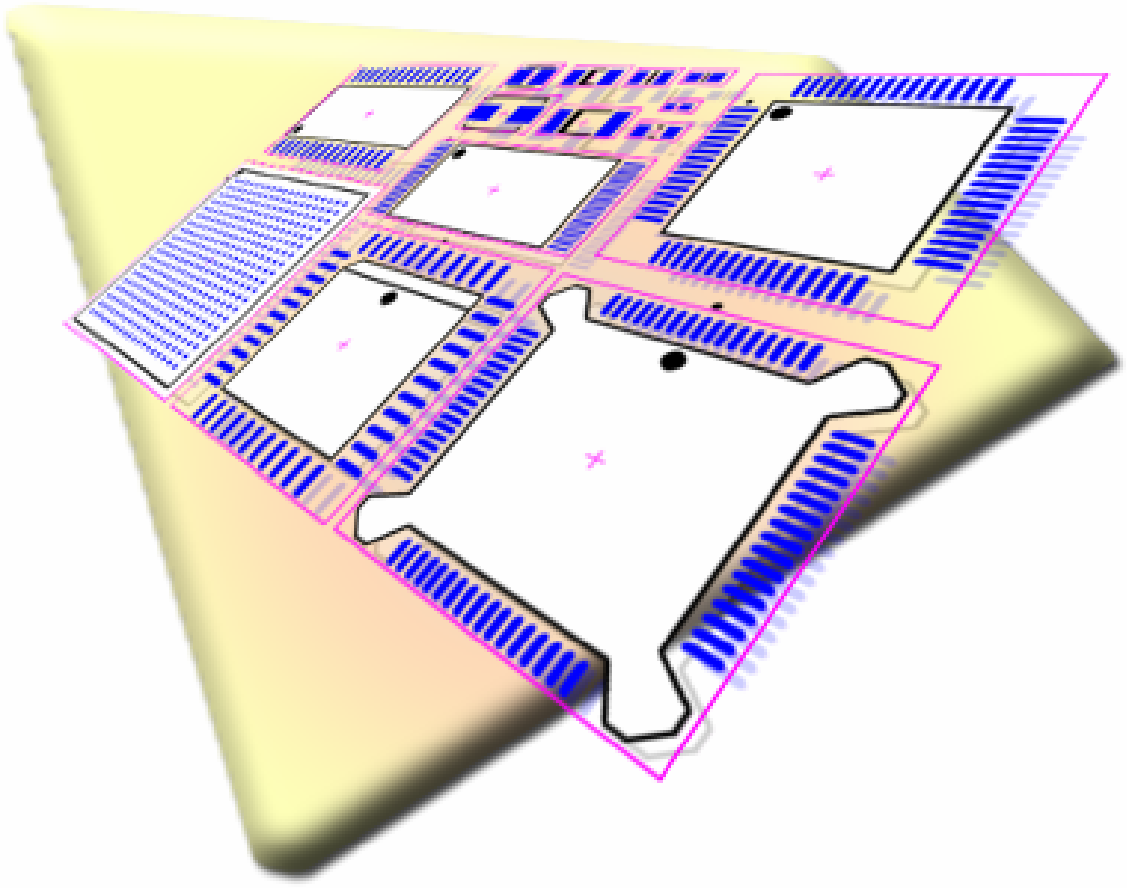
PCB Libraries IPC-7351 LP Suite

User Guide

Version 2.500 - 3/7/05



IPC
ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES®

PCB Libraries™
NEW DIMENSIONS FOR EDA

CONTENTS

SECTION **PAGE**

1 INTRODUCTION 1-1
1.1 Basic Features 1-1
2 INSTALLATION 2-2
2.1 .net Framework 2-2
2.2 Large and Small Fonts 2-3
2.3 Administration Rights 2-4
2.4 PCBL LP Suite Folders 2-4
2.5 PCBL LP Suite Updates 2-4
3 SETTING UP USER PREFERENCES 3-5
3.1 What is a .DAT file? 3-5
3.2 Creating Multiple .DAT Files 3-7
3.3 Setting Search Preferences 3-8
3.4 Setting Calculator Preferences 3-9
3.5 Setting Wizard Preferences 3-12
4 SEARCH LIBRARY 4-18
4.1 Opening the Search Library Menu 4-18
5 SMD CALCULATOR GUI 5-1
5.1 Opening the Calculator GUI 5-1
5.2 Loading a Library 5-2
5.3 QFP Calculator GUI 5-3
5.4 Using the Viewer Feature 5-4
5.5 Tolerance and Settings 5-5
5.6 Environmental Variables 5-6
5.7 Land Pattern Data 5-8
5.8 Pan and Zoom in the Browser 5-10
6 SMD CALCULATOR SEARCH 6-1
6.1 Searching for Existing Components 6-1
7 SMD CALCULATOR EDIT EXISTING PART 7-1
7.1 Edit an Existing Part 7-1
8 SMD CALCUALTOR CREATE NEW PART 8-1
8.1 Create New Land Pattern 8-1
9 SMD CONSTRUCTION WIZARD 9-1
9.1 Create PADS™ Library Part 9-1
9.2 Create a PADS™ ASCII File 9-5
9.3 Non-PADS™ Users Update an Existing Library File 9-5
10 CREATING ATTRIBUTES 10-7
10.1 What are Attributes? 10-7
10.2 Creating IPC-7351 Standard Part Attributes 10-7
10.3 Creating Non-Standard (Mfr Specific) Part Attributes 10-8

10.4 Creating Connector Part Attributes....10-9
11 PLATED THROUGH HOLE CALCULATOR....11-10
11.1 Through-hole Calculator....11-10
11.2 Through Hole Components....11-12
12 VIA CALCULATOR....12-1
12.1 Calculating BGA Vias....12-1
13 Quality Control....13-5
13.1 Send Your .P file to PCB Libraries....13-5
14 HELP FEATURE....14-1
14.1 Help Contents....14-1
14.2 Help Index....14-3
14.3 Help Find....14-4

1 INTRODUCTION

1.1 Basic Features

The IPC-7351 LP Calculator was co-developed by IPC and PCB Libraries for the sole purpose of creating accurate CAD land patterns for PCB design layout.

The use of the LP Calculator will help standardize CAD Land Pattern Names (which will help standardize schematic symbol land pattern references), Zero Component Orientations, Land Pattern Origins, Pad Sizes, Pad Spacing, Pad-stack Data, Pad Shape, Placement Courtyard boundaries, Silkscreen Outlines & Polarity Markings, Assembly Drawing Outlines & Polarity Markings, Post Assembly Inspection Dot Usage, Reference Designator Height and Width and 3D Component Outline Data for mechanical drafting software that does 3D component rendering.

The basis for this tutorial is to familiarize IPC-7351 LP Calculator Wizard Users with the basic fundamental features of the software program.

Reviewing this tutorial will quickly bring you up to speed with the use of the IPC-7351 LP Calculator Wizard software product.

There are 12 basic features that we will cover in this tutorial:

1. Setting up “User Preferences”
2. How to use the “Search Library” feature
3. Basic explanation of the “SMD Calculator GUI”
4. How to use the “SMD Calculator” to Search for existing components
5. How to use the “SMD Calculator” to edit existing components
6. How to use the “SMD Calculator” to create new components
7. How to use the “SMD Wizard” to auto-generate new PADS library parts and how to export PADS ASCII data to import into various CAD tools
8. Using the Multi-part Wizard
9. How to Create Attributes
10. How to use the “PTH Calculator”
11. How to use the “Via Calculator”
12. How to use the “Help” feature

Even though we will only cover one or two component families in this guide, all references in this tutorial are applicable to every component family. This tutorial should be used as a “Quick Start” to expedite your learning curve on the use of the IPC-7351 LP Calculator.

2 INSTALLATION

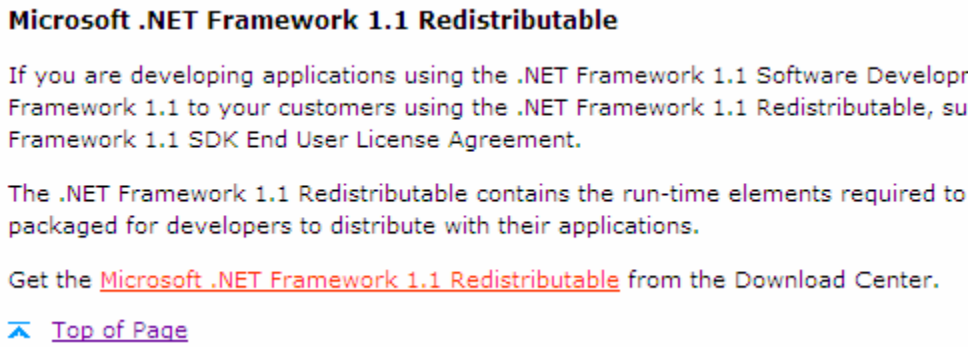
2.1 .net Framework

The IPC-7351 LP Suite was written using Microsoft VB.net software. The LP Suite requires your computer to have Microsoft .net Framework (dotnetfx.exe) installed in order for VB.net software to work. The VB.net software program is fairly new and the only OS that has it already installed is Windows XP Professional. If you are not using Windows XP Professional you will have to download the free Framework.net from this Microsoft URL:

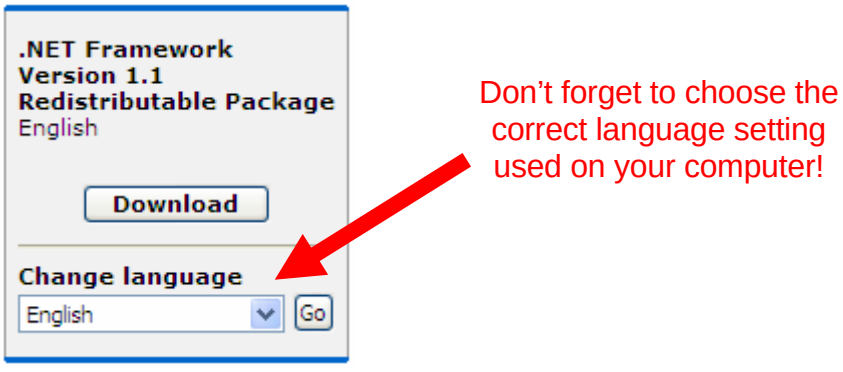
http://www.microsoft.com/downloads/details.aspx?familyid=262D25E3-F589-4842-8157-034D1E7CF3A3&displaylang=en

Download or Order the CD for the .NET Framework

Select the Download Link and Scroll down to this line.



After you select the Redistributable Link, select the Download Button.



Put the .net Framework **DOTNETFX.EXE** file anywhere you want on your computer and then install the .net Framework Program.

Now you are ready to install the IPC-7351 LP Wizard

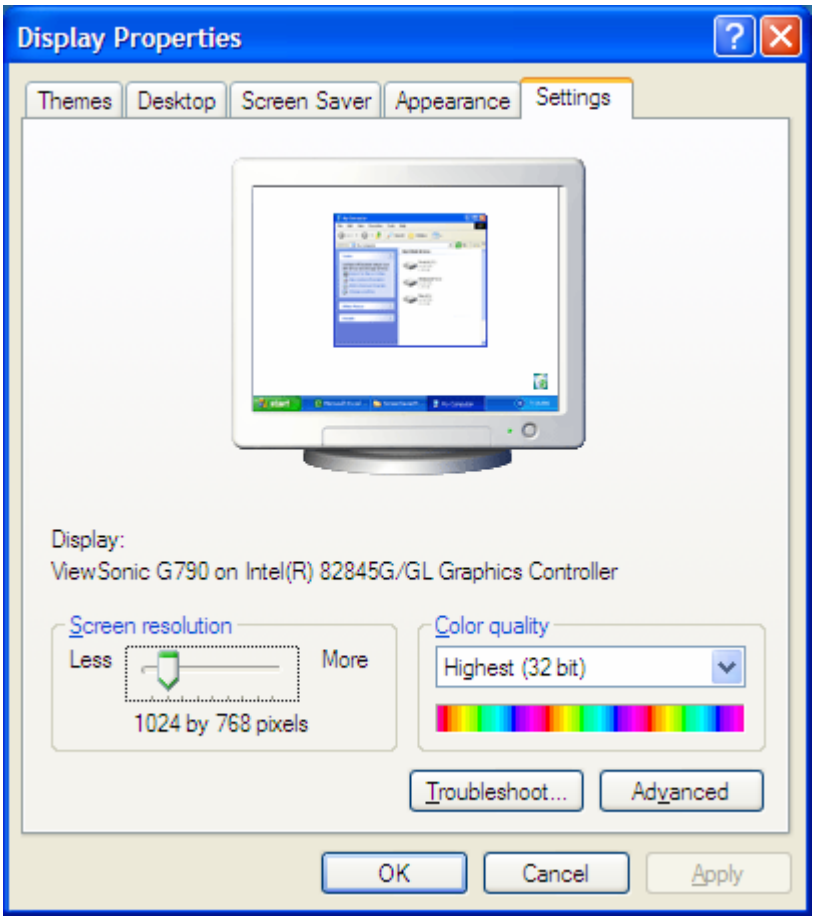
2.2 Large and Small Fonts

Your Desktop Settings have multiple Font settings. Win2000 has “Large and Small” Fonts. Windows XP has “Normal and Large Fonts”.

The LP Suite program works best with “Normal or Small Fonts”, but if you choose to Large Fonts, depending on you screen resolution, scroll bars could appear in some of the LP Suite Dialog Boxes making it slightly more difficult to navigate.

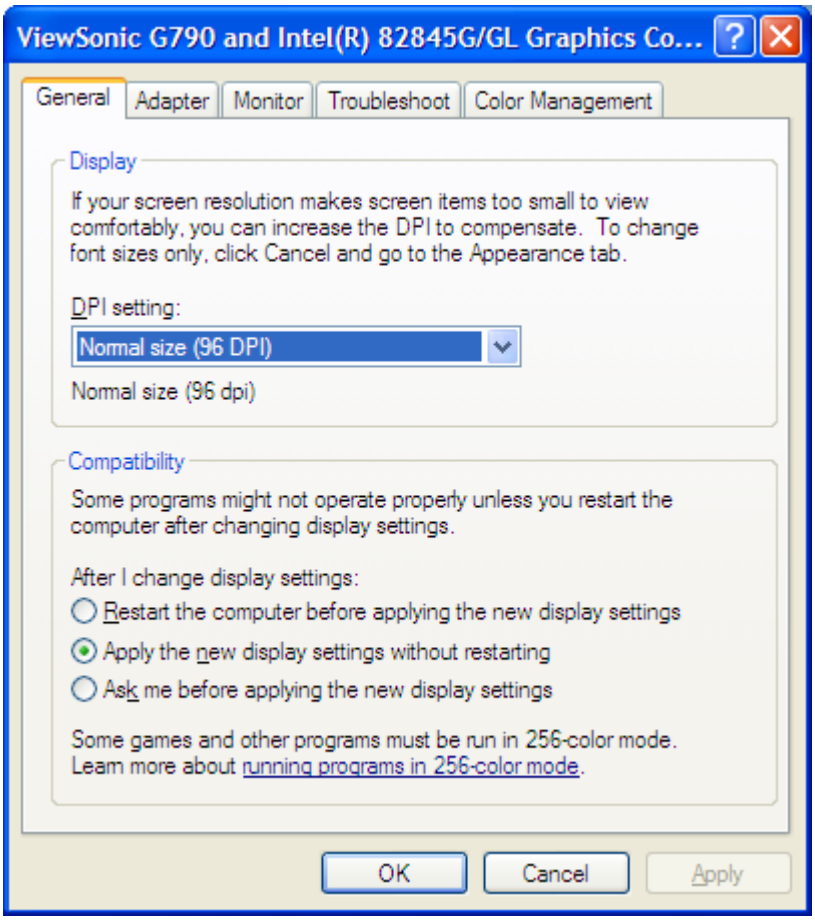
If you are using Large Fonts and would like to change them to Normal or small fonts, simply Right Mouse Button directly on your computer desktop and select “**Properties**”.

This will bring up your Display Properties. Select the “**Settings Tab**”.



Next select the “**Advanced Button**” and then select the “**General Tab**”.

In the Display area there is a DPI setting pull down which will allow you to change your Font Setting. The picture below shows the setting at “**Normal Size**”.



2.3 Administration Rights

If you work for a large company that has an internal IT department, we have experienced situations where the IT department sets up all the corporation computers without "**Administration Rights**".

This means that you will probably not be able to install the IPC-7351 LP Suite program without assistance from your IT department because they are the "**Administrators**" of your computer.

2.4 PCBL LP Suite Folders

When you install the IPC-7351 LP program, the default directory folder is: **C:\Program Files\PCB Libraries LP**. You can change this path if you choose to. The only files that the LP Suite installation puts on your computer go in this folder.

2.5 PCBL LP Suite Updates

When installing a new version of the LP program you **do not** need to un-install a previous version. The LP Suite Installation will simply write over an existing install.

3 SETTING UP USER PREFERENCES

3.1 What is a .DAT file?

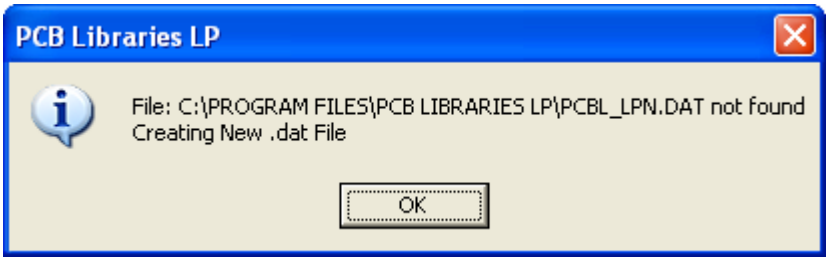
The LP Wizard comes stock with three .DAT files

1. PCBL_LPL.DAT – Least Environment Land Pattern
2. PCBL_LPN.DAT – Nominal Environment Land Pattern (Default .DAT file)
3. PCBL_LPM.DAT – Most Environment Land Pattern

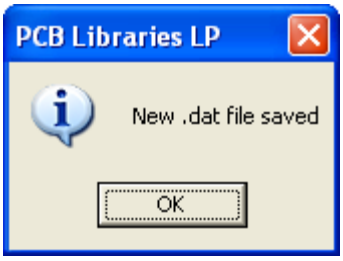
A .DAT file stores all "**User Preferences**" including you default library name and directory path. The .DAT file also stores your CAD Tool Preference.

If you are running several different CAD Wizard Interfaces, you will probably want to create several custom .DAT files and name them after each CAD Tool.

The IPC-7351 LP program installation does not include a .DAT file, but one is created automatically the **first time** you run the program. When starting the program for the first time, the User will be prompted by a message that states a .DAT file does not exist and that one is being created.



After selecting the OK button and second message appears confirming that the file was created.



The new **PCBL_LPN.DAT** file is initially set to the PCB Libraries "**Default**" settings. Then the User can modify the settings to customize their product usage.

If you create an elaborate **"CUSTOM".DAT** file then we highly recommend that you back up the file to a safe place.

Each time you install a LP program update, the installation will not affect your .DAT file because the installer does not contain a .DAT file.

If the LP program is not reacting normally or the Search Engine is not sorting data correctly, chances are that you possibly corrupted your .DAT file. To fix the problem, just delete the PCBL_LPN.DAT file and when you restart the program it will automatically create a new PCBL_LPN.DAT file.

We are going to review all the Preferences that affect the PCBL_LPN.DAT file.

You can manually edit the PCBL_LPN.DAT file using Wordpad or Notepad.

This is what the PCBL_LPN.DAT file looks like:

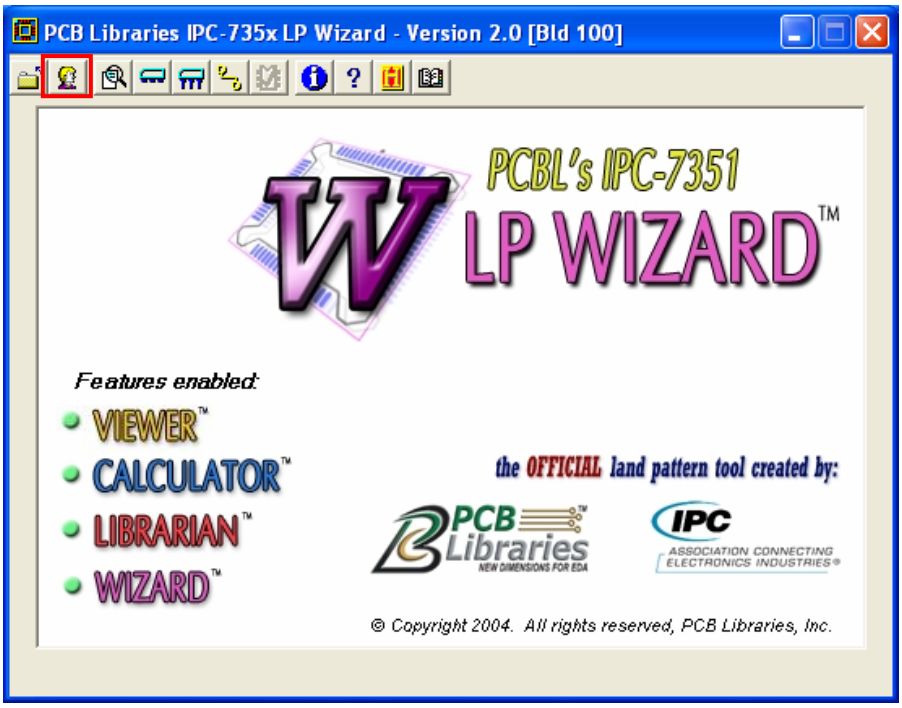
```
PCB Libraries LP User Profile
User=USR
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\SMN7351.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\SM_MFR.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\TH.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\CONNECTORS.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\AMP.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\BERG.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\CUI-STACK.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\HIROSE.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\JST.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\KYCON.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\MOLEX.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\SAMTEC.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\SWITCHCRAFT.p
LibFile=C:\Program Files\PCB Libraries LP\CAD Data\Wizard.p
BGA150=0;1000
BGA127=0;1000
BGA100=0;1000
BGA80=0;1000
BGA75=0;1000
BGA65=0;1000
BGA50=0;1000
BGA40=0;1000
BGA30=0;1000
BGA25=0;1000
CAE=0;1;500;-100;400;500;50;50;100;200;200
CAE10=0;1;700;-50;500;500;50;50;100;200;200
CHP1=0;1;100;-50;0;150;50;50;100;200;200
CHP2=0;1;350;-50;0;250;50;50;100;200;200
LCC=0;1;550;-150;-140;250;50;50;100;200;200;200;200
MELF=0;1;400;100;50;250;50;50;100;200;200
MLD=0;1;500;150;-50;250;50;50;100;200;200
PLC127=0;1;350;-200;30;250;100;50;50;200;200;200;200
QFN=0;1;300;0;-40;250;50;50;50;100;200;200;200
QFPGW100=0;1;350;350;30;250;50;50;50;100;200;200;200
QFPGW62=0;1;350;350;-20;250;50;50;50;100;200;200;200
QFPMGW100=0;1;350;350;30;250;50;50;50;100;200;200;200
QFPMGW62=0;1;350;350;-20;250;50;50;50;100;200;200;200
SOGW100=0;1;350;350;30;250;50;50;50;100;200;200;200
SOGW62=0;1;350;350;-20;250;50;50;50;100;200;200;200
SOMGW100=0;1;350;350;30;250;50;50;50;100;200;200;200
```

```
SOMGW62=0;1;350;350;-20;250;50;50;50;100;200;200;200
SOJ127=0;1;350;-200;30;250;100;50;50;200;200;200;0
SON=0;1;300;0;-40;250;50;50;50;100;200;0;0
SOTGW100=0;1;350;350;30;250;50;50;50;100;200;200;200
SOTGW62=0;1;350;350;-20;250;50;50;50;100;200;200;200
SOTMGW100=0;1;350;350;30;250;50;50;50;100;200;200;200
SOTMGW62=0;1;350;350;-20;250;50;50;50;100;200;200;200
TO=0;1;350;350;30;250;50;50;100;200;200;200
WizAttrib=PCBL.Component
WizAttrib=PCBL.Process
WizAttrib=Geometry.Height
WizAttrib=Description
WizAttrib=Standard Name
WizAttrib=Also Known As
WizAttrib=Manufacturer
WizAttrib=Mfr Part Number
WizAttrib=On-Line-Link
WizLayr=26;20;25;27;
WizLibDir=C:\padspwr\Lib
WizLibNam=Wizard
WizCADNam=C:\Program Files\PCB Libraries LP\CAD Data\Wizard.p
WizSettings=S;Y;N;A;T;T;V50
```

3.2 Creating Multiple .DAT Files

Using the “Open User Profile” you can either create a new .DAT file or open an existing one. This way you can have various .DAT files for different projects or customers. To create a new User Profile .DAT file, select the first Tool Bar button on the main menu.

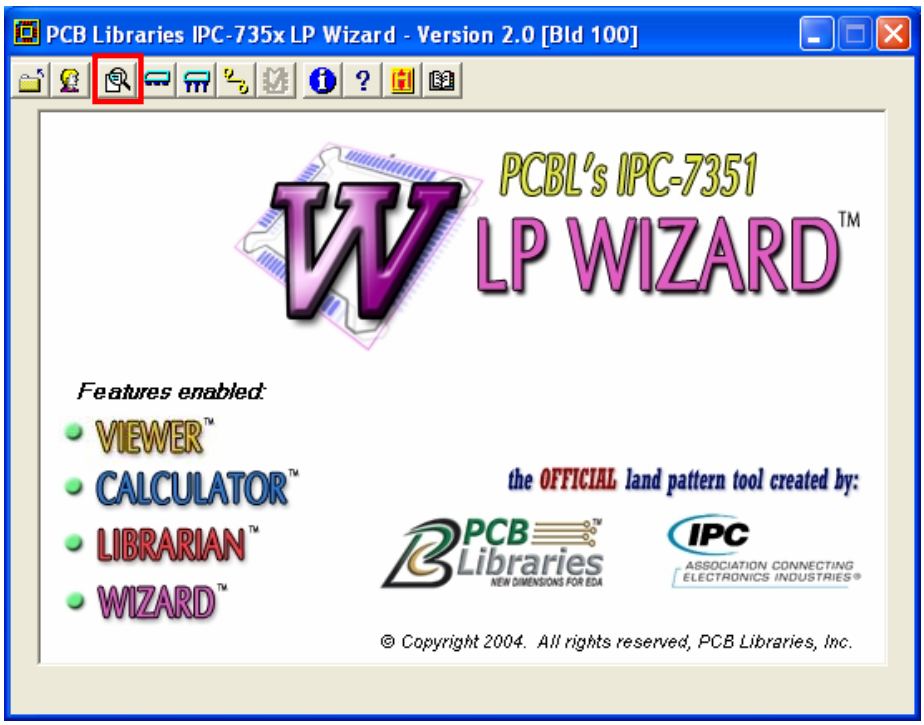
The main point is that if you create custom .DAT files for various Environments or CAD Tools, you must load the correct .DAT file at the beginning of every session.



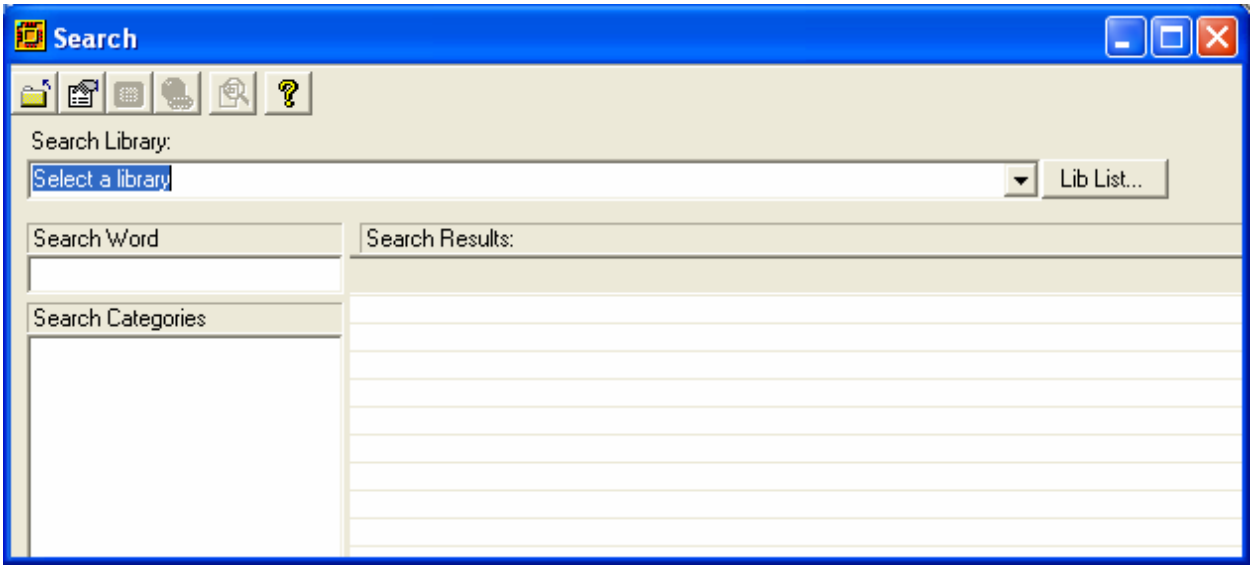
3.3 Setting Search Preferences

Depending on what CAD tool you use, you must setup your "**Default Preferences**"

Open the Search Library Menu, Select the "**Search Library**" **Tool Bar** Button.



Next select the **Preferences** **Tool Bar** Button



The .P files are your Library Attributes used for searching parameters. The more detail you put in the Attributes, the more "Keywords" you'll have to search with. For more detail, see **Chapter 10** – "**Creating Attributes**".

