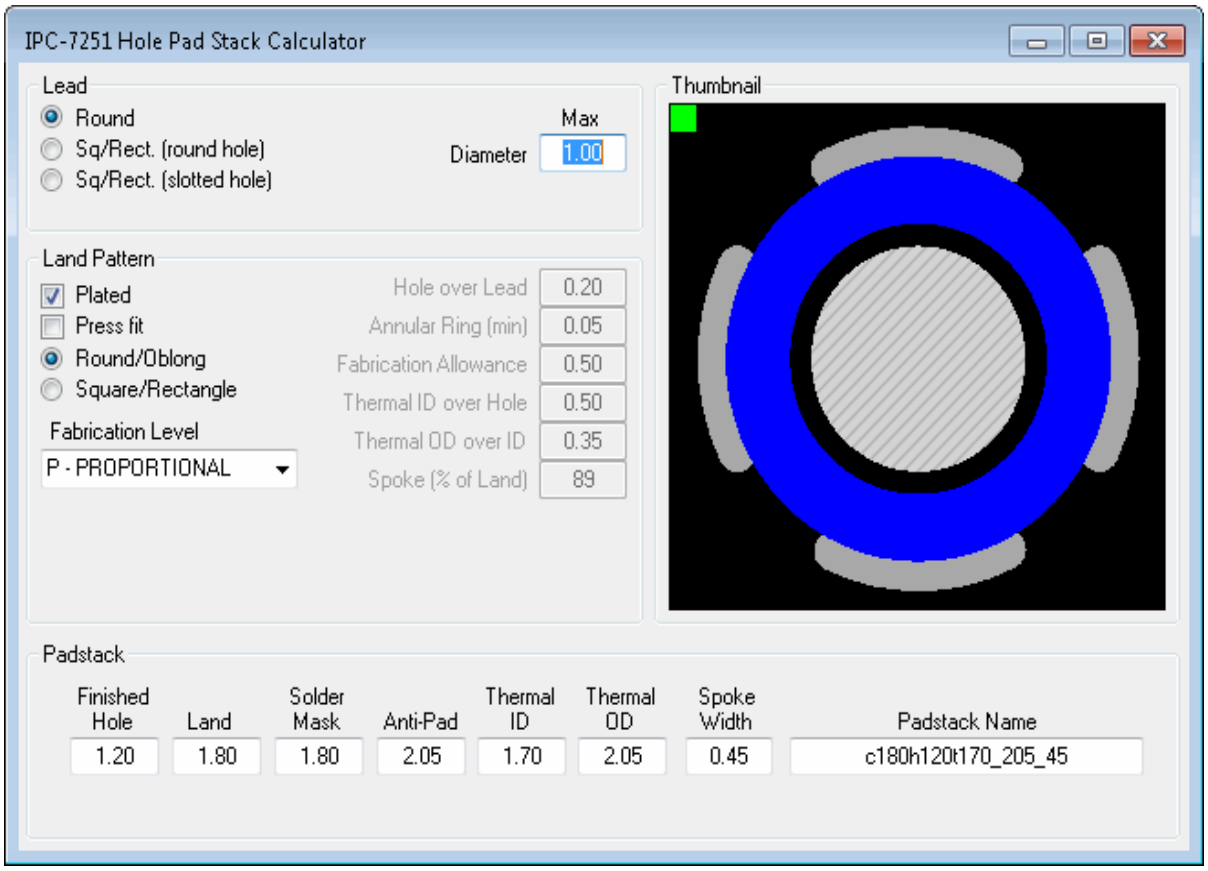
- **Exact** — the hole will fit the target size without any clearance.
- **IPC Most** — the hole will fit the target size using IPC fabrication level A.
- **IPC Nominal** — the hole will fit the target size using IPC fabrication level B.
- **IPC Least** — the hole will fit the target size using IPC fabrication level C.
- **ANSI Tight** — the hole will fit the target size using the standard for an ANSI Tight clearance.
- **ANSI Loose** — the hole will fit the target size using the standard for an ANSI Loose clearance.

The calculated **Hole Diameter** is displayed. When used in conjunction with a land pattern calculator that allows a hole size to be specified (Shrouded Header Hole Size (d) for example) this value can be dragged and dropped in the Calculator.

Hole Pad Stack Calculator

The Hole Pad Stack Calculator provides a quick and easy way to determine the various elements for a hole and each layer element.

Figure 18. Hole Pad Stack Calculator



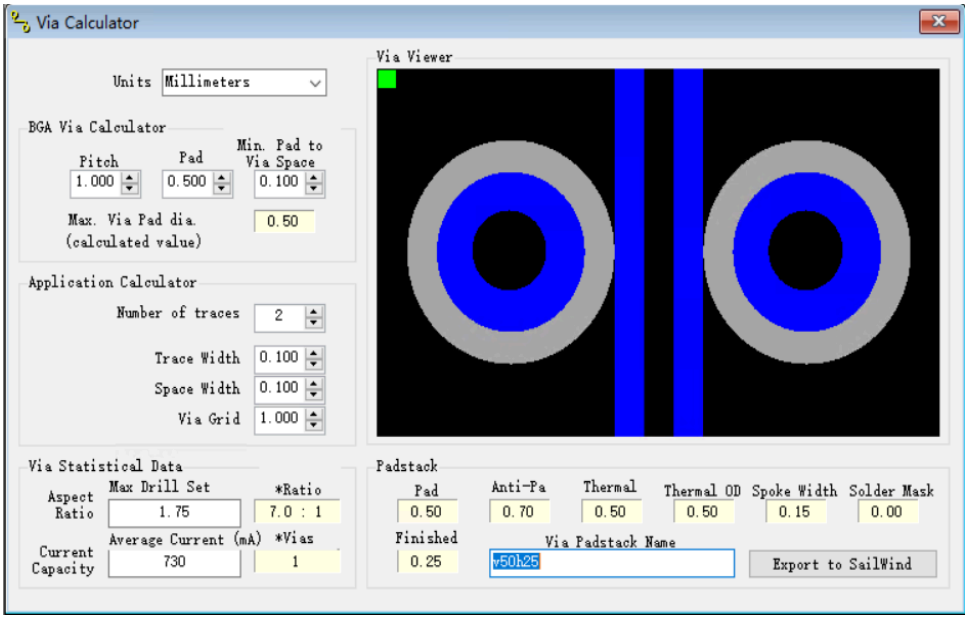
This calculator is similar to the Hole Size Calculator to the extent that the hole size can be calculated based on the size and shape of the object the that hole will accommodate and the fabrication level of the application. Additionally the hole size can be directly specified.

Sizes for All pad layers, anti pad and thermal relief are calculated, displayed and may be copied into other applications.

Via Calculator

Choosing the right via for a particular application takes a number of considerations into account: The desired design via spacing grid; the trace and space widths; the number of traces required to pass between vias; the thickness of the material; the amount of power the via may be required to carry.

The Via Calculator is designed to accept a few of the foregoing as user inputs and generate the remaining as outputs as well as calculating the proper Padstack for the resulting via.



Inputs to the BGA Via Calculator

- **Pitch** (center-to-center spacing of BGA pads).
- **Pad (Diameter)**
- **Minimum Pad-to-Via Spacing —** A typical default value for this entry is automatically applied based on the BGA Pitch/Pad combination. BGA calculations are automatically transferred to the Application Calculator where additional configurations may be tried.

Inputs to the Application Calculator

Calculations made to the BGA via calculator will be transferred to the application calculator but changes made to the application calculator do not effect the BGA calculator.

- **Number of Traces between Vias**
- **Trace Width**
- **Space Width**
- **Via Grid** (center-to-center spacing of vias).

Outputs from the Via Calculator

- **Aspect Ratio** (material thickness divided by hole diameter) — is the relationship between the hole and the thickness of the material the hole will penetrate. This is important because affects the ability to properly plate the barrel of the via in the board fabrication process. The aspect ratio is calculated by dividing the material thickness by the hole diameter.

  When any via calculation is initiated the aspect ratio is calculated to a value of 7:1. Ratios greater than this value should be avoided. Changing the thickness recalculates the aspect ratio without effecting the Via Pad stack

- **Current Capacity** (approximate) — is the maximum amount of current that may be applied to the number of “Vias req’d”. When any calculation is initiated the power capacity is recalculated for a single via. Changing the Current recalculates the number of required vias without effecting the Via Pad stack.

  This value is based the premise that each hole will be plated with a minimum copper thickness of 1 mil. The calculator then converts the cross section of the plated area of the hole to an equivalent wire and applies the standard formula $I = k * (T ^{.44}) * (A ^ {.725})$ where I = current in Amps, k = debating constant 0.024 for inner layers and T = temperature rise in C, 10 being typical.

- **Padstack information —** the correct via for most applications may be found by changing the inputs and evaluating the outputs. Then use the Padstack information to define the via in the design database.

  - **Finished Hole** — This will be the diameter of the finished hole after plating the barrel to .054 mm (1mil) of copper.
  - **Pad** — This is the pad diameter for all routing layers.
  - **Antipad** — This is the hole clearance diameter for unconnected plane layers.
  - **Thermal ID/OD** — If you choose not to "flood over" vias ID will be the inner diameter and OD will be the outer diameter of plane thermal voids.
  - **Spoke Width** — This the width of 1 (of four) of the conductive elements spanning the ThermalID/OD void.
  - **Solder Mask** — For finished holes of .4mm or larger , a Solder mask equaling the hole diameter + .15mm is added the via.
  - **Via Padstack Name** — This name is based on the hole and pad size.

Via Viewer

The Via Viewer displays the Via Hole (Black), Pad, (Blue), Antipad (Gray), and Traces (Blue) in there true proportional size. Thermals are not shown.

When an "Antipad Encroaches on Trace" message is generated the location of the problem is indicated by a White circle at the encroachment location.

When the Via BGA calculator is used as the source of the calculation BGA pads are represented by six (Gray, partial) circles in their true proportion to the vias.

Calculating Vias

Depending upon factors such as the routing density and minimum clearance rules, via calculations can sometimes be complex. Use the Via Calculator dialog box to calculate vias.

