# Contents

# 1 Introduction

BOM Builder is a software tool used to prepare electronic component data for automated assembly. This tool is design for use by circuit designers who design circuits and who are responsible for the realization of the final assemblies. Additional features within BOM Builder support component selection, schematic to PCB translation and purchasing functions. Control is provided for reuse of existing component data, the incorporation of new component and the control of design variations. BOM Builder eliminates the need for schematic library attributes and allows PCB decal definitions to be easily linked with purchasing database systems.

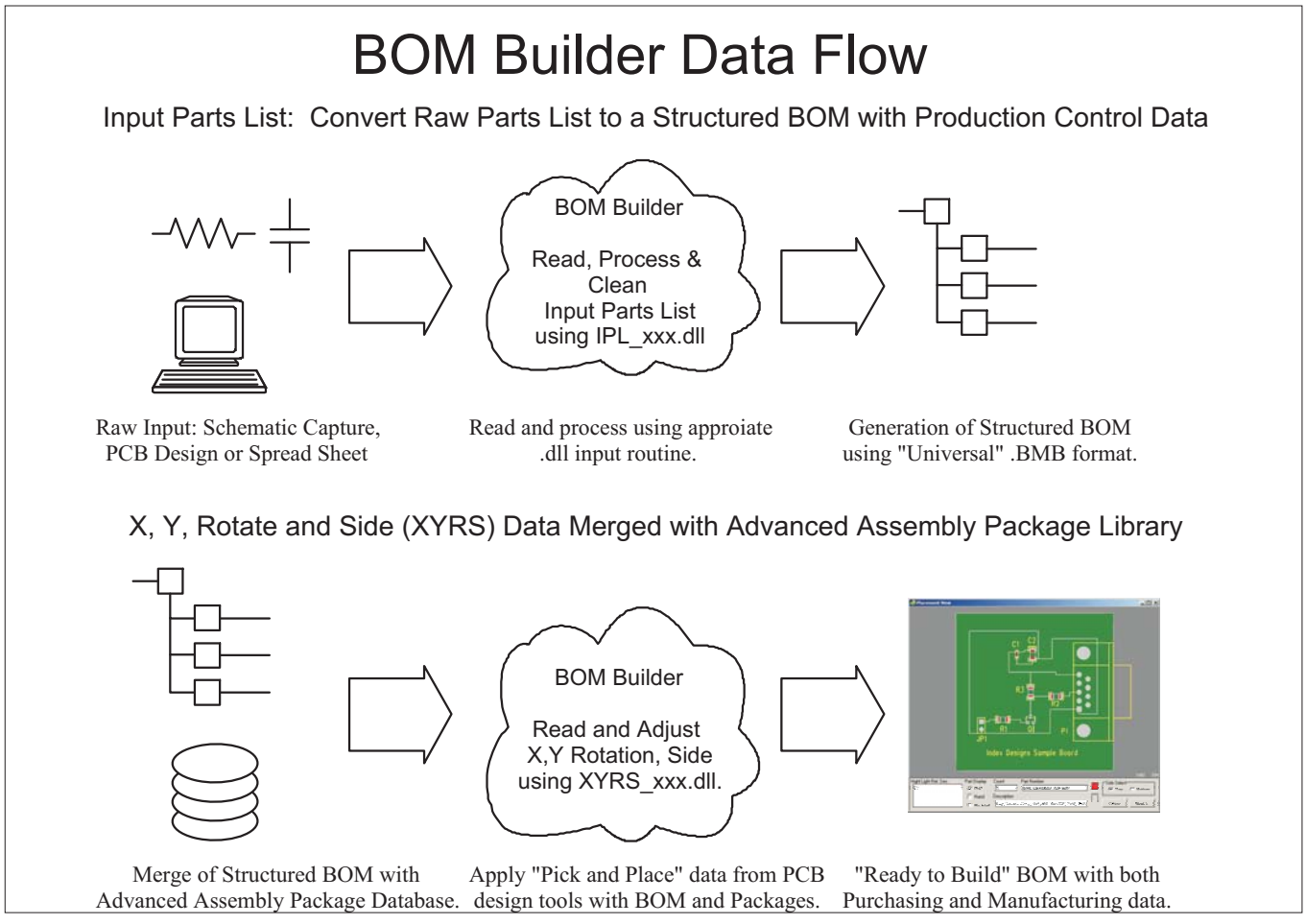
BOM Builder provides the required link to pass data to automated assembly. Index Designs provides data libraries making BOM Builder a unique tool for preparing assembly data.

## 1.1 Overview

The basic idea is quite simple, BOM Builder is a tool that collects a "Parts List" from a schematic report or spread sheet BOMs and associates that data with physical packages and their X/Y placement on a PCB surface. Several additional data items are collected and merged with this list:

- *Similar parts are merged into Line Items in preparation for Pick and Place setup.*
- *STUFF and NO STUFF status is provided to support construction of Product Variations.*
- *X, Y Rotations and Side (TOP or BOT) status (XYRS Information) for Pick and Place.*
- *Indicators for Through Hole, Machine and Hand Placed Components.*

With a parts list and all the component data present the next questions is "Is All The Data Right"? BOM Builder is very unique in that it presents all this data as a completed assembly image. Using a custom Gerber Reader and package graphic libraries, BOM Builder show the resulting data as a final board im-

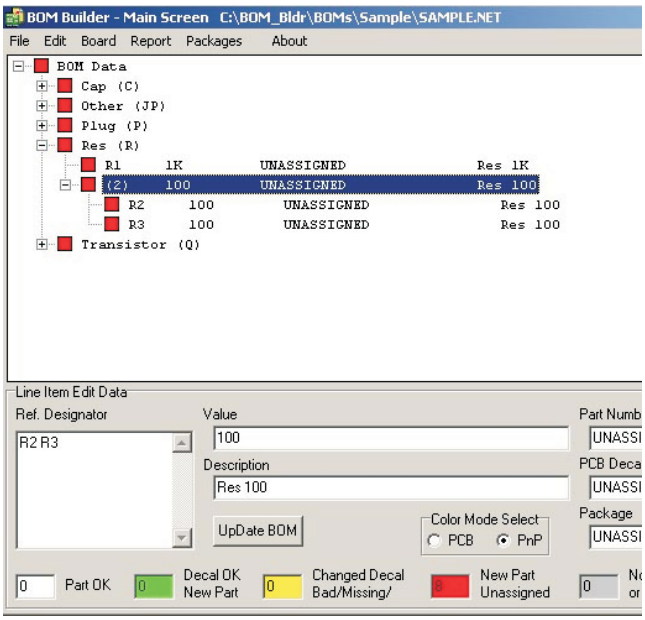


age. This image allows the "Form and Fit" of the various components to be checked BEFORE an actual bare PCB is fabricated.

The final result of this process is a set of production documents with all the data required to drive the manufacturing process. Data can be viewed and checked before bare PCBs are fabricated or purchased.

## 1.2 Input Data Overview

BOM Builder automatically sorts components into Line Items and displays the result in a multi-level tree structure. Software logic identifies line items by analyzing the Reference Designators, Component Values and, if they are provided, Part Numbers. As Line Items are formed from the input "Parts List" it is very common that errors are detected. Many BOMs are constructed by hand using Excel or other spread sheet programs. Such BOMs are often filled with errors and data inconsistencies. BOM Builder includes logic that detects input errors, provides a list, and allows an operator to quickly identify and correct the errors. If two or more parts have the same Part Number but they have different Values or Descriptions then "Line Item Errors" are reported. If Part Numbers are not provided BOM Builder will sort by Value and PCB Decal. Errors will also be generated if differences in Descriptions is detected. Accurate formation of Line Items is crucial for automated assembly processes. Errors detected while a job is "ON THE MACHINE" are very expensive. BOM Builder reduces the number of such errors.



Structured BOM.