The Library .P files are originally placed in the default folder:

C:\Program Files\PCB Libraries LP\CAD Data>

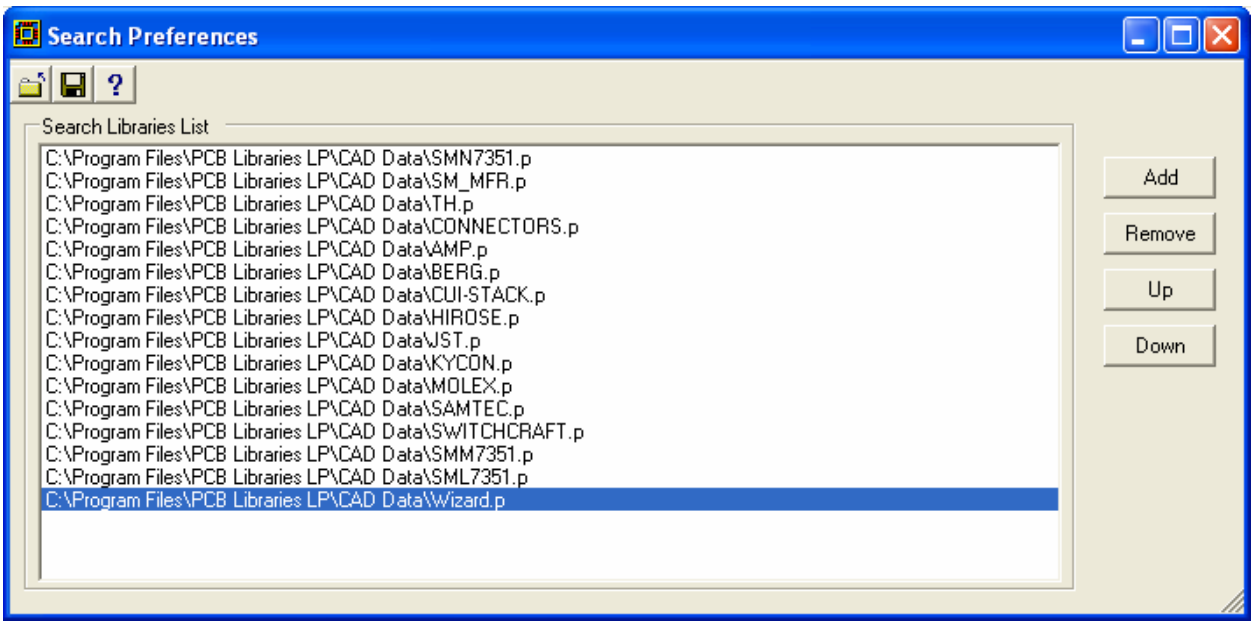
You can relocate these library .P files to any folder that you want, but remember that PCB Libraries is constantly updating the master files and when you install LP Suite updates the new library .P files will always be placed in the same folder. The folder "search path" for the library files is stored in the .DAT file with all your "**User Preferences**".

Use this "Search Preferences" menu to load "**Add**" all your library .P files once you have them in the folder of your choice.

Similarly you can "**Remove**" any Library Files from your search list.

You can also arrange the order of your libraries by selecting a library and then select the "**Up**" & "**Down**" buttons. Users tend to put the most used libraries first.

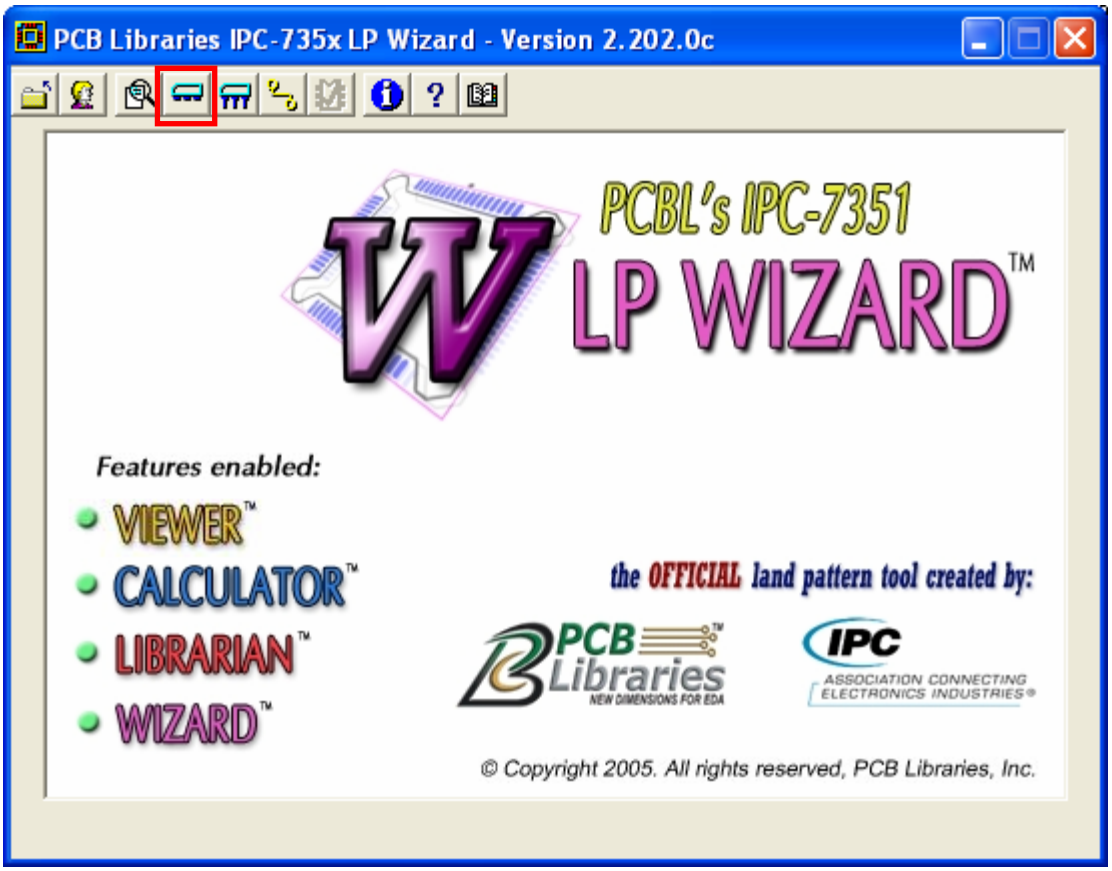
Once your libraries are loaded and in the correct order select the "**Save Preferences**" **Tool Bar** Button and then select the **Close** Button.



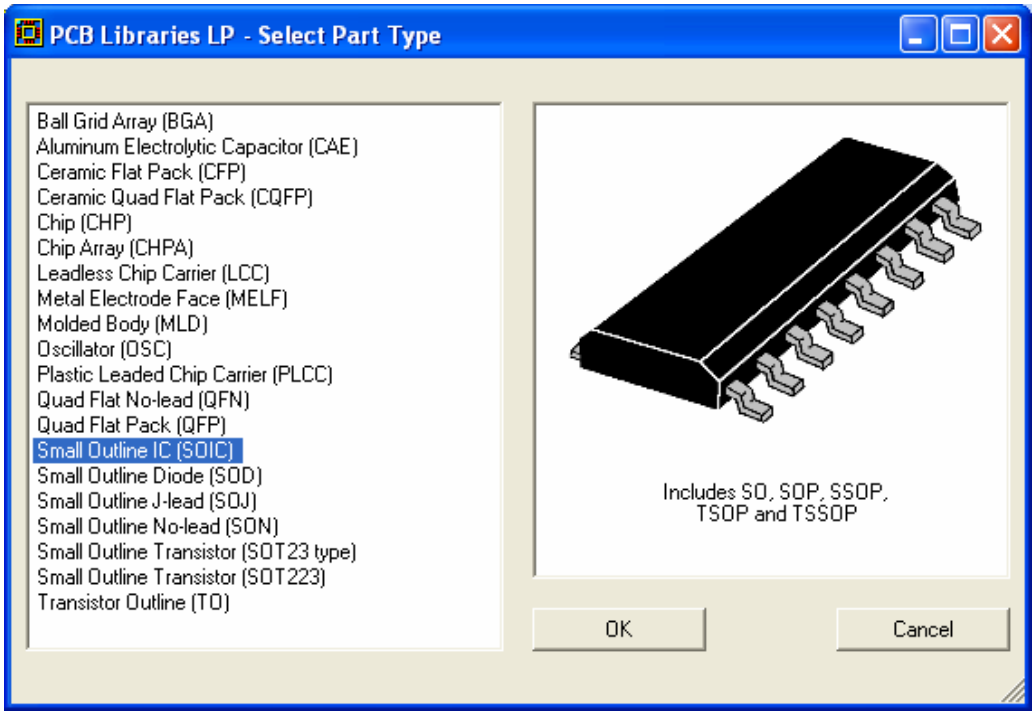
3.4 Setting Calculator Preferences

Every Component family has their own unique preferences. We are going to review how to change the preferences for the SOIC component family. The same procedure can be preformed in any of the component families.

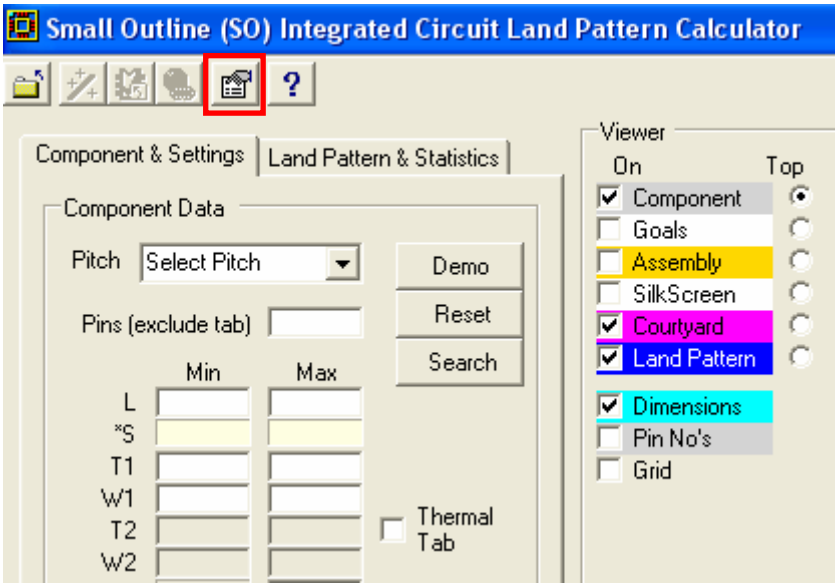
Select the **SMD Calculator** Tool Bar button (In the **Red** Box below)



The “Select Part Type” menu will appear. Select the Small Outline (SOIC) line and “Double Click” or select the “OK” button to open the SOIC menu.



Select the **Preferences** **Tool Bar** Button to open the SOIC Preferences Menu



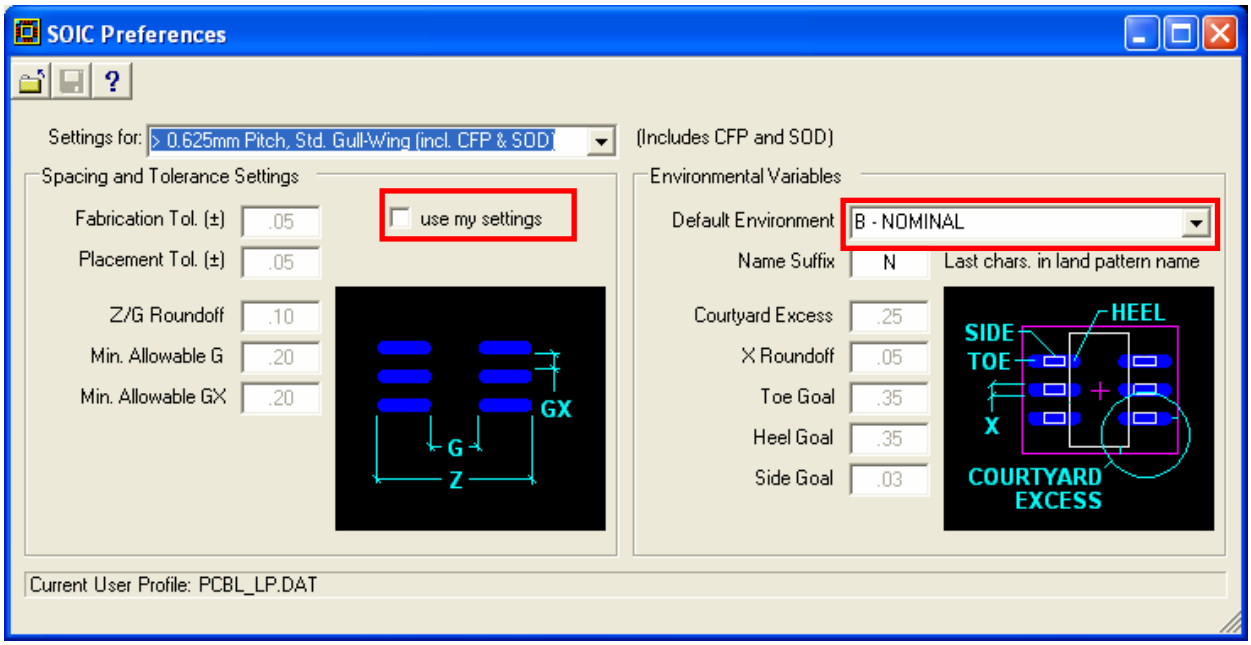
Select the "**use my settings**" check box to change the fabrication and assembly tolerances and the Minimum Allowable G & Gx dimensions.

Then in the Environmental Variables use the pull-down bar to select "**USER**". Then you can change the User Preferences for Toe, Heel and Side Goal, Courtyard Excess and X Roundoff (X = the pad width).

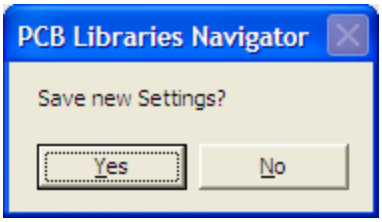
Caution: If you change any of these numbers, you automatically fall out of compliance with the IPC-7351 specification and you are on your own.

Select the **Help** **Tool Bar** Button if you need "Help"

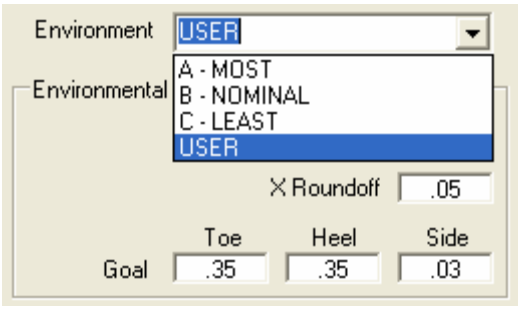
"**Save Preferences**" **Tool Bar** Button and then select the **Close** Button



If you do not select the **Save Preferences** **Tool Bar** Button the menu below will appear when closing this menu



For this exercise select "**NO**"



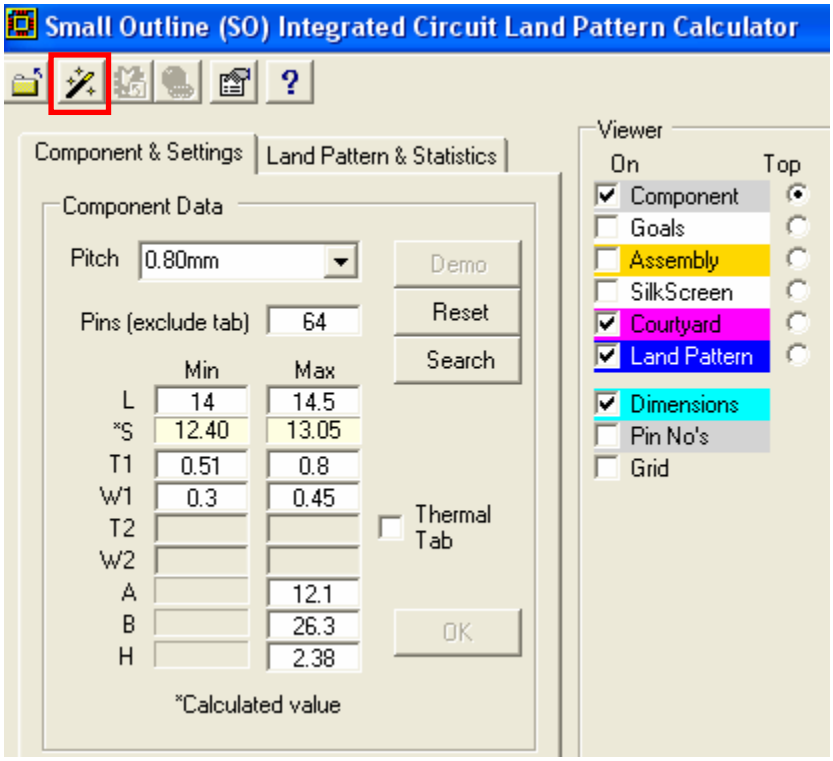
In the Calculator Select "**USER**" Preferences

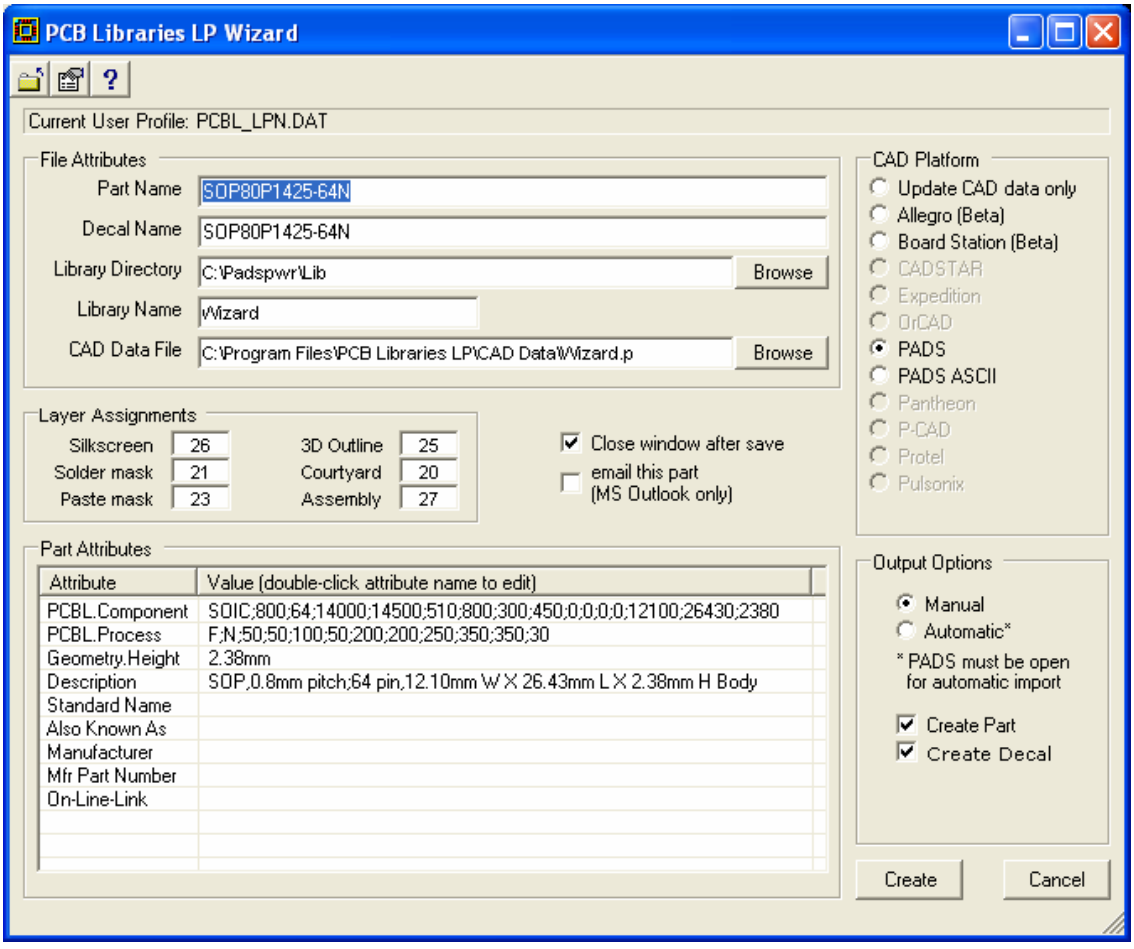
3.5 Setting Wizard Preferences

Setting the Wizard Preferences allows you to establish your Default Attributes, Library Name and Library Path.

In the SOIC Calculator select the **"Demo Button"** and the Wizard Button will become enabled.

Select the **Wizard** **Tool Bar** Button


Small Outline (SO) Integrated Circuit Land Pattern Calculator
Component & Settings
Land Pattern & Statistics
Component Data
Pitch
0.80mm
Demo
Reset
Search
Pins (exclude tab)
64
Min
Max
L
14
14.5
*S
12.40
13.05
T1
0.51
0.8
W1
0.3
0.45
T2
W2
A
12.1
B
26.3
H
2.38
Thermal Tab
OK
*Calculated value
Viewer
On
Top
Component
Goals
Assembly
SilkScreen
Courtyard
Land Pattern
Dimensions
Pin No's
Grid

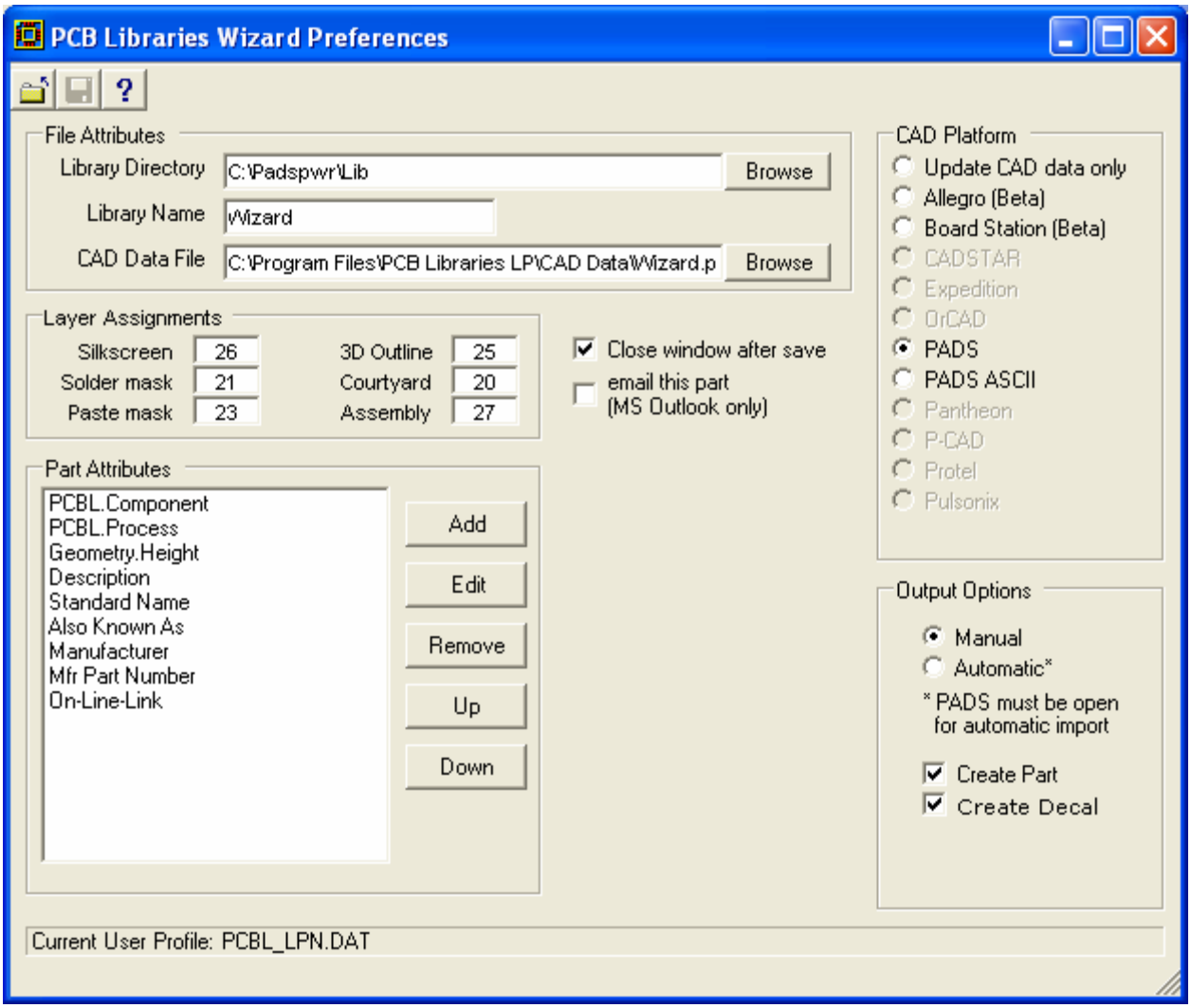

PCB Libraries LP Wizard
Current User Profile: PCBL_LPN.DAT
File Attributes
Part Name
SOP80P1425-64N
Decal Name
SOP80P1425-64N
Library Directory
C:\Padspwr\Lib
Browse
Library Name
Wizard
CAD Data File
C:\Program Files\PCB Libraries LP\CAD Data\Wizard.p
Browse
Layer Assignments
Silkscreen
26
Solder mask
21
Paste mask
23
3D Outline
25
Courtyard
20
Assembly
27
Close window after save
email this part (MS Outlook only)
CAD Platform
Update CAD data only
Allegro (Beta)
Board Station (Beta)
CADSTAR
Expedition
OrCAD
PADS
PADS ASCII
Pantheon
P-CAD
Protel
Pulsonix
Part Attributes
Attribute
Value (double-click attribute name to edit)
PCBL.Component
SOIC;800;64;14000;14500;510;800;300;450;0;0;0;0;12100;26430;2380
PCBL.Process
F;N;50;50;100;50;200;200;250;350;350;30
Geometry.Height
2.38mm
Description
SOP,0.8mm pitch;64 pin,12.10mm W X 26.43mm L X 2.38mm H Body
Standard Name
Also Known As
Manufacturer
Mfr Part Number
On-Line-Link
Output Options
Manual
Automatic*
* PADS must be open for automatic import
Create Part
Create Decal
Create
Cancel

Select the **Preferences** **Tool Bar** Button to open the Wizard Preferences Menu
First let's set the "**Library Directory**" path and "**Library Name**".

Select the "**Automatic**" Button if you are a PADS User for direct to PADS Import.

If you do not use PADS, select the "**Manual**" button, uncheck "**Create Decal**" button

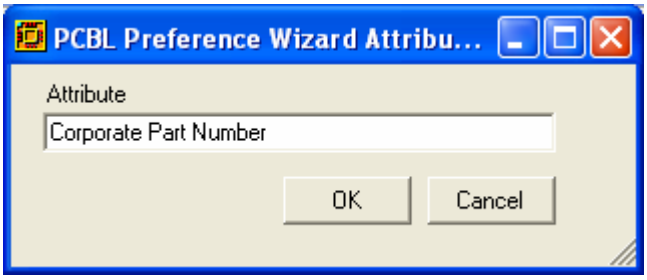
If you want to save all your component dimensional data, select the "**Update CAD data only**" so that the IPC-7351 LP Librarian can continually update your USER Library with all new component information that you enter into the calculator.

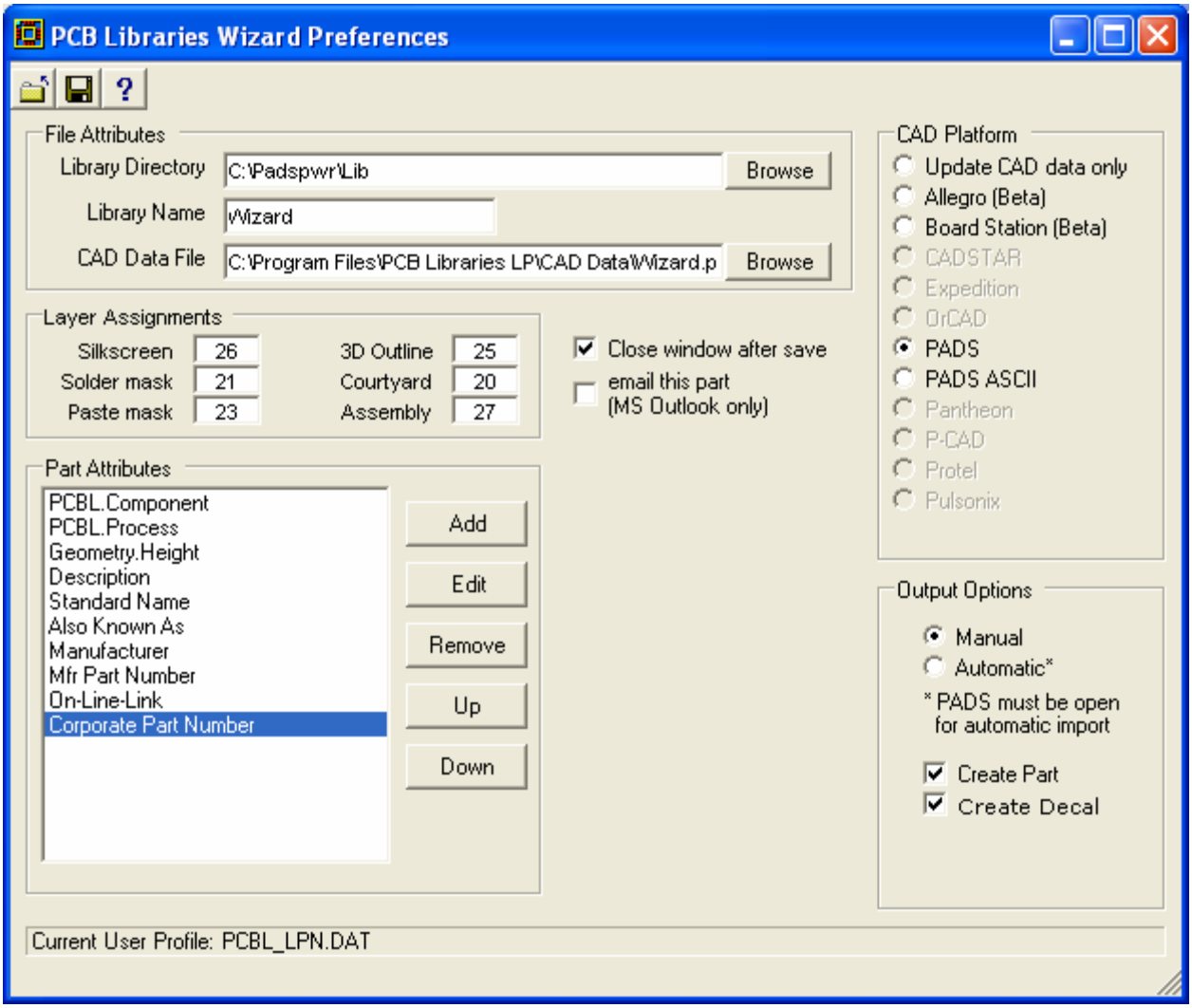


In the Part Attributes window select the "**Add**" button to add a new custom attribute

A good example is "**Corporate Part Number**" which would be used for your company's part numbering system. Once you create an Attribute, it will appear every time you build a new CAD land pattern.