Procedure

1. On the main toolbar, click the **Calculate** button then click Via Calculator.
2. In the “Via Calculator” on page 95, if the application is for a BGA use the BGA calculator first to select an appropriate via for the BGA.
3. Adjust Number of Traces, Trace and Space Widths, and Grid to the design constraints then click on the **Calculate** button to calculate and display the results.
4. Use the Statistical entries to gather information about the calculated via and use the Padstack data to create the padstack in your design database.

Chapter 7
Naming Conventions

The following topics describe the naming convention used by the SailWind Land Pattern creator and by the SailWind Extended libraries.

Disclaimer

The land pattern calculators currently included with the SailWind products perform calculations based upon recommendations contained within the IPC-7351B *Generic Requirements for Surface Mount Design and Land Pattern Standard*.

The naming conventions used are also based on IPC-7351B as well as the IPC-7251 *Generic Requirements for Through-Hole Design and Land Pattern Standard* (draft). The SailWind Land Pattern Creator-generated SailWind Layout Extended Libraries were built to comply with the current versions of these standards in effect at the time the libraries were created.

Chengdu Paizi Interconnect Electronics Technology Co., Ltd. provides these libraries as a design aid to our customers to help them become familiar with the library construction process and are provided "as is" with no warranty by Paizi. It is the responsibility of the designer to verify that these parts are in compliance with the manufacturer's recommendations and your company's specific design rule requirements.

If parts are found to be in error, please submit a Service Request at market@pzeda.com so that the parts can be individually reviewed and potentially fixed or removed from the libraries.

IPC-7351B Naming Convention for Standard SMT Land Patterns

The IPC-7351B Specification provides an extensive and comprehensive method of describing SMT land patterns through a systematic naming convention.

Table 12. Surface Mount Land Patterns

Component, Category	Land Pattern Name
Ball Grid Arrays	**BGA** + Pin Qty + **C** or **N** + Pitch P + Ball Columns **X** Ball Rows _ Body Length **X** Body Width **X** Height

Table 12. Surface Mount Land Patterns (continued)

Component, Category	Land Pattern Name
BGA w/Dual Pitch	BGA + Pin Qty + C or N + Col Pitch X Row Pitch P + Ball Columns X Ball Rows _ Body Length X Body Width X Height
BGA w/Staggered Pins	BGAS + Pin Qty + C or N + Pitch P + Ball Columns X Ball Rows _ Body Length X Body Width X Height
	BGA Note: C = Collapsing; N = Non-collapsing Balls
Capacitors, Chip, Array, Concave	CAPCAV + Pitch P + Body Length X Body Width X Height - Pin Qty
Capacitors, Chip, Array, Flat	CAPCAF + Pitch P + Body Length X Body Width X Height - Pin Qty
Capacitors, Chip, Non-polarized	CAPC + Body Length + Body Width X Height
Capacitors, Chip, Polarized	CAPCP + Body Length + Body Width X Height
Capacitors, Chip, Wire Rectangle	CAPCWR + Body Length + Body Width X Height
Capacitors, Molded, Non-polarized	CAPM + Body Length + Body Width X Height
Capacitors, Molded, Polarized	CAPMP + Body Length + Body Width X Height
Capacitors, Aluminum Electrolytic	CAPAE + Base Body Size X Height
Ceramic Flat Packages	CFP127P + Lead Span Nominal X Height - Pin Qty
Column Grid Array, Circular Lead	CGA + Pin Qty + C + Pitch P + Pin Columns X Pin Rows _ Body Length X Body Width X Height
Column Grid Array, Square Lead	CGA + Pin Qty + S + Pitch P + Pin Columns X Pin Rows _ Body Length X Body Width X Height
Crystals (2 leads)	XTAL + Body Length X Body Width X Height
Dual-in-Line Packages (Butt Mount)	DIP + Pitch P + Lead Span Nominal X Height - Pin Qty
Dual Flat No-lead	DFN + Body Length X Body Width X Height – Pin Qty
Diodes, Chip	DIOC + Body Length + Body Width X Height
Diodes, Molded	DIOM + Body Length + Body Width X Height
Diodes, MELF	DIOMELF + Body Length + Body Diameter
Diodes, Side Concave, 2 Pin	DIOSC + Body Length X Body Width X Height - Pin Qty

Table 12. Surface Mount Land Patterns (continued)

Component, Category	Land Pattern Name
Fuses, Molded	FUSM + Body Length + Body Width X Height
Inductors, Chip	INDC + Body Length + Body Width X Height
Inductors, Molded	INDM + Body Length + Body Width X Height
Inductors, Precision Wire Wound	INDP + Body Length + Body Width X Height
Inductors, Chip, Array, Concave	INDCAV + Pitch P + Body Length X Body Width X Height - Pin Qty
Inductors, Chip, Array, Flat	INDCAF + Pitch P + Body Length X Body Width X Height - Pin Qty
Land Grid Array, Circular Lead	LGA + Pin Qty + C + Pitch P + Pin Columns X Pin Rows _ Body Length X Body Width X Height
Land Grid Array, Square Lead	LGA + Pin Qty + S + Pitch P + Pin Columns X Pin Rows _ Body Length X Body Width X Height
Land Grid Array, Rectangle Lead	LGA + Pin Qty + R + Pitch P + Pin Columns X Pin Rows _ Body Length X Body Width X Height
LED's, Molded	LEDM + Body Length + Body Width X Height
LED's, Side Concave, 2 Pin	LEDSC + Body Length X Body Width X Height - Pin Qty
LED's, Side Concave, 4 Pin	LEDSC + Pitch P + Body Length X Body Width X Height - Pin Qty
Oscillators, Side Concave	OSCSC + Pitch P + Body Length X Body Width X Height - Pin Qty
Oscillators, J-Lead	OSCJ + Pitch P + Body Length X Body Width X Height - Pin Qty
Oscillators, L-Bend Lead	OSCL + Pitch P + Body Length X Body Width X Height - Pin Qty
Oscillators, Corner Concave	OSCCC + Body Length X Body Width X Height
Plastic Leaded Chip Carriers	PLCC + Pitch P + Lead Span L1 X Lead Span L2 Nominal X Height - Pin Qty
Plastic Leaded Chip Carrier Sockets Square	PLCCS + Pitch P + Lead Span L1 X Lead Span L2 Nominal X Height - Pin Qty
Quad Flat Packages	QFP + Pitch P + Lead Span L1 X Lead Span L2 Nominal X Height - Pin Qty
Ceramic Quad Flat Packages	CQFP + Pitch P + Lead Span L1 X Lead Span L2 Nominal X Height - Pin Qty
Quad Flat No-lead	QFN + Pitch P + Body Width X Body Length X Height - Pin Qty + Thermal Pad

Table 12. Surface Mount Land Patterns (continued)

Component, Category	Land Pattern Name
Pull-back Quad Flat No-lead	PQFN + Pitch P + Body Width X Body Length X Height - Pin Qty + Thermal Pad
Quad Leadless Ceramic Chip Carriers	LCC + Pitch P + Body Width X Body Length X Height - Pin Qty
Quad Leadless Ceramic Chip Carriers (Pin 1 on Side)	LCCS + Pitch P + Body Width X Body Length X Height - Pin Qty
Resistors, Chip	RESC + Body Length + Body Width X Height
Resistors, Molded	RESM + Body Length + Body Width X Height
Resistors, MELF	RESMELF + Body Length + Body Diameter
Resistors, Chip, Array, Concave	RESCAV + Pitch P + Body Length X Body Width X Height - Pin Qty
Resistors, Chip, Array, Convex, E-Version (Even Pin Size)	RESCAXE + Pitch P + Body Length X Body Width X Height - Pin Qty
Resistors, Chip, Array, Convex, S-Version (Side Pins Diff)	RESCAXS + Pitch P + Body Length X Body Width X Height - Pin Qty
Resistors, Chip, Array, Flat	RESCAF + Pitch P + Body Length X Body Width X Height - Pin Qty
Small Outline Diodes, Flat Lead	SODFL + Lead Span Nominal + Body Width X Height
Small Outline IC, J-Leaded	SOJ + Pitch P + Lead Span Nominal X Height - Pin Qty
Small Outline Integrated Circuit, (50 mil Pitch SOIC)	SOIC127P + Lead Span Nominal X Height - Pin Qty
Small Outline Packages	SOP + Pitch P + Lead Span Nominal X Height - Pin Qty
Small Outline No-lead	SON + Pitch P + Body Width X Body Length X Height - Pin Qty + Thermal Pad
Pull-back Small Outline No-lead	PSON + Pitch P + Body Width X Body Length X Height - Pin Qty + Thermal Pad
Small Outline Transistors, Flat Lead	SOTFL + Pitch P + Lead Span Nominal X Height - Pin Qty
SOD (Example: SOD3717X135 = JEDEC SOD123)	SOD + Lead Span Nominal + Body Width X Height
SOT89 (JEDEC Standard Package)	SOT89

Table 12. Surface Mount Land Patterns (continued)

Component, Category	Land Pattern Name
SOT143 & SOT343 (JEDEC Standard Package)	SOT143 & SOT343
SOT143 & SOT343 Reverse (JEDEC Standard Package)	SOT143R & SOT343R
SOT23 & SOT223 Packages (Example: SOT230P700X180-4)	SOT + Pitch P + Lead Span Nominal X Height - Pin Qty
TO (Generic DPAK - Example: TO228P970X238-3)	TO + Pitch P + Lead Span X Height - Pin Qty

IPC-7351B Land Pattern Naming Convention Notes

- All dimensions are in Metric Units
- All Lead Span and Height numbers go two places past the decimal point and “include” trailing Zeros
- All Lead Span and Body Sizes go two place before the decimal point and “remove” leading Zeros
- All Chip Component Body Sizes are one place to each side of the decimal point
- Pitch Values are two places to the right & left of decimal point with no leading Zeros but include trailing zeros

Naming Convention Special Character Use for Land Patterns

- The _ (underscore) is the separator between pin Qty in Hidden & Deleted pin components
- The – (dash) is used to separate the pin qty.
- The X (capital letter X) is used instead of the word “by” to separate two numbers such as height X width like “Quad Packages”.