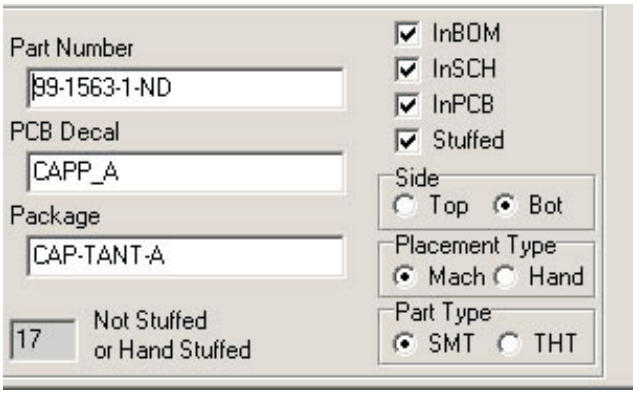
Sorting of components into "Line Items" is a fundamental requirement for both manufacturing and purchasing functions.

The number of line items in a project directly affects assembly machine setup as each component type requires a separate machine feeding location. Purchasing departments also require a list of required components in line item format. The number of BOM line items directly effects the effort required to purchase material and the effort of kitting components for the assembly process. Setup costs dominate low volume assembly costs and line item counts are the key contributor. Good engineering practices will always attempt to reduce the Line Item counts.

The concept of X, Y, Rotation and Side data being used for assembly is easily visualized. The need for Stuffed, Machine and Hand assembly data is not as obvious until the entire assembly process is considered. A kit of components must be sorted such that the various types of components arrive at the proper workstations. None of the Through Hole parts are required at the pick and place machine. Since some SMT parts can not be machine placed they must be hand placed BEFORE the solder reflow process. Through Hole part mounting is accomplished AFTER both top and bottom SMT (Machine and Hand) at yet another workstation. The cost for Hand and Through Hole assembly is much higher than SMT machine assembly and these counts are very important when computing assembly cost.

BOM Builder provides several Manufacturing Controls along with controls for BOM generation and PCB design. It is very common for parts to exist independently in the schematic, PCB design or BOM.

These controls identify how component data is used and allows data to be transferred to various company departments in an automated manner.



Manufacturing Controls

A fundamental problem with many schematic capture and PCB design tools is they do not consider the various types of data required for assembly. It is very common for a single schematic and PCB design to be used for a number of different assembly variations. In each variations of the design it is the number and types of components installed which is different. BOM Builder provides the ability to construct these product variations from a single set of schematic and PCB inputs. Since each BOM Builder variation maintain a complete design database it is always possible to "Back Check" a BOM to the current CAD data. The ability to verify a variation against CAD design tools is not possible using spread sheets or other techniques.

## 1.3 Output Data Overview

Outputs from BOM Builder include PCB Design data, BOMs for both purchasing and production, and assembly documentation for assembly and test. Most important is the ability of Index Designs to take the final .BMB file and directly drive a SMT manufacturing process. Advantages of using the .BMB to drive our automated manufacturing process include:

- *Eliminate the need for schematic design libraries and CAD librarians.*
- *Eliminate errors associated with manual programming of Pick and Place equipment.*
- *Allows usage of cut tape or loose parts.*
- *BOMs sorted per Assembly step. Through Hole parts are directed to Through Hole work stations, SMT parts go to the Pick and Place.*
- *Low cost stencil and solder paste dispensing.*
- *Eliminate random errors associated with hand assembly.*

Although generation of data for automated assembly is very important, BOM Builder can also prepare data for PCB design. Data concerning PCB footprints, electrical values and company part numbers is typically inserted into schematic through schematic library attributes. The preparation of these library attributes is time consuming, expensive and error prone.

BOM Builder eliminates the need to populate these CAD libraries with data from purchasing, manufacturing and MRP (Material Resource and Planning) systems. Instead, BOM Builder supports the annotation of circuit components using data from external software systems using “Point and Click” with a graphic interface.The result in a significant reduction in CAD library maintenance while quickly generating accurate BOMs for manufacturing. Changes in MRP, purchasing or manufacturing data do not require changes in library attributes, BOM Builder simply reads the new data and makes it available to circuit designers, synchronization is automatic.

BOM Builder does not require active connections to advanced “Data Servers” or SQL engines. Software within BOM Builder sorts tables, forms indexes, cross links tables and provides all internal data serving requirements. Raw data can simply be transferred as text files using local area networks, CDs or web browsers. Engineer can exchange simple files which include all required data.

## 1.4 Using Dynamic .DLLs

BOM Builder uses dynamically loaded .DLL files to read user CAD and component data. There are three major .DLL function and one internal Microsoft Access connection:

- *Input Part List IPL_xxxx.dll*
- *X,Y Rotate, Side XYRS_xxxx.dll*
- *MRP Data MRP_xxxx.dll*
- *Internal links to Parts & Vendors data in Microsoft Access format.*

Various .dll routines are supplied for each function and for various CAD systems. For example IPL_Protel.dll is used to read Input Parts List from Protel while the XYRS_Layout_Insert.dll is used to read XYRS data from OrCAD Layout Component Insertion reports. Additional .dlls are provided for reading spread sheet data (.CSV format) and simple text files. BOM Builder stores the selected .dll routines in the output .BMB file. Multiple BOMs from multiple CAD systems are easily managed using this technique.

For all IPL_xxx.dlls the required data consists of Reference Designator, Value, Part Number, PCB Decal and Description. Only the Reference Designator is required though a structured BOM can not be built if all the other fields are blank. Typical usage involves using the Reference Designator and part Value as contains in the original schematic.

Note: Reference designators must consist of ALPHA characters followed by NUMERIC characters. Reference designators such as R1, C9999 and SWIT5 are accepted. Reference Designators such as C5A, GND, +5 or A1_5 are not accepted.

Data for the XYRS_xxx.dlls requires a Reference Designator, X Location, Y Location, Rotation, Side and a PCB Decal. Only the PCB Decal is optional, all other fields must exist. Each .dll routine will process it's associated file format and will return the extracted data to BOM Builder in the proper format.

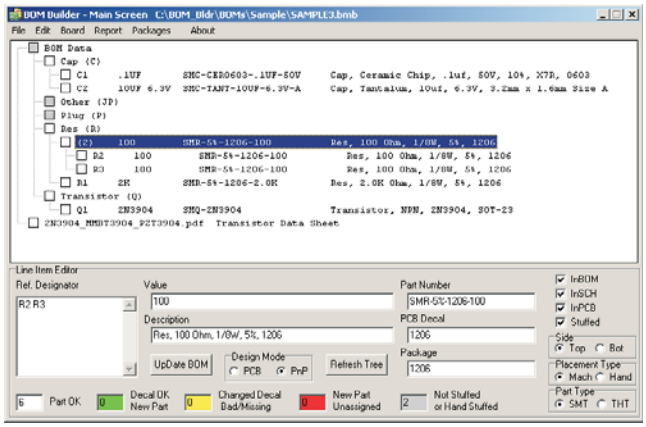
BOM Builder can populate it's internal search engines with user part data by reading data through the MRP_xxx.dll routine. Internal code provides support for reading and writing Parts & Vendors data structures.

## 1.5 Main Screen

The BOM builder main screen is show below and is divided into top and bottom sections.

- *The Main Tree in the top section show a structured view of line items. Color coding in the Main Tree view gives an indication of Existing, New and Unknown parts.*
- *A Line Item Editor area is at the screen bottom. This area allows the user to Add, delete or modify the nature of line items.*
- *The Update button will force all the Line Item Editor fields into the shown reference designators. A Tree Refresh button can be used to redraw the Main Tree when required.*
- *Along the very bottom are the counts of components that fall into various categories.*

File SAMPLE3.BMB file has been loaded below and line items are shown. The line item consisting of 2 resistors R2 and R3 is selected. Each part is a 100 ohm 1206 resistor and each is to be include in the BOM and the PCB design. Both are to be placed by pick and place on the top side and each is a SMT type component. When doing compares to the schematic these parts will be verified since each is checked as being "In The Schematic".




Main Screen

While the various control flags might look like a significant amount of "Extra" work most will be automatically set as data is read into BOM Builder.