Select the **OK** button when the new attribute is entered.

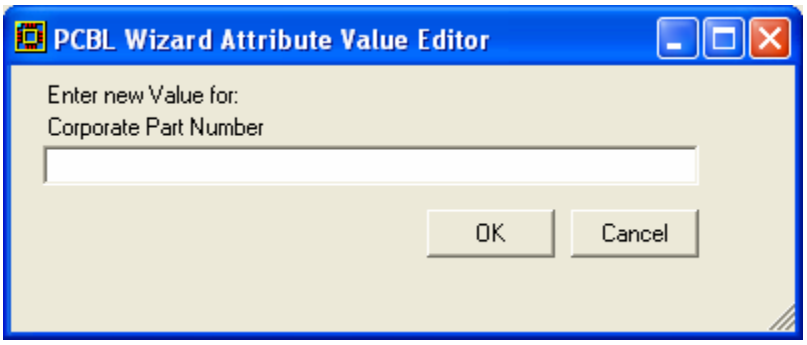




Select the Close Tool Bar button and the “Save New Settings” box will appear.

When you want to enter a value to an attribute **Double Click on the Attribute** you want to edit “Corporate Part Number”.

When you Double Click on any Attribute, the “Enter new Value for:” dialog box will appear



The default Attributes cannot be removed because the IPC-7351 LP Wizard depends on their existence to function properly. The **Description**, **Manufacturer** and **Mfr Part Number** are self explanatory but some Attributes need an explanation.

The “**PCBL.Component**” attribute is the Component Dimensions to view the Component in the Calculator Viewer. Once this attribute is defined, you can create a new decal automatically.

The “**PCBL.Process**” attribute is the IPC-7351 Tolerance Dimensions and Settings used to define the rules (Solder Joint Goals) and the environment of the Land Pattern.

The “**Geometry.Height**” attribute will create the side view of the component and provide the 3D model data for Solid Works and PRO Engineer mechanical drafting software.

The “**Standard Name**” attribute is the official JEDEC or EIA component package name. Many component manufacturers list the JEDEC package name in their component data sheet PDF file. This allows you to search for the Standard JEDEC part number to locate the correct land pattern data.

The “**Also Known As**” (AKA) attribute is the alternate name of the Land Pattern. This Attribute really helps Users with transitioning to the new IPC-7351 Land Pattern Naming Convention. The User can enter the old legacy land pattern name in this field and use it as a search “Keyword” until you become familiar with the Standard IPC-7351 Land Pattern Naming Convention.

The “**On-Line-Link**” attribute is to access the component manufacturer’s PDF file. This Attribute is invaluable to all Users who want quick access to the original component manufacturer’s data sheet. This also helps track component obsolescence. When the component manufacturer’s web-link to a datasheet no longer exists, most of the time it’s because the component manufacturer discontinued making the part.

The basic concept behind keeping good attributes is so you can easily locate all the parts that are currently in the library and all the parts that you create in the future.

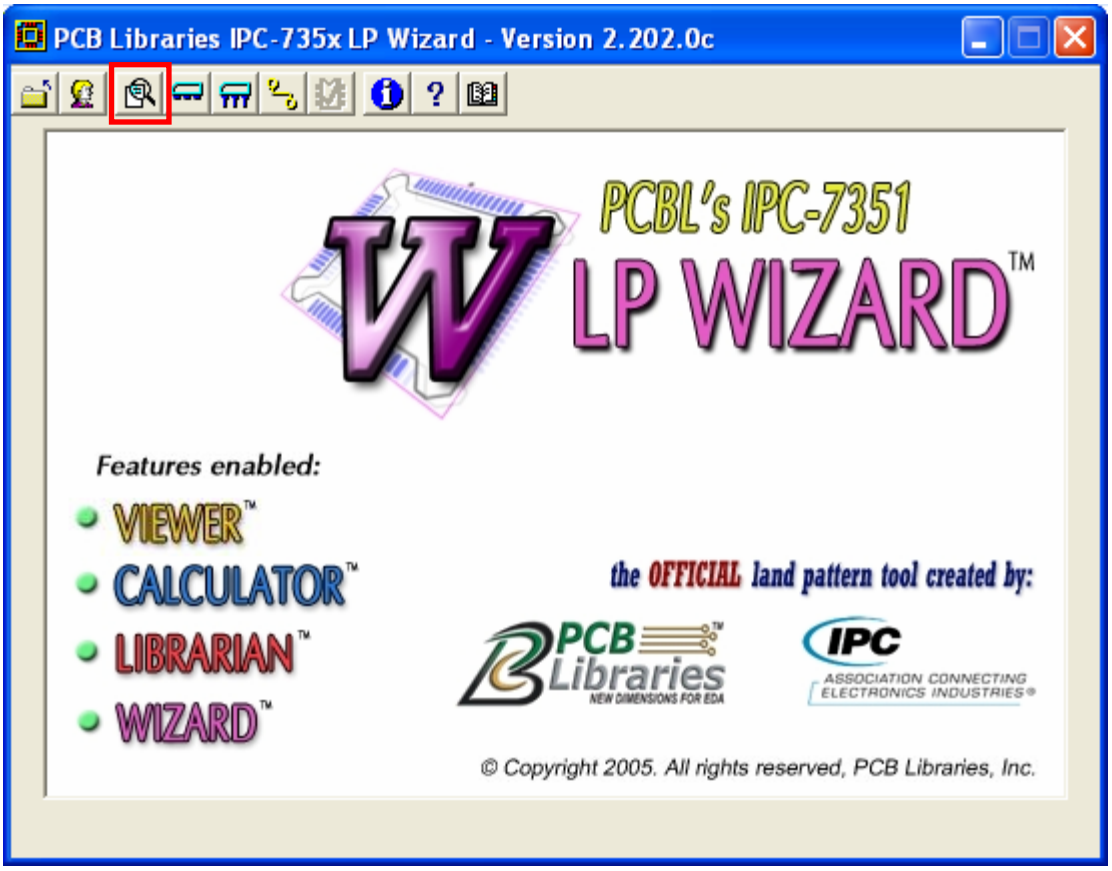
This is not only for your personal benefit, but it will help others co-workers and/or customers quickly locate data that they are looking for when using the free IPC-7351 LP Viewer.

The IPC-7351 LP Viewer is your “Library Documentation” and the better your “Attributes” are defined the better your library documentation will be.

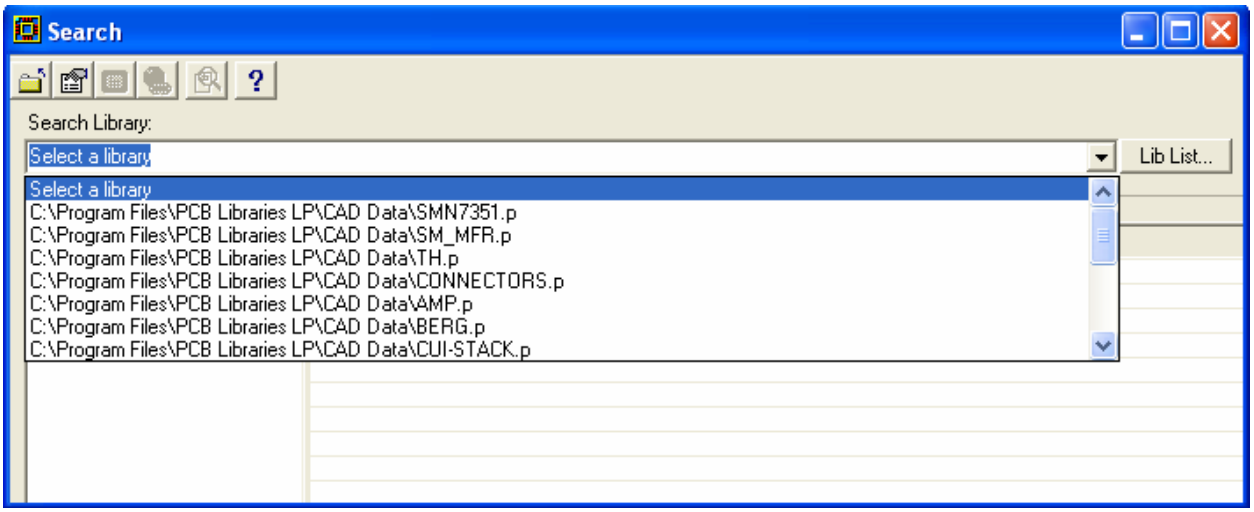
4 SEARCH LIBRARY

4.1 Opening the Search Library Menu

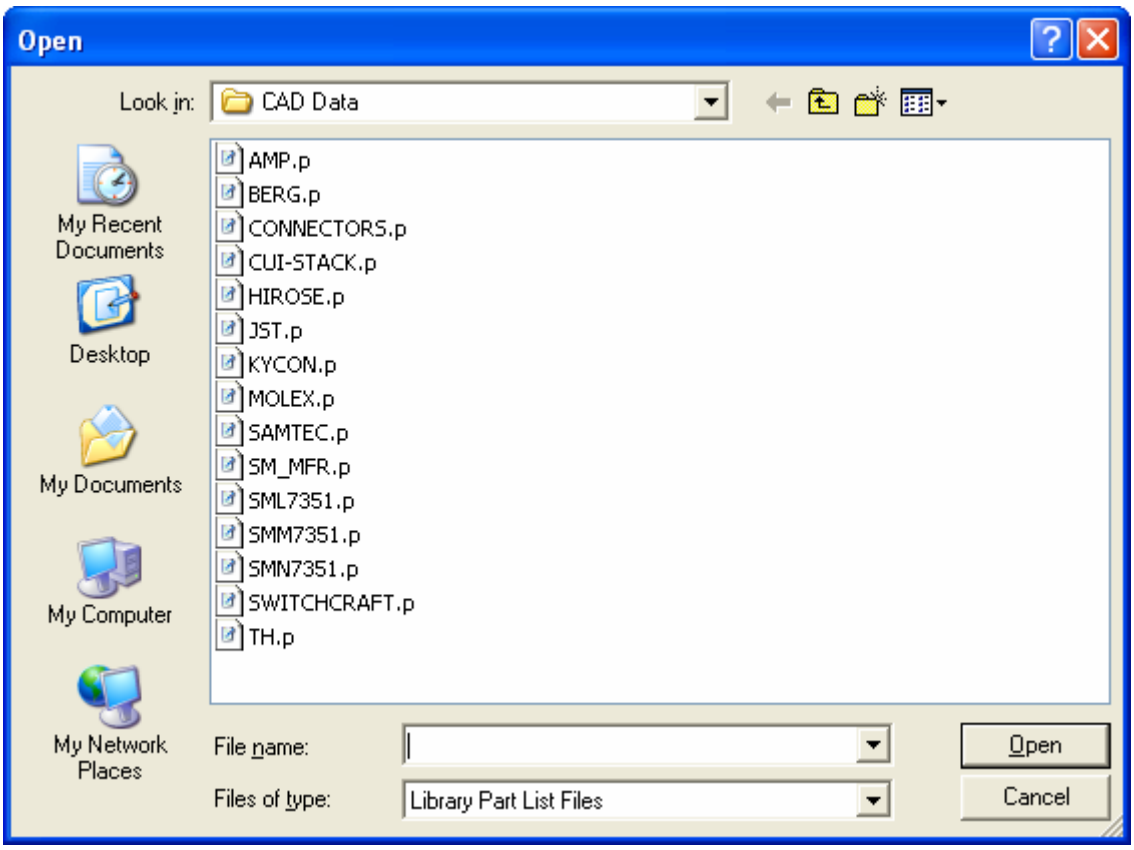
To open the Search Library Menu, Select the "**Search Library**" **Tool Bar** Button



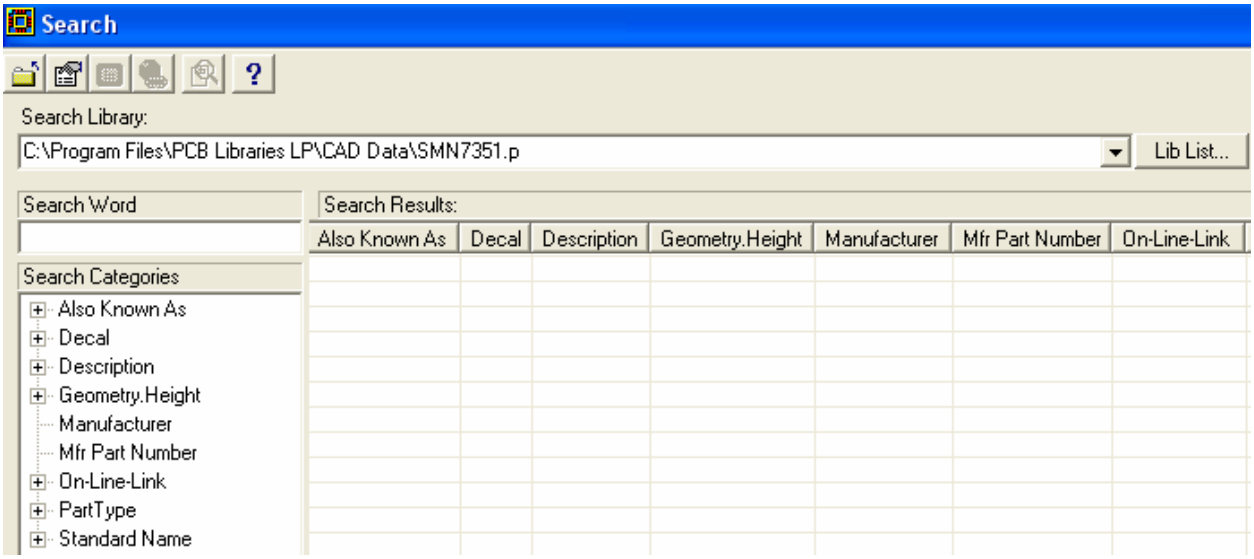
Select the pull down button "**Select a Library**" and select the file **SMN7351.p**



Or you can alternatively use the "**Lib List**" Button to activate the Library List and set the default search path for your Library .P files. Select "**SMN7351.p**" & "**Open**" button.



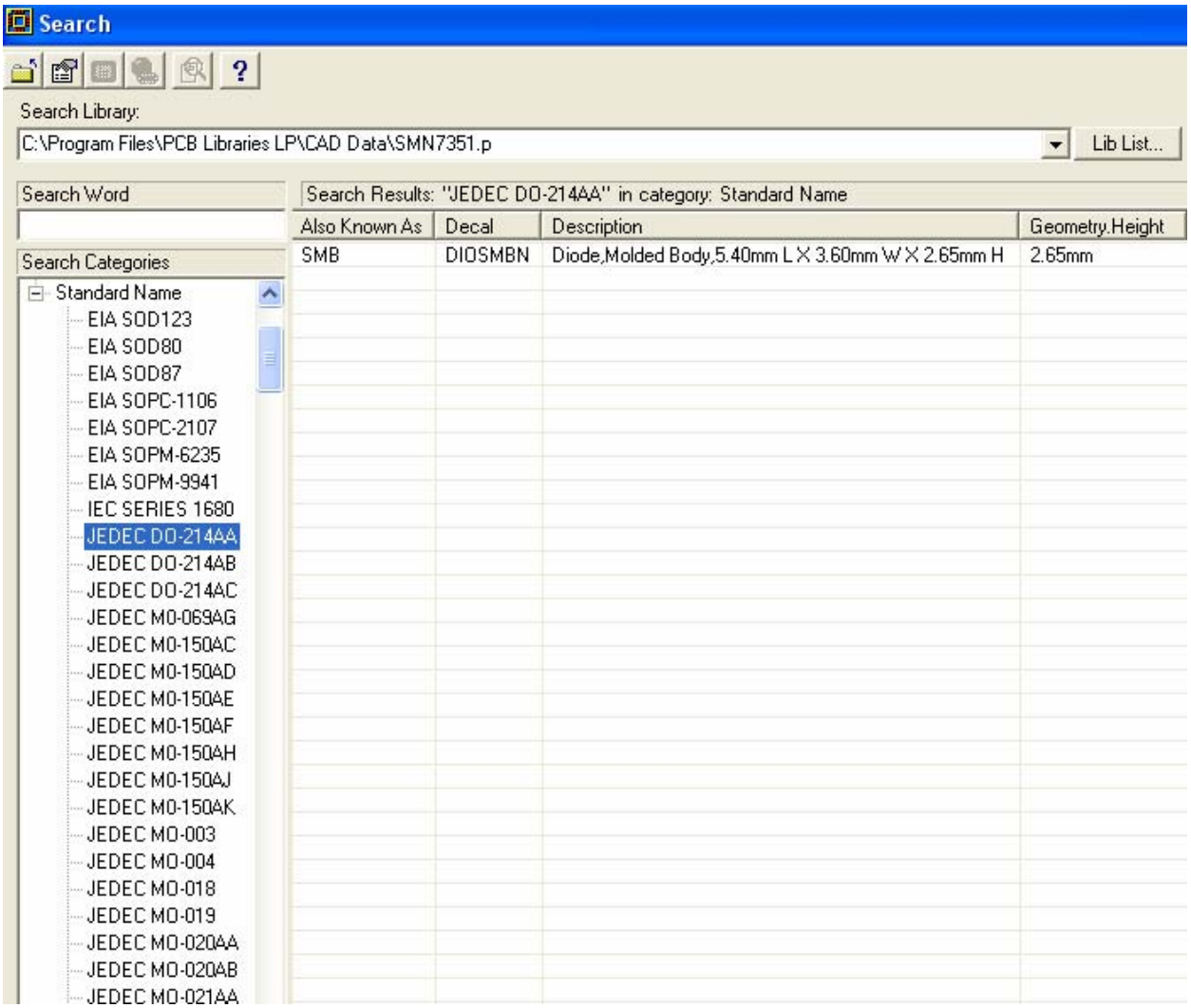
You can Search by any of the below hierarchical descriptions.



In this example we'll select the "**Standard Name**" section. After you select the Standard Name, select the first item (EIA) DO-214AA & then select the **Search Tool Bar** and the contents of the search will show up in the "Search Results" fields.

Or you can select any item using the "**Double Click**" feature to load Search Results.

For this exercise Double Click on item **EIA DO-214AA**.

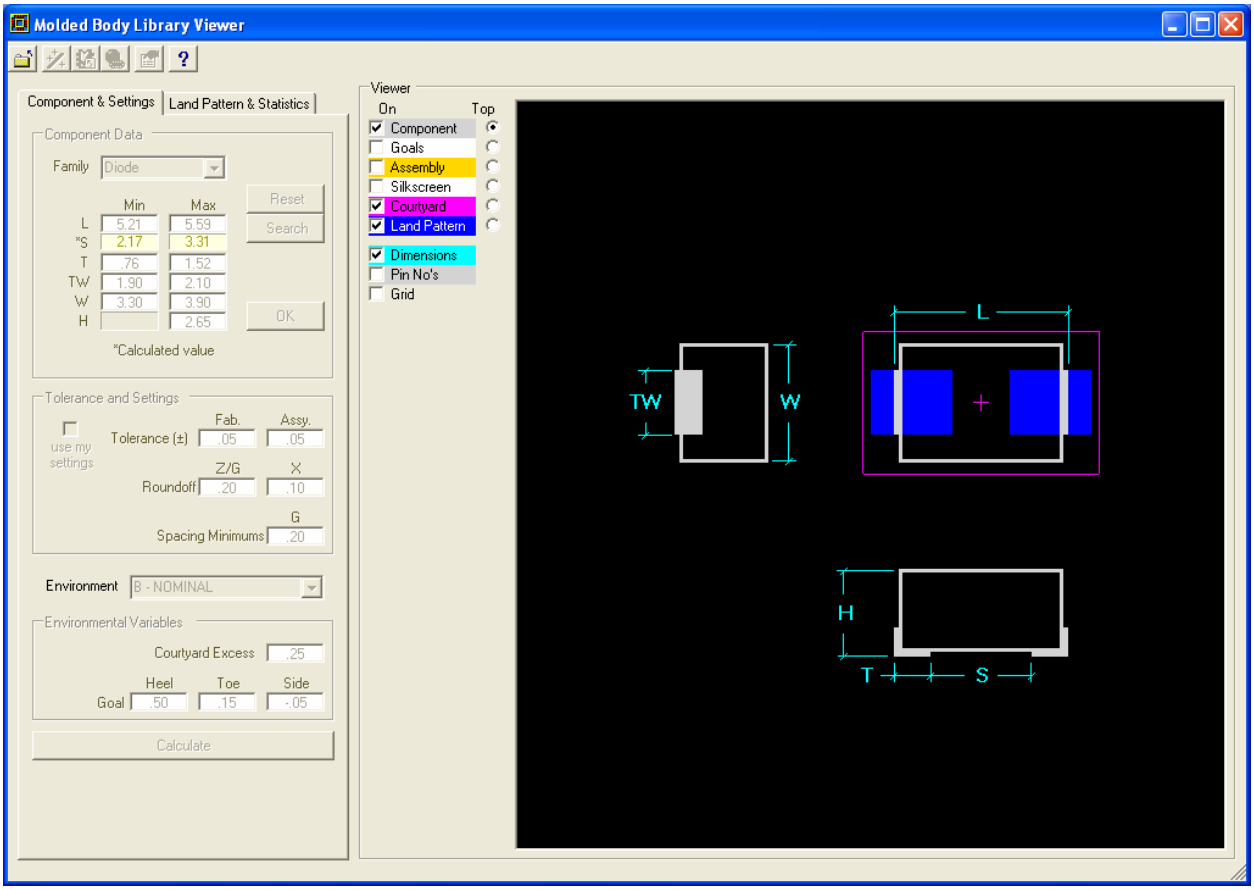


You can also use the **Search** **Tool Bar Button** technique for searching for all parts.

You can also enter a "**Search Word**" and then select the **Search** **Tool Bar** Button. The Search Word feature works when you select any "Search Category" and the search will be limited to that category. If you do not select a Category, then the search will look through all the Categories.

Use the **Viewer** **Tool Bar** Button to open the LP Viewer or **Double Click** on the text line to see the component and land pattern dimensional data.

Use the "**Viewer / Top**" buttons to toggle through the various dimensions.

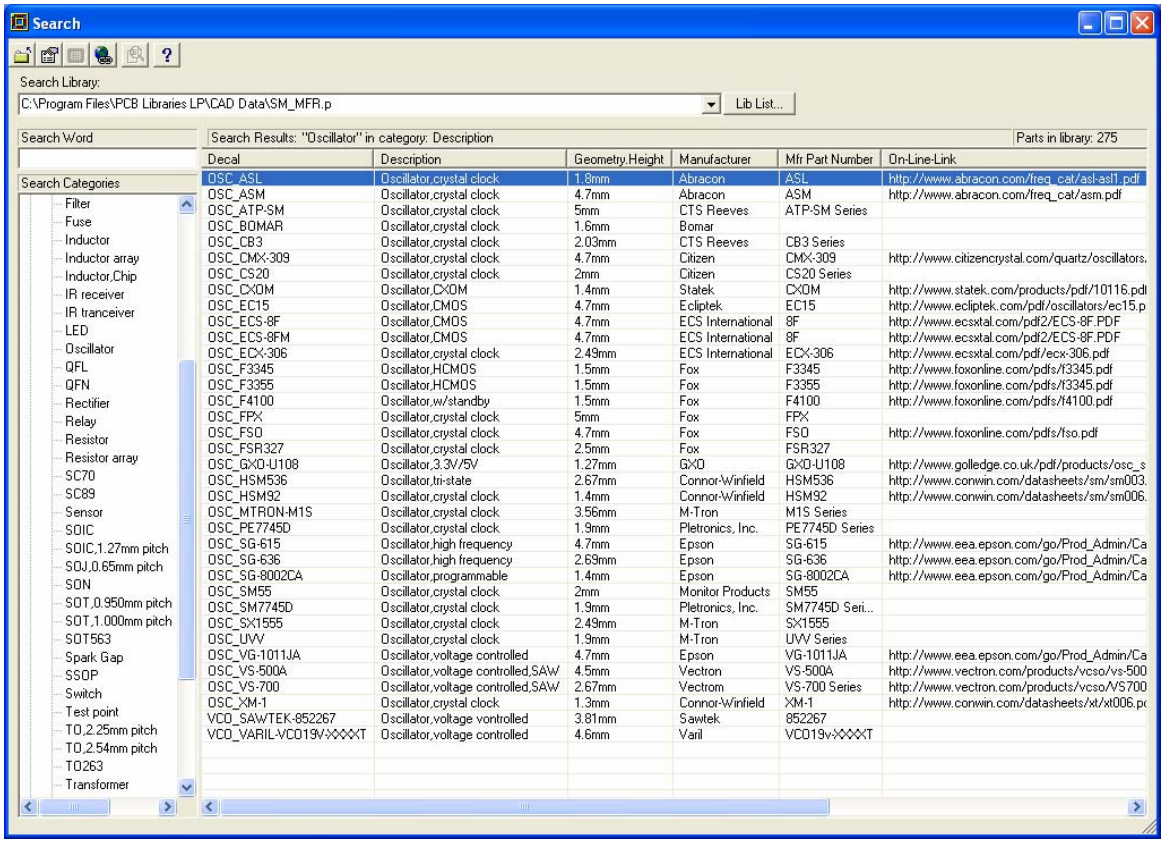


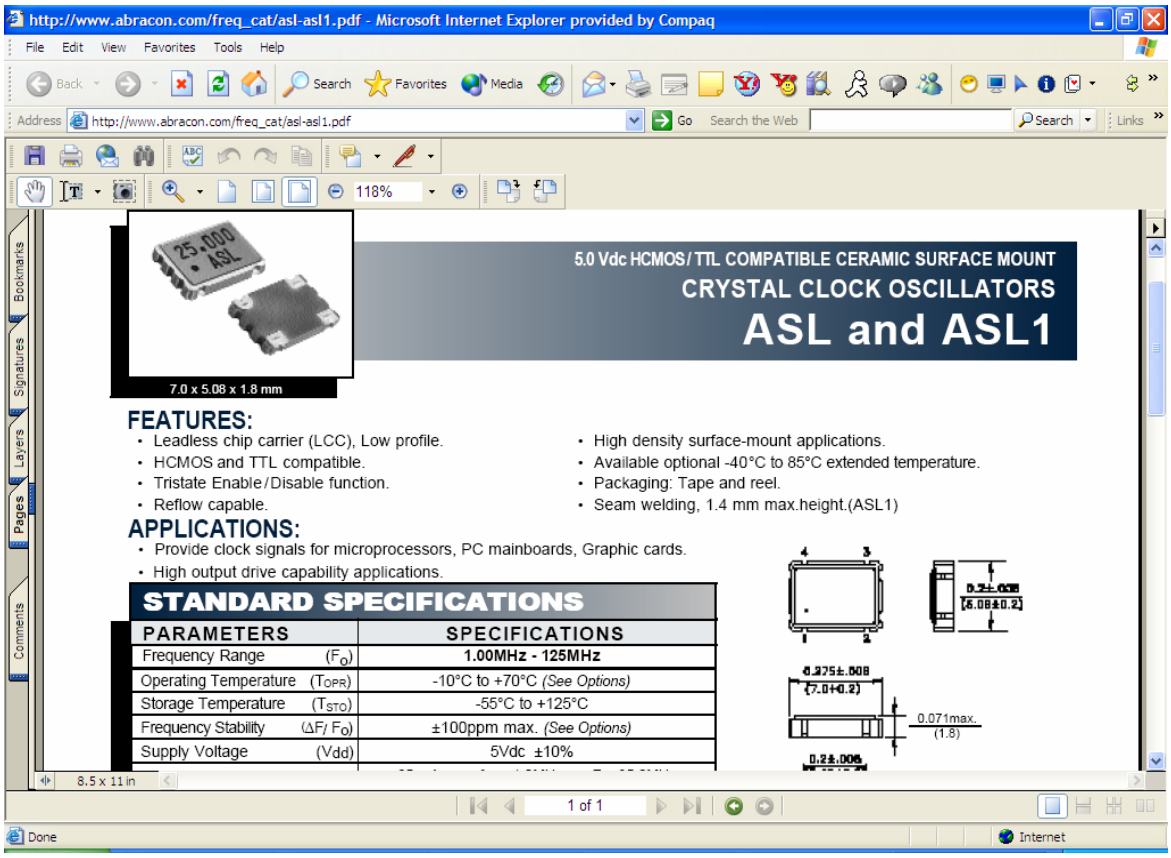
Close the Viewer menu when you want to search for another component.