IPC-7351B Suffix Naming Convention for Land Patterns

Common SMT Land Pattern to Describe Environment Use

This is the last character in every name.

Note:
This excludes the BGA component family as they only come in the Nominal Environment Condition

- M = Most Material Condition (Level A)
- N = Nominal Material Condition (Level B)
- L = Least Material Condition (Level C)

Alternate Components that do not follow the JEDEC, EIA or IEC Standard

- A = Alternate Component (used primarily for SOP & QFP when Component Tolerance or Height is different)
- B = Second Alternate Component

Reverse Pin Order

- -20RN = 20 pin part, Reverse Pin Order, Nominal Environment

Hidden Pins

- -20_24N = 20 pin part in a 24 pin package. The pins are numbered 1 – 24 the hidden pins are skipped. The schematic symbol displays up to 24 pins.

Deleted Pins

- -24_20N = 20 pin part in a 24 pin package. The pins are numbered 1 – 20. The schematic symbol displays 20 pins.

JEDEC and EIA Standard parts that have several alternate packages

- AA, AB, AC = JEDEC or EIA Component Identifier

GENERAL SUFFIXES

- _HS = Land Pattern with Heat Sink attachment requiring additional holes or pads

  Example: TO254P1055X160_HS-6N
- _BEC = Base, Emitter and Collector (Pin assignments used for three pin Transistors)

  Example: SOT95P280X160_BEC-3N
- _SGD = Source, Gate and Drain (Pin assignments used for three pin Transistors)

  Example: SOT95P280X160_SGD-3N
- _213 = Alternate pin assignments used for three pin Transistors

  Example: SOT95P280X160_213-3N

Suggested Naming Convention for Non-Standard SMT Land Patterns

Table 13. Non-standard SMT Land Patterns

Component, Category	Land Pattern Name
Amplifiers	AMP_ Mfr.'s Part Number
Batteries	BAT_ Mfr.'s Part Number
Capacitors, Variable	CAPV_Mfr.'s Part Number
Capacitors, Chip, Array, Concave (Pins on 2 or 4 sides)	CAPCAV_Mfr Series No. - Pin Qty
Capacitors, Chip, Array, Flat (Pins on 2 sides)	CAPCAF_Mfr Series No. - Pin Qty
Capacitors, Miscellaneous	CAP_Mfr.'s Part Number
Crystals	XTAL_Mfr.'s Part Number
Diodes, Miscellaneous	DIO_Mfr.'s Part Number
Diodes, Bridge Rectifiers	DIOB_Mfr.'s Part Number
Ferrite Beads	FB_Mfr.'s Part Number
Fiducials	FID + Pad Size X Solder Mask Size
Filters	FIL_Mfr.'s Part Number
Fuses	FUSE_Mfr.'s Part Number
Fuse, Resettable	FUSER_Mfr.'s Part Number
Inductors, Miscellaneous	IND_Mfr.'s Part Number
Inductors, Chip, Array, Concave (Pins on 2 or 4 sides)	INDCAV_Mfr Series No. - Pin Qty
Inductors, Chip, Array, Flat (Pins on 2 sides)	INDCAF_Mfr Series No. - Pin Qty
Keypad	KEYPAD_Mfr.'s Part Number
LEDS	LED_Mfr.'s Part Number
LEDS, Chip	LED_Mfr.'s Part Number
Liquid Crystal Display	LCD_Mfr.'s Part Number
Microphones	MIC_Mfr.'s Part Number

Table 13. Non-standard SMT Land Patterns (continued)

Component, Category	Land Pattern Name
Opto Isolators	OPTO_Mfr.'s Part Number
Oscillators	OSC_Mfr.'s Part Number - Pin Qty
Quad Flat Packages w/Bumper Corners, Pin 1 Side	BQFP + Pitch P + Lead Span L1 X Lead Span L2 Nominal X Height - Pin Qty
Quad Flat Packages w/Bumper Corners, 1 Center	BQFPC + Pitch P + Lead Span L1 X Lead Span L2 Nominal X Height - Pin Qty
Resistors, Chip, Array, Concave (Pins on 2 or 4 sides)	RESCAV_Mfr Series No. - Pin Qty
Resistors, Chip, Array, Convex Type E (Pins on 2 sides)	RESCAXE_Mfr Series No. - Pin Qty
Resistors, Chip, Array, Convex Type S (Pins on 2 sides)	RESCAXS_Mfr Series No. - Pin Qty
Resistors, Chip, Array, Flat (Pins on 2 sides)	RESCAF_Mfr Series No. - Pin Qty
Relays	RELAY_Mfr.'s Part Number
Speakers	SPKR_Mfr's Part Number
Switches	SW_Mfr.'s Part Number
Test Points, Round	TP + Pad Size (1 place left of decimal and 2 places right of decimal, Example TP100 = 1.00mm)
Test Points, Square	TPS + Pad Size (1 place left of decimal and 2 places right of decimal)
Test Points, Rectangle	TP + Pad Length X Pad Width (1 place left of decimal and 2 places right of decimal)
Thermistors	THERM_Mfr.'s Part Number
Transceivers	XCVR_ Mfr.'s Part Number
Transducers (IRDA's)	XDCR_Mfr.'s Part Number
Transient Voltage Suppressors	TVS_Mfr.'s Part Number
Transient Voltage Suppressors, Polarized	TVSP_Mfr.'s Part Number
Transistor Outlines, Custom	TRANS_Mfr.'s Part Number

Table 13. Non-standard SMT Land Patterns (continued)

Component, Category	Land Pattern Name
Transformers	XFMR_Mfr.’s Part Number
Trimmers & Potentiometers	TRIM_Mfr.’s Part Number
Tuners	TUNER_Mfr.’s Part Number
Varistors	VAR_Mfr.’s Part Number
Voltage Controlled Oscillators	VCO_Mfr.’s Part Number
Voltage Regulators, Custom	VREG_Mfr.’s Part Number

Through-hole Naming Convention

The IPC-7251 Specification provides an extensive and comprehensive method of describing through-hole land patterns through a systematic naming convention.

IPC-7251 Naming Convention for Through-Hole Land Patterns

- The land pattern naming convention uses component dimensions to derive the land pattern name.
- The first 3 – 6 characters in the land pattern name describe the component family.
- The first number in the land pattern name refers to the Lead Spacing or hole to hole location to insert the component lead.
- All numbers that follow the Lead Spacing are component dimensions.

These characters are used as component body identifiers that precede the value and this is the priority order of the component body identifiers –

- P = Pitch for components with more than two leads
- W = Maximum Lead Width (or Component Lead Diameter)
- L = Body Length for horizontal mounting
- D = Body Diameter for round component body
- T = Body Thickness for rectangular component body
- H = Height for vertically mounted components
- Q = Pin Quantity for components with more than two leads

- R = Number of Rows for connectors
- A, B & C = the fabrication complexity level as defined in the IPC-2221 and IPC-2222

Notes:

- All component body values are in millimeters and go two places to the right of the decimal point and no leading zeros.
- All Complexity Levels used in the examples are “B”.

Table 14. Through Hole Land Patterns

Component, Category	Land Pattern Name
Capacitors, Non Polarized Axial Diameter Horizontal Mounting	CAPAD + Lead Spacing + W Lead Width + L Body Length + D Body Diameter **Example:** CAPAD800W52L600D150B (Lead Spacing 8.00; Lead Width 0.52; Body Length 6.00; Body Diameter 1.50)
Capacitors, Non Polarized Axial Rectangular	CAPAR + Lead Spacing + W Lead Width + L Body Length + T Body thickness + H Body Height **Example:** CAPAR800W52L600T50H70B (Lead Spacing 8.00; Lead Width 0.52; Body Length 6.00; Body Thickness 0.50; Body Height 0.70)
Capacitors, Non Polarized Axial Diameter Vertical Mounting	CAPADV + Lead Spacing + W Lead Width + L Body Length + D Body Diameter **Example:** CAPADV300W52L600D150B (Lead Spacing 3.00; Lead Width 0.52; Body Length 6.00; Body Diameter 1.50mm)
Capacitors, Non Polarized Axial Rectangular Vertical Mounting	CAPARV + Lead Spacing + W Lead Width + L Body Length + T Body Thickness + H Body Height **Example:** CAPARV300W52L600T50H70B (Lead Spacing 8.00; Lead Width 0.52; Body Length 6.00; Body Thickness 0.50; Body Height 0.70)
Capacitors, Non Polarized Radial Diameter	CAPRD + Lead Spacing + W Lead Width + D Body Diameter + H Body Height **Example:** CAPRD200W52D300H550B (Lead spacing 2.00; lead width 0.52; Body Diameter 3.00; Height 5.50)
Capacitors, Non Polarized Radial Rectangular	CAPRR + Lead Spacing + W Lead Width + L Body Length + T Body thickness + H Body Height **Example:** CAPRR200W52L50T70H550B (Lead spacing 2.00; lead width 0.52; Body Length 0.50; Body thickness 0.70; Height 5.50)

Table 14. Through Hole Land Patterns (continued)