## 1.6 Assembly View

A unique feature of BOM builder is the ability to show how parts will fit on the surface of the final PCB. Using the Index Designs library of Pick and Place package models and an internal Gerber Viewer, BOM Builder can show how packages will fit before the actual PCB is fabricated.

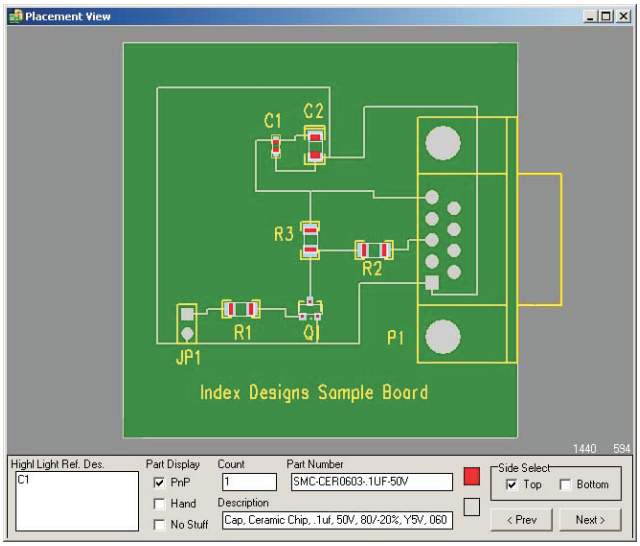
In addition to simply viewing the BOM data the user can use the Assembly View to:

- *Adjust component position and rotation.*
- *To query a mount location and determine the RedDes, Value, Desc. and Part Number.*
- *Identify the line item to which the part belongs and locate similar parts.*

- *Verify polarized component orientation.*
- *Verify component fit and PCB Decals.*
- *Generate "Smart" .PDF containing BOM data and PCB graphics.*

A very important function of the assembly view is to identify how component leads fit on the copper and solder paste deposits. Copper is shown in grey while solder paste is shown in light blue. Package models from Index Designs show metal lead faces in RED or BLUE depending on the displayed side. The ability to visually inspect how a component lead is positioned in paste and on copper is critically important to product reliability and assembly yield. Further the setup and programming of automatic assembly equipment is 100% verified.

The ability to automate the collection of assembly data from schematics, PCB designs, CAD tools or Spread Sheets and to combine this data with a library of component models is unique. The ability to drive a automated assembly line with this data is truly a unique service.




## 1.7 BOM Builder Does Not:

BOM Builder does not replace the need for an external component database system. BOM Builder simply organizes external data into a format that allows a rapid association between BOM line items and external data. BOM Builder maintains these links in a local database and can exchange these data structures using simple text files. This process eliminates the need for extensive attributes within CAD tool libraries. BOM builder is a “Steal the Data as Required” environment that keeps a record of what was “Stolen” and allows sharing of the “Stolen” but organized data.

BOM Builder will operate in a stand alone manner without a component database. Index Designs provides a limited database of standard parts for companies without internal systems.

## 1.8 Installing

Installing is very simple, simply download and run the distribution from www.aapcb.com. If BOM Builder is already installed it MUST be removed first. Use the "Remove Programs" function in Co"Control Panel". If you have made any changes to the c:\BOM_Bldr\Data\BOM_Bldr.ini file backup this file first. Also, if custom PACKAGES have been created backup the Package Data (PACKAGES - BACKUP PACKAGES) then save the c:\BOM_Bldr\Data\BOM_Bldr.ini file.

# 2 Input Parts Lists

Schematic capture programs are the preferred source of part data. Spread sheet are often used but most spread sheet users generate these list by hand and the data is often inconsistent or simply in error. Other source of data include PCB design software though once again this data may not represent what is truly required by the schematic designer. In either case the required data consists of:

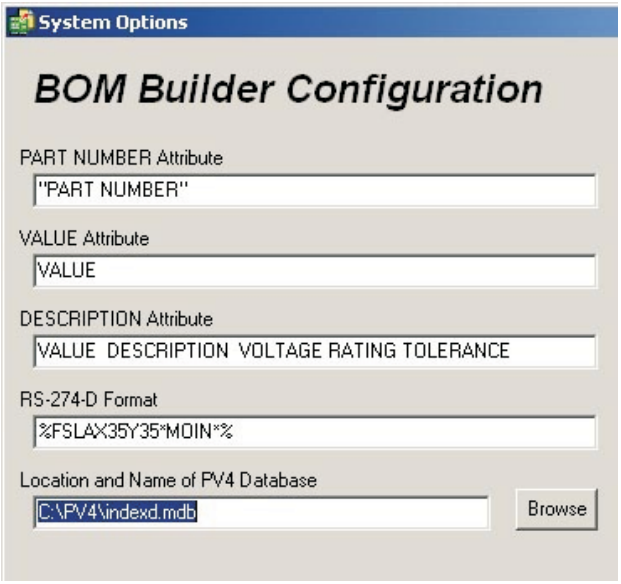
- *RefDes: A simple combination of up to 5 letters followed by a number between 1 and 9999. Valid examples are C1 and RN5. Invalid examples are U5A, J1-1 and +5.*
- *Value: A short text string that provides basic information about a component. Examples are 10K, .1uf and LM555.*
- *Part Number: Optional. A company part number is very useful as it gives exact definition about the nature of a component*
- *Description: Optional. Provides useful information for assembly, purchasing and test. BOM Builder will generate default Descriptions.*
- *PCB Decal: Optional. If available from the schematic it will assist with forming line items. PCB Decal is provided my most XYRS input .dlls and is not required in the parts list.*

The above data items provide the information that is used to form line items. If a Part Number is provided parts with identical Part Numbers will be merged into a single line item. If parts with identical Part Numbers but different Values or Descriptions are detected BOM Builder to report "Line Item Errors". The nature of these errors should be resolved.

## 2.1 PADs .ASC Format

BOM Builder contain internal logic to read in PADs .ASC files which are generated by the Mentor PowerPCB design tool. This type of .ASC file IS NOT the same as a .ASC netlist file generated by most schematic capture programs. A PowerPCB .ASC file contains all the XYRS data, Decal names, Values and other component attributes. These attributes may or may not contain useful data depending on the quality of the schematic and PCB libraries.

Three fields in the System Options screen allow the selection of PADs attributes to be used as input for various BOM Builder fields.



System Options Screen

The above setting will result in the .ASC reader using attribute "PART NUMBER" for BOM Builder part numbers while the VALUE, DESCRIPTION, VOLTAGE RATING and TOLERANCE attributes are all merged for use as a Description. This information is only valid if the PCB designer has carefully prepared component attributes. This is seldom the case.

As the .ASC file is read two conditions are detected and the user must indicate how to proceed. These conditions are:

- *New reference designators are detected.*
- *Should PCB decals be read.*

A PCB design may contain many items that are not found in a final BOM. Examples of these are mounting and tooling holes. If a BOM Builder file already has been edited to where it contains only the required parts then the user does not want the reader to read in new reference designator. If the user is starting a new BOM Builder file then reading of all the reference designators is required.

## 2.2 Schematic Input

BOM Builder reads parts list information from a schematic using one of two different input functions.

- *FILE - OPEN OTHER - PADs NETLIST*
- *Use of IPL_xxxx.dll routines*

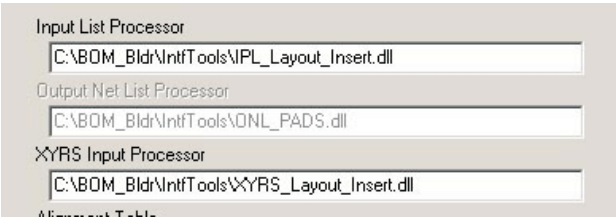
## 2.3 PADs Netlist

The first is a PADs netlist format which has component information in the *PARTS* section. Like the .ASC reader attributes will be pulled from the netlist according to the attribute names assigned in the Systems Options screen. Many schematic capture programs will generate PADs netlists but few will populate this netlist with component attributes.