Each Library has its own independent Search Engine because the attributes may vary. An example of this is SMN7351 library contains only “Standard Component Packages” and therefore will not have the ON-LINE-LINK attribute. But most parts in the SMN_MFR Library will have the ON-LINE-LINK attribute because each part is manufacturer specific and the component manufacturers URL is available so you can download the latest PDF file datasheet.

Open the SMN_MFR library and search for an **Oscillator**. Select Keyword / Oscillator / the **Search** **Tool Bar** Button and every Oscillator in the library will appear in the Search Results Window.

Select any one of the Crystals and then select the **URL Link** **Tool Bar** Button or **Double Click** on the Line Text and the LP Viewer will open the manufacturers PDF file from the component manufacturer’s website. Use this same feature on any component in any of the 10 Connector Libraries and the “TH” Through-hole library.

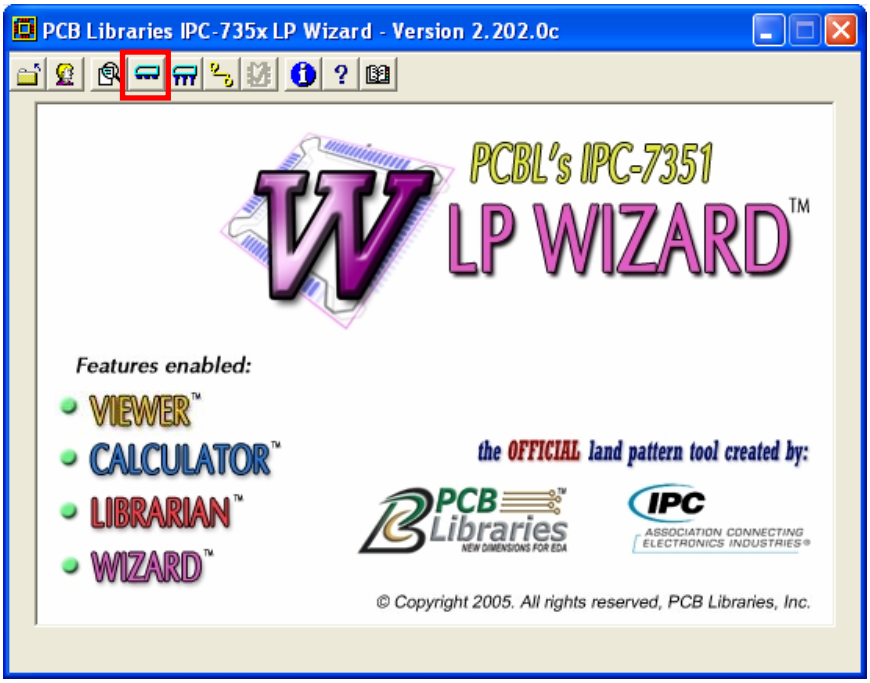

Search
Search Library:
C:\Program Files\PCB Libraries LP\CAD Data\SM_MFR.p
Lib List...
Search Word
Search Categories
Search Results: "Oscillator" in category: Description
Parts in library: 275
Decal
Description
Geometry.Height
Manufacturer
Mfr Part Number
On-Line-Link
OSC_ASL Oscillator,crystal clock 1.8mm Abracon ASL http://www.abracon.com/freq_cat/asl-asl1.pdf
OSC_ASM Oscillator,crystal clock 4.7mm Abracon ASM http://www.abracon.com/freq_cat/asm.pdf
OSC_ATP-SM Oscillator,crystal clock 5mm CTS Reeves ATP-SM Series
OSC_BOMAR Oscillator,crystal clock 1.6mm Bomar
OSC_CB3 Oscillator,crystal clock 2.03mm CTS Reeves CB3 Series
OSC_CMX-309 Oscillator,crystal clock 4.7mm Citizen CMX-309 http://www.citizencrystal.com/quartz/oscillators.
OSC_CS20 Oscillator,crystal clock 2mm Citizen CS20 Series
OSC_CXOM Oscillator,CXOM 1.4mm Statek CXOM http://www.statek.com/products/pdf/10116.pdf
OSC_EC15 Oscillator,CMOS 4.7mm Ecliptek EC15 http://www.ecliptek.com/pdf/oscillators/ec15.p
OSC_ECS-8F Oscillator,CMOS 4.7mm ECS International 8F http://www.ecsxtal.com/pdf2/ECS-8F.PDF
OSC_ECS-8FM Oscillator,CMOS 4.7mm ECS International 8F http://www.ecsxtal.com/pdf2/ECS-8F.PDF
OSC_ECX-306 Oscillator,crystal clock 2.49mm ECS International ECX-306 http://www.ecsxtal.com/pdf/ecx-306.pdf
OSC_F3345 Oscillator,HCMOS 1.5mm Fox F3345 http://www.foxonline.com/pdfs/f3345.pdf
OSC_F3355 Oscillator,HCMOS 1.5mm Fox F3355 http://www.foxonline.com/pdfs/f3345.pdf
OSC_F4100 Oscillator,w/standby 1.5mm Fox F4100 http://www.foxonline.com/pdfs/f4100.pdf
OSC_FPX Oscillator,crystal clock 5mm Fox FPX
OSC_FSO Oscillator,crystal clock 4.7mm Fox FSO http://www.foxonline.com/pdfs/fso.pdf
OSC_FSR327 Oscillator,crystal clock 2.5mm Fox FSR327
OSC_GXO-U108 Oscillator,3.3V/5V 1.27mm GXO GXO-U108 http://www.golledge.co.uk/pdf/products/osc_s
OSC_HSM536 Oscillator,tri-state 2.67mm Connor-Winfield HSM536 http://www.conwin.com/datasheets/sm/sm003.
OSC_HSM92 Oscillator,crystal clock 1.4mm Connor-Winfield HSM92 http://www.conwin.com/datasheets/sm/sm006.
OSC_MTRON-M1S Oscillator,crystal clock 3.56mm M-Tron M1S Series
OSC_PE7745D Oscillator,crystal clock 1.9mm Pletronics, Inc. PE7745D Series
OSC_SG-615 Oscillator,high frequency 4.7mm Epson SG-615 http://www.eea.epson.com/go/Prod_Admin/Ca
OSC_SG-636 Oscillator,high frequency 2.69mm Epson SG-636 http://www.eea.epson.com/go/Prod_Admin/Ca
OSC_SG-8002CA Oscillator,programmable 1.4mm Epson SG-8002CA http://www.eea.epson.com/go/Prod_Admin/Ca
OSC_SM55 Oscillator,crystal clock 2mm Monitor Products SM55
OSC_SM7745D Oscillator,crystal clock 1.9mm Pletronics, Inc. SM7745D Seri...
OSC_SX1555 Oscillator,crystal clock 2.49mm M-Tron SX1555
OSC_UVV Oscillator,crystal clock 1.9mm M-Tron UVV Series
OSC_VG-1011JA Oscillator,voltage controlled 4.7mm Epson VG-1011JA http://www.eea.epson.com/go/Prod_Admin/Ca
OSC_VS-500A Oscillator,voltage controlled,SAW 4.5mm Vectron VS-500A http://www.vectron.com/products/vcso/vs-500
OSC_VS-700 Oscillator,voltage controlled,SAW 2.67mm Vectron VS-700 Series http://www.vectron.com/products/vcso/VS700
OSC_XM-1 Oscillator,crystal clock 1.3mm Connor-Winfield XM-1 http://www.conwin.com/datasheets/xt/xt006.p
VCO_SAWTEK-852267 Oscillator,voltage vontrolled 3.81mm Sawtek 852267
VCO_VARIL-VC019V-XXXXT Oscillator,voltage controlled 4.6mm Varil VC019v-XXXXT
Filter
Fuse
Inductor
Inductor array
Inductor,Chip
IR receiver
IR tranceiver
LED
Oscillator
QFL
QFN
Rectifier
Relay
Resistor
Resistor array
SC70
SC89
Sensor
SOIC
SOIC,1.27mm pitch
SOJ,0.65mm pitch
SON
SOT,0.950mm pitch
SOT,1.000mm pitch
SOT563
Spark Gap
SSOP
Switch
Test point
TO,2.25mm pitch
TO,2.54mm pitch
TO263
Transformer


http://www.abracon.com/freq_cat/asl-asl1.pdf - Microsoft Internet Explorer provided by Compaq
File Edit View Favorites Tools Help
Address http://www.abracon.com/freq_cat/asl-asl1.pdf
5.0 Vdc HCMOS/TTL COMPATIBLE CERAMIC SURFACE MOUNT
CRYSTAL CLOCK OSCILLATORS
ASL and ASL1
7.0 x 5.08 x 1.8 mm
FEATURES:
• Leadless chip carrier (LCC), Low profile.
• HCMOS and TTL compatible.
• Tristate Enable/Disable function.
• Reflow capable.
• High density surface-mount applications.
• Available optional -40°C to 85°C extended temperature.
• Packaging: Tape and reel.
• Seam welding, 1.4 mm max.height.(ASL1)
APPLICATIONS:
• Provide clock signals for microprocessors, PC mainboards, Graphic cards.
• High output drive capability applications.
STANDARD SPECIFICATIONS
PARAMETERS SPECIFICATIONS
Frequency Range (F0) 1.00MHz - 125MHz
Operating Temperature (TOPR) -10°C to +70°C (See Options)
Storage Temperature (TSTO) -55°C to +125°C
Frequency Stability (ΔF/ F0) ±100ppm max. (See Options)
Supply Voltage (Vdd) 5Vdc ±10%
0.071max.
(1.8)
1 of 1
Done
Internet

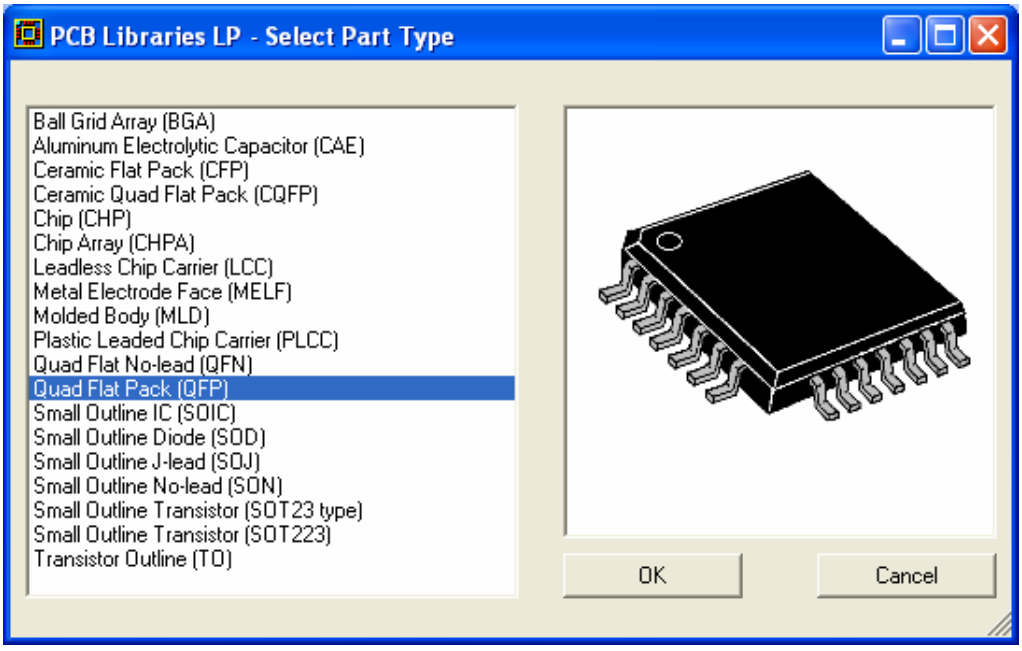
5 SMD CALCULATOR GUI

5.1 Opening the Calculator GUI

To open any Calculator Menu select the "**SMD Calculator**" Tool Bar button. There currently are 18 different SMD component families.



In this illustration we're selecting the QFP component family.



Remember that every feature that is explained in this session is applicable to every component family.

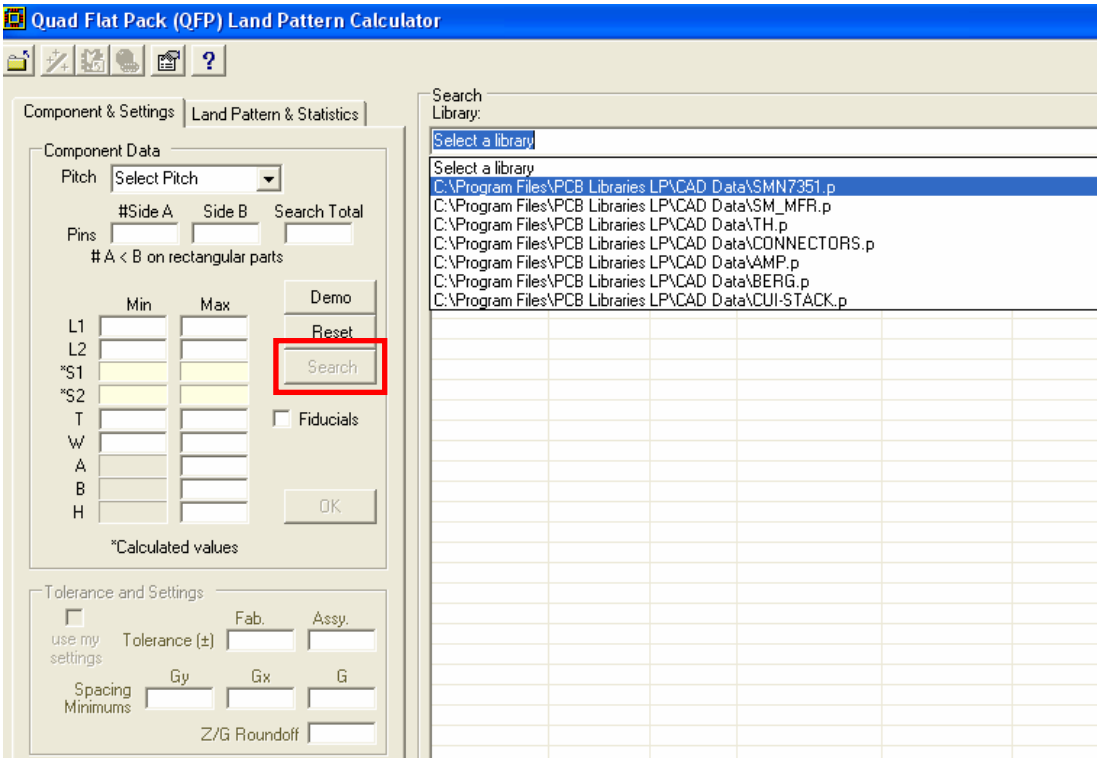
The Calculator GUI is broken up into 5 sections:

1. The Component Dimensional Data
2. The Tolerance and Settings
3. The Environmental Variables
4. The Land Pattern Dimensions and Land Pattern Name
5. The Browser Window

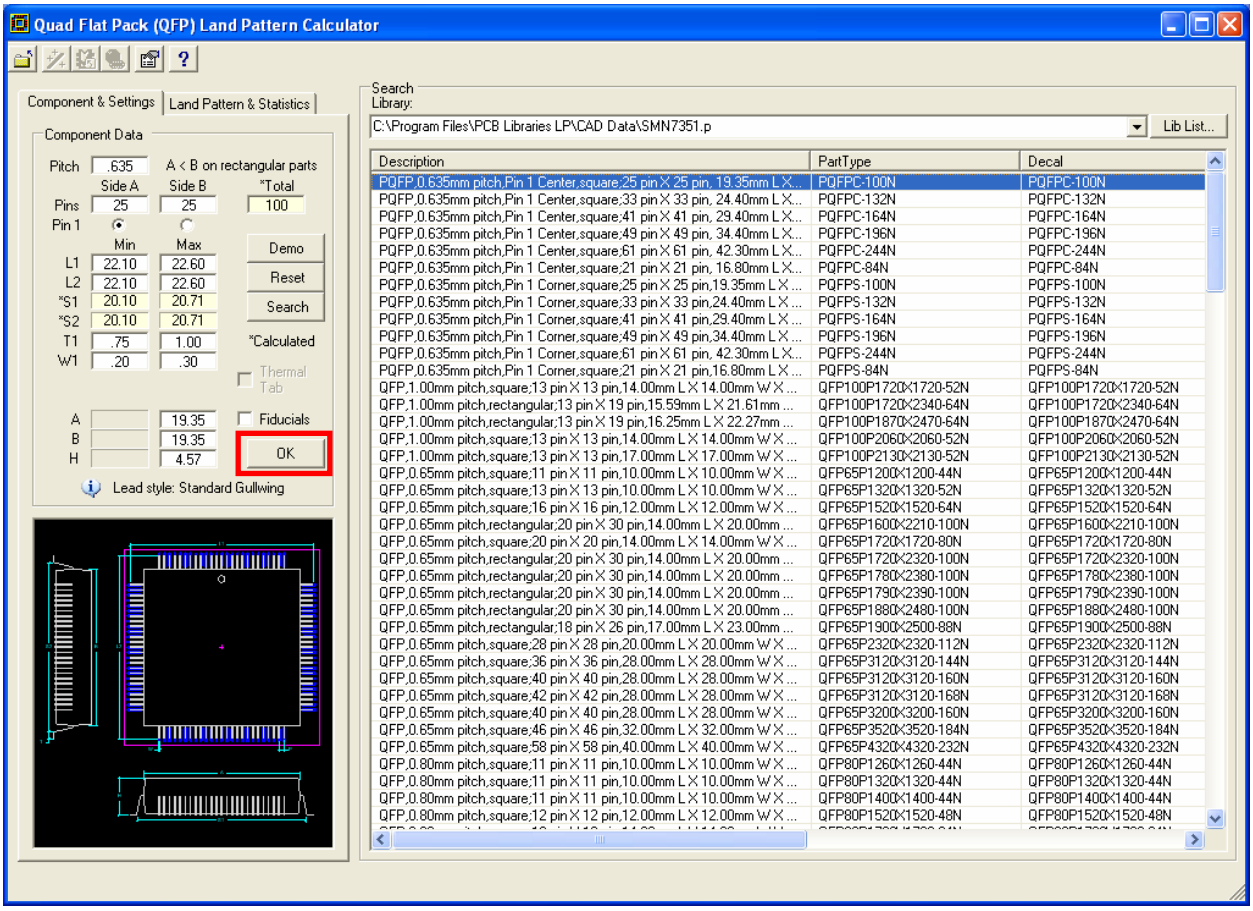
5.2 Loading a Library

To access existing Library Data, we first must load a library into the program.

Select the "**Search**" pull down button or select the "**Lib List**" button and select SMN7351.p file.



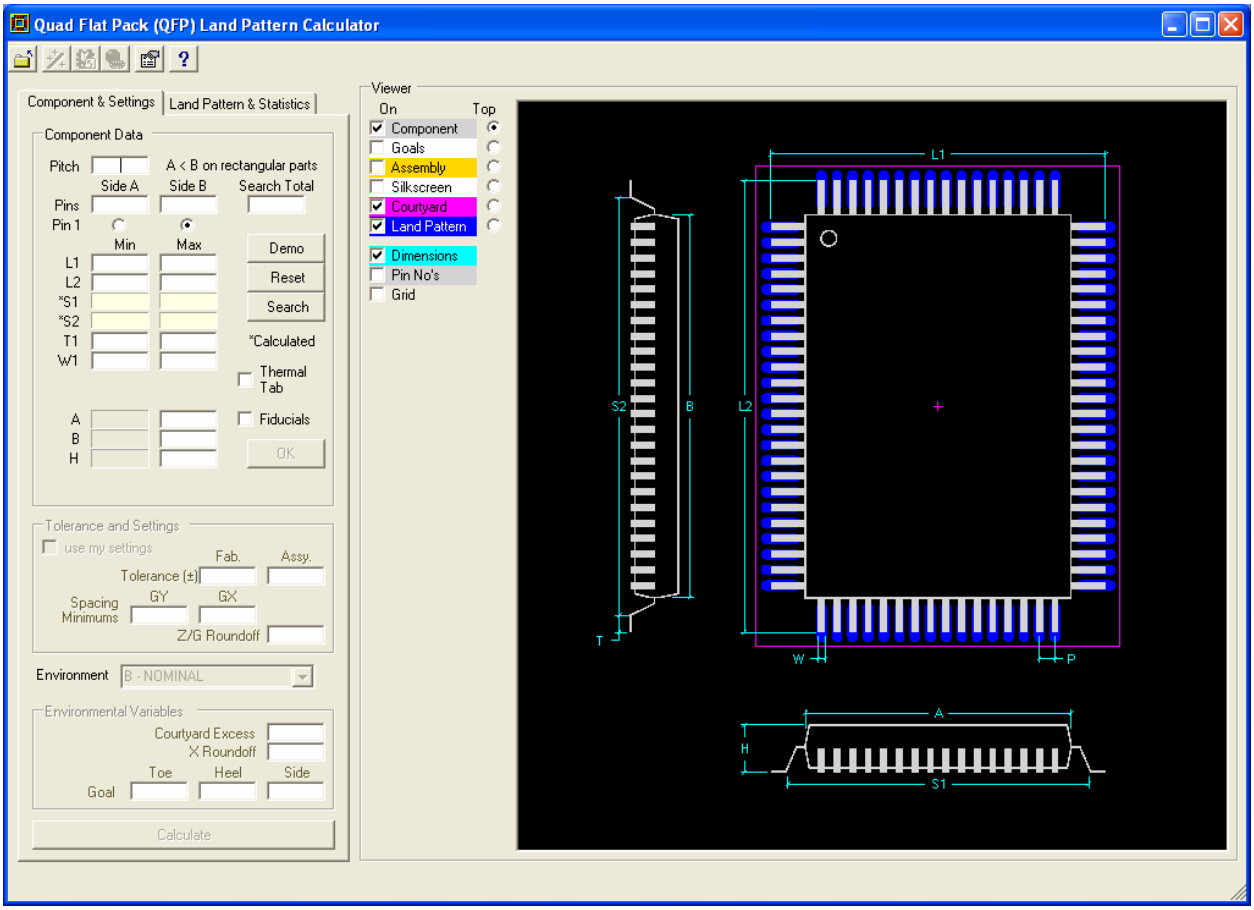
This is what should appear (234 QFP components). Select any item and then select the **OK** button to load the component and land pattern data into the Viewer.



5.3 QFP Calculator GUI

On the right side is the “Component Dimensional Data” and includes:

1. A pull-down to select the “**Pin Pitch**” (1, 0.8, 0.65, 0.635, 0.5, 0.4 & 0.3mm)
2. Number of Pins on “**Side A**” (top & Bottom) and on “**Side B**” (left & right)
3. “**Search Total**” is the total Pin Quantity of the component. “**Search Total**” may also display “***Total**”. “**Search Total**” is the total Pin Quantity to search for if a search is to be made. “***Total**” is the calculated number of pins based on the “Side A” & “Side B” entries.
4. The other blank fields are for the various component dimensions
5. The “**Demo Button**” will load dimensions from an existing QFP part
6. The “**Reset Button**” removes or clears all dimensional data
7. The “**Search Button**” is used for searching for exiting QFP components that are stored in the program memory
8. The “**Fiducial**” check box is used when your QFP requires Local Fiducials



Under the Component Data is the “Tolerance and Settings”.
Under the Tolerance and Settings is the “Environmental Variables”.
Land Pattern & Statistics “Land Pattern Dimensions” and “IPC-7351 Land Pattern Name”

5.4 Using the Viewer Feature

For this tutorial, select the “**Reset & Demo Button**” in the Component Data box.

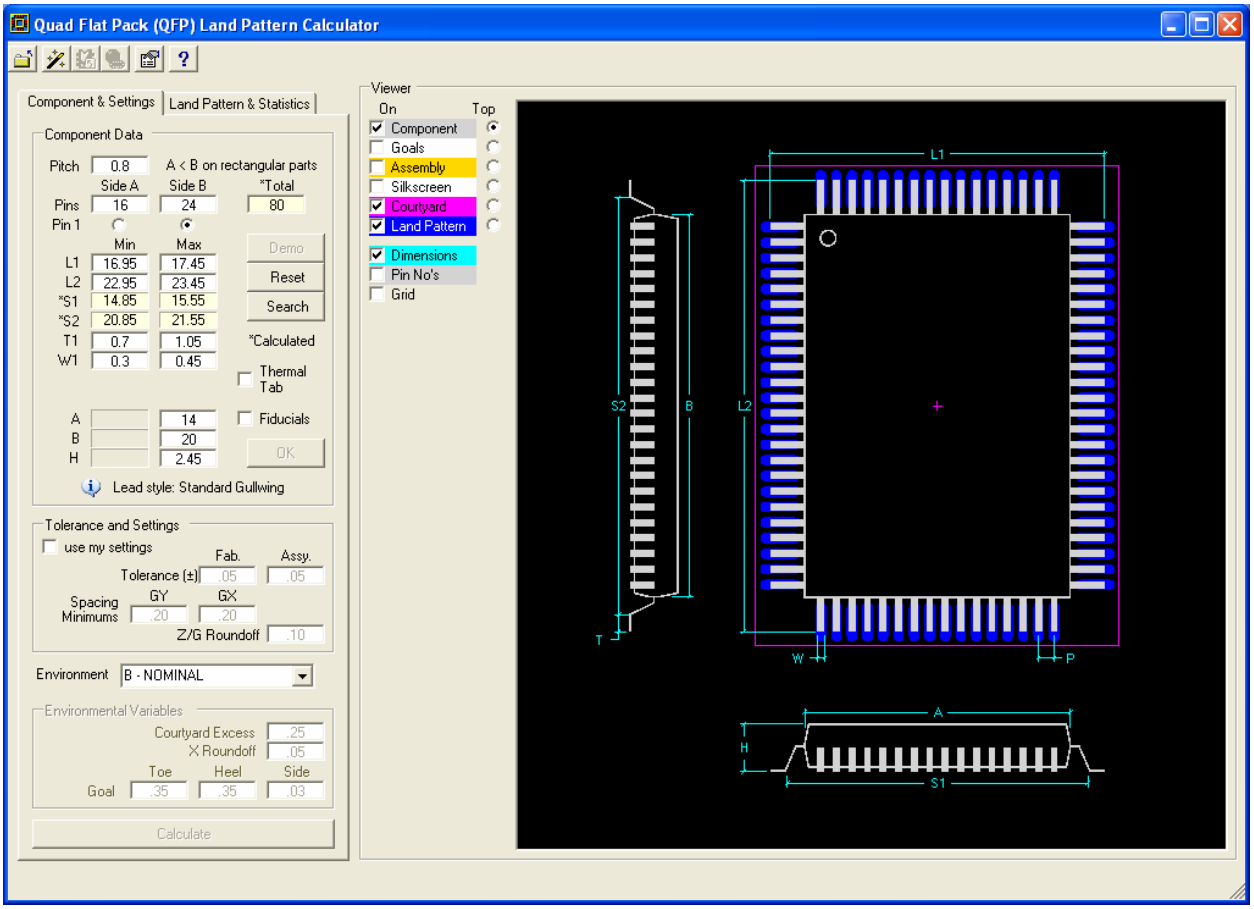
The **Viewer** feature turns on/off dimensional data for the items listed in the Viewer

Select any button on the right side of the menu under the word “**Top**” the new dimensional data will appear in the browser.

Selecting the Dimensions or Pin No’s check box will toggle those features.

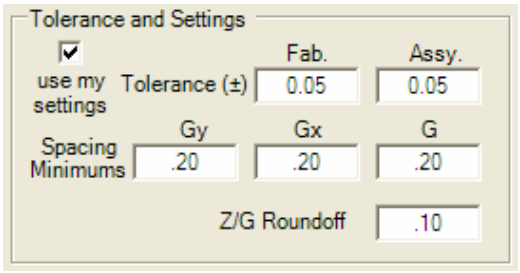
Note: The “**Goals**” selection illustrates where the metal portion of the component lead falls on the solder pad. The Goal dimensional data includes Gx & Gy which represents the minimum spacing between pads. The default minimum spacing setting for all component pins is 0.2mm. If the IPC formula creates a spacing less than 0.2mm a warning will appear notifying you that the “Pads are being Trimmed”. The importance of maintaining a 0.2mm pad to pad spacing is to allow for a solder mask sliver to be manufactured in-between the pads to avoid solder bridging.

Select every "**Top**" item button so you can see every dimension for every feature in the browser. When you select the "Land Pattern & Statistics Tab" the dimensions will change.



5.5 Tolerance and Settings

The Tolerance and Settings menu allow you to use the IPC default values or customize your own personal settings for creating each Land Pattern.



The Fabrication Tolerance allows for copper etch-back on the outer layers. The Assembly Tolerance allows for pick & place allowances. These are both set to 0.05mm and calculated in a formula along with the component terminal tolerance.

CL = Component Lead Tolerance provided by each component manufacturer

F = Fabrication Tolerance

P = Placement (Assembly) Tolerance

As you can see in the table below, the tolerances do not accumulate but are squared, added and then square root of the total.

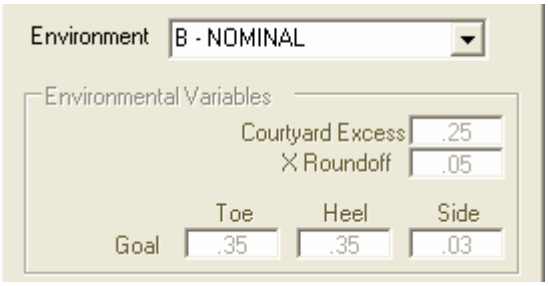
Land Pattern Equations

- $Z_{max} = L_{min} + 2J_T + \sqrt{C_L^2 + F^2 + P^2}$
- $G_{min} = S_{max} - 2J_H - \sqrt{C_L^2 + F^2 + P^2}$
- $X_{max} = W_{min} + 2J_S + \sqrt{C_L^2 + F^2 + P^2}$

- ***where***
  - ***Z is the overall length of land pattern;***
  - ***G is the distance between lands of the pattern;***
  - ***X is the width of land pattern;***

5.6 Environmental Variables

The Environmental Variables take into consideration the Solder Joint Goals for a Three Tier Library System. Each Tier has different Toe, Heel and Side Goals.