Component, Category	Land Pattern Name
Capacitors, Non Polarized Radial Disk Button	CAPRB + Lead Spacing + W Lead Width + L Body Length + T Body thickness + H Body Height **Example:** CAPRB200W52L50T70H550B (Lead spacing 2.00; lead width 0.52; Body Length 0.50; Body thickness 0.70; Height 5.50)
Capacitors, Polarized Axial Diameter Horizontal Mounting	CAPPA + Lead Spacing + W Lead Width + L Body Length + D Body Diameter **Example:** CAPPAD800W52L600D150B (Lead Spacing 8.00; Lead Width 0.52; Body Length 6.00; Body Diameter 1.50)
Capacitor, Polarized Radial Diameter	CAPPR + Lead Spacing + W Lead Width + D Body Diameter + H Body Height **Example:** CAPPRD200W52D300H550B (Lead spacing 2.00; lead width 0.52; Body Diameter 3.00; Height 5.50)
Diodes, Axial Diameter Horizontal Mounting	DIOAD + Lead Spacing + W Lead Width + L Body Length + D Body Diameter **Example:** DIOAD800W52L600D150B (Lead Spacing 8.00; Lead Width 0.52; Body Length 6.00; Body Diameter 1.50)
Diodes, Axial Diameter Vertical Mounting	DIOADV + Lead Spacing + W Lead Width + L Body Length + D Body Diameter **Example:** DIOADV300W52L600D150B (Lead Spacing 8.00; Lead Width 0.52; Body Length 6.00; Body Diameter 1.50)
Dual-In-Line Packages	DIP + Lead Span + W Lead Width + P Pin Pitch + L Body Length + H Component Height + Q Pin Qty **Example:** DIP762W52P254L1905H508Q14B (Lead Span 7.62; Lead Width 0.52; Pin Pitch 2.54; Body Length 19.05; Body Height 5.08; Pin Qty 14)
Dual-In-Line Sockets	DIPS + Lead Span + W Lead Width + P Pin Pitch + L Body Length + H Component Height + Q Pin Qty **Example:** DIPS762W52P254L1905H508Q14B (Lead Span 7.62; Lead Width 0.52; Pin Pitch 2.54; Body Length 19.05; Body Height 5.08; Pin Qty 14)
Transistor Outline, Flange Mount, Horizontal	TO + Pin Pitch P + Body Length X Body Width X Height Max – Pin Qty **Example:** TO170P2207X1028X470-5A (1.70 Pin Pitch; 22.07 Body Length; 10.28 Body Width; 4.70 Height; 5 pins; Fabrication Level A)

Table 14. Through Hole Land Patterns (continued)

Component, Category	Land Pattern Name
Transistor Outline, Flange Mount, Vertical	TO + Pin Pitch P + Body Length X Body Width X Height Max – Pin Qty **Example:** TO127P817X1028X2084-5A (1.27 Pin Pitch; 8.17 Body Length; 10.28 Body Width; 20.84 Height; 5 pins; Fabrication Level A)
Transistor Outline, Cylindrical	TO + Pin Pitch P + Body Diameter X Height Max – Pin Qty **Example:** TO508R895X660-4A (5.08 Pin Radius; 8.95 Body Diameter; 6.60 Height; 5 pins; Fabrication Level A)
Headers, Vertical	HDRV + total Pins + W Lead Width + P Row Pitch (+ X Column Pitch [if different]) + _ Row s + X Pins per Row + _ Body Length + X Body Thickness + X Component Height + Fabrication Level **Example 1: (**2.54mm pitch;0.635mm lead width,20 pins,2 rows,10 pins per row,25.40mm L X 2.54mm W X 8.38mm H body) HDRV20W64P254_2X10_2540X254X838P Example 2: (2.54mm pitch; 0.635mm lead width, 50 pins, 3 rows, 25 pins per row, 63.50mm L X 2.54mm W X 8.38mm H body - 3 rows by 25 pins with 25 missing ping pins) HDRV50W64P254_3X25_6350X254X838P
Headers, Right Angle	HDRRA + total Pins + W Lead Width + P Row Pitch (+ X Column Pitch [if different]) + _ Row s + X Pins per Row + _ Body Length + X Body Thickness + X Component Height + Fabrication Level Example: (2.54mm pitch; 0.635mm lead width, 20 pins, 2 rows, 10 pins per row, 25.40mm L X 2.54mm W X 5.08mm H body) HDRRA20W64P254_2X10_2540X254X508P
Inductors, Axial Diameter Horizontal Mounting	INDAD + Lead Spacing + W Lead Width + L Body Length + D Body Diameter **Example:** INDAD800W52L600D150B (Lead Spacing 8.00; Lead Width 0.52; Body Length 6.00; Body Diameter 1.50)
Inductors, Axial Diameter Vertical Mounting	INDADV + Lead Spacing + W Lead Width + L Body Length + D Body Diameter **Example:** INDADV300W52L600D150B (Lead Spacing 3.00; Lead Width 0.52; Body Length 6.00; Body Diameter 1.50)
Jumpers, Wire	JUMP + Lead Spacing + W Lead Width **Example:** JUMP500W52B (Lead Spacing 5.00; Lead Width 0.52)
Mounting hole	Mounting hole for ANSI size 6 with flat washer, tight fitting, non-plated; 3.85mm dia. hole, 8.7mm land, with 6 vias

Table 14. Through Hole Land Patterns (continued)

Component, Category	Land Pattern Name
	Example: MTGNP870H385V6P MTG + NP (non-plated) + Land Size + H + Hole Size + V + No. of vias + Fab Level
Mounting hole	Mounting hole for Metric size M3.5 pan head, tight fitting, plated; 3.85mm dia. hole, 7.35mm land **Example:** MTGP735H385Z735P MTG + P (plated) + Land Size + H + Hole Size + Z + Anti-pad size + Fab Level
Mounting hole	Mounting hole for size 2.75 mm, loose fitting, plated; 2.9mm dia. hole, 4mm land **Example:** MTGP400H290Z400P MTG + NP (plated) + Land Size + H + Hole Size + Z + Anti-pad size + Fab Level
Mounting hole	Example – clearance hole - Mounting hole for size 2.25 mm, tight fitting, non-plated; 2.6mm dia. hole,1.3mm land **Example:** MTGNP130H260Z130P MTG + NP (non-plated) + Land Size + H + Hole Size + Z + Anti-pad size + Fab Level
Oscillators	OSC + Lead Span + W Lead Diameter + P Pin Pitch + L Body Length + H Component Height + Q Pin Qty **Example for 8 pin Oscillator:** OSC762W46P762L1320H600Q8B (Lead Span 7.62; Lead Diameter 0.46; Pin Pitch 762; Body Length 13.20; Body Height 6.00; Pin Qty 8) **Example for 14 pin Oscillator:** OSC762W53P1524L2080H508Q14B (Lead Span 7.62; Lead Diameter 0.53; Pin Pitch 762; Body Length 20.80; Body Height 508; Pin Qty 14)
Pin Grid Arrays	PGA + Pin Qty + P Pitch + C Pin Columns + R Pin Rows + L Body Length X Body Width + H Component Height **Example:** PGA84P254C10R10L2500X2500H300B (Pin Qty 84; Pin Pitch 2.54; Columns 10; Rows 10; Body Length 25.00 X 25.00; Component Height 3.00)
Resistors, Axial Diameter Horizontal Mounting	RESAD + Lead Spacing + W Lead Width + L Body Length + D Body Diameter **Example:** RESAD800W52L600D150B (Lead Spacing 8.00; Lead Width 0.52; Body Length 6.00; Body Diameter 1.50)
Resistors, Axial Diameter Vertical Mounting	RESADV + Lead Spacing + W Lead Width + L Body Length + D Body Diameter **Example:** RESADV300W52L600D150B

Table 14. Through Hole Land Patterns (continued)

Component, Category	Land Pattern Name
	(Lead Spacing 3.00; Lead Width 0.52; Body Length 6.00; Body Diameter 1.50)
Resistors, Axial Rectangular Horizontal Mounting	RESAR + Lead Spacing + W Lead Width + L Body Length + T Body thickness + H Body Height **Example:** RESAR800W52L600T50H70B (Lead Spacing 8.00; Lead Width 0.52; Body Length 6.00; Body Thickness 0.50; Body Height 0.70)
Single-In-Line Packages	SIP + Body Width + W Lead Width + P Pin Pitch + L Body Length + H Component Height + Q Pin Qty **Example:** SIP150W52P254L1905H508Q8B (Body Width 1.5; Lead Width 0.52; Pin Pitch 2.54; Body Length 19.05; Body Height 5.08; Pin Qty 8)
Test Points	Test Point; 0.635mm lead width, round, 2.54mm Diameter X 5.84mm H body height. **Example (round test point with round or square lead):** TPCW64D254H584P TP + C + W + Lead Width + D + Body Diameter + H + Height + Fab Level
Test Points	Test Point; 0.635mm lead width, square, 2.54mm W X 5.84mm H body. **Example (square test point with round or square lead):** TPRW64L254H584P TP + R + W + Lead Width + L + Body Size + H + Height + Fab Level
Test Points	Test Point; 1.57mm W X 0.635mmT lead width, rectangular, 2.54mm L X 0.635mm W X 3.30mm H body. **Example (rectangular test point with rectangular lead):** TPRW157X64L254T64H330P
Wire	PAD + Wire Width **Example:** PAD52

Suggested Naming Convention for Non-standard PTH Land Patterns

Table 15. Non-standard PTH Land Patterns

Component, Category	Land Pattern Name
Amplifiers	AMP_ Mfr.’s Part Number
Batteries	BAT_ Mfr.’s Part Number
Bridge Rectifiers	DIOB_Mfr.’s Part Number

Table 15. Non-standard PTH Land Patterns (continued)

Component, Category	Land Pattern Name
Converters	CONV_Mfr.'s Part Number
Crystals	XTAL_Mfr.'s Part Number
Ferrite Beads	FB_Mfr.'s Part Number
Filters	FIL_Mfr.'s Part Number
Fuses	FUSE_Mfr.'s Part Number
Fuse, Resettable	FUSER_Mfr.'s Part Number
Heat Sinks	HSINK_Mfr.'s Part Number
Inductors	IND_Mfr.'s Part Number
LEDs	LED_Mfr.'s Part Number
Liquid Crystal Display	LCD_Mfr.'s Part Number
Microphones	MIC_Mfr.'s Part Number
MOV	MOV_Mfr.'s Part Number
Opto Isolators	OPTO_Mfr.'s Part Number
Oscillators	OSC_Mfr.'s Part Number
PAD	PAD + Pad Size X Hole Size + H
Photo Detectors	PHODET_Mfr.'s Part Number
Regulators	REG_Mfr.'s Part Number
Relays	RELAY_Mfr.'s Part Number
Shield, off the shelf	SHIELD_Mfr.'s Part Number
Shield, Custom	SHIELD + Body Length X Body Width
Speakers	SPKR_Mfr's Part Number
Stiffners	STIF_Mfr's Part Number
Switches	SW_Mfr.'s Part Number
Thermistors	THERM_Mfr.'s Part Number
Transducers (IRDA's)	XDCR_Mfr.'s Part Number
Transient Voltage Suppressors	TVS + Mfr.'s Part Number

Table 15. Non-standard PTH Land Patterns (continued)

Component, Category	Land Pattern Name
Transient Voltage Suppressors, Polarized	TVSP + Mfr.'s Part Number
Transistor Outlines (JEDEC Standard Package)	TO- JEDEC Number
Transistor Outlines, Custom	TRANS_Mfr.'s Part Number
Transformers	XFMR_Mfr.'s Part Number
Trimmers & Potentiometers	TRIM_Mfr.'s Part Number
Tuners	TUNER_Mfr.'s Part Number
Varistors	VAR_Mfr.'s Part Number
Voltage Controlled Oscillator	VCO_Mfr.'s Part Number
Voltage Regulators (JEDEC Standard Package)	TO- JEDEC Number

Connector Naming Convention

The IPC-7x51 Specification provides an extensive and comprehensive method of describing connector land patterns through a systematic naming convention.