PADs netlists are read using the Main Menu sequence FILE - OPEN OTHER - PADS NETLIST. While a PADs netlist can also be read using .dll routines the OPEN OTHER - PADS NETLIST allows use of named attributes when collecting data. See section 2.1 PADs .ASC format for a description of how this logic operates.

## 2.4 IPL_xxx.dll Routines

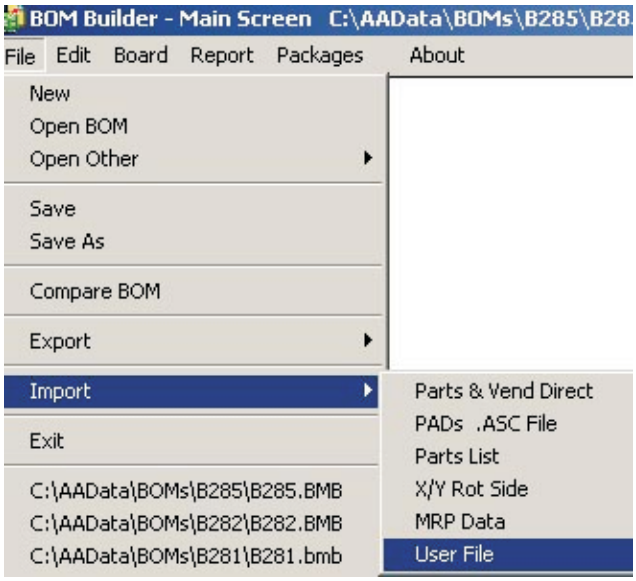
The second method is through a user selected IPLxxxx.DLL file. There are many different IPL_xxx.dll files each of which supports the reading of a specific report from a specific schematic capture program. A single .dll is selected by using a DOUBLE CLICK on that items field.



BOM Options .dll Selection

In the above example .dll routines for reading "OrCAD Layout Component Insertion Reports" are selected for both IPL (Input Parts List) and XYRS (X, Y, Rotation and Side) import functions. When the user selects the generic Import Parts List the IPL_Layout_Insert.dll will be used. When the user selects the generic XYRS Import the XYRS_Layout_Insert.dll will be used.

Selecting the "Parts List" menu item will cause BOM Builder to load the IPLxxx.dll and read in a parts list using that .dll. Selecting the "X/Y Rot Side" button will result in BOM Builder using the XYRS_xxxx.dll to read in XYRS data.



Generic Import Functions

The menus items are generic while the BOM Options setting select a specific .dll routine. The selected routines, as well as all the other BOM Options are saved in the final .BMB file.

Within the System Options screen ( EDIT - SYSTEM OPTIONS) is a table used to associate component descriptions with reference designators. When a net list is read the leading ALPHA CHARACTERS of the reference designator are stripped off and the “Ref. Des. To Type List” is search using the stripped alpha characters. If a match is found the “Component Type” text is appended to the description field.

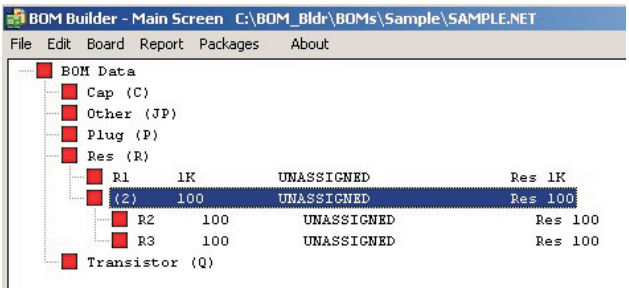
## 2.5 Import Example

The best description is an example. After starting BOM Builder use the sequence:

FILE - OPEN OTHER - PADS NETLIST

This sequence will open a dialog box showing a SAMPLE directory. Open this directory and click on the SAMPLE.NET file. (A SAMPLE.PDF files is also located in the SAMPLE directory and it shows the source schematic.) This .NET file was constructed in OrCAD Capture using the PAD output format with {Value} in the FOOTPRINT PROPERTY STRING.

When the SAMPLE.NET file is read a structured BOM tree will be shown in the upper portion of the Main Screen with several red elements. This tree is a view showing the various “BOM Line Items” which are contained in the schematic. Identical items have been grouped together. Click on the red “Res (R)” line which will drop open and then click on the red “(2) 100” line which will also open. As “BOM Line Items" in the tree are selected detail about these parts is shown in various fields located in the Line Item Editor section towards the screen bottom. This bottom area is where a user edits BOM line items and where the Update BOM button can be clicked which will update the BOM database with Line Item Editor data.



Read of SAMPLE.NET

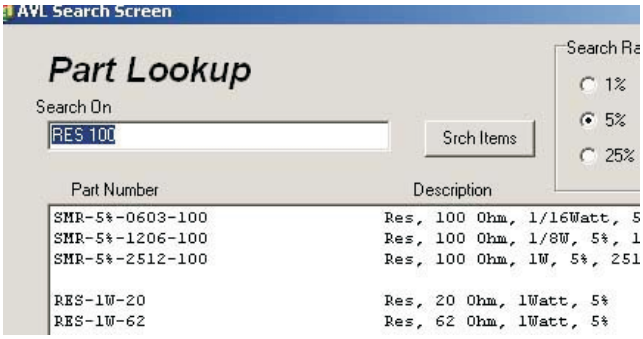
When schematic data is read, the Reference Designator and Value are used to provide the initial data concerning a circuit assembly. The key items are reference designators and component values for electrical parts. The resulting “Parts List” is the starting point for BOM generation.

Many companies generate extensive CAD libraries which include additional data (attributes) about these parts for use in secondary processes. Examples of these attributes include Company Part Number, PCB Decal, Vendor Numbers and assembly information. BOM Builder eliminates the need for these libraries.

## 2.6 Part Selection

With SAMPLE.NET read in all parts are shown as RED since most fields are undefined. Click on the “(2) 100” line item under the Res branch and the Line Item Editor at the screen bottom will show that R2 and R3 are selected and the current Description as Res 100. A DOUBLE CLICK on the Description field will start the search engine with results shown in the “Part Lookup” screen.

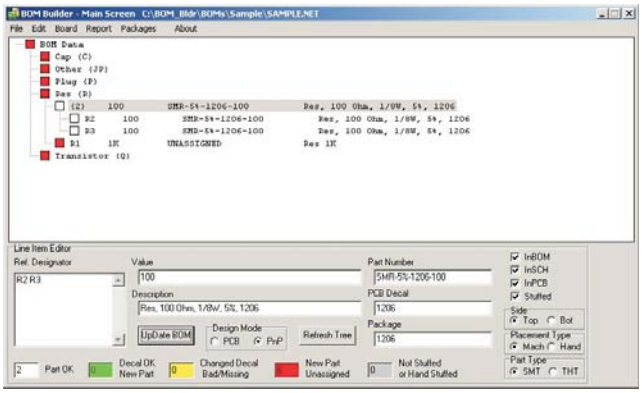
The top matches (three in this case) are 100 ohm resistors in different sizes. Parts are grouped by the



Results of Search on Res 100

quality of the match. In this case the first 3 items matched with equal priority.

In the Search Results List DOUBLE CLICK on the 1206 part and the main tree screen is shown with new information about the selected component in the Line Item Editor area. Click on the “Update BOM” button to save the new data for parts R2 and R3. Notice the R2 - R3 line item are now "White Coded" since complete engineering data is now linked. Also notice that both Package and PCB Decal contain “1206”, the Part Number is “SMR-5%-1206-100” and the Description is quite detailed.