The Environment pull-down has four selections: A – Most, B – Nominal, C – Least, and USER, which will allow you to set your own Goals for each Land Pattern.

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 the user should consider product qualification testing based on the conditions shown in Table 3-13.

Density Level A: Maximum (Most) Land Protrusion –

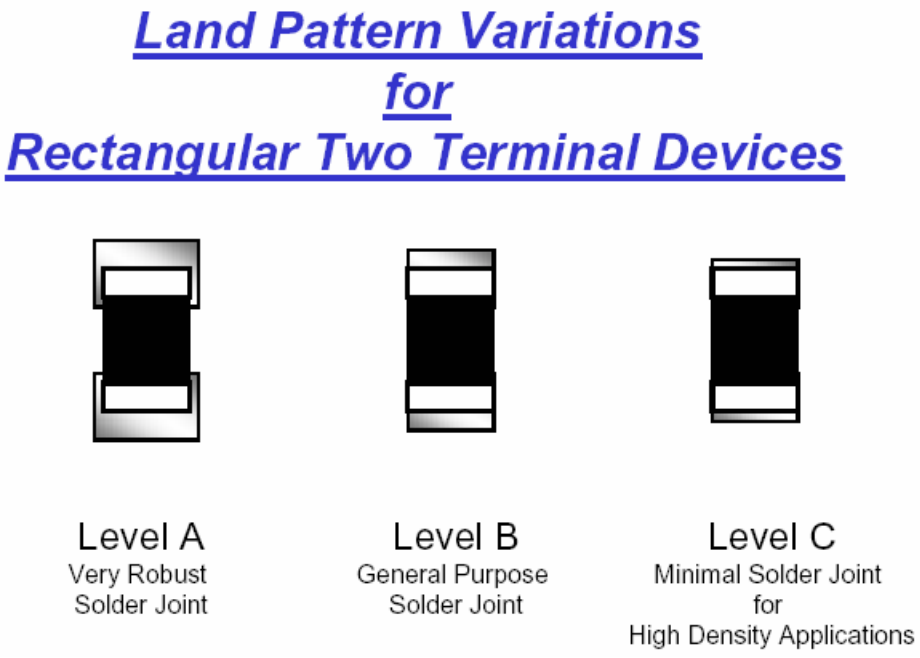
For low-density product applications, the 'maximum' land pattern condition has been developed to accommodate wave or flow solder of leadless chip devices and leaded gull- wing devices. The geometry furnished for these devices, as well as inward and “J”-formed lead contact device families, may provide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

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.

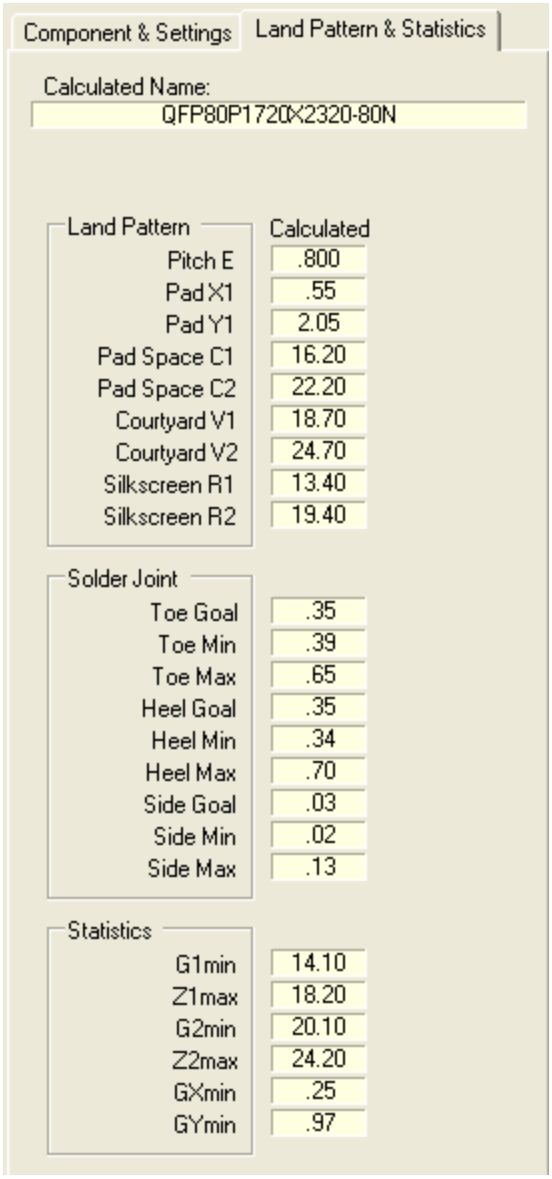
Density Level C: Minimum (Least) Land Protrusion –

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.



5.7 Land Pattern Data

The Land Pattern Data is derived directly from the component dimensions.



The Land Pattern Name is also derived from the component dimensions and in this example it's broken into 6 parts:

1. The component family prefix – **QFP**
2. The component pin pitch E = .80mm for "**80P**"
3. The component lead span nominal X = 17.20mm for "**1720**"
4. The component lead span nominal Y = 23.20mm for "**2320**"
5. The total component Pin Qty = 80 pins for "**80**"
6. The Environmental Variable = "**N**" for nominal
7. End result = **QFP80P1720X2320-80N**

See the document "IPC-7351 Land Pattern Naming Convention.pdf" for the complete naming convention for every component family or use the "Help" menu.

Users may wish to modify names that would otherwise be duplicated without allowing for variables that aren't accounted for by this convention…height, for example.

When you change any existing component data or enter in new component data, the "**Calculate Button**" must be selected to calculate new land pattern dimensional data.

The R1 and R2 dimensions are for the Silkscreen

The V1 and V2 dimensions are for the Placement Courtyard

The C1 and C2 dimensions are for the Pin Center to Center spacing

The X is the Pad Width

The Y is the PAD Length

The E is the Pin Pitch

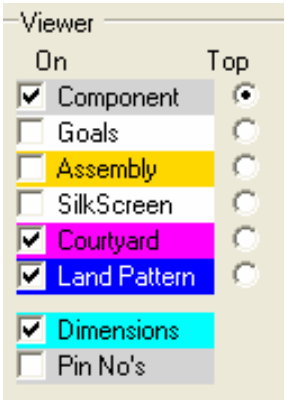
The Zmax is the Outer most dimension for the solder pad

The Gmin is the Inner most dimension for the solder pad

The Gx is the minimum pin to pin spacing between the pins

The Gy is the minimum pin to pin spacing in the QFP corners

5.8 Pan and Zoom in the Browser

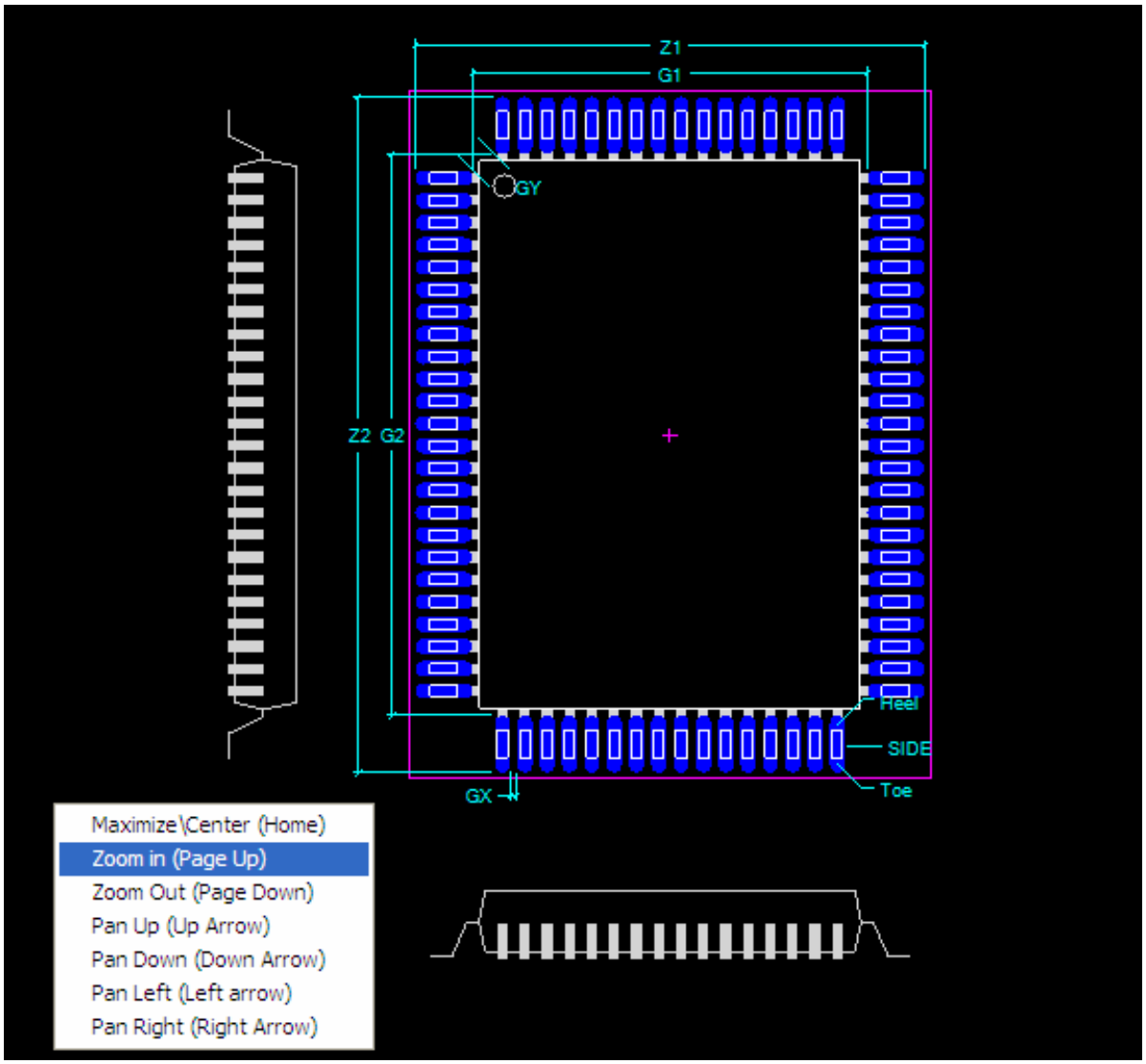


To turn on the Solder Joint Goals select the "**Goals – Top**" Button.

The Browser has a Pan and Zoom feature that is User friendly.

Use the RMB (Right Mouse Button) or the Numeric Key Pad on your keyboard.

For this session, use the **Pg Up**, **Pg Dn**, **Left**, **Right**, **Top**, **Bottom Arrows** & **Home**



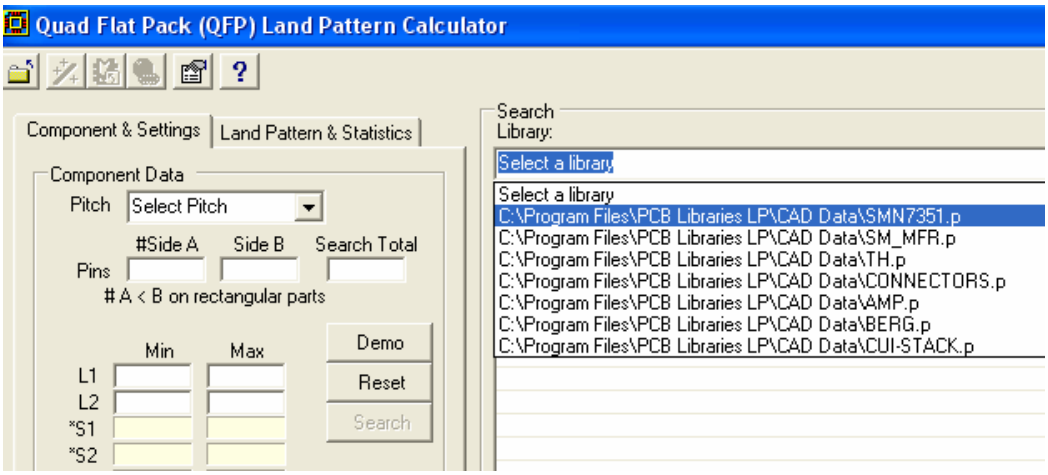
The above example is using the Right Mouse Button in the Browser Area.

6 SMD CALCULATOR SEARCH

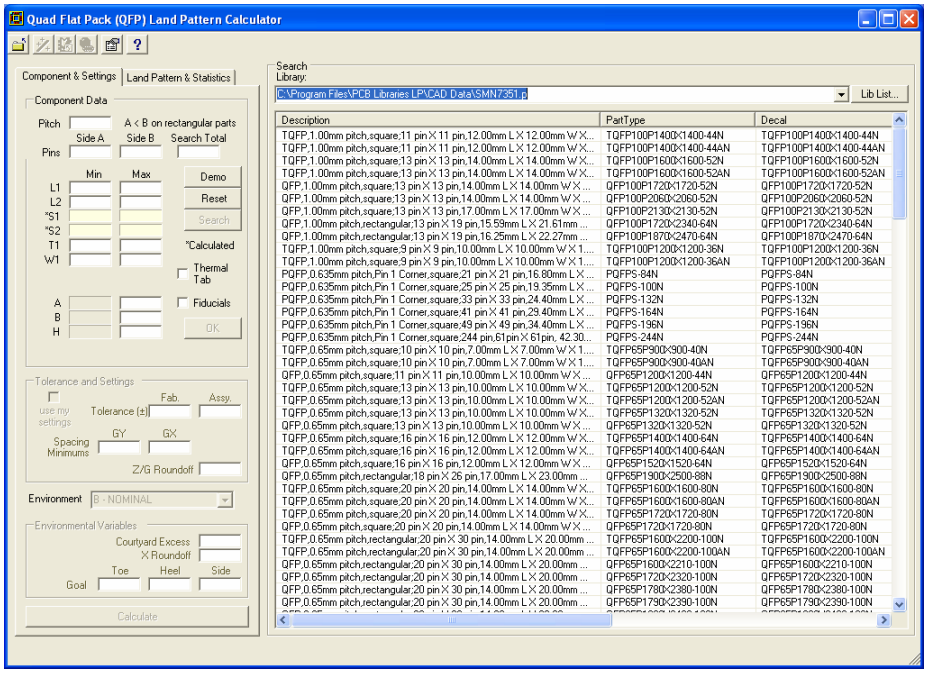
6.1 Searching for Existing Components

Select the "**Reset Button**" to clear all the dimensional data.

The Calculator has a powerful search engine that allows the user to enter data into the component fields and then select the "**Search Button**" and then select the "**Select a Library Pull Down**" button to load a library into the program.



You can select any one of the 200+ QFP's in the Search Window and then select OK and the program will load the selected component and land pattern data.



In the next example we'll enter some basic data to minimize the search.

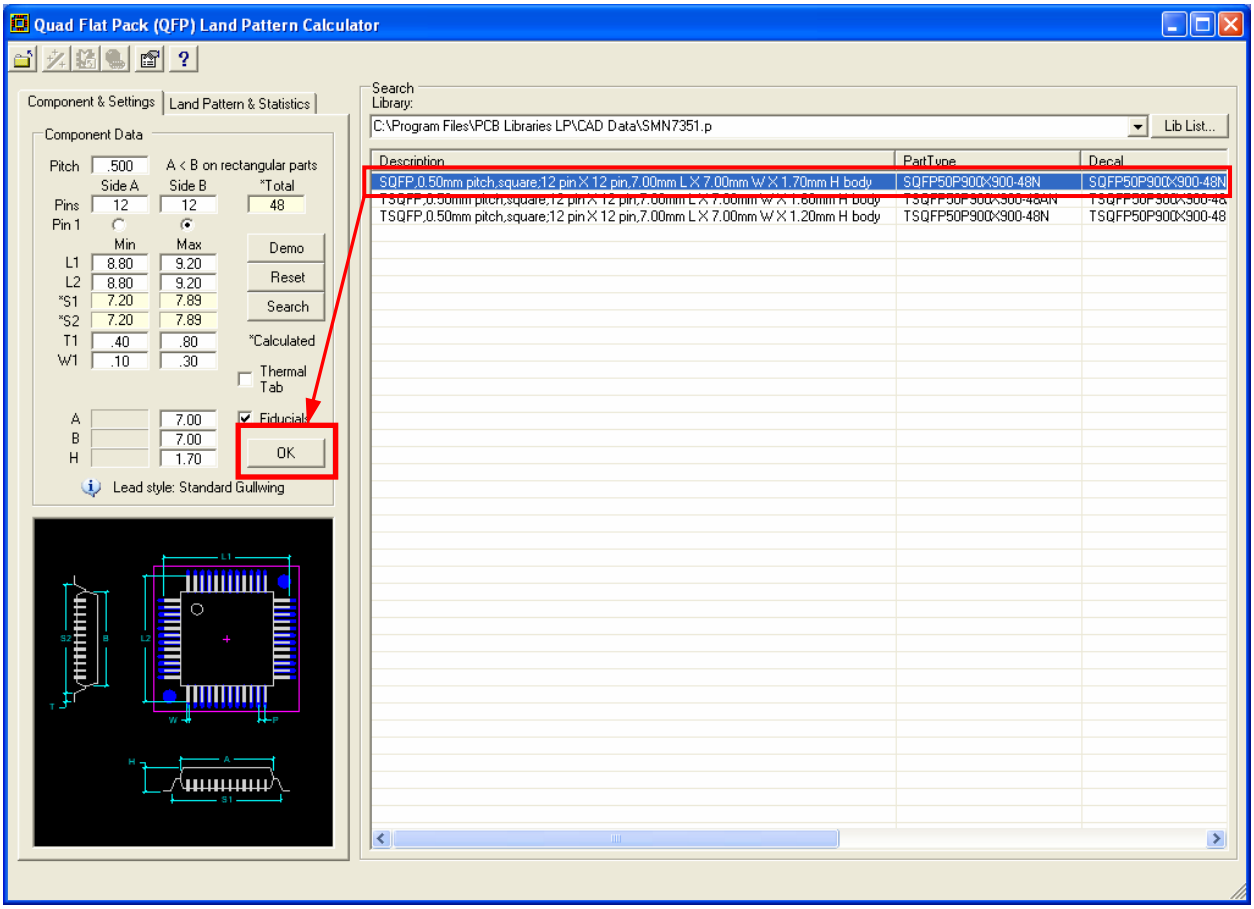
Select the "**Reset Button**" to clear the Search Window.

Use the Pull-Down "**Pitch**" and select **0.5mm** Pitch

Enter the number **48** in the "**Search Total**" Pin Qty

Now select the "**Search Button**" and three QFP parts that match the search parameters will appear in the Search Box.

Note that only items possessing the characteristics of 0.50mm pitch and 48 total pins have been displayed. Click one of the lines with the left mouse button and observe that all dimensional data for the selected part are displayed in the component group box.



Select one of the three items (for this example select the first item) the SQFP.

Next, select the "**OK Button**" and the program will load the component and land pattern dimensional data for which ever component you selected. You can also "**Double Click**" on the line item and it will load the data into the LP Viewer.

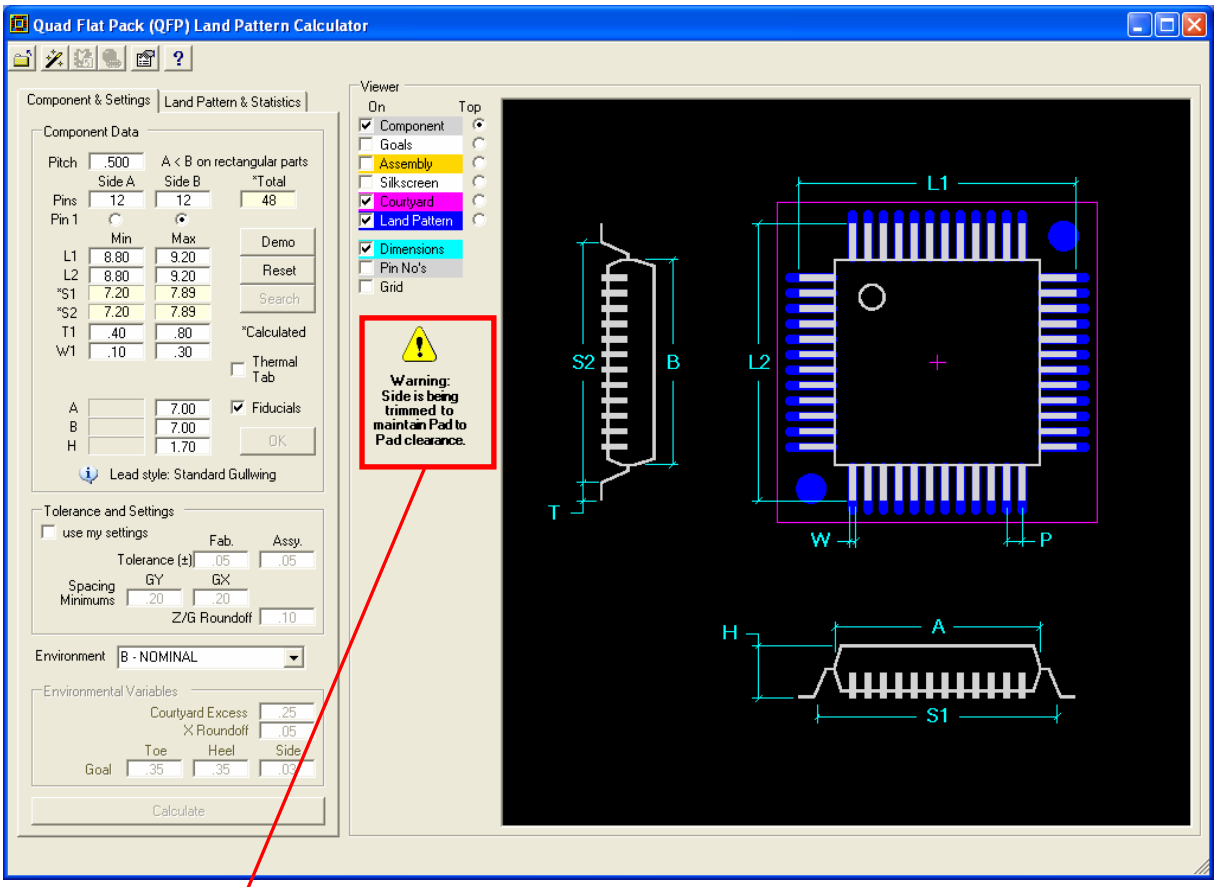
The more data you enter, the finer the search will be.

Example: If you would have entered **1.7** in the "**H**" (height) field only one item would have appeared in the Search Window.

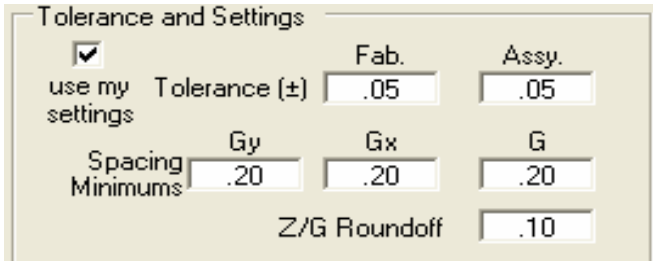
If no component is found, then simply enter all the component data and select the "**OK Button**" when completed.

Then select the "**Calculate Button**" in the Land Pattern section and the land pattern data will appear.

Use this component data for the next session 7 "Edit Existing Part".



This "Warning: Side is being trimmed to maintain Pad to Pad clearance." message appears when the size of the pad width violates the rules that are set in the Tolerance and Settings section.



7 SMD CALCULATOR EDIT EXISTING PART

7.1 Edit an Existing Part

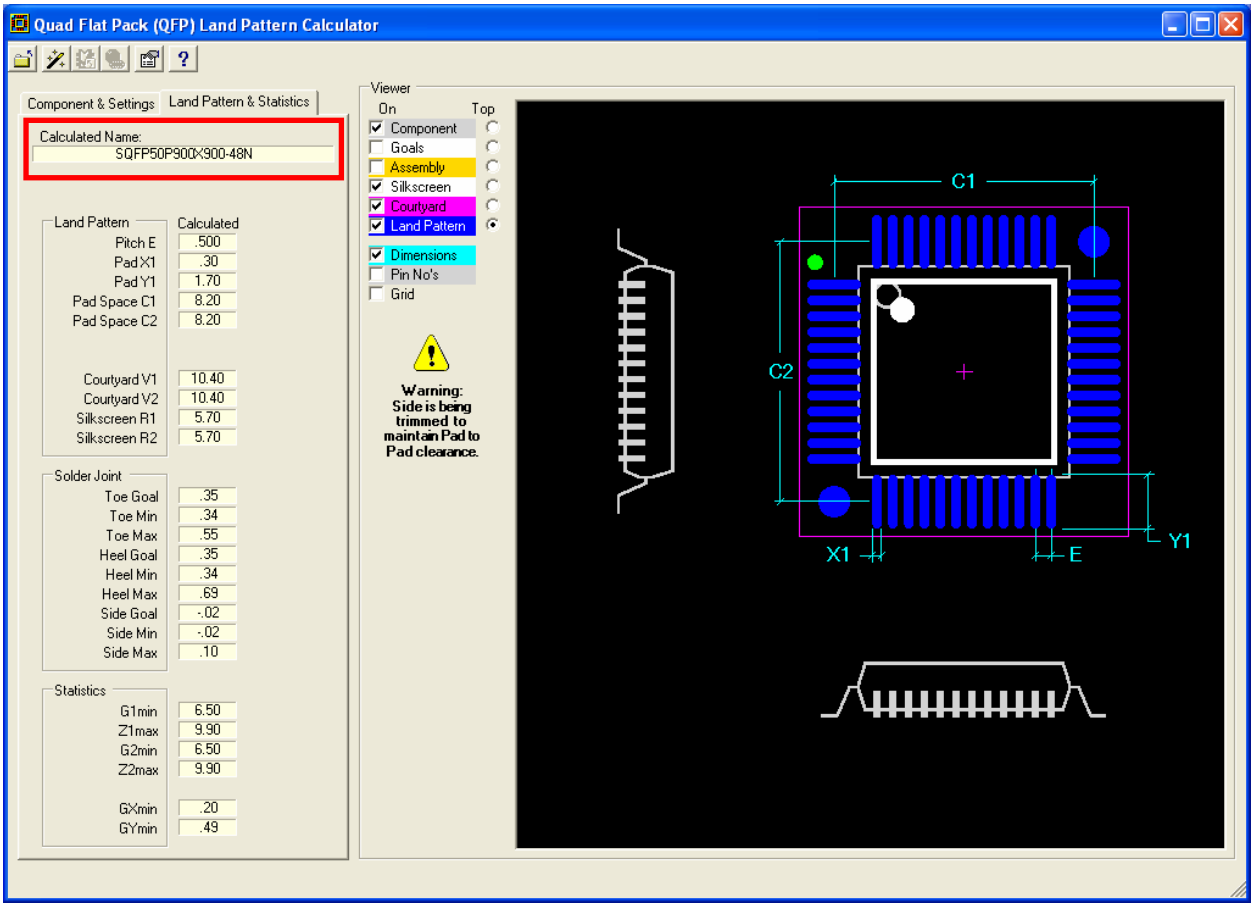
If you can find a component close to the one you are looking for, all you have to do is modify the numbers that are different and select the "**OK Button**".

In the example below we are going to change the "**L1 and L2**" Min from **8.80** to **9**. Then select the "**OK Button**" then select the "**Calculate Button**".

See the Land Pattern Name change from:

SQFP50P900X900-48N to **SQFP50P910X910-48N**

Due to the tighter tolerance of the overall lead span, the nominal dimension changed from 9.00mm to 9.10mm.



Use this feature to make existing component dimensional data changes to create new land pattern data.