IPC-7x51 Naming Convention for Connector Land Patterns

Table 16. Connector Land Patterns

Component, Category	Land Pattern Name
3M™	3M_Part Number
AGILENT™	AGILENT_Part Number
AIRBORNE™	AIRBORNE_Part Number
AMPHENOL™	AMPHENOL_Part Number
AVX™	AVX_Part Number
BERG™	BERG_Part Number
BLOCKMASTER ELECTRONICS™	BLOCKMASTER_Part Number
CUI-STACK™	CUI-STACK_Part Number
E.F. JOHNSON™	JOHNSON_Part Number
ERNI	ERNI_Part Number

Table 16. Connector Land Patterns (continued)

Component, Category	Land Pattern Name
FCI ELECTRONICS™	FCI_Part Number
FUJITSU™	FUJITSU_Part Number
HIROSE™	HIROSE_Part Number
ITT CANNON™	ITT_Part Number
JALCO™	JALCO_Part Number
JWT™	JWT_Part Number
JST™	JST_Part Number
KEYSTONE™	KEYSTONE_Part Number
KYCON™	KYCON_Part Number
LEMO™	LEMO_Part Number
MILL-MAX™	MILL-MAX_Part Number
MOLEX™	MOLEX_Part Number
NEUTRIK™	NEUTRIK_Part Number
PHOENIX™	PHOENIX_Part Number
PULSE™	PULSE_Part Number
RIA™	RIA_Part Number
SAMTEC™	SAMTEC_Part Number
SIEMENS™	SIEMENS_Part Number
SPEEDTECH™	SPEEDTECH_Part Number
STEWART™	STEWART_Part Number
SULLINS™	SULLINS_Part Number
SWITCHCRAFT™	SWITCHCRAFT_Part Number
TYCO™	TYCO_Part Number
YAMAICHI™	YAMAICHI_Part Number

Sectional Breakdowns and Zero Orientation

The IPC-7351B Specification is further divided into sectional breakdowns and also contains information related to the zero orientation of both surface mount and through-hole land patterns.

IPC-7351 Surface Mount Land Patterns Sectional Breakdown

IPC-735* Component Family Breakdown

- IPC-7351 = IEC 61188-5-1, Generic requirements - Attachment (land/joint) considerations – General Description
- IPC-7352 = IEC 61188-5-2, Sectional requirements - Attachment (land/joint) considerations – Discrete Components
- IPC-7353 = IEC 61188-5-3, Sectional requirements - Attachment (land/joint) considerations – Gull-Wing leads, two sides (SOP)
- IPC-7354 = IEC 61188-5-4, Sectional requirements - Attachment (land/joint) considerations – J leads, two sides (SOJ)
- IPC-7355 = IEC 61188-5-5, Sectional requirements - Attachment (land/joint) considerations – Gull-Wing leads, four sides (QFP)
- IPC-7356 = IEC 61188-5-6, Sectional requirements - Attachment (land/joint) considerations – J leads, four sides (PLCC)
- IPC-7357 = IEC 61188-5-7, Sectional requirements - Attachment (land/joint) considerations – Post leads, two sides (DIP)
- IPC-7358 = IEC 61188-5-8, Sectional requirements - Attachment (land/joint) considerations – Area Array Components (BGA)
- IPC-7359 = NO IEC Document, Sectional requirements - Attachment (land/joint) considerations – No Lead Components (LCC)

IPC-7351 Surface Mount Land Pattern Zero Orientation

- Chip Capacitors, Resistors and Inductors (RES, CAP and IND) – **Pin 1 (Positive) on Left**
- Molded Inductors (INDM), Resistors (RESM), Molded Polarized Capacitors (CAPMP) – **Pin 1 (Positive) on Left**
- Precision Wire-wound Inductors – **Pin 1 (Positive) on Left**
- MELF Diode – **Pin 1 (Cathode) on Left**
- SOD Diodes – **Pin 1 (Cathode) on Left**

- Aluminum Electrolytic Capacitors – **Pin 1 (Positive) on Left**
- SOT Devices (SOT23, SOT23-5, SOT223, SOT89, SOT143, etc.) – **Pin 1 Upper Left**
- TO252 & TO263 (DPAK Type) Devices – **Pin 1 Upper Left**
- Small Outline Gullwing ICs (SOIC, SOP, TSOP, SSOP, TSSOP) – **Pin 1 Upper Left**
- Ceramic Flat Packs (CFP) – **Pin 1 Upper Left**
- Small Outline J Lead ICs (SOJ) – **Pin 1 Upper Left**
- Quad Flat Pack ICs (PQFP, SQFP) – **Pin 1 Upper Left**
- Ceramic Quad Flat Packs (CQFP) – **Pin 1 Upper Left**
- Bumper and Plastic Quad Flat Pack ICs (BQFPC, PQFPC Pin 1 Center) – **Pin 1 Top Center**
- Plastic Leaded Chip Carriers (PLCC) – **Pin 1 Top Center**
- Leadless Chip Carriers (LCC) – **Pin 1 Top Center**
- Leadless Chip Carriers (LCCS Pin 1 on Side) – **Pin 1 Upper Left**
- Quad Flat No-Lead ICs (QFN) QFNS & QFNRV, QFNRH – **Pin 1 Upper Left**
- Ball Grid Arrays (BGA) – **Pin A1 Upper Left**

IPC-7251 Through-hole Land Patterns Sectional Breakdown

IPC-725* Component Family Breakdown

- IPC-7251 = Generic requirements – Attachment (land/joint) considerations – General Description
- IPC-7252 = Sectional requirements – Attachment (land/joint) considerations – Discrete Components (Axial & Radial)
- IPC-7253 = Sectional requirements – Attachment (land/joint) considerations – Dual-In-Line Package (DIP)
- IPC-7254 = Sectional requirements – Attachment (land/joint) considerations – Three Leaded Semiconductor
- IPC-7255 = Sectional requirements – Attachment (land/joint) considerations – Pin Grid Array
- IPC-7256 = Sectional requirements – Attachment (land/joint) considerations – Unique Multiple function Parts

- IPC-7257 = Sectional requirements – Attachment (land/joint) considerations – Connectors & Headers
- IPC-7258 = Sectional requirements – Attachment (land/joint) considerations – Single-In-line Package (SIP) Resistor Networks
- IPC-7259 = Sectional requirements – Attachment (land/joint) considerations – Mounting Hardware

IPC-7251 Through-hole Land Pattern Zero Orientation

- Axial Lead Capacitors, Resistors, Diodes and Inductors (RES, CAP, DIO and IND) – **Pin 1 (Positive or Cathode) on Left**
- Radial Lead Capacitors (CAP) – **Pin 1 (Positive) on Left**
- Dual-in-line Packages (DIP) – **Pin 1 Left – Upper**
- Three Leaded Semiconductor – **Pin 1 Left – Upper**
- Pin Grid Array (PGA) – **Pin 1 Left – Upper**
- Unique Multiple function Parts – **Pin 1 Left – Upper**
- Connectors & Headers (HDR) – **Pin 1 Left – Upper**
- Single-In-line Package (SIP) Resistor Networks – **Pin 1 Left – Upper**

Padstack Naming Convention

The padstack consists of combinations of letters and numbers that represent shape, or dimensions of lands on different layers of printed boards or documentation. The name of the padstack needs to represent all the various combinations. These are used in combination with the land pattern conventions defined herein according to the rules established in the IPC-2220 Design standards.

The first part of the padstack convention consists of a land (pad) shape. There are six basic land shape identifiers

Note:
All alphabetical characters are “lower case”. This helps discriminate numeric values.

Basic Land Shape Letters

c = Circular

s = Square

r = Rectangle

b = Oblong

u = User Defined (Irregular Shape)

d = D Shape (Square on one end and Circular on the other end)

Note:
The “b” was used for Oblong because the letter “o” can easily be confused with the character zero “0”.

Padstack Defaults

The following are assumed defaults to keep the default padstack name short and simple. Any deviations from these padstack defaults require the use of special modifiers.