"White Coding" of R2 and R3

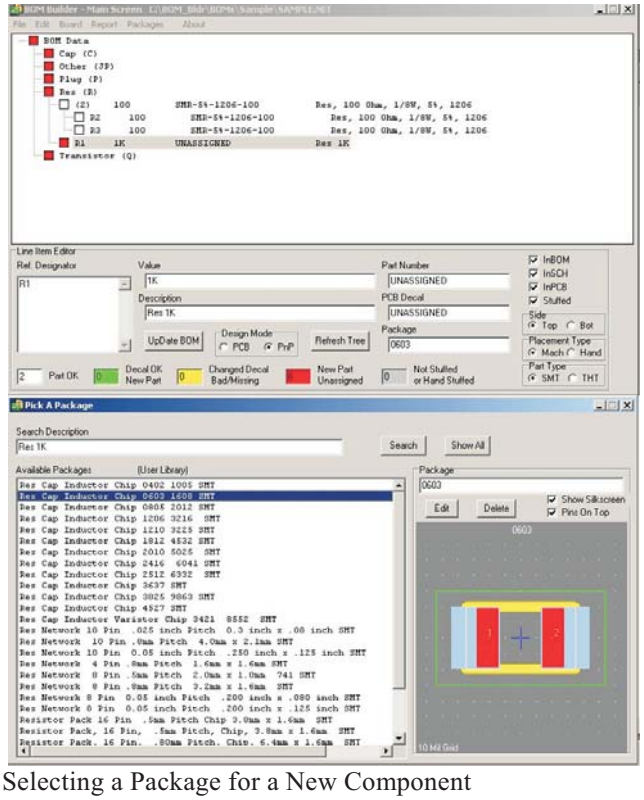
Annotation of component using this technique is a very powerful and unique function of BOM Builder. Just as different schematics and PCB tools are interfaced to BOM Builder through the use of .dll routines, the data which drive the search engine can be read from a user databases through .dll routines. With this ability to rapidly associate data between line items and a company database the need to place attributes in schematic libraries is eliminated.

BOM Builder eliminates the need for these attributes by allowing an engineer to “Steal” data from external data sources AFTER the schematic is complete. Schematic generation is much faster and data more accurate since part assignment is a “Post” schematic design process. Engineers can concentrate on connectivity with support staff assisting with most part assignments. BOM Builder encourages data reuse by providing a means for collecting information about new components.

## 2.7 New Part Creation

When designing a circuit assembly new parts must be introduced into the engineering process. BOM Builder provides a means for identifying new components and allowing a project to continue until new part numbers are generated.

Click on the “R1 1K” line item in the main tree which shows R1 and Res 1K in the Line Item Editor fields. Part Number, PCB Decal and Package are all UNASSIGNED. DOUBLE CLICK on the “Package” field which will open the Pick A Package screen and start the package search engine. Move the windows as shown below



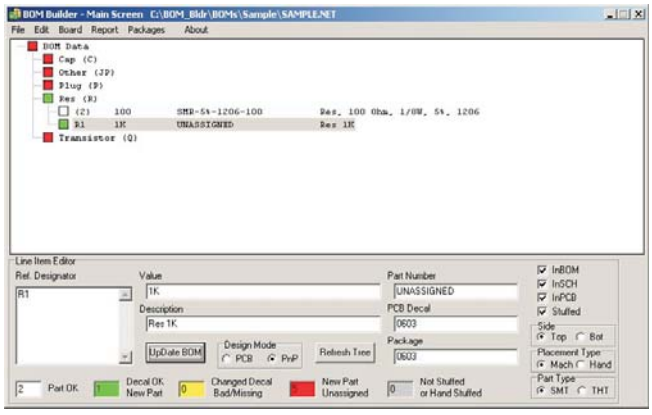

Selecting a Package for a New Component

You might need to click on the "Search" button to execute the search. Click on the: “Res Cap Inductor Chip 0603 1608 SMT” line in the “Available Packages” list. You will see 0603 filed into the Package field in the line item editor and at the same time the "Part Type" field will change to SMT.

At this point you have selected a new package for R1 but the BOM has not yet been updated. Click on the "UpDate BOM" button in the Line item editor. Several actions will take place when UpDate BOM is clicked.

- *A User Table is searched for any PCB Decal names that have been associated with the 0603 package.*
- *Either a matching PCB Decal name or the Package name is filled into the PCB decal Field.*
- *The BOM database is updated from the fields in the Line Item Editor area.*
- *R1 becomes "Green Coded". All the required data is present for PCB design and manufacturing. Purchasing data is not available as no Part Number has been assigned. (New parts don't have numbers.)*

The resulting screen is shown below.




Green Coded Part after Update BOM

BOM Builder allows construction of line items which are “Technically Incomplete” yet it allows the design and manufacturing process to continue. The GREEN indicates the PCB process can continue but there are open issues with this line item.

With the GREEN “R1 1K” item selected click on the STUFFED check box (which will remove the check mark) then click on the “Update BOM” button. This identifies that this part is not to be stuffed and causes the color to change to grey. Additional check boxes allow parts to be independently marked as IN BOM, IN PCB Design or included IN SCHEMATIC checking. BOM Builder includes logic for comparing different BOM versions or comparing

new schematics to existing BOMs. The " In SCH" bit is used in this checking logic. There is also a "Part Type" field which allows a part to be marked as SMT or Through Hole (THT). This setting, along with the Top and Bottom side status, is typically set automatically. There are occasions where manual control of these and other flags is required.

PCB Assembly techniques require that material be sorted by technology (SMT or THT) and side (Top or Bottom). Key to efficient and timely assembly is having the correct material at the correct work stations. Machine operators and programmers need exact listings of the material that must be processed.

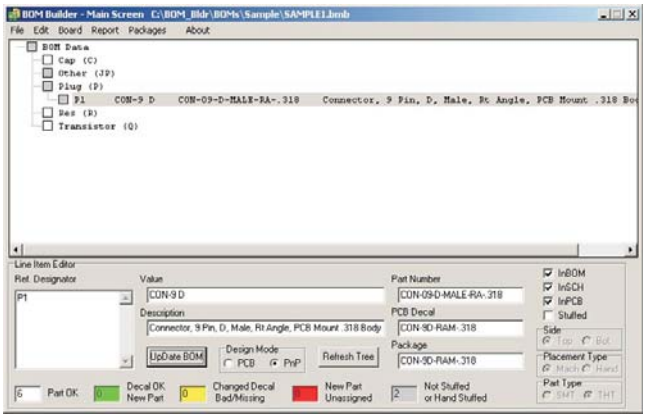
At the bottom of the main tree screen is a "Design Mode" group of radio button. Click on the PCB (PCB Design Mode) button and coded parts turn to Yellow. While these parts have PCB decals assigned there are no corresponding entries in the User Decal Table. The Yellow indicates that there may be a problem with the PCB decal assignments. The color of the line items is different for different design phases. PCB is associated with PCB Design while PnP (Pick N Place) is associated with manufacturing processes. Also notice that when a "Design Mode" button is clicked the count of component for each color state is recalculated and shown in the bottom color boxes.

Continue selecting each line item in the main tree view and then naming each part by double clicking on each Description then selecting the below parts from the part search engine results. The final part assignments will be:

C1 SMC-CER0603-.1UF-50V
C2 SMC-TANT-10UF-6.3V-A
P1 CON-09-D-MALE-RA-.318
JP1 POST-025-1X2
R1 SMR-5%-0603-1.0K
R2 SMR-5%-1206-100
R3 SMR-5%-1206-100
Q1 SMQ-2N3904

Select P1 and change it to be HAND stuffed, do the same for JP1. These parts are through hole devices and they can not be automatically placed. Unless these parts are marked as HAND they will show as yellow in the PnP color mapping mode.

Save the final BOM as SAMPLE1.BMB using the FILE - SAVE AS menu. Edit the name of the file to SAMPLE1.bmb in the SAMPLE directory. This output .BMB file has all the linked data, setup and options recorded in a single text file format.



SAMPLE1.BMB Final Status

## 2.8 Part List Summary

The most important aspect of reading the input parts list is getting the Value and Descriptions correct. If a Part Number is read IT MUST BE CORRECT. It is this information that drives the assembly process and bad data is guaranteed to deliver miss-stuffed assemblies.