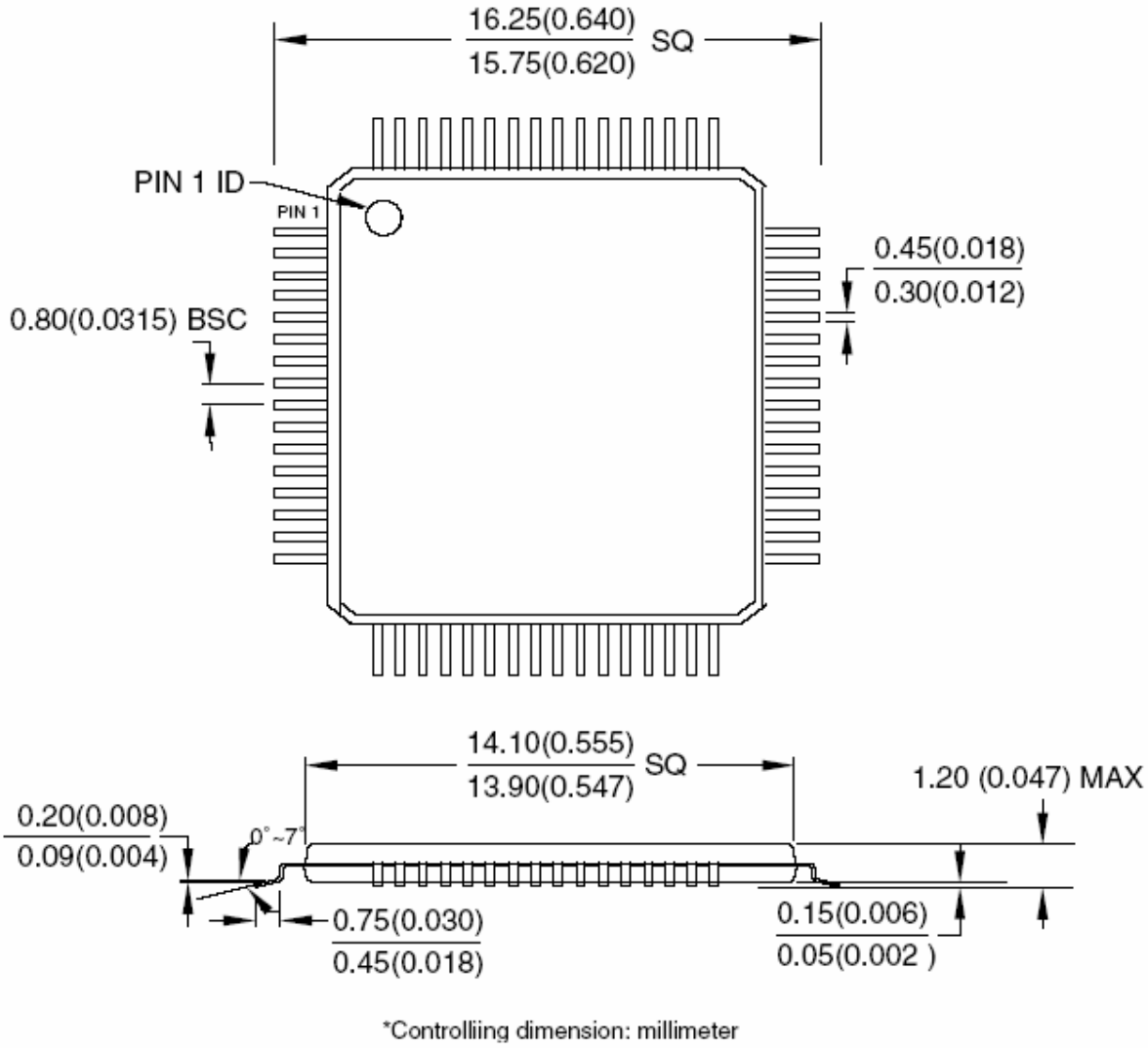
8 SMD CALCUALTOR CREATE NEW PART

8.1 Create New Land Pattern

If you cannot find an exact component data match and the new data is totally different than any existing component in the LP Wizard, you must build a new land pattern from scratch.

To do this, you must have all the necessary component dimensional data for all the LP Wizard Component Fields. See the datasheet below.

64-lead, Thin (1.0 mm) Plastic Quad Flat Package
(TQFP), 14x14mm body, 2.0mm footprint, 0.8mm pitch.
Dimensions in Millimeters and (Inches)*
JEDEC STANDARD MS-026 AEB

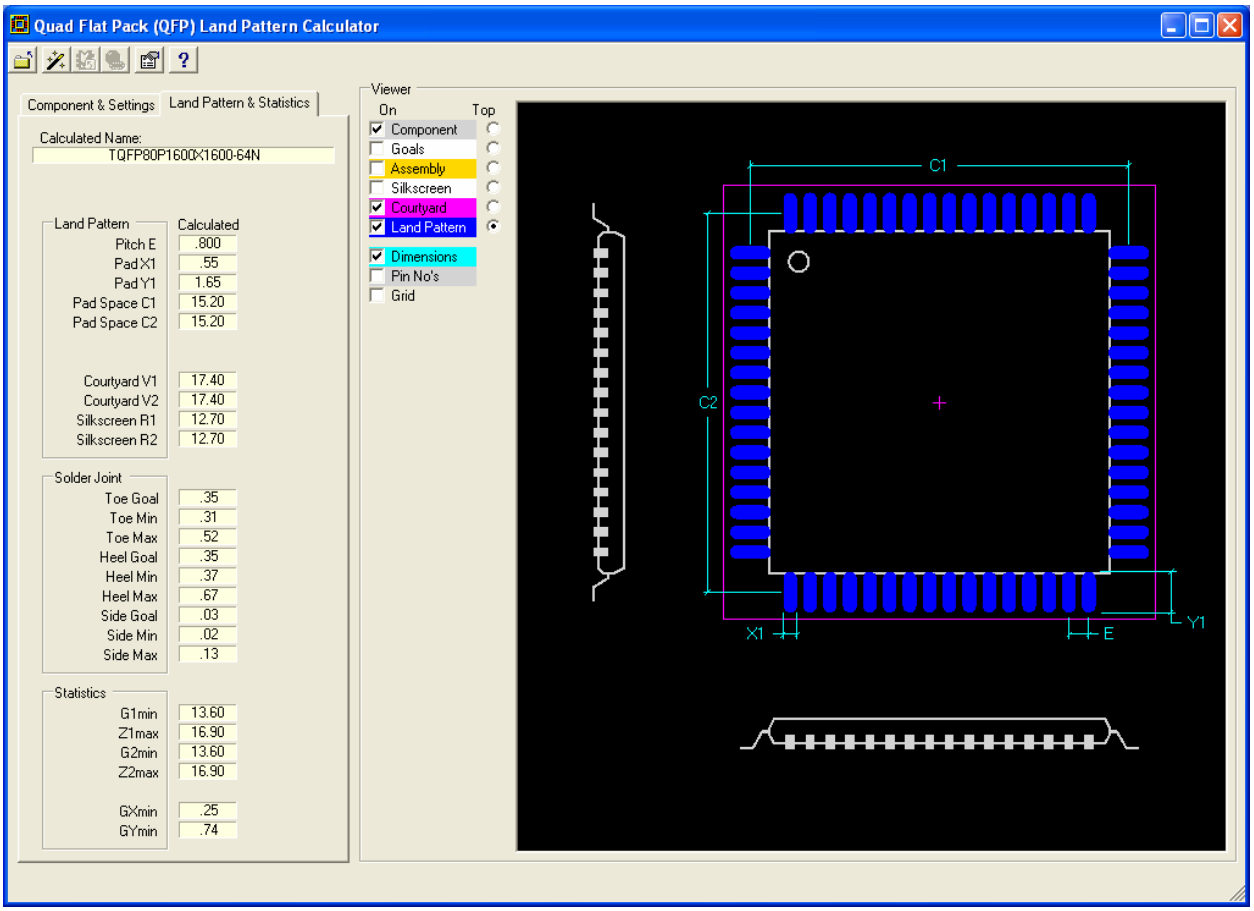


*Controlliing dimension: millimeter

Enter all the component dimensions into the Component Fields.

Select the "**OK Button**" in the Component Data section and it will automatically calculate the Land Pattern data.

Select the "**Land Pattern & Statistics**" Tab at the Top of the Component Data section (next to the "Component & Settings" Tab to view the Land Pattern data.



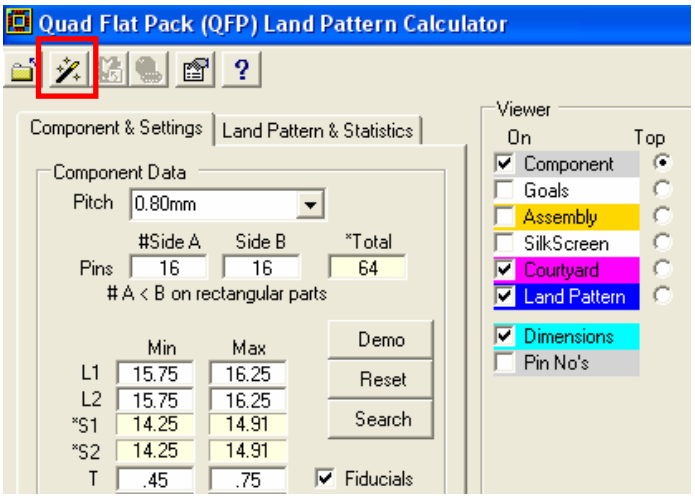
Use this new QFP to create a new library part in the next **Session 9 SMD Construction Wizard**, either directly into a PADS™ library or create a PADS™ ASCII file to export into any other CAD tool.

9 SMD CONSTRUCTION WIZARD

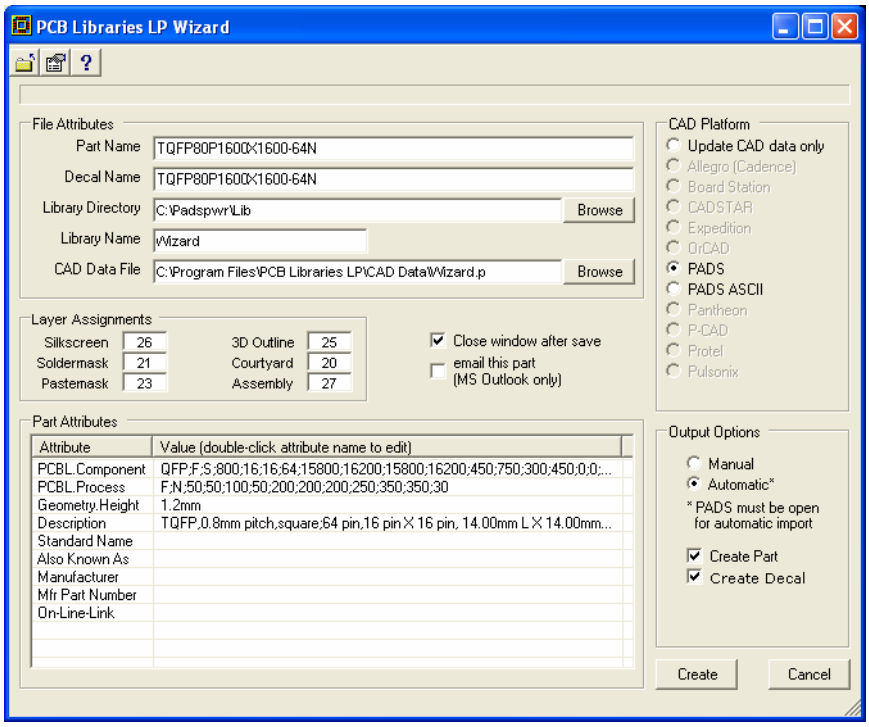
9.1 Create PADS™ Library Part

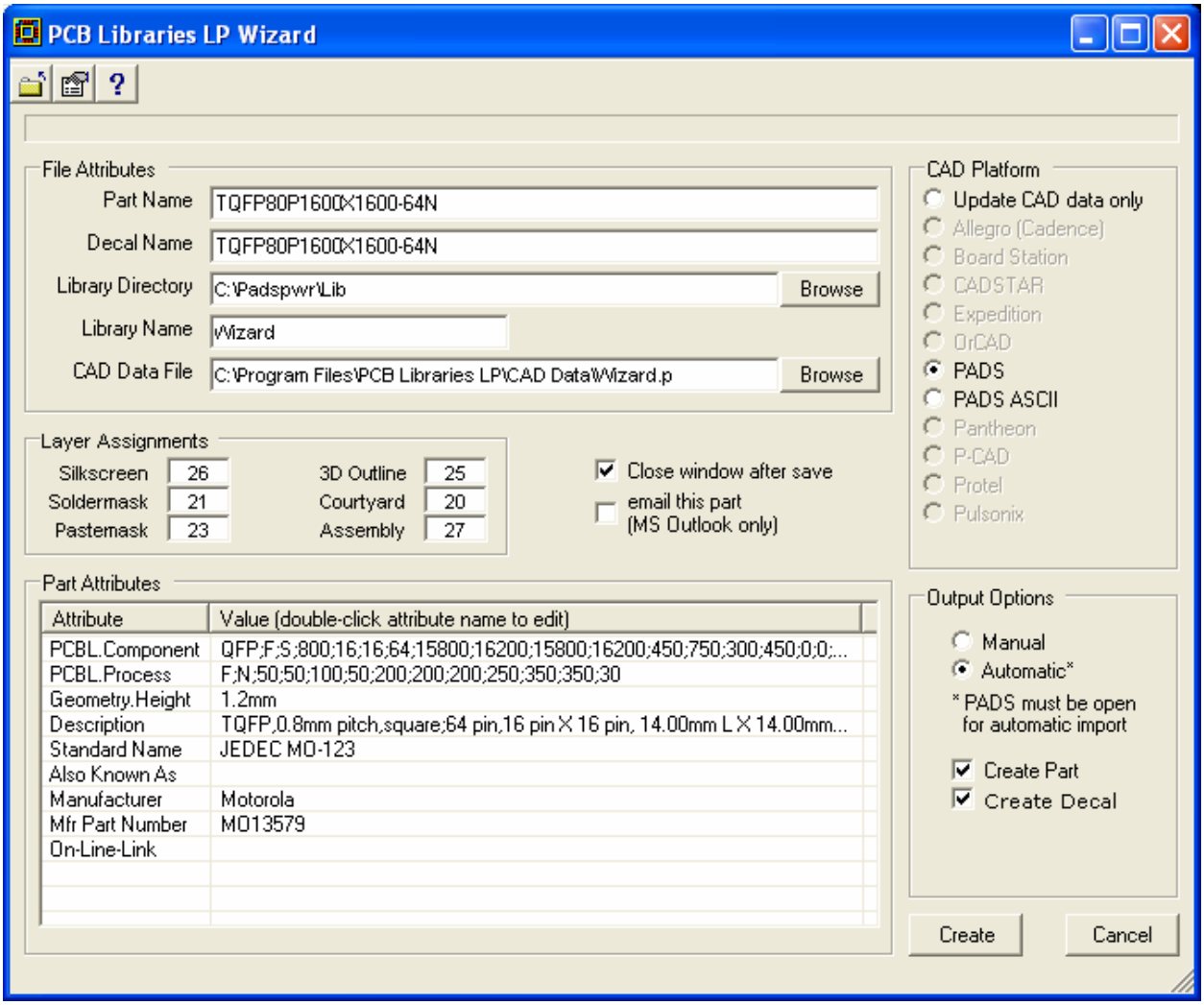
Once you have entered the correct component data and calculated the land pattern data you can auto-generate a new PADS™ Decal or Part-Type

In the Land Pattern Calculator, select the **Wizard** **Tool Bar Button**



Double Click on any Attribute Name to Edit and Customize the Attributes



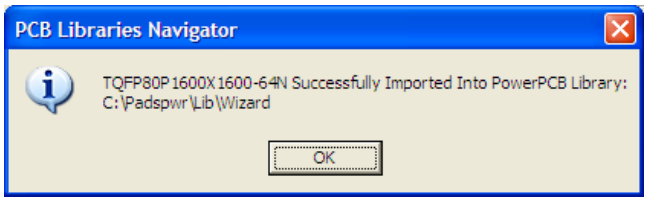


You can now enter in the component manufacturer and part number into the attribute fields and you can set you library path and the name of your existing library. If the library name does not exist, the LP Wizard will create a new library with your library name. For more detail, see Chapter 10 – "**Creating Attributes**".

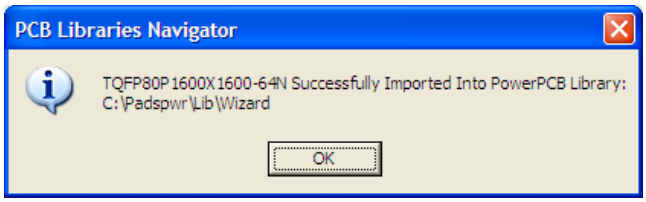
You can create PADS™ .d (decal) and .p (part-type) files by selecting the **Manual Button**

You can insert the new library part directly into PADS™ by selecting the **Automatic Button**. Note: PADS™ must be open to use this option.

We are going to use the Automatic option for this session. After you enter the Manufacturer, the Part Number, the correct Library Path and Library Name have been entered select the "**OK Button**" and a menu box will appear.



Select **OK**



Select **OK**

Then in PADS™ and select File / Library / Pull-down to Wizard

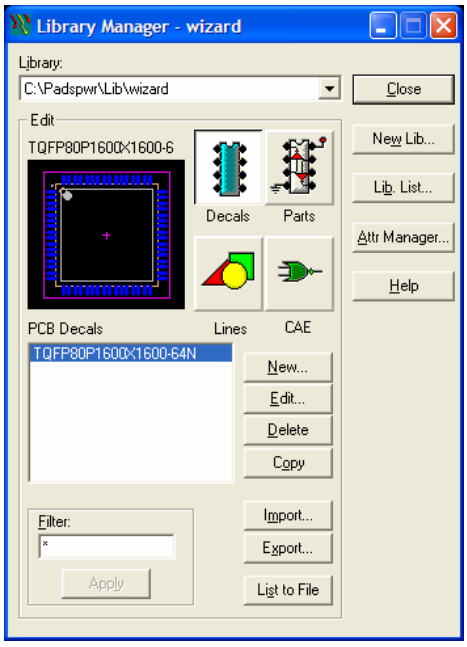
The Filter should be set to * (see picture on next page)

Select the Decal Button and the new part will appear in the browser window.

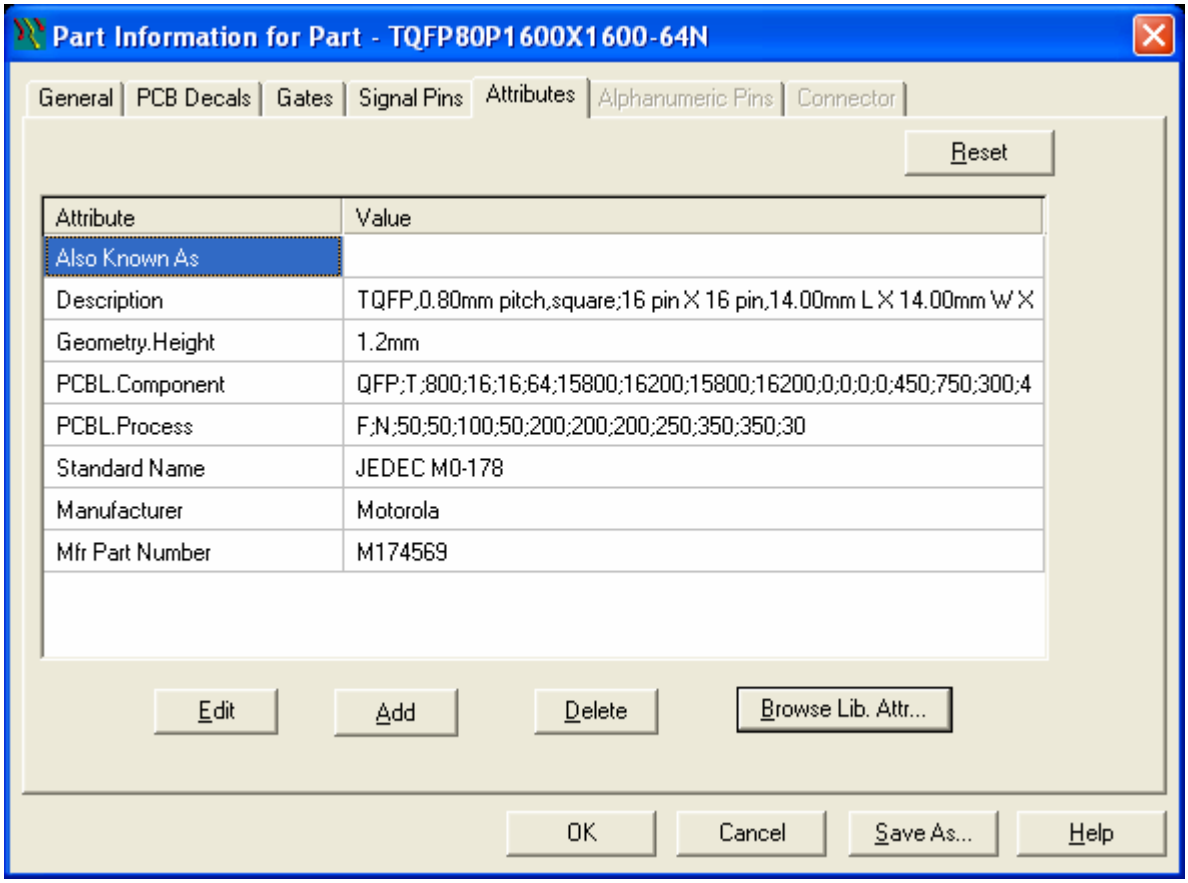
Select **Edit Button** to bring the decal into the PADS-PowerPCB decal Editor.

To see all the component elements on the correct layers you will need the **Decaledt.asc** file that is available at this URL:
http://www.pcblibraries.com/downloads/PADS-prog.asp

This is the PADS™ Library Manager



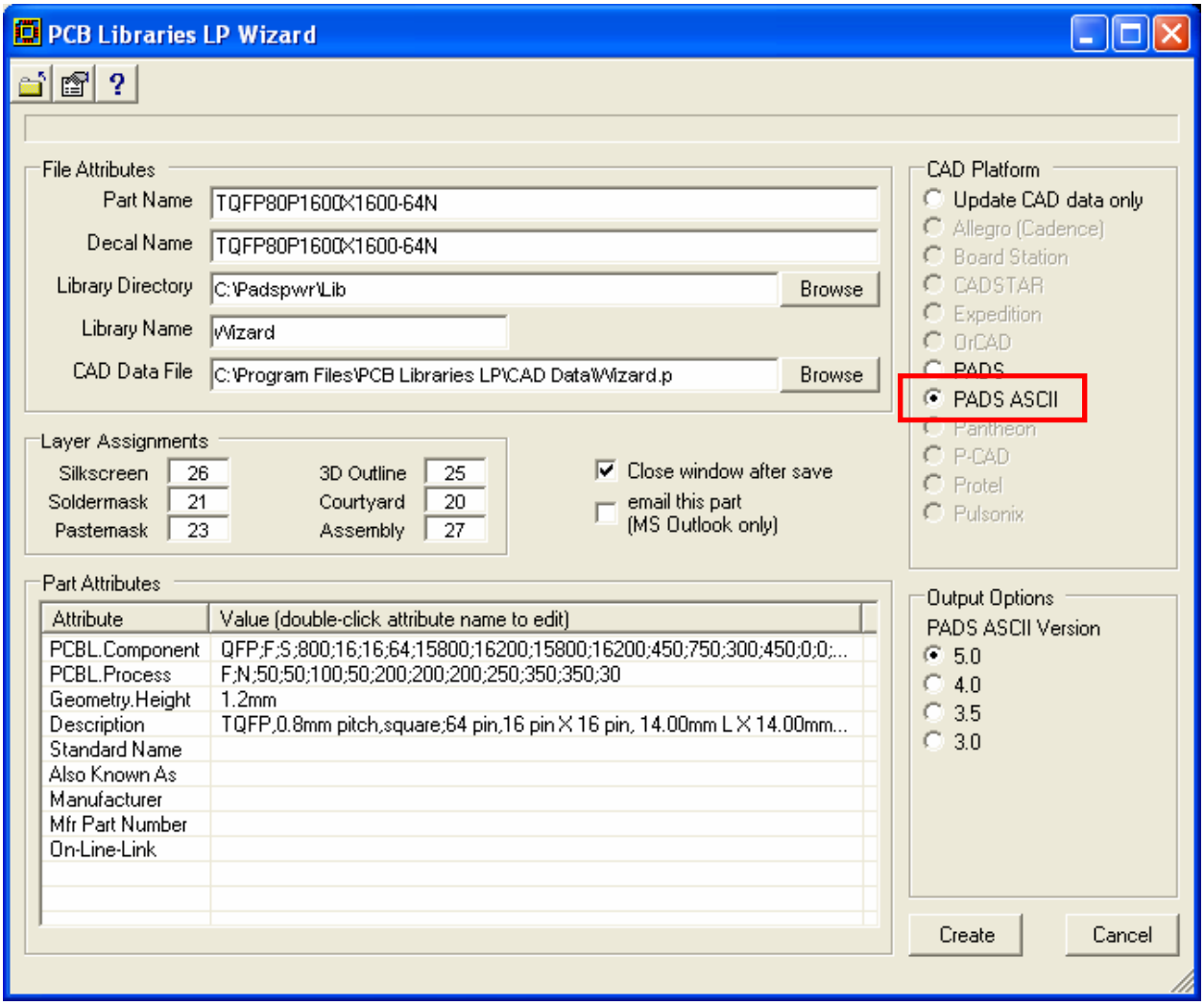
PADS™ Layout Part-Type Attributes



9.2 Create a PADS™ ASCII File

Most CAD tools can import PADS™ ASCII files. Ask your local reseller which version of PADS™ your CAD tool can import.

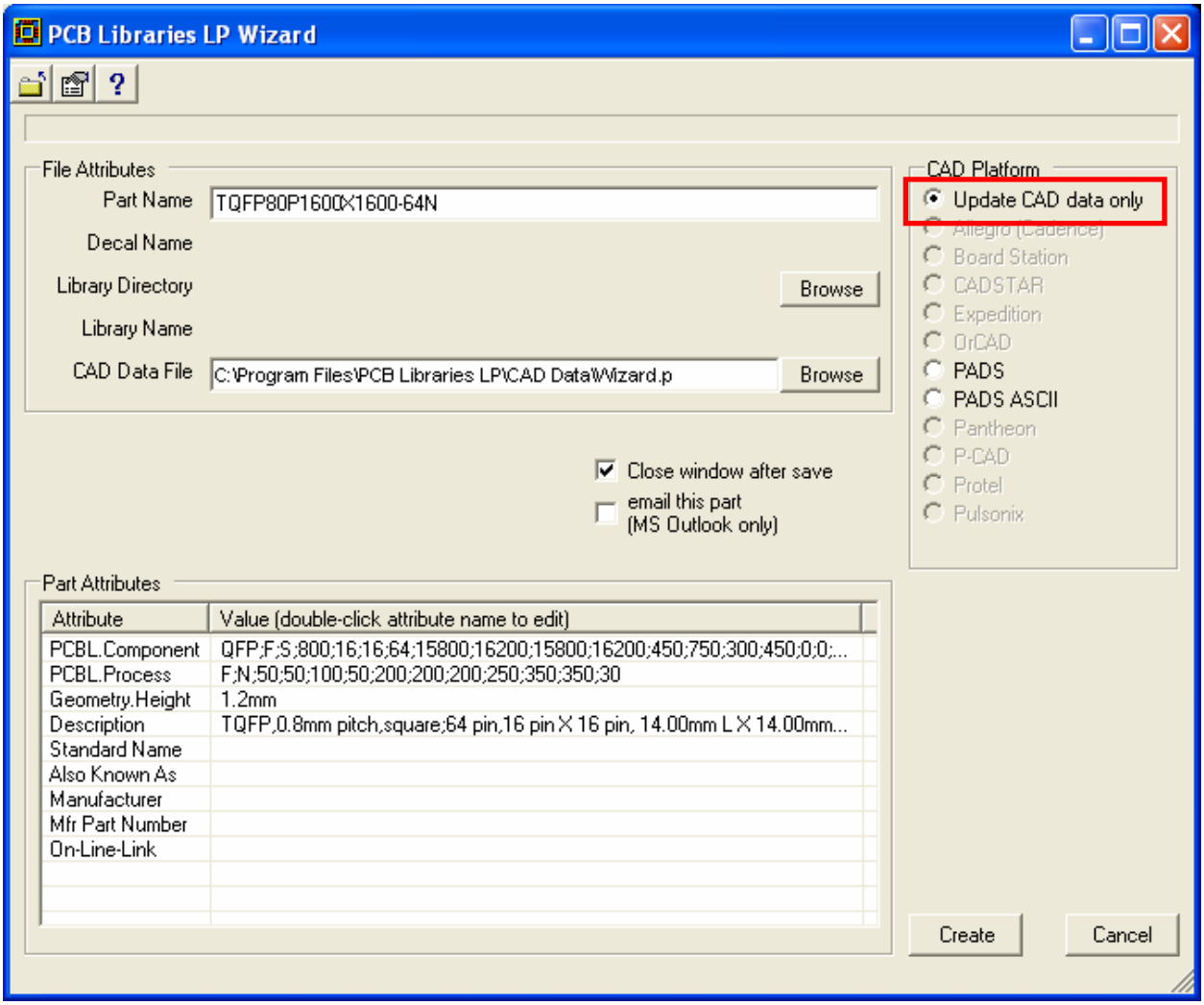
Here is the way to export PADS ASCII data. Select **Export to PADS ASCII** button



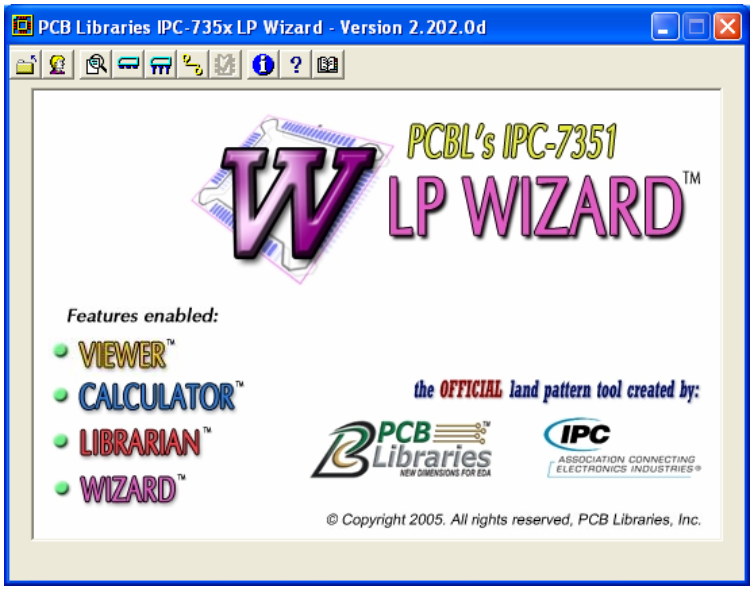
This will add the new part to your text library .P file so you can recall the part data at a later date. You can change the library name and path. You can edit and update existing parts to add additional information.

9.3 Non-PADS™ Users Update an Existing Library File

If you use Cadence Allegro or any other CAD tool and choose not to create a PADS™ ASCII file but want to store your new component data in a library .P file, select the directory path of the library and select **Update CAD data only** button.



After this exercise, close all windows and go back to the main menu



10 CREATING ATTRIBUTES

10.1 What are Attributes?

Attributes hold vital statistical and descriptive data that every land pattern needs so other users can quickly identify the component characteristics. Attributes help organize data used to search for existing library parts, to aid in web-linking to manufacturers websites and store dimensional data for land pattern creation.