- Solder Mask is 1:1 scale of the land size (*see Solder Mask Scale note below)
- Paste Mask is 1:1 scale of the land size
- The Assembly Layer land is 1:1 scale of the land size
- Inner Layer Land is the same shape as the outer layer land
- The Primary and Secondary lands are the same size
- The inner layer land shapes are Circular
- Vias are Circular
- Thermal ID, OD and Spoke Width sizes follow the IPC Level A, B or C
- Plane Clearance Anti-pad size follows the IPC Level A, B or C
- Thermals have 4 spokes
- Mounting Holes are Circular

Restriction:
Illegal characters that cannot be used (Microsoft requirement) include “ ”, ; : / \ [ ] ( ) . { } * & % # $! @ ^ =

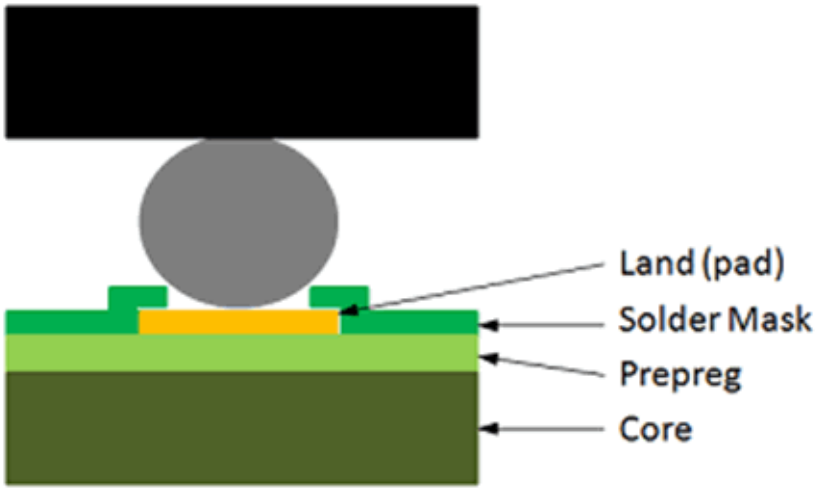
Solder Mask Scale

Every board fabricator’s ability to register solder mask is different. The 1:1 scale solder mask default compensates for the variation, and so long as manufacturers are building to standard specifications such as the IPC-6012 that states you can’t have misregistration of the solder mask. It’s important that when you are creating a CAD library that will be used for various trace/space combinations, that you leave the responsibility of the solder mask swell up to the fabrication CAM operator when they are panelizing your

Gerber or ODB++ data. By having all of the solder mask sizes 1:1 scale of the land (pad) size, you are providing the manufacturer with a known starting point for them to work with.

Exception: Creating solder mask defined lands for BGAs. IPC does not recommend solder mask defined BGA CAD library parts but some companies use this technique for very fine pitch parts that require a small diameter land size. In this case, the solder mask acts as an adhesive to secure the land to the PCB Prepreg to withstand drop testing for hand held electronic products. It has been proven in drop tests for hand held electronic devices that a fine pitch BGA solder joint is more secure than the land attachment to the Prepreg. i.e.: during drop testing, a fine pitch BGA pad will rip away from the PCB Prepreg material before the BGA solder joint fails. See Figure 1 as an example of a solder mask defined BGA land.

Figure 19. Solder Mask Defined BGA Land



Solder mask defined lands are also used for Flexible circuit boards for the same reason, to hold the land (pad) to the PCB surface to prevent the land from ripping away from the PCB material. When you use solder mask defined lands you must indicate which parts deviate from the 1:1 scale solder mask rule in the fabrication drawing notes to notify the CAM operator not to swell these solder mask features.

PAD Stack Constructions

The following sample padstack names are based on the IPC-7251/7351 Padstack Naming Convention. Examples utilizing the padstack naming convention (all values are in metric units)

Note:
Every number goes two places to the right and as many as needed to the left of the decimal. Examples: 1150 = 11.50 mm or 11500 µm, 150 = 1.50 mm or 1500 µm, 15 = 0.15 mm or 150 µm

- **c150h90**—where “c” denotes a Circular land with a 1.50 diameter and H denotes a hole size of 0.90
- **v50h25**—where a “v” denotes a via with a 0.50 land (default Circular land) and H denotes a 0.25 hole
- **s150h90**—where “s” denotes a 1.50 Square land and H denotes a hole size of 0.90
- **s350**—where “s” denotes a square SMT land size of 3.50

- **r200_100**—where “r” denotes a Rectangular SMT land 2.00 land length X 1.00 land width
- **b300_150**—where “b” denotes a SMT Oblong land size of 3.00 X 1.50
- **b400_200h100**—where “b” denotes an Oblong land size of 4.00 length X 2.00 width and 1.00 hole
- **d300_150**—where “d” denotes land with one circular end and one square end (looks like a D) 3.00 X 1.50
- **v30h15l1-3**—where “v” denotes a 0.30 blind via with 0.15 Hole; 1 is the starting layer, 3 is the end layer
- **r200_100r5**—Rounded Rectangular 2mm X 1mm X 0.05mm radius corners
- **r200_100c10**—Chamfered Rectangular 2mm X 1mm X 0.1mm chamfered corners
- **v30h15l3-6**—where “v” denotes a 0.30 buried via with 0.15 Hole; 3 is the starting layer, 6 is the end layer

The through hole “IPC-7251 Padstacks.xls” file should be used as the basis for a new chart in IPC-7251 and IPC-7351B.

Note:
Supporting paragraphs containing formulae, document the math involved. It is assumed that the padstack has the same value as the mounted layer size and shape for ñ

- Inner Layer
- Opposite Side
- Solder Mask
- Solder Paste
- Assembly Layers

It is also assumed that the “Plane Clearance” and “Thermal Relief” data follows the through-hole convention guidelines defined in the IPC-2221 and IPC-2222 standards.

Modifiers that are used when padstack features are different than the defaults

These are the “Variants” or “Modifiers” that go after the basic padstack naming convention. These are used when you need to change the padstack default values either by a different dimension or a different shape. In instances where shapes are different this becomes a two letter code with the modifier first followed by the land shape letter.

- **n** = Non-plated Hole
- **z** = Inner Layer land dimension if different than the land on primary layer
- **x** = Special modifier used alone or following other modifiers for lands on opposite side to primary layer land dimension
- **t** = Thermal Relief; if different than IPC standard padstack - tid_od_sw for 4 spoke default
- **m** = Solder Mask if different than default 1:1 scale of land
- **p** = Solder Paste if different than default 1:1 scale of land
- **a** = Assembly surface land if different than default 1:1 scale of land
- **y** = Plane Clearance (Anti-pad) if the value is different than the Thermal OD
- **o** = Offset Land Origin
- **k** = Keep-out
- **r** = Radius for Rounded Rectangular Land Shape
- **c** = Chamfer for Chamfered Rectangular Land Shape

Shape change is the last letter in the string prior to the dimension.

Other usage of the padstack naming convention

USE of letter v: Vias can be named using the pad stack naming convention. Because most vias use lands that are circular in shape, the letter V will be used in place of the letter C in the padstack naming convention. If this is not true the modifiers can be added after the letter V to signify shape or dimensional changes to this default.

USE of letter w: In addition to Vias the padstack naming convention can also be used for defining mounting holes. The letter W shall be used to define the mounting hole characteristics and any associated lands used for the surface lands (either plated or un-plated)

Examples of double character modifiers

ts = Thermal Square; if different than the top side land shape and dimensions

sw = Thermal spoke width

zs = Inner Layer Land Shape is Square (**Note:** The default is circular)

m0 = No Solder Mask

mxc = Solder Mask Opposite Side Circular

mx0 = Solder Mask Opposite Side No Solder Mask

xc = Opposite Side Circular

vs = Via with Square land

hn = Non-plated Hole

Modifier Example for Through-hole

s150h90zs150 = where “s” is Square 1.50 land with 0.90 Hole with 1.50 inner (Z) Layer Square land

c150h90zc150 = where “c” is Circular 1.50 land with 0.90 Hole with 1.50 inner (Z) Layer Circular land

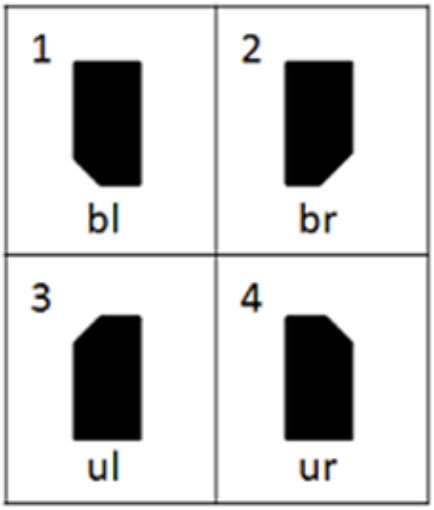
Modifier Examples for Vias

vs50h25 = where “vs” denotes a 0.50 Square Via with a 0.25 Hole

v50h25xs70 = where “v” is 0.50 Circular Via with 0.25 Hole and 0.70 Square land on opposite side

Modifiers for Chamfered and Rounded Corners

Chamfered & Rounded corner modifiers are used to indicate which corner(s) are modified. Order of precedence has been given to the first 4 modifiers.



Modifiers:

- **bl** = bottom left
- **br** = bottom right
- **ul** = upper left
- **ur** = upper right
- **ulr** = upper left & right
- **blr** = bottom left & right

- **ubl** = upper and bottom left
- **ubr** = upper and bottom right

Rounded and Chamfered lands in “one corner” Modifier Examples

- **r100_200rbl50** = rectangular land 1.00 x 2.00 with 0.50 radius for rounded corner in bottom left corner
- **r100_200rbr50** = rectangular land 1.00 x 2.00 with 0.50 radius for rounded corner in bottom right corner
- **r100_200rul50** = rectangular land 1.00 x 2.00 with 0.50 radius for rounded corner in upper left corner
- **r100_200rur50** = rectangular land 1.00 x 2.00 with 0.50 radius for rounded corner in upper right corner
- **r100_200cbl50** = rectangular land 1.00 x 2.00 with 0.50 chamfer for chamfer corner in bottom left corner
- **r100_200cbr50** = rectangular land 1.00 x 2.00 with 0.50 chamfer for chamfer corner in bottom right corner
- **r100_200cul50** = rectangular land 1.00 x 2.00 with 0.50 chamfer for chamfer corner in upper left corner
- **r100_200cur50** = rectangular land 1.00 x 2.00 with 0.50 chamfer for chamfer corner in upper right corner

Note:
Chamfered and Rounded Rectangular with all four corners chamfered does not need a corner modifier.