Part Numbers are a key to data reuse. Accurate procurement, manufacturing, PCB foot print selection and the programming of automatic assembly equipment rely on having verified data. Maintaining a single accurate component database requires a sginificant effort that few organizations master. Keeping multiple independent databases for schematic entry, PCB design, purchasing and manufacturing is extremely difficult. BOM Builder is a unique tool for merging data from multiple sources.

# 3 Reading Gerber Images

A major feature of BOM Builder is giving users the ability to visualize and verify BOM and physical assembly data. Just as a PCB design tool shows how traces will be rendered into Gerber fies (then to copper) BOM Builder shows how components are "rendered" on a Pick and Place. The "Assembly View" screen is actually a "Software Pick and Place" engine that shows physical package data on an image of the final PCB. Generation of that PCB image requires of several Gerber files, the construction of graphic data for each layer and the alignment of those layers into a color graphic image.

Three Gerber files are required for each side (Top and Bottom) of the PCB that contains machine or hand mounted components. These required layers are Copper, Solder Paste and Silkscreen. Reading of Gerber data is complicated by the need to align the layers and locate actual components. The general steps are:

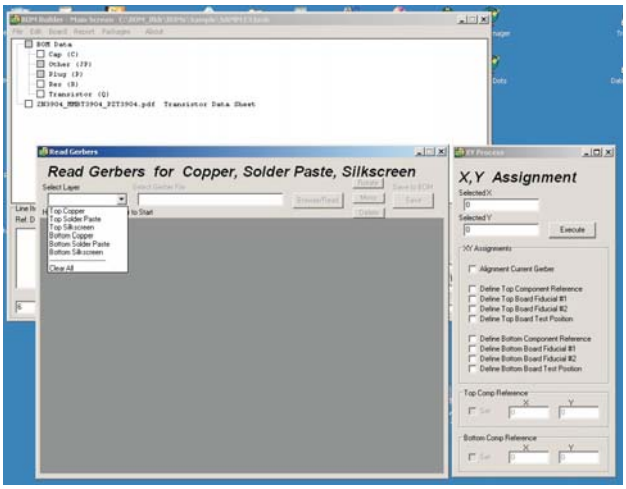
- *Align Gerbers: While many CAD tools will output Gerber data relative to the same X,Y location some uneducated users still "Center" Gerber plots. This results in random offsets which requires all layers to be manually align. This would not be a problem except these same few "Designers" fail to place a consistent object in all the layers to enable manual alignment.*
- *Mirror or Rotate: All Gerber files should be provided as "View Through the Top". The orientation must be the same as the component location (XYRS) information.*
- *Set Component X,Y Reference:The 0,0 point for the component XYRS data must be defined. Often this is 0,0 but not always.*

BOM Builder reads Gerber data and generates internal display lists for graphic display. RS-274-X polygons are rendered as "Cross Hatch" area and "Clear" polygons are ignored.

## 3.1 Selecting Layers - Read

Selecting the BOARD - READ GERBER menu opens two windows, Read Gerber and X,Y Assignment. These screens are shown below.

The drop down list in the Select Layer control allows the user to select a layer for data transfer. When a layer is selected any Gerber data that already exists for that layer will be placed in the grey graphic screen. The mouse can be used to click and select graphic items from that screen. As items are



Select Gerber layer

select their X Y locations are placed in the X,Y Assignment screen and the selected item is highlighted in white. Pressing the Delete Key will delete a selected item. Line items can be selected by clicking on a center or endpoint. Round items are selected at their center.

Click on the BROWSE READ button to read an external RS-274-X Gerber file. Note that a layer must be selected before BROWSE READ can be selected. Very limited support is provided for RS-274-D files as special .dll files are required for reading the aperture tables. See file c:\BOM_Bldr\IntfTools\Gerb_Apertures.txt for details.

Once read the image will be fitted to the screen and a blue cross will show the 0,0 if this location is fitted on screen. Buttons allow an image to be mirrored or rotated. The SAVE button will copy the graphic image into the BOM Builder database and the Gerber image can then be saved with the BOM.

## 3.2 Align Gerber Layers

The X,Y Assignment screen will accept X and Y parameters and will use these to modify the current Gerber image or database according to a series of check boxes. These check boxes control how the X Y values are used to midify Top and Bottom side settings.

- *Align Current Gerber*
- *Define Component 0,0 Reference Point*

- *Define Board Fiducials 1 and 2*
- *Define Board Test Position*

In order for BOM Builder to overlay the Copper, Solder Paste and Silkscreen Gerber files into a single image, the various Gerber Layers must be aligned. When the Align Gerber check box is checked, and the user clicks the Execute button, the currently selected X Y location is subtracted from Gerber X Y locations. This results in the selected X Y location becoming the new Gerber 0,0 location. (Do not confuse this with the Component X Y Reference Point.) Unaligned gerbers files can be easily aligned if there is a common graphic object in each layer.

Each time the Align Current Gerber is checked and processed a copy of the selected X Y location is saved in the BOM Options screen in a Gerber Offsets area. Also in that area is a Enable flag. If the Enable flag is checked then these X Y offsets will be automatically applied to Gerber files as they are read in. Since most Gerber files are constructed using identical offsets this automatic usage simplifies the reading of multiple files. If Gerber Files have been "Centered" (which is a very bad idea) then the user should uncheck the Enable flag and preform alignment manually.

Once a gerber file has been read and adjusted it must be saved to the currently selected layer by clicking the SAVE button. This saved gerber image will be saved with other BOM data and will eventually be written into the final .BMB file.

The Component X,Y Reference location is very important. When XYRS information is read from PCB CAD tool files it is this point which is used as the graphic 0,0 reference point. If this point is not set correctly components can not be correctly located on the board image. BOM Builder contains "Angle, Offset and Translation" (AOT) logic that will adjust for XYRS data not being defined for the component centroid. In order for this logic to operate correctly the Component Reference Point must be set correctly. This reference location will be marked with a red + sign.

If the Component Reference Point is not set correctly any attempts to use AOT logic or Group Moves will fail. The Component 0,0 reference point must be accurately defined.

Each PCB side with machine mounted parts must include the definition of two board alignment fiducials. When a check box is checked and the Execute button is clicked the selected X Y values will be used for the corresponding fiducial X,Y location. Fiducial 1 will be drawn as a small circle with a + sign, fiducial 2 will be draw using a small circle. Both makers are drawn in red.

Automatic placement equipment requires a location where the board height can be safely sensed. This test position must be clear of components and large holes. The Board Test Position sets this location and it is marked with a small circle and X.

At the bottom of the X,Y Assignment window is a flag and X,Y field for both the Top and Bottom sides. As the Component Reference point is set for each side the associated check box will be set. The X,Y location from the Gerber object will be placed in the X and Y fields. The 0,0 of a Gerber is related to the Component Reference Point through these X Y locations. The display of these locations can help when unknown Gerber files and XYRS files must be debugged.

## 3.3 Read Gerber Example

The C:\BOM_Bldr\SAMPLE directory contains a set of Gerber files generated from PowerPCB. Also included is a file containing all the component XYRS data.

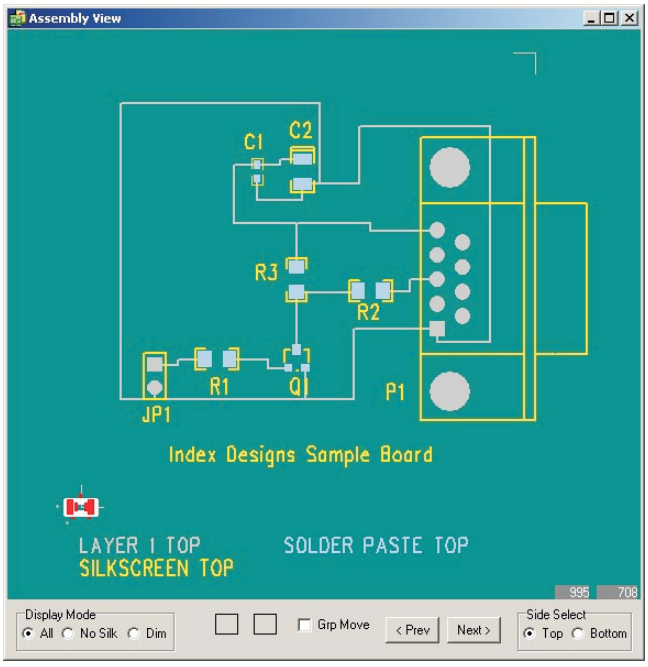
- *ART01.PHO (Top Side Copper)*
- *SST0126.PHO (Top Side Silkscreen)*
- *SMD0123.PHO (Top Side Solder Paste)*

By reading these gerbers and position text into BOM Builder an image of the PCB can be generated. The following steps will read these Gerber files and construct a board image.