There are three distinct sets of Attributes that need to be defined to maximize the benefits of the IPC-7351 LP Wizard. It is very important that you properly set up your Attributes to get the best benefits from the IPC-7351 LP Wizard.

10.2 Creating IPC-7351 Standard Part Attributes

For Industry Standard Parts there are six attributes that should be defined.

Description – The description attribute for Standard Parts should be formatted so that the most important information is first, and then secondary and so on.

The order of importance for the Standard Part "Description" attribute is:

1. The Component Family Abbreviation
2. The Pin Pitch
3. The Package Shape (Square or Rectangle)
4. The Pin Quantity
5. Other Physical Characteristics

Example BGA Description: BGA,1.00mm pitch (square);100 Pin,10 columns X 10 rows,blocked columns & rows,11mm X 11mm X 3.5 H

The information before the **;** (semicolon) is presented in the preliminary search and the other data appears when you expand the search.

Standard Name – This refers to the JEDEC, IEC or EIA Part Number.

Example: JEDEC DO-214

Geometry.Height – This is always in metric units and the value should always be followed by lower case mm. This data is used to generate a 3D Model mechanical drawing.

Example: 1.6mm

Also Known As – The AKA Attribute refers to Alternate part names.

Example for a 3216 Chip Component: 1206

PCBL.Component – This Attribute is automatically filled in every time you build a new Land Pattern. It is the Component dimensional data that was inserted in the Component Data fields in the Calculator. It's also used to reconstruct the component data in the LP Viewer so other people can view the component data. It is also used to recreate the Land Pattern data.

PCBL.Process – This Attribute is automatically filled in every time you build a new Land Pattern. The PCBL Process data refers to the "Environment Level" that was used to generate the Land Pattern. The Environment Level can be Least, Nominal, Most or User Defined.

10.3 Creating Non-Standard (Mfr Specific) Part Attributes

For Non-standard Parts there are five attributes that should be defined. The two Non-standard libraries are – Through-hole and Surface Mount Mfr specific.

Description – The description attribute for Standard Parts should be formatted so that the most important information is first, and then secondary and so on.

The order of importance for the Standard Part "Description" attribute is:

1. The Component Family Spelled Out (No Abbreviation). Example: BATTERY
2. A Second Keyword that describes the Component Family. Example: HOLDER
3. A Third Keyword that describes the Component Family. Example: COIN
4. Continue until all Keywords are in the Description.
5. Use UPPER CASE characters and delimit the Keywords with commas.

   Example: Battery holder;coin,PC

Manufacturer – Spell out the component manufacturer's name (do not abbreviate)

Example: Texas Instruments

Mfr Part Number – Component Manufacturer's full part number

Example: 74FC357

Geometry.Height – This is always in metric units and the value should always be followed by lower case mm. This data is used to generate a 3D Model mechanical drawing.

Example: 1.6mm

On-Line-Link – A URL website link to the Component Manufacturers PDF, .GIF or .TIF datasheet or an on-line page that contains the component dimensional data.

Example: http://www.anadigics.com/products/addrefs/Datasheet/ABA3100.pdf

10.4 Creating Connector Part Attributes

For Connector Parts there are five attributes that should be defined.

Description – The description attribute for Connector Parts should be formatted so that the most important information is first, and then secondary and so on.

The order of importance for the Connector "Description" attribute is:

1. Pin Pitch – Example: 2mm pitch
2. Row Count – Example: 2 row
3. Type – Example: header or connector
4. Break – Example: ; (semicolon)
5. Pin Quantity – Example: 100 pin,
6. Angle – Example: r/a, or vertical,
7. Pin Style – Example: SMT, or PTH,
8. Sex – Example: receptacle, or plug,
9. Style – Example: shrouded,
10. Pin Data – Example: SQ post, or RND post
11. Polarity – Example: w/alignment pins,
12. Other – Example: w/locking clips

Example: 2.54mm pitch 2 row header;10 pin,vertical,SMT,plug,SQ post,w/locking clips

Note: The data before the **;** (semicolon) will appear in the "**Search Categories**" window. The data after the semicolon will appear in the Master "**Search Results**" window.

Manufacturer – Spell out the component manufacturer's name (do not abbreviate)

Example: Airborne, Inc.

Mfr Part Number – Component Manufacturer's full part number

Example: RM222-010-291-0256

Geometry.Height – This is always in metric units and the value should always be followed by lower case mm. This data is used to generate a 3D Model mechanical drawing.

Example: 7.62mm

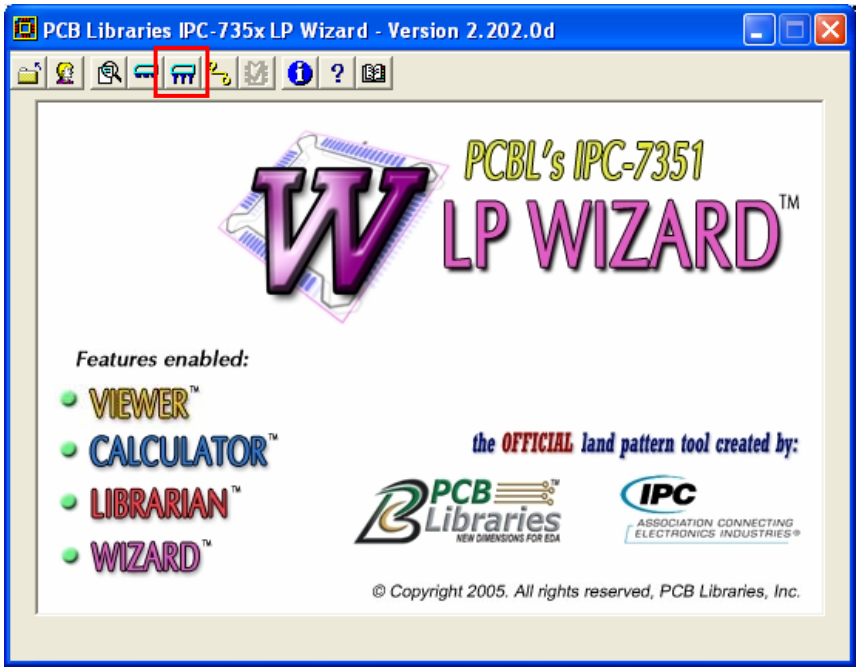
On-Line-Link – A URL website link to the Component Manufacturers PDF, .GIF or .TIF datasheet or an on-line page that contains the component dimensional data.

Example: http://www.airborn.com/docs/viewAttpub.asp?id=2497

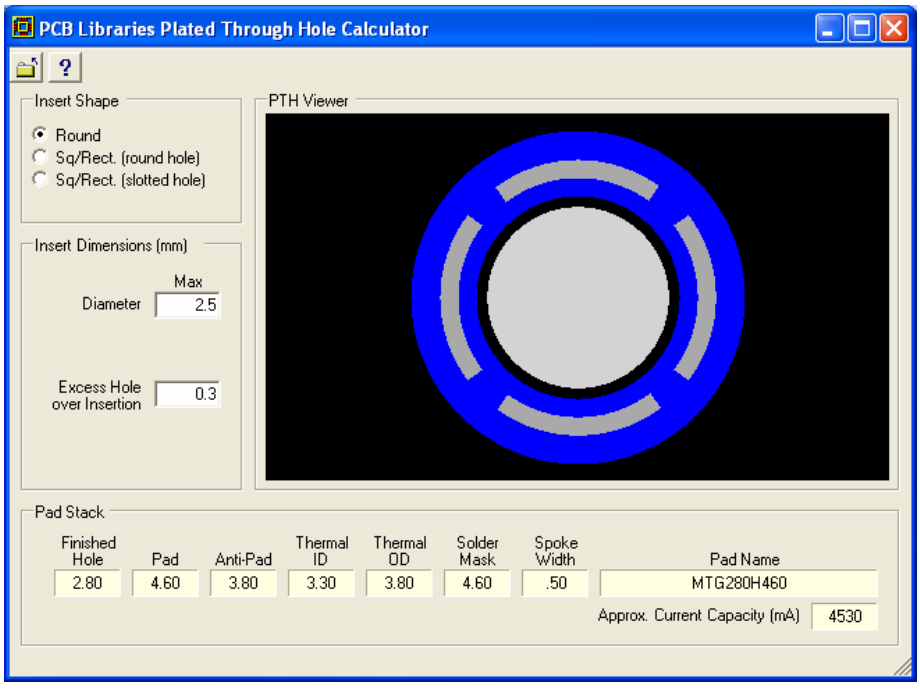
11 PLATED THROUGH HOLE CALCULATOR

11.1 Through-hole Calculator

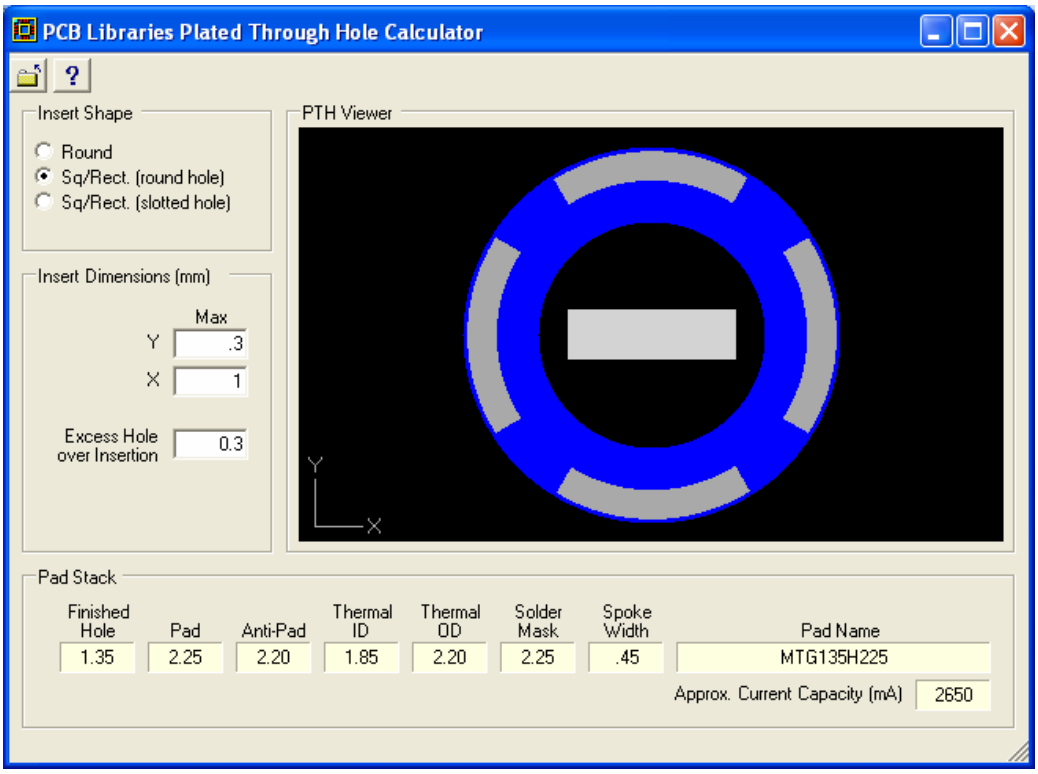
At the main menu select the **Plated Thru-hole Calculator** tool bar button.



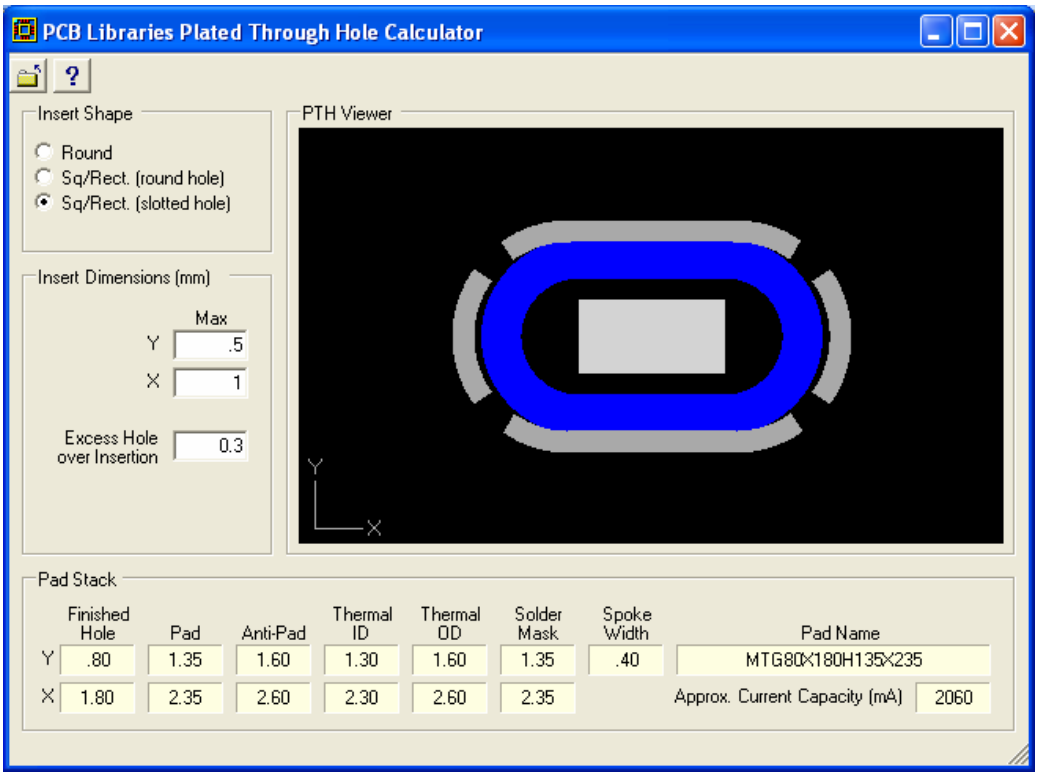
This Main Plated Through-hole Menu Calculator will appear



You can select a Square Component Lead



You can select a Rectangle Component Lead



11.2 Through Hole Components

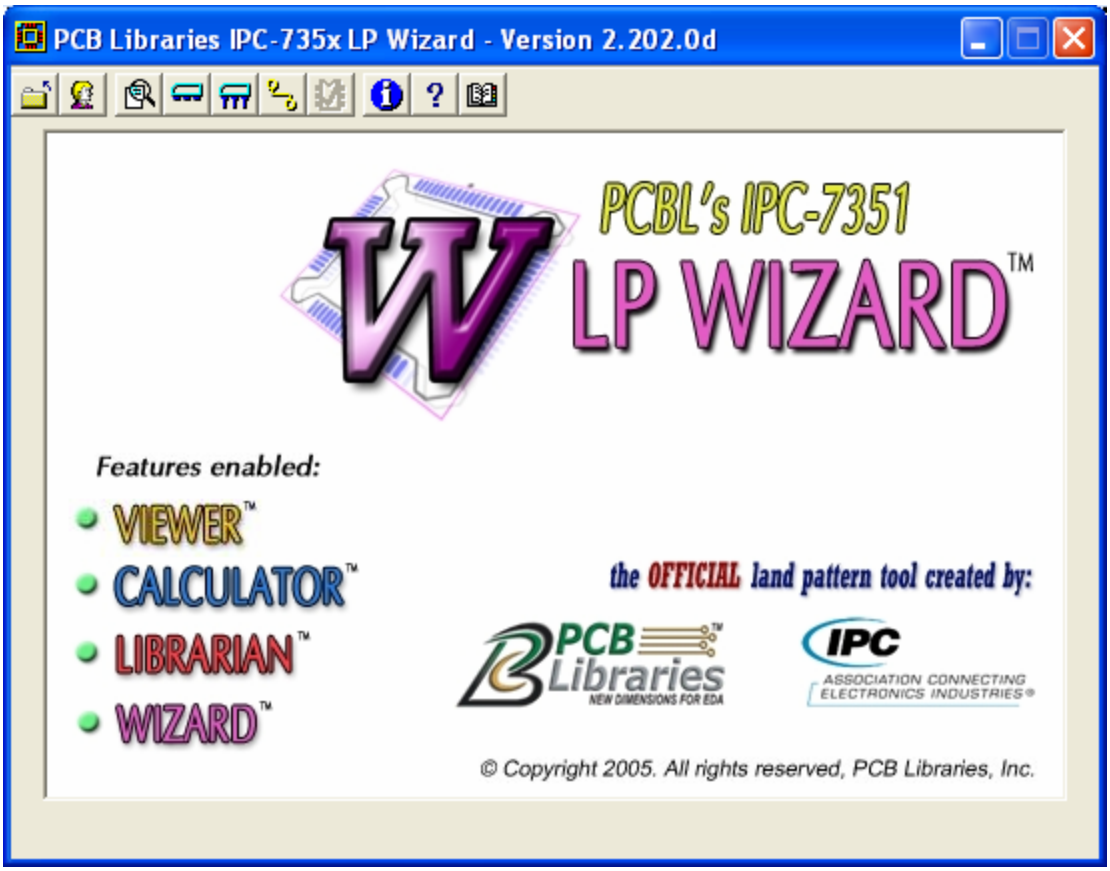
At this time, Through-hole Components are under development.

The **Plated Thru-hole Calculator** is used to create Through-Hole component padstack data.

Soon there will be Axial and Radial Lead Components, DIP & SIP Components and then connectors and many more. We are working with IPC to develop a 3-Tier Through-hole library concept to satisfy all industry requirements.

We are considering a maximum lead size in a minimum hole, nominal hole and maximum hole. Each will have a corresponding pad size to accommodate the current carrying capacity of the component lead size.

Close all windows and return to the Main Menu

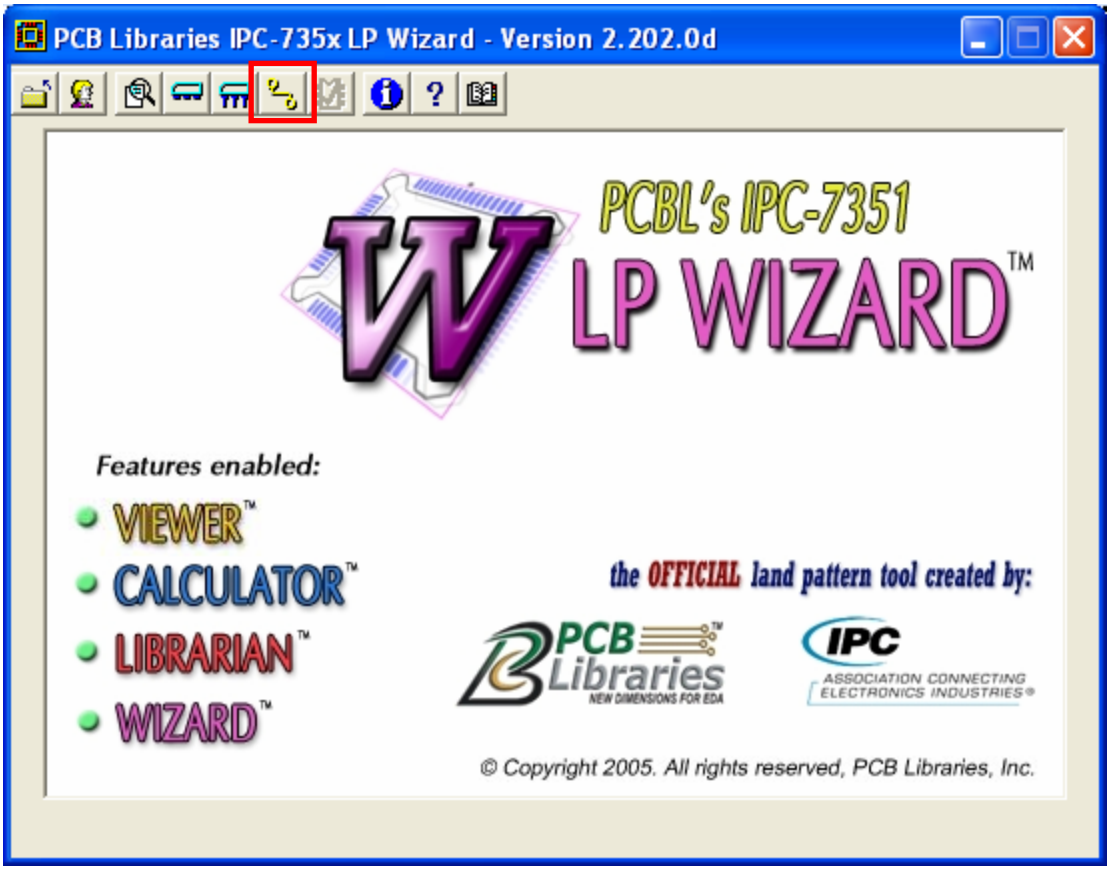


12 VIA CALCULATOR

12.1 Calculating BGA Vias

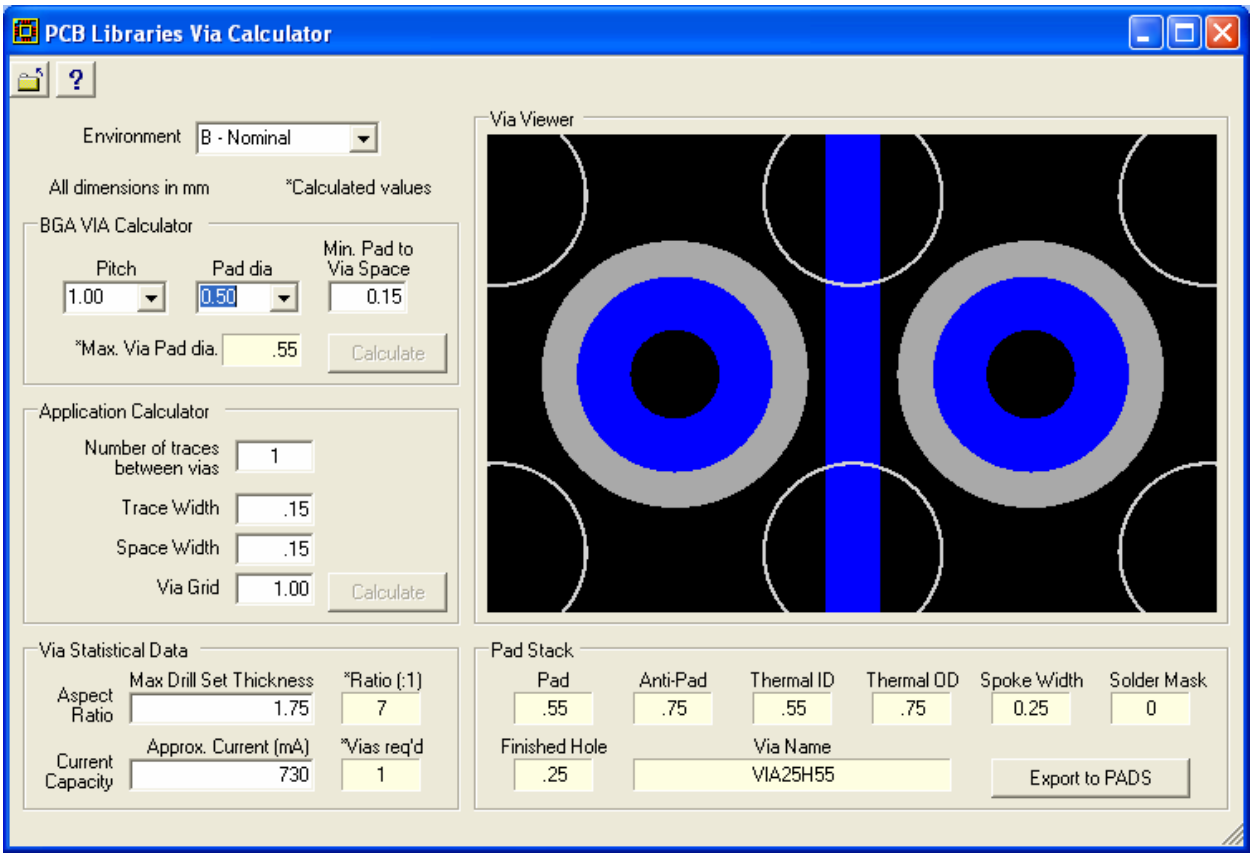
The Via Calculator was created out of the demand for calculating the perfect via size for every BGA pin Pitch or Via Fanout Grid.

At the Main Menu select the **Via Calculator** Tool Bar button.



The Via Calculator is actually two calculators in one. The BGA Via Calculator calculates the best size for a via based on BGA pitch, pad diameter and minimum allowable pad-to-via spacing. Once this information is entered, click the Calculate button in the BGA Calculator Group box. Two vias will be displayed surrounded by white circles that represent the adjacent BGA pads.

The Application Calculator is used when parameters, other than BGA, will be used to determine the via size. Note that white circles denoting BGA pads do not appear when using the application calculator.



To demonstrate, select a BGA pitch from the **Pitch pull down menu**. Note that only pad sizes appropriate that pitch are available in the Pad diameter pull down selection.

Next, select a **Pad Diameter**. Note that a minimum value is suggested for **Min. Pad to Via Space**. You can alter this value if desired but generally it will be the absolute minimum that should be applied.

Finally click on the **Calculate button** in the **BGA Group Box.** A ***maximum*** via pad diameter is calculated and displayed in the BGA Group Box. Additionally the **Size** and **Space** for **one Trace** is calculated and applied to entries in the **Application Calculator**. In this case the BGA pitch determines the **Via Grid**.

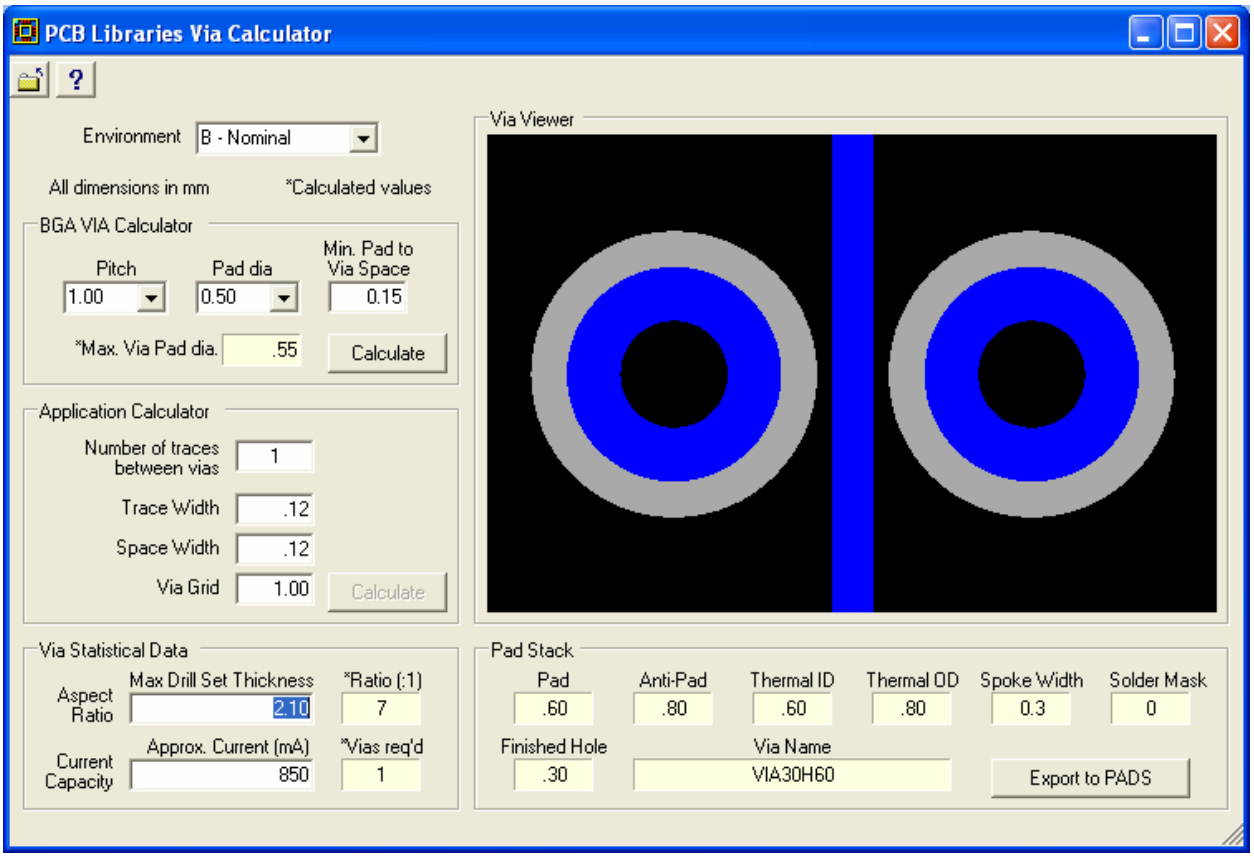
Via Statistical Data is calculated to produce the **Aspect Ratio** for a maximum Ratio of 7 to 1 and **Power Capacity** is calculated for 1 via.

All information about the **Pad Stack** is calculated and displayed.

Data generated by the BGA Calculator can be modified in the Application Calculator to create different solutions with varying number of traces, widths, spacing or grid. Additionally, the statistical data may be changed to produce a new aspect ratio based on board thickness or calculate the number of vias required to accommodate current requirements.

Warnings may appear to alert users when good design practice has been violated.

Note: Entries in the BGA calculator will fall through into the Application Calculator but the Application Calculator has no effect on the BGA Calculator.