Modifier Examples with Rounded Rectangle Land Shape:

- **r200_100r50** = rectangular land 2.00 x 1.00 with 0.50 radius for rounded corners in all 4 corners
- **r200_100r50** = rectangular land 2.00 x 1.00 with 0.50 radius for rounded corners in all 4 corners

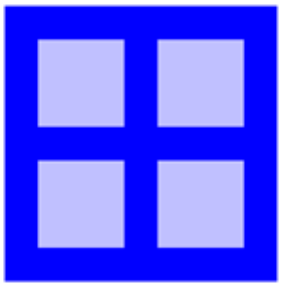
Modifier Examples with Rounded Rectangle Land Shape:

- **r200_100culr50** = rectangular land 2.00 x 1.00 with 0.50 chamfer for chamfered corners in 2 corners
- **r200_100c50** = rectangular land 2.00 x 1.00 with 0.50 chamfer for chamfered corners in all 4 corners

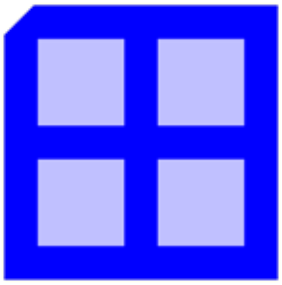
Thermal Pad Examples for QFN, SON, QFP and SOP

Thermal pads can have a combination of chamfered and rounded corners however the typical application is 2 variations. The most prominent is a chamfered corner located near pin 1 and the second is a chamfered corner located near pin 1 with the other 3 corners rounded. These two variations are the default.

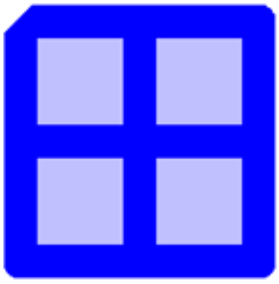
- **s480p4s152** = 4.80mm square land with 4 paste mask squares 1.52mm each



- **u480p4s152cul50** = 4.80mm square land with 4 paste mask squares 1.52mm each with 0.50mm chamfer in upper left corner



- **u480p4s152cul50r25** = 4.80mm square land with 4 paste mask squares 1.52mm each with 0.50mm chamfer in upper left corner with 0.25mm corner radius



Examples of a padstack with Circular land with hole using various modifiers

- **c150h90** = Default padstack with a 1.50 circular land with a 0.90 hole (no modifiers used)
- **c150hn90** = Default padstack with a 1.50 circular land with a 0.90 non-plated hole (no modifiers used)
- **c150h90z140** = Inner layer land is smaller than external lands 1.40 or 0.10 smaller
- **c150h90z140x170** = Opposite side land is larger than top side land 1.70 or 0.20 larger
- **c150h90z140x170m165mx185** = Solder mask opening for top and bottom lands 0.15 larger for each
- **c150h90z140x170m165mX185a200** = Assembly drawing land in 0.50 larger than 1.50 primary land
- **c150h90z140x170m165mx185a200y300** = Plane clearance anti-pad diameter is 3.00
- **c150h90z140x170m165mx85** = Solder mask encroachment on opposite land by 0.65 smaller
- **c150h90m165** = adding a solder mask opening of 1.65 diameter or 0.15 larger than land
- **c150h90t150_180_40** = Thermal ID 1.50, OD 1.80, Spoke Width 0.40, Anti-pad 1.80
- **c150h90t150_180_40y200** = Anti-pad 2.00 (because the size is different than the Thermal OD)
- **c150h90t150_180_80_2** = Spoke Width 0.80 with 2 Spokes
- **c150h90m165t150_180_40** = Solder Mask 1.65

Examples of a padstack with Oblong land with Slotted Hole

- Sample - b = Oblong Land Shape then “X” dimension (length) then Underscore _ “Y” dimension (width)
- **b400_200h300_100** = Oblong land 4mm length X 2mm width with slotted hole size 3mm X 1mm
- **b400_200hn300_100** = Oblong land 4mm X 2mm with non-plated slotted hole size 3mm X 1mm

Examples of a SMT padstack land using various modifiers

- **b300_150** = Default padstack with a 3.00 length and 1.50 width land (no modifiers used)
- **b300_150m330_180** = Solder Mask is 0.30 larger than the land
- **b300_150m330_180p240_140** = Solder Paste is smaller by 0.10 width and 0.60 length
- **b300_150b-50** = Oblong Land 3.0mm X 1.5mm w/Offset Origin negative 0.5mm
- **r400_200po430_230** = Rectangle SMT land 4.00 X 2.00 with a Oblong Solder Paste size of 4.30 X 2.30

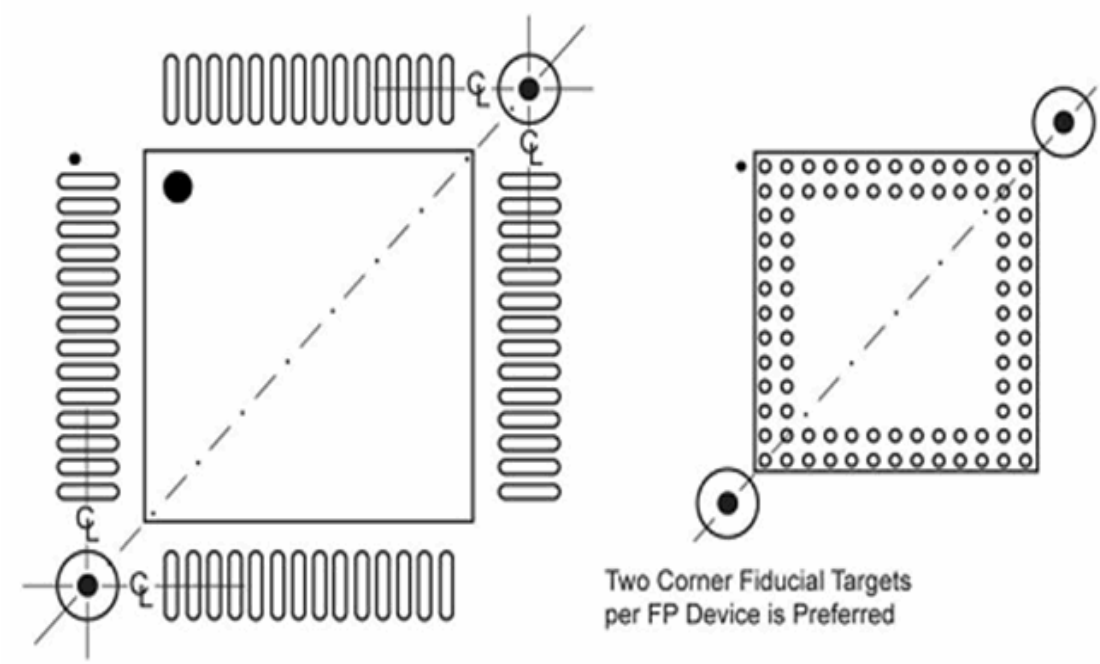
Example of a Mounting Hole

- **w700h400z520m720** = This is a Plated Through Mounting hole for a #6-32 screw using a 4.00 diameter hole and having a circular 7.00 land on the primary and secondary side of the board, with a solder mask clearance that is 0.20 larger than the 7.20 land. The internal lands are smaller that the external and are also circular 5.20 in diameter.
- **w700hn400z520m720** = Non-plated version

Example of a Local Fiducial for Fine Pitch SMT Components

- **c100m200k200** = Circular Land 1.00 with Solder Mask 2.00 with Keep-out 2.00
- **s100m200k200** = Square Land 1.00 with Solder Mask 2.00 with Keep-out 2.00

Figure 20. Local Fiducial Application on Fine Pitch Components



Example of Proportional Plated Through-hole padstack

- **c150h100** = 1.5mm circular pad with 1mm hole with 1.5mm solder mask with 1.5mm plane clearance with 1.5mm assembly outline with Thermal Relief w/4 spokes 0.4mm width with ID 1.5mm and OD 1.8mm

Example of Proportional Non-plated Through-hole padstack

- **c100h150** = 1mm circular pad with 1.5mm hole with 1.5mm solder mask with 2.35mm plane clearance with 2.1mm keep-out

Land Pattern Name Syntax

Land Pattern Names are generally derived by combining the name of the component category with a dimensional aspect of the component and the number of pins followed by a suffix derived from the environmental application.

If additional characteristics are required they should be added as an alphabetical character, or combinations of characters that preceded the suffix.

For a complete description of the naming process, you should get the "IPC-7351 Land Pattern Naming Convention.pdf" file first.

Sample Name: QFP65P + Lead Span L1 X Lead Span L2 Nominal - Pin Qty

The + (plus sign) stands for “in addition to” (no space between the prefix and the body size)

The - (dash) is used to separate the pin qty.

The X (capital letter X) is used instead of the word “by” to separate two numbers such as height X width like “Quad Packages”.

SUFFIXES

Environmental Application.

Suffixes for every common SMT land pattern describe the environment application (This is the last character in every name)

Note:
This excludes the BGA.

Three land pattern geometry variations are supplied for each of the device families; maximum land protrusion (Density Level A), median land protrusion (Density Level B) and minimum land protrusion (Density Level C). Before adapting the minimum land pattern variations you should consider product qualification testing based on the conditions shown in Table 3-13.

- **M - Density Level A: Maximum (Most) Land Protrusion**—For low-density product applications, the 'maximum' land pattern condition has been developed to accommodate wave or flow solder of leadless chip devices and leaded gull- wing devices. The geometry furnished for these devices, as well as inward and “J”-formed lead contact device families, may provide a wider process window for reflow solder processes as well.

- **N - Density Level B: Median (Nominal) Land Protrusion**—Products with a moderate level of component density may consider adapting the 'median' land pattern geometry. The median land patterns furnished for all device families will provide a robust solder attachment condition for reflow solder processes and should provide a condition suitable for wave or reflow soldering of leadless chip and leaded gull-wing type devices.

- **L - Density Level C: Minimum (Least) Land Protrusion**—High component density typical of portable and hand-held product applications may consider the 'minimum' land pattern geometry variation. Selection of the minimum land pattern geometry may not be suitable for all product use categories. The use of classes of performance (1, 2, and 3) is combined with that of component density levels (A, B, and C) in explaining the condition of an electronic assembly. As an example, combining the description as Levels 1A or 3B or 2C, would indicate the different combinations of performance and component density to aid in understanding the environment and the manufacturing requirements of a particular assembly.

Alternate Components that do not follow JEDEC, EIA or IEC standard.

- A - Alternate Component (used primarily for SOP & QFP when Component Tolerance or Height is different) AA, AB, etc. - JEDEC or EIA Component Identifier (Used primarily on Chip Resistors, Inductors and Capacitors).

- B - Second Alternate Component

Reverse Pin Order.

- -20RN - 20 pin part, reverse pin order, Nominal environment.

Hidden Pins.