- *Open the SAMPLE1.BMB file created in Chapter 2.*
- *Use BOARD - READ GERBER to open the Gerber Read screen.*
- *Use Layer Select to select Top Copper.*
- *Click the Browse/Read button and navigate to the c:\BOM_Bldr\BOMs\Sample directory and Open the ART01.PHO file.*

- *In the X,Y Alignment screen enter 1000 in both the Selected X and Selected Y fields or click on the lower left corner of the lower left board edge mark. (A 1000 x 1000 mil offset was used when creating this Gerber.) Check the Align Gerbers check box and click Execute.*
- *Save the Copper Gerber image to the BOM using the Save button.*
- *Use Layer Select to select the Top Solder Paste. Use Browse/Read to Open file SMD0123.PHO. This file was automatically aligned so save to the BOM using the Save button.*
- *Use Layer Select to select Top Silkscreen. Use the Browse/Read to Open file SST0126.PHO. This was automatically aligned so save to the BOM using the Save button.*
- *Close the Read Gerber window.*
- *Use FILE - SAVE AS to save the file as SAMPLE2.bmb in the c:\BOM_Bldr\Sample directory.*

At this point the board Gerber files have been read into the BOM. Use the BOARD - ASSEMBLY VIEW to view the merged gerbers along with all the parts sitting at location 0,0. The resulting merged image is shown below.




Sample2.bmb Gerber read Parts at 0,0

The above image shows the Copper in gray, Silkscreen in yellow and Solder Paste in light blue. All components are located at 0,0 since the XYRS information has not yet been read.

# 4 XYRS & Placement Data

Data concerning component locations and rotations is probably the most important type of data used during automated assembly. Without good data everything else is just "Hand Assembly". Most machine operators "Hand Adjust" placement data using an actual PCB as the target. Nothing could be more prone to errors. Silkscreens which direct hand placement are often removed from pads and vias if the silkscreen even exists at all. Camera views on machines are limited and working with a limited view can result in errors. BOM Builder contains the tools to "Get it Right" at design time which improves data accuracy and reduces assembly time.

As important as XYRS data is to the assembly process there are no standards for CAD data input or data being sent out to assembly equipment. Combined with these issues are another set of problems:

- *Stuffing Options*
- *Mount Order*
- *Missing Orientation Marks*
- *Missing Fiducials*
- *Different Rotation Directions*
- *Package Variations*
- *Machine Orientation*

The generation of XYRS data for parts that are machine mounted must be sorted by side. A single sorting scheme is not sufficient to drive automated assembly. Various pieces of data must be provide for complete and accurate assembly. Incomplete data results in delays and errors.

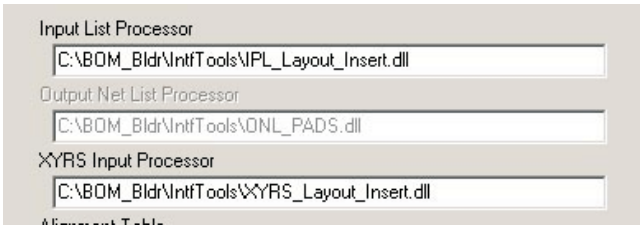
Index Designs has addressed these issues through the construction of standard package data and by coding BOM Builder to record and reuse rotation data. This process consists of:

- *Reading CAD XYRS Data*
- *Adjusting Positions*
- *Recording and Reuse of Adjustment Data*

## 4.1 XYRS .DLL Routines

The use of a XYRS_xxxx.dll requires that the XYRS data format be known. A number of .dll routines are provided for various CAD programs. New .dll routines are easily written if the CAD data format is known.

A single XYRS_xxxx.dll routine is selected through the BOM Options screen. A DOUBLE CLICK on the XYRS Input Processor field will allow the user to select the appropriate XYRS_xxxx.dll file.



BOM Options Fields for .dll Selection

When the main menu "IMPORT - X/Y Rot Side" item is clicked, the above pre-selected .dll file will be used to read several pieces of data.

- *Reference Designator*
- *X and Y Location*
- *Rotation*
- *Side*
- *PCB Decal*

Since XYRS data is generated by PCB CAD tools the actual PCB Decal associated with each reference designator is often available. PCB Decal data is key to alignment reuse. Most PCB designers are very careful about PCB decal assignments since one wrong decal and a new PCB fabrication cycle is required. PCB decals are very similar to ndex Designs Package libraries and these decal assignments serve as the "Key Link" between manufacturing data and the PCB design. When using the "Group Move" function to adjusting component positions this correction data is associated with theses "Key Links". These linkages are collected while BOM Builder is operating and they are saved in Alignment, Offset and Translation (AOT) files.

AOT files can be read and saved automatically or manually. Several options in the BOM Builder .ini file controls this operation. Adjustment data from these .AOT files are used when reading XYRS data from a PCB CAD tool. The result is BOM Builder "Learns" about the PCB decals from a specific CAD tool and only new components require adjustment. Existing components are correctly placed automatically using existing data.

## 4.2 Read XYRS Example

Use the FILE - OPEN BOM to read in the file: c:\BOM_Bldr\Sample\sample2.bmb . Use the EDIT - BOM OPTIONS to show the BOM Options Screen. DOUBLE CLICK on the XYRS Input Processor field and a file open dialog box is displayed with a number of XYRS_xxx.dll files shown. Using this dialog box select the XYRS_PADSXYPositions.dll file then click OPEN. This file name will then be shown in the XYRS Input Processor field. Close the BOM Option screen using the upper right X button.

When the BOARD - ASSEMBLY VIEW screen is opened all the components can be seen at the 0,0 location. Use the Main Screen FILE - IMPORT - X/Y Rot Side to read in PCB position data from:

c:\BOM_Bldr\BOMs\Sample\POSITIONS.txt

Click OK on the "XYRS Unit Select Form" and several action will take place.

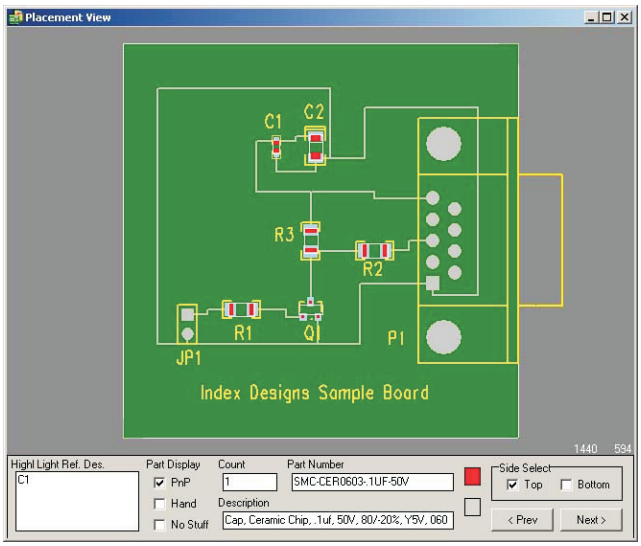
- *XYRS Data is Read*
- *Component Locations in the Assembly View screen are updated.*
- *A XYRS_Log.txt result file is created and opened in the default system editor for .txt files.*

As XYRS data is read the incoming reference designators are checked against existing BOM components. If a new item is detected in the XYRS data it is flagged. If a BOM component does not receive XYRS data then this is noted at the end of the log text. Missing or extra items could indicate errors with the Input Parts List.