For a new demo, use these settings:

Environment: **Least**

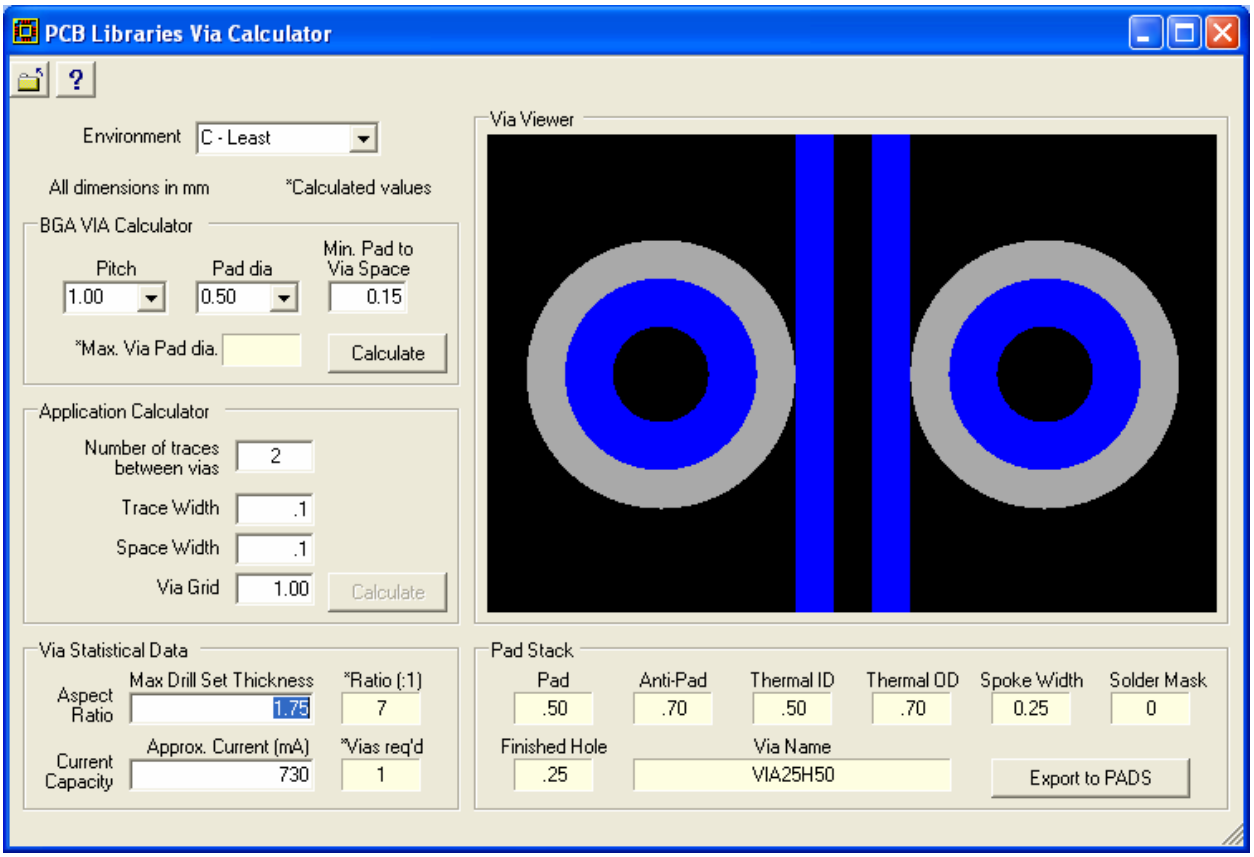
BGA Pitch **1.00mm**

Number of traces between vias: **2**

Trace Width **.1mm**

Trace Space **.1mm**

Select the **Calculate Button** and watch the Pad Stack values change



The Blue Pad is the via annular ring and the Grey is the Plane Anti-pad. The goal is to never have the trace cross over an anti-pad to insure a clean return path on the reference plane.

Notes:

The Least Anti-Pad = 0.45mm over the hole size

The Nominal Anti-Pad = 0.50mm over the hole size

The Maximum Anti-Pad = 0.55mm over the hole size

The Minimum Hole size = 0.20mm

Note: If you are Using PADS™, you can use the "Transfer" Button to import the Via Padstack Data directly into the PADS / Setup / Padstacks / Via menu.

Return to the Main Menu for the next session

13 Quality Control

13.1 Send Your .P file to PCB Libraries

PCB Libraries, Inc. offers a full range of quality control checks on all CAD Land Patterns created by the LP Librarian or LP Wizard. Let PCB Libraries, Inc. help you quality control your library data. That's right - PCB Libraries, Inc. will quality check your LP Librarian or LP Wizard .P file data to the manufacturer's component dimensional data PDF file for no charge! At the same time, you will help build the world's largest standard CAD library to eliminate duplication, and automate electronic product development!

If you are using PADS Layout, we will also check the final Land Pattern Decal and Part-Type to ensure maximum quality.

To send in your library data for quality control, please:

1. Name the .P file with the respective Land Pattern name.

   Example: **SOP80P1425-64N.p**

   In the **SOP80P1425-64N.p** file, please enter as many Attributes as possible:

   a. **PCBL.Component:** Component Dimensional Data created by the Calculator
   b. **PCBL.Process:** Environment Dimensional Data created by the Calculator
   c. **Description:** Either created by the Calculator or from the component data sheet
   d. **Manufacturer:** Texas Instruments
   e. **Mfr Part Number:** XYZ123
   f. **Standard Name:** JEDEC MO-169AB
   g. **Also Known As:** TSSOP64
   h. **On Line Link:** http://www.ti.com/XYZ123.pdf

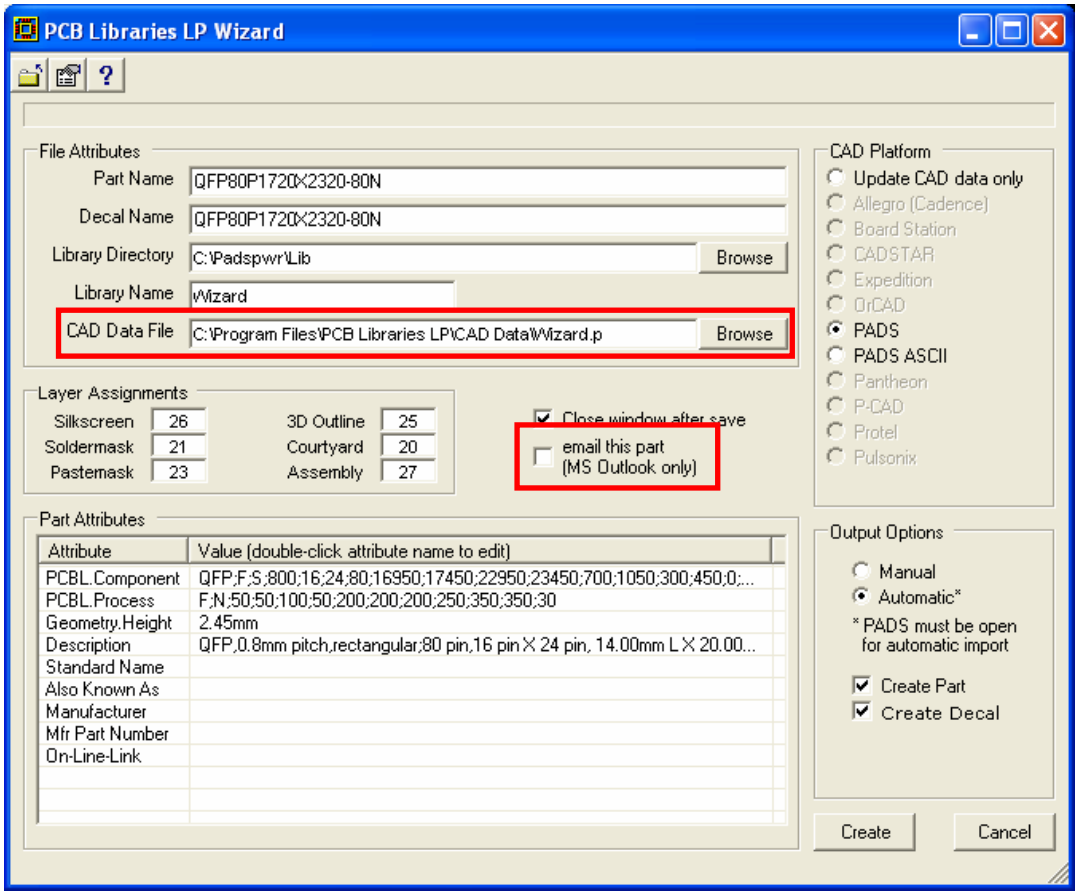
PCB Libraries, Inc. will return either:

1. Confirmation that all the data was correctly entered into the LP Librarian or LP Wizard, or
2. The corrected data and updated files along with a QC report of what was corrected

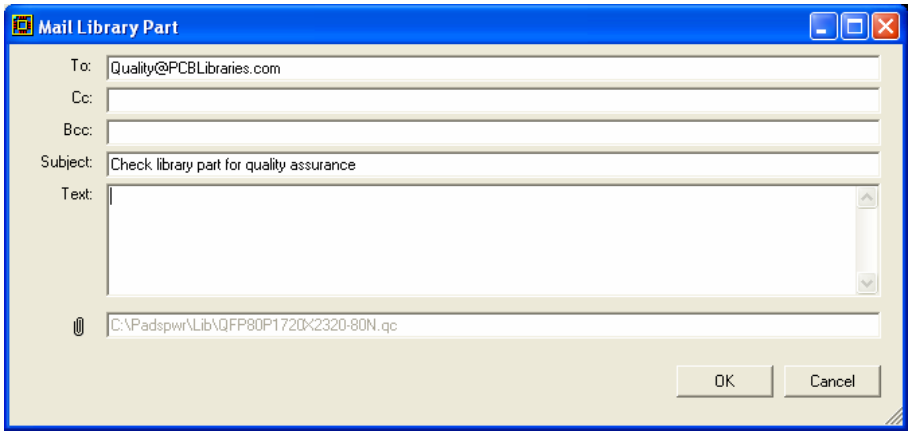
What will PCB Libraries, Inc. do with the data?

1. Add the new .P file to the IPC-7351 library in the 3-Tier format and update the free download center to share the data with the world.
2. Add the PADS Layout Land Pattern Decals and Part-Types to the master IPC-7351 libraries and make them available for everyone to download.

Send the PCBL .P file to: Quality@PCBLibraries.com by selecting the **email this part** and copy / paste the Land Pattern name in the "**CAD Data File**" folder. See below.



The following dialog box will open and all you have to do is select the **OK** button.



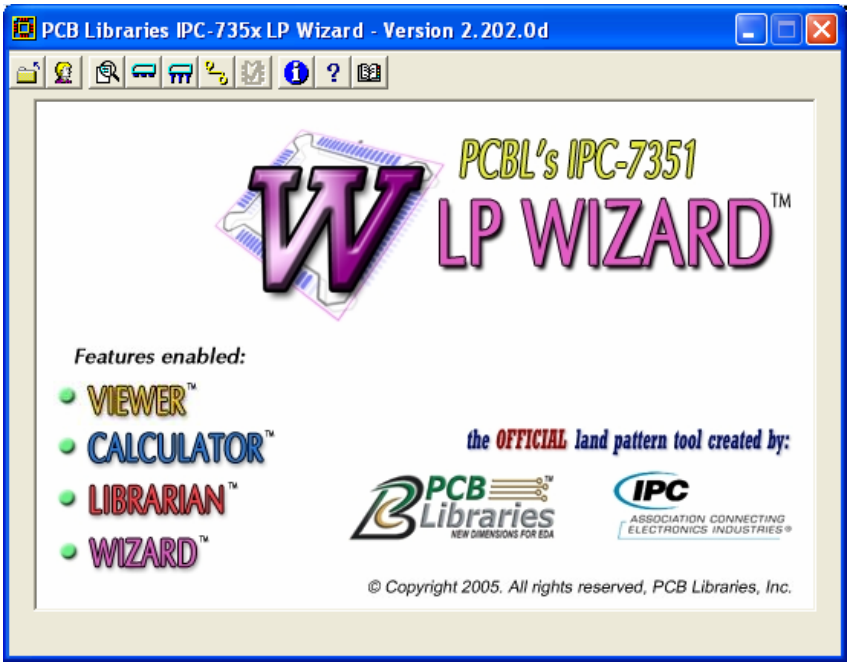
Let PCB Libraries, Inc. perform quality control checks all your library data and help us eliminate worldwide duplication of effort by sending us all of you library .P files.

Any questions regarding this service please contact – Tom@PCBLibraries.com

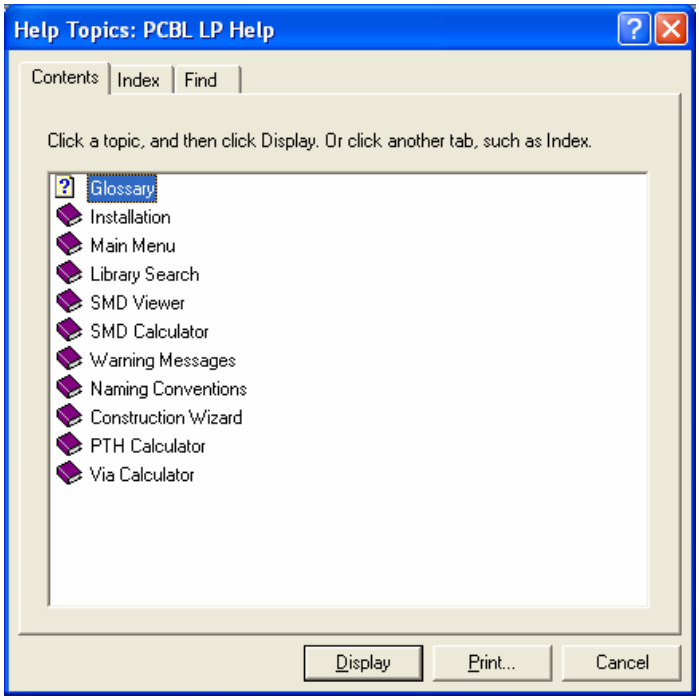
14 HELP FEATURE

14.1 Help Contents

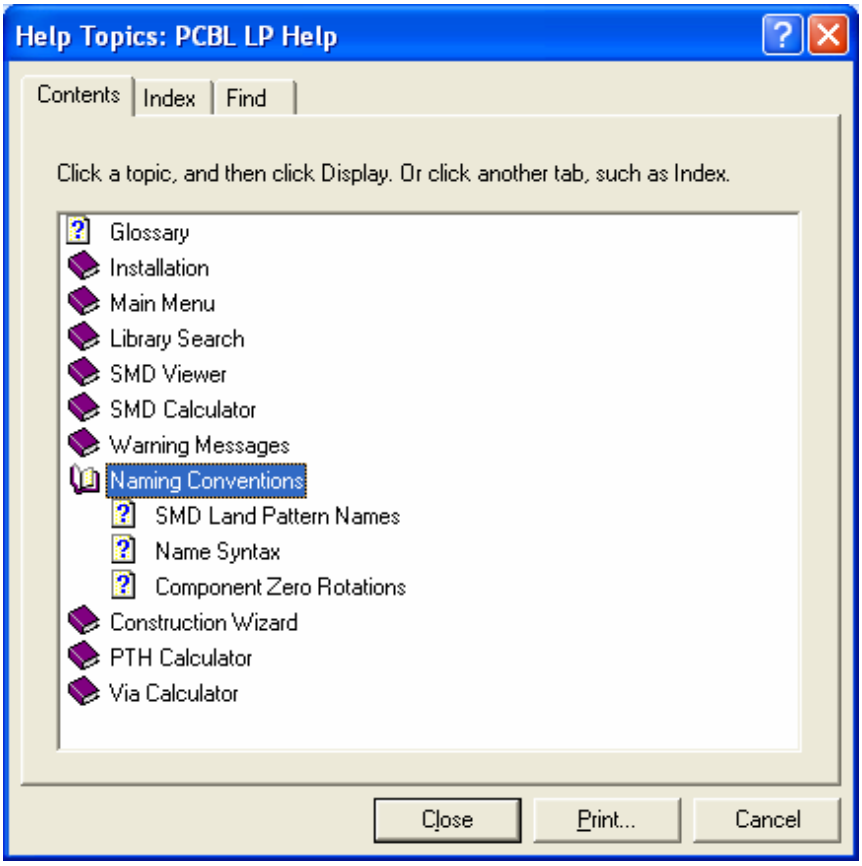
At the Main Menu select the Help Tool Bar Button

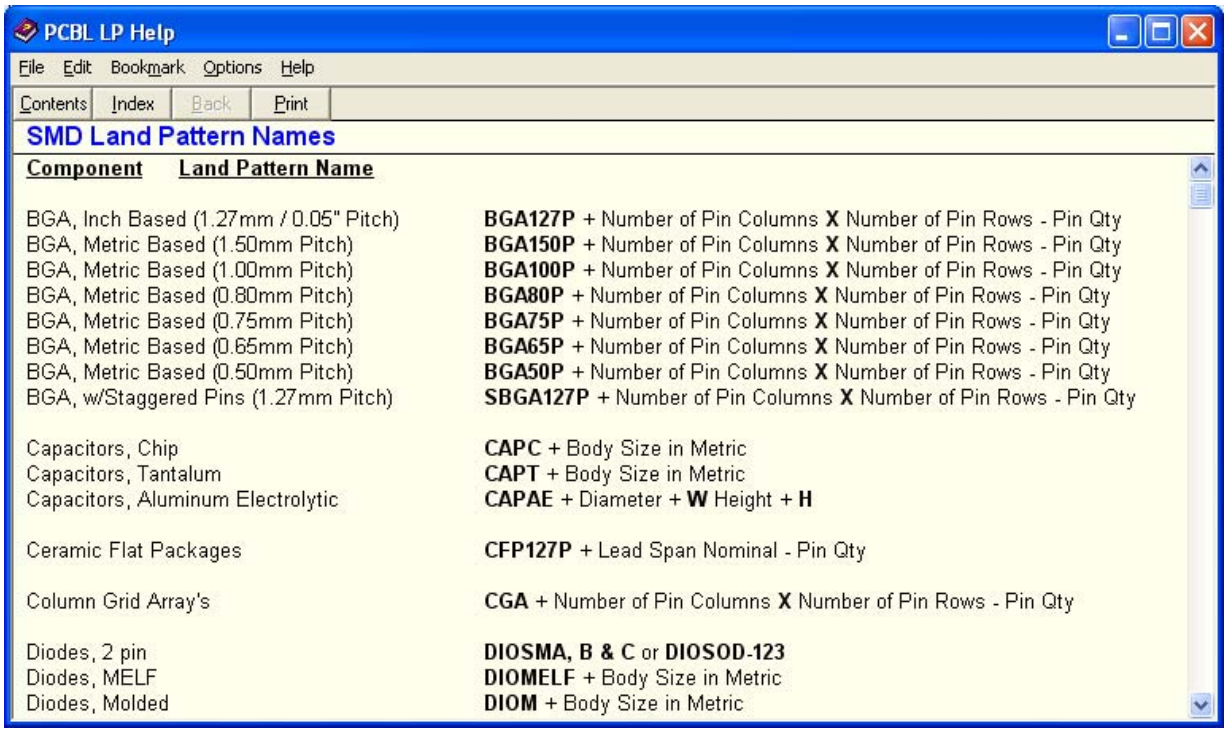


The Help Menu is broken into three sections – Contents, Index and Find



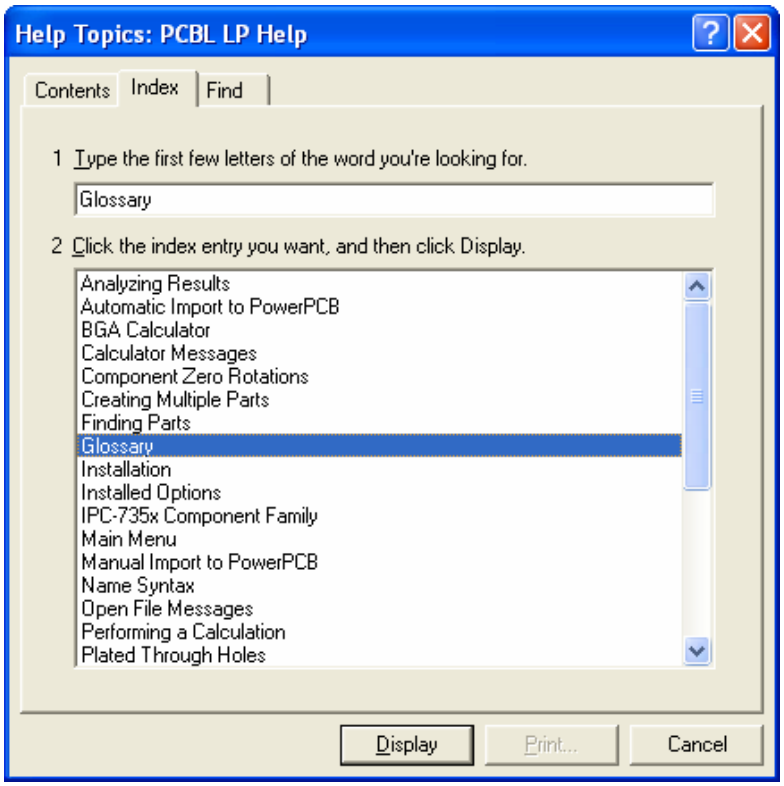
In the **Contents** Section select "**Naming Conventions**" and then select **SMD Land Pattern Names**

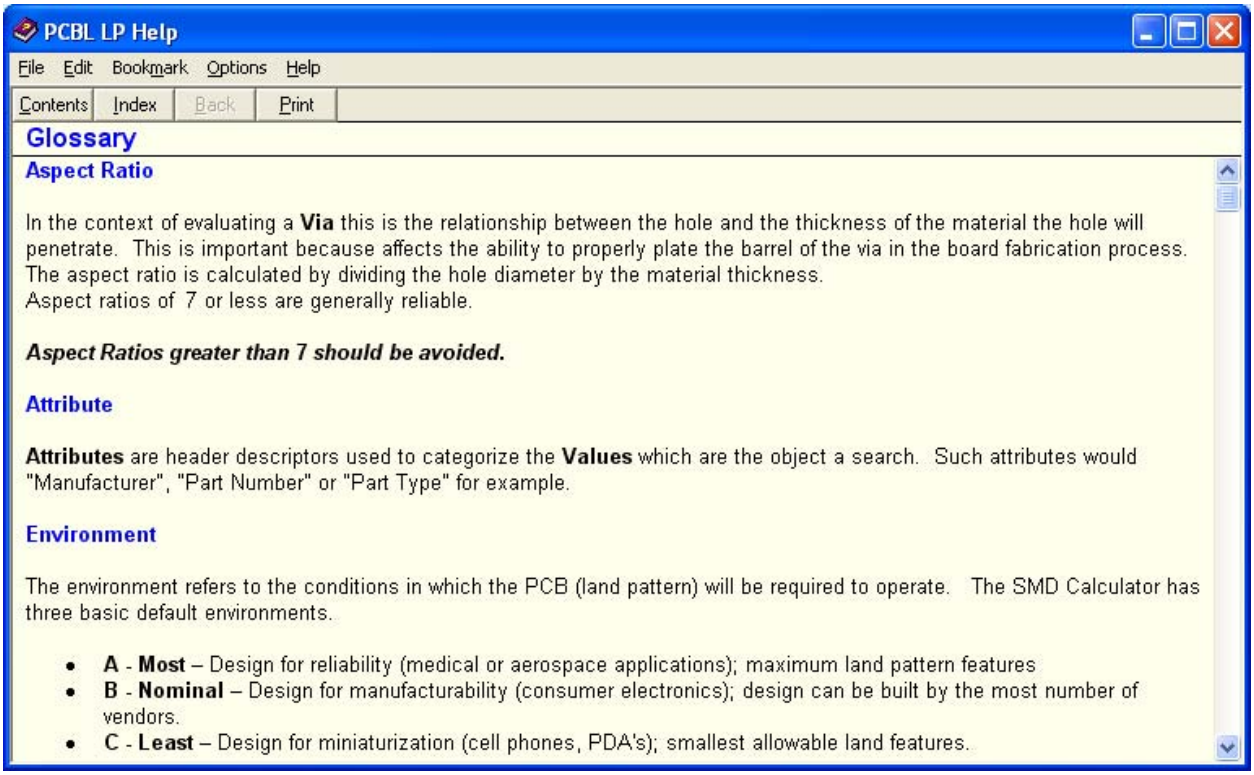




14.2 Help Index

Type in a “Keyword” or you can select from the list. Select “**Glossary**” & “**Display**”

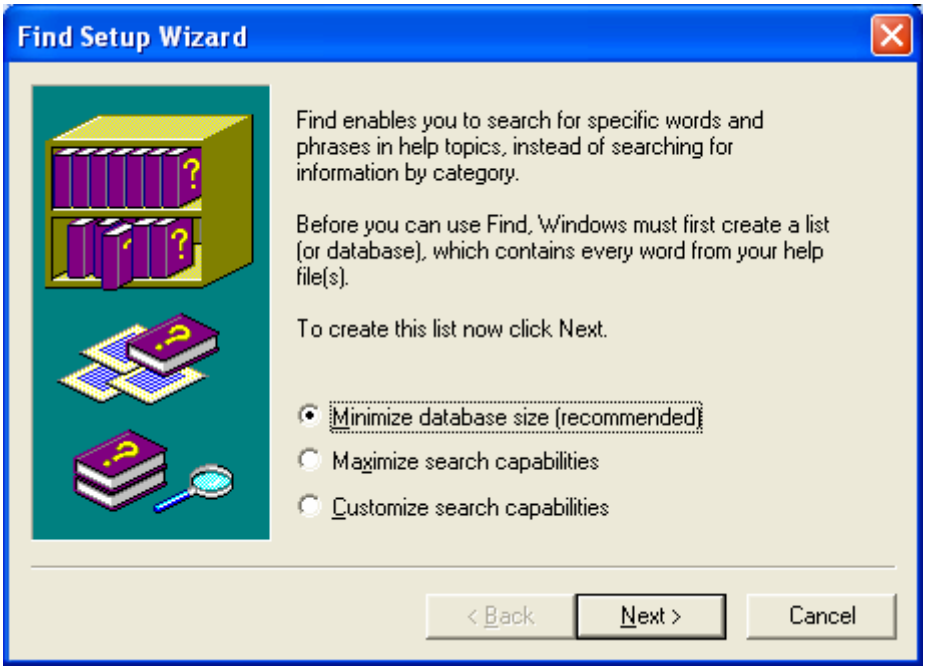




14.3 Help Find

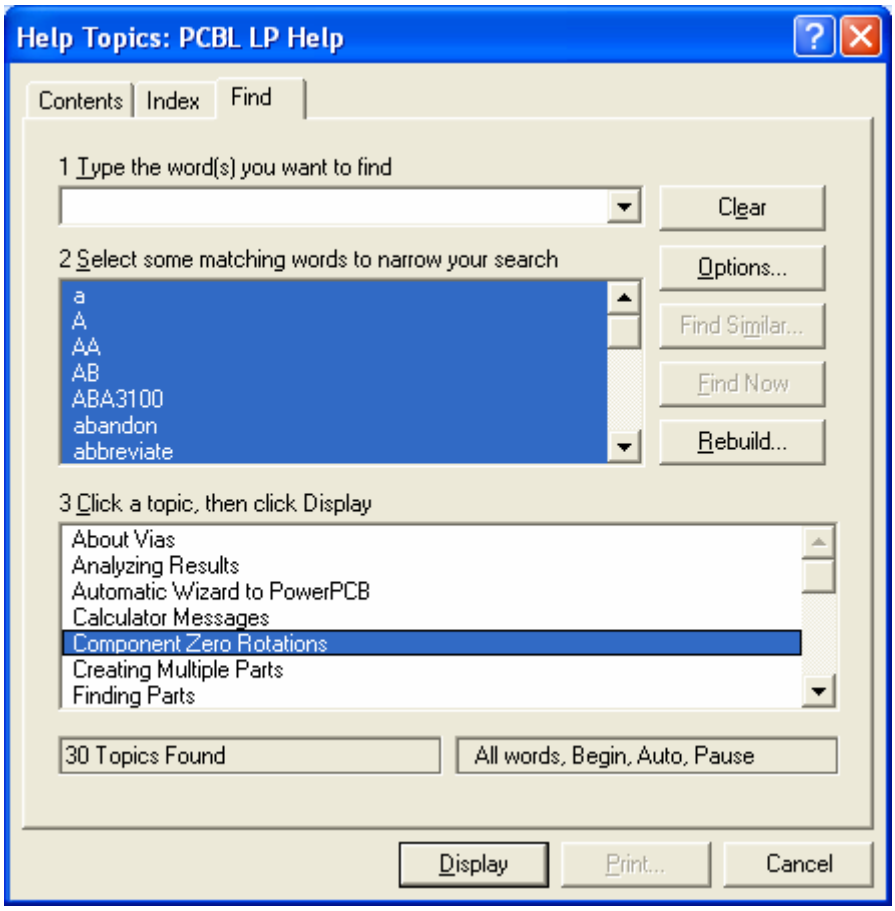
Using Help Find is another way to locate key features of the LP Suite program

Select the "**Minimize database size (recommended)**" and then select "**Next**"



In the "**Find Setup Wizard**" select the "**Finish**" Button to start creating the word list

In the Help Topics select the word "**Display Options**" and then select "**Display**"



Select the Display Button to retrieve the help data

Close all Windows and Start Building Library Parts

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

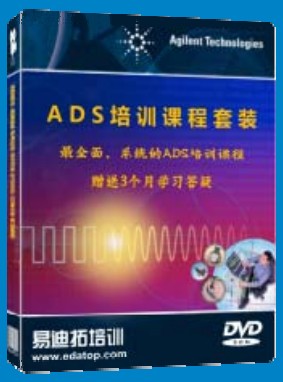
易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

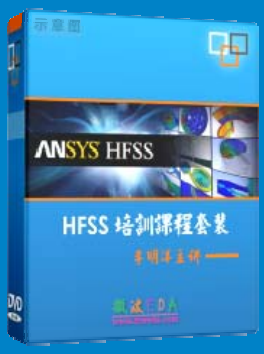
课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装



该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...

课程网址： http://www.edatop.com/peixun/ads/13.html



HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

CST 学习培训课程套装

该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com