- -20_24N - 24 pin locations with 20 pins populated. Pin numbering follows pin locations (pins number 1 to 24).

Deleted Pins.

- -24_20N - 24 pin locations with 20 pins populated. Pin numbering follows populated pins (pin number 1 to 20).

General Suffixes

HS—Land Pattern with Heat Sink attachment requiring additional holes or pads.
Example:TO254P1055_HS-6N

BEC—Base, Emitter and Collector (Pin assignments used for three pin Transistors)

SGD—Source, Gate and Drain (Pin assignments used for three pin Transistors)

213—Alternate pin assignments used for three pin Transistors

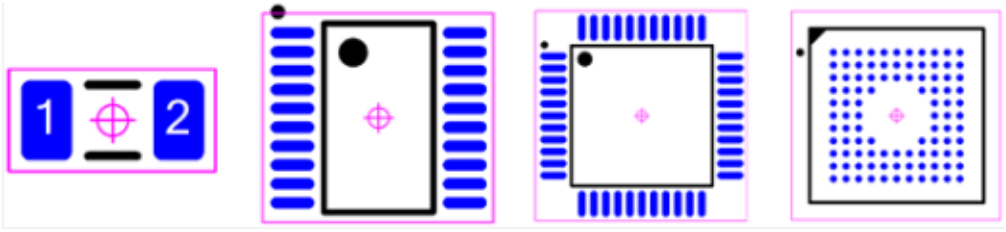
Component Zero Rotations

Each land pattern described in the IPC-7351B and IPC-7251 Specification contains recommendations for zero rotation of the base land patterns. This helps to provide a baseline to assembly operations so that pick and place machines can properly orient their feeders and deliver components to the machine in a predictable orientation.

Pin # 1 Orientations

Pin 1 locations (top view) are typically oriented to the top left of the land pattern.

Figure 21. IPC-7351 Zero Orientation with Pin 1 Upper Left



Glossary

Aspect Ratio

In the context of evaluating a **Via** this is the relationship between the hole and the thickness of the material the hole will penetrate. This is important because affects the ability to properly plate the barrel of the via in the board fabrication process. The aspect ratio is calculated by dividing the hole diameter by the material thickness.

Aspect ratios of 7 or less are generally reliable.

Tip
Aspect Ratios greater than 7 should be avoided.

Attribute

Attributes are general descriptors used to categorize the **Values** which are the object a search. Such attributes would "Manufacturer", "Part Number" or "Description" for example.

Courtyard

The area that provides an electrical and mechanical clearance around the component body and land pattern boundaries.

Courtyard Excess

The distance between the courtyard boundary and its defining boundary.

Environment

The environment refers to the conditions in which the surface mount PCB (land pattern) will be required to operate. The SMD Calculator has four basic default environments.

Note:
For Surface Mount Devices, "L", "M", and "N" are used as standard name suffixes to denote environments of "Least", "Most" and "Nominal", respectively.

- **Most**—**Density Level A: Maximum (Most) Land Protrusion**—For low-density or high-reliability product applications, the 'maximum' land pattern condition has been developed to accommodate wave or flow solder of leadless chip devices and leaded gull- wing devices. The geometry furnished for these devices, as well as inward and “J”-formed lead contact device families, may provide a wider process window for reflow solder processes as well.

- **Nominal—Density Level B: Median (Nominal) Land Protrusion**—Products with a moderate level of component density may consider adapting the 'median' land pattern geometry. The median land patterns furnished for all device families will provide a robust solder attachment condition for reflow solder processes and should provide a condition suitable for wave or reflow soldering of leadless chip and leaded gull-wing type devices.

- **Least**—**Density Level C: Minimum (Least) Land Protrusion**—High component density typical of portable and hand-held product applications may consider the 'minimum' land pattern geometry variation. Selection of the minimum land pattern geometry may not be suitable for all product use categories. The use of classes of performance (1, 2, and 3) is combined with that of component density levels (A, B, and C) in explaining the condition of an electronic assembly. As an example, combining the description as Levels 1A or 3B or 2C, would indicate the different combinations of performance and component density to aid in understanding the environment and the manufacturing requirements of a particular assembly.

- **User**—**Non Standard Density: User Defined Land Protrusion**—For user specific applications.

These environments determine the default Goals for Toe Fillet, Heel Fillet and Side Fillet. The defaults are recommendations based on studies by IPC regarding surface mount solder joint reliability. These goals will vary depending on component type size.

Error Handler

A red disk that may appear next to a user-entered value indicating an unacceptable entry. Moving the cursor over the disk will display a pop-up 'tool-tip' that will explain the nature of the error.

Fabrication Level

The fabrication level refers to the conditions in which the through-hole PCB (land pattern) will be required to operate. The PTH Calculator has five basic default fabrication levels.

Note:
For Through-Hole Devices, "C", "A", and "B" are used as standard name suffixes to denote fabrication levels of "Least", "Most" and "Nominal", respectively.

- **Most—Density Level A: Maximum (Most) Land Protrusion**—For low-density or high-reliability product applications.

- **Nominal**—**Density Level B: Median (Nominal) Land Protrusion**—For medium-density or 'consumer market' reliability product applications.

- **Least**—**Density Level C: Minimum (Least) Land Protrusion**—For applications where size or available space must be considered in addition to reliability.

- **Proportional**—**Non Standard Density: Formula Defined Land Protrusion**—Legacy approach to through-hole design where most dimensions are linear relative to the hole size as opposed to fixed depending on hole size.

- **User**—**Non Standard Density: User Defined Land Protrusion**—For user specific applications.

These fabrication levels determine the default Goals for: hole-relative-to-lead; land-relative-to-hole; thermal-relative-to hole; and spoke-relative-to-land.

Fabrication Tolerance

This figure describes how accurately the fabrication process can produce a land pattern feature of a given size. Within plus or minus .05 mm is typical for most applications but may want to increase for high volume applications.

Goals

Toe, Heel and Side goals are the minimum desired **protrusions** of the land beyond the terminal which the calculator attempts to solve for.

Heel

Denotes the beginning of a terminal. The Heel is usually close to the main body of a part.

Note:
On Inverted “L” (molded body) and J-Lead IC's (SOJ & PLCC), the Heel is away from the body while the Toe is closer to the body.

IPC 735x Family

- **IPC-7351** = IEC 61188-5-1, Generic requirements- Attachment (land/joint) considerations—General Description
- **IPC-7352** = IEC 61188-5-2, Sectional requirements - Attachment (land/joint) considerations—Discrete components
- **IPC-7353** = IEC 61188-5-3, Sectional requirements - Attachment (land/joint) considerations—Gull-wing leads, two sides
- **IPC-7354** = IEC 61188-5-4, Sectional requirements - Attachment (land/joint) considerations—J leads, two sides
- **IPC-7355** = IEC 61188-5-5, Sectional requirements - Attachment (land/joint) considerations—Gull-wing leads, four sides
- **IPC-7356** = IEC 61188-5-6, Sectional requirements - Attachment (land/joint) considerations—J leads, four sides
- **IPC-7357** = IEC 61188-5-7, Sectional requirements - Attachment (land/joint) considerations—Post (DIP) leads, two sides
- **IPC-7358** = IEC 61188-5-8, Sectional requirements - Attachment (land/joint) considerations—Area Array Components (BGA)
- **IPC-7359** = NO IEC Document, Sectional requirements - Attachment (land/joint) considerations—No Lead Components (LCC, QFN, SON)

Land Pattern

The contact area on the PCB to which any given component is soldered.

Periphery

The excess pad over or under the outer dimensions of the terminal it supports. The term 'periphery' is generally applied in conditions where a terminal is flush with the bottom of a component thus having no vertical element. The periphery is equal on all sides of such a terminal/pad relationship.

Placement Tolerance

This figure describes how accurately the placement process can locate a component on any given land pattern. Within plus or minus .05 mm is typical for most applications but may want to increase for high volume applications.

Process

This is a catchall name referring to the **Fabrication** and **Assembly** processes.

Protrusion

This is the amount by which the Land Pattern extends beyond the **Terminal** (the terminal being the contact area between the component lead and land pattern).

Protrusions are categorized as Toe, Heel and Side.

- **Toe Fillet**—refers to the amount of land between the end of a terminal and the end of its land pattern. The Toe is usually away from the body of the part.
- **Heel Fillet**—refers to the amount of land between the start of a terminal and the other end of its land pattern. The Heel is usually close to the body of a part.
- **Side Fillet**—refers to land between the terminal and pad on the remaining two sides.

Note:
On Inverted “L” (molded body) and J-Lead IC's (SOJ & PLCC), the Heel is away from the body while the Toe is closer to the body.

Round off

Round off values control the resolution of certain land pattern features. As features become smaller it may become advantageous to adjust the rounding factors in order to improve the solder joint while maintaining spacing clearances.

For example:

- With a round off value of 0.05, 2.13 would round to 2.15
- With a round off value of 0.10, 2.13 would round to 2.10
- With a round off value of 0.20, 2.13 would round to 2.20

Note:
Round off's can not equal zero

Side

Denotes two sides between the Toe and Heel.

Solder Joint Analysis

Solder Joint Analysis is the process of evaluating the relationship between the terminal, which is the contact area of the lead, and the land pattern.

Surface mount components don't rely on the solder joint just for electrical connectivity. In most cases this joint is what holds the component to the PC board. For this reason the joint must also be capable of resisting stress. Its ability to do this depends on getting the right amount of solder in the right place with respect to the terminal and the land pattern.

Conversely, a land pattern that's bigger than it needs to be wastes board space that could be used for routing.

The table of Solder Joint Analysis gives you important feedback about the relationship between land and terminal. The Calculator takes all component, process and environmental variables into account then reports the minimum and maximum protrusions of the land for the Toe, Heel and Side.

Terminal

The contact area between the component lead and land pattern.

Toe

Denotes the end of a terminal. The Toe is usually away from the main body of a part.

Note:
On Inverted “L” (molded body) and J-Lead IC's (SOJ & PLCC), the Heel is away from the body while the Toe is closer to the body.

Value

Values are the object any search. Values are unique for each part and are organized according to their general description or **Attribute**.