The resulting Assembly View screen is shown below. In this example, the X,Y locations for the various components will all represented the true component centroids.

If PCB Decals had been constructed with origins at pin 1 locations or with rotation orientations different from the Index Designs libraries, the location of the shown components would be offset.

Use the main menu FILE - SAVE AS to save the complete database as SAMPLE3.bmb. All Gerber files, part images, BOM line items are saved in the .BMB file. When the file SAMPLE3.BMB is opened the board can be viewed. Using BOM Builder users can view the BOM, component placements and stuffing options BEFORE a bare board is fabricated. This feature can detect bad PCB decals that could result in expensive re-spins and delays.



Assembly View After XYRS Read

## 4.3 XYRS Adjustments

Close the XYRS_Log.txt editor and check the Group Move (GRP Move) check box in the Assembly View screen. If required, adjust the size and positions or the Main Screen and Assembly View Screen such that both can be seen at the same time.

Moving the mouse cursor over a component in the Assembly View will display information about that component. A LEFT CLICK on a Assembly View component will show that component in the main screen. Clicking on items in the Main Screen will select that component in the Assembly View. The ESC key will cause the Assembly View to again show all component.

PAGE UP and PAGE DOWN will zoom in and out of the Assembly View. When zoomed in a mouse RIGHT CLICK will pan the display. When the mouse is over a component a CTRL RIGHT CLICK will rotate that component AND ALL OTHERS of the same type if GRP MOVE is checked. CTRL LEFT CLICK will pick up a parts and attached it to the mouse cursor. CTRL RIGHT CLICK will rotate the attached part. A LEFT CLICK will drop the cursor component at the current location. If GRP Move is checked all other components of the same PCB_Decal and Package will be relocated by he same amount.

Keyboard arrow keys can also be used to move components. Zoom in and LEFT CLICK on Q1, information about Q1 is shown in the upper right. Press and hold the ALT key. While the ALT is down the arrow keys will move Q1 in 1mil increments. If the ALT and SHIFT keys are held down each arrow key press will move the component by 5 mils.

Do not press and hold the arrow keys down, the automatic key rate might be faster than the board is redrawn.

## 4.4 Adjusting Component Reference Points

As previous mentioned the Component 0,0 Reference Point is very important. The ability to automatically adjust component positions using Group Move and AOT logic depends on BOM Builder knowing the reference point for component values read from CAD tools data.

While any "Good" PCB design will have this point clearly marked some PCB Designers forget to include this datum. In these cases where it is not possible to set the reference point using something in a Gerber file it is possible to set these points using the Assembly View editor.

If there is a single component where the centroid of the component is known that component can be "Placed" and the Component Reference Point set. Typical components that work with this scheme are large QFP packages. Any package will work if the PCB Design has constructed the PCB decal with the 0,0 at the decal center.

Using CTRL ALT LEFT CLICK a part can be picked up and moved just as if a CTRL LEFT CLICK had been used. The difference is when the component is dropped. Dropping the part in this case does not move the part, instead software computes the X,Y differences and applies this to the Component Reference Point. The result is the "Moved" component moves to the correct location and all other parts move at the same time. It is not parts being moved only the reference point. A dialog box will ask if these offsets should be used for both side, the answer is typically yes.

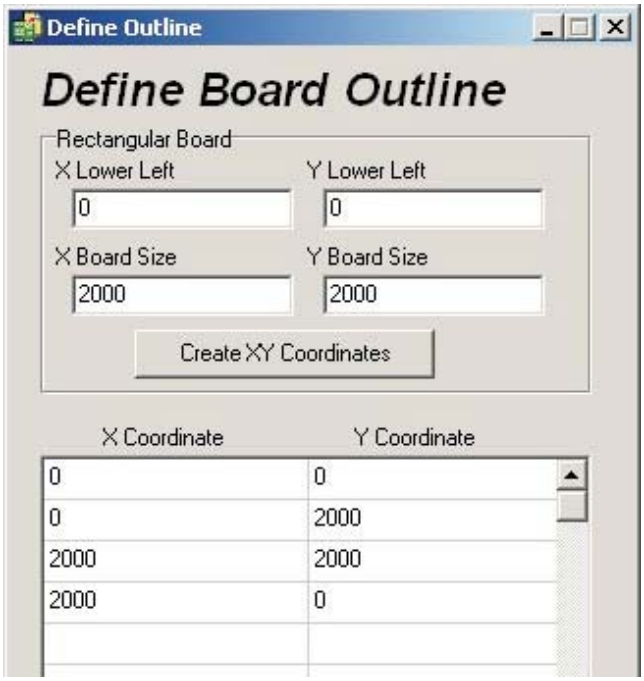
There is one more scheme for adjusting the Component Reference Points which is done using arrow keys. A ^R (CTRL R) will start a Reference Point Mode of operation where ALT Arrow keys will adjust the Component Reference Point for the currently viewed side. Adjustments are in 1 mil increments and holding down the SHIFT key will move in 5 mil increments.

## 4.5 Board Outline

BOM Builder allows the definition of a Board Outline. Currently this outline serves only as a cosmetic function, the centering and color control for the Assembly View screen. This outline can be defined in one of two manners:

- *Using the BOARD - DEFINE OUTLINE Main Menu function*
- *Using the ^B function in Assembly View*

With the SAMPLE3 file loaded use the BOARD - DEFINE OUTLINE sequence to open the Define Outline screen. The top portion of the screen is a short cut for generating outlines for rectangular and square boards. The bottom section is a sequence of X,Y values that make up the points of the board outline.




Outline Dimensions for SAMPLE Board

When the "Create Coordinates" button in the top section is clicked the bottom section will be populated with a set of X,Y values. The "Create Board Outline" button at the screen bottom will actually

generate the outline from the sequence of X,Y locations.

The SAMPLE board is 2.0 inch x 2.0 inch in size and the numbers can be entered in mils or mm. Any number with a decimal point is consider mm. Any number without a decimal point is considered mils.

All X,Y values are referenced to the Component Reference Point. The user must know the lower left location and the board size. If the Component Reference Point is not correct the outline will not be drawn in in the correct position.

With the SAMPLE3 file loaded open the Assembly View window and notice the tick marks in each board corner. Also notice the 0,0 reference point in the lower left. With the Assembly View window selected press a ^B to enter Board Outline Mode.

Using the mouse LEFT CLICK the lower left corner then LEFT CLICK the upper left corner. A pink line will be drawn showing the first segment of the new board outline. Mouse clicks will pick the nearest point in either of the 3 Gerber layers. The closest point is used for the X,Y location of a outline point..Continue by selecting the upper right, then lower right corners. Press another ^B to end the Board Outline Mode and the completed image is displayed.

Save the final image to SAMPLE3.BMB.

## 4.6 Manual Placement

If a board is relatively small and XYRS data is not available then parts must be hand located. The Assembly View contain logic and a "Placement Sequence List" to assist with this tedious chore. The placement list allows a user to zoom in on a section of the board and then enter a list of components that can be viewed at that zoom factor.

With the Assembly View window selected enter a ^P (CTRL P) to open the "Placement Sequence" window. Enter a number of reference designators with a space between each reference designator. Press the ENTER key when done. This list of reference designators will be displayed in the lower left.

Press the ESC key and the component for the first reference designator will be attached to the cursor. LEFT CLICK will drop this component while CTRL RIGHT CLICK will rotate the component. When the component is dropped it is removed from the Placement Sequence List. Pressing the ESC key will take the next component from the list and that part can be placed.

This logic allows a user to ZOOM in to a good working level and then place components without a lot of ZOOM and PAN operations.

## 4.7 Verifying Placements

Verification of component placement and orientation is critical for accurate assembly. BOM Builder contains links between the Main Tree Structured BOM and the Assembly View screen for "Cross Probing" type operation. When both of these screens are positioned on the computer desktop such that both are visible component checking is simplified.

A click on a Main Tree line item will highlight the selected components in the Assembly View. Since components within a single line item can be located on different sides of a board there are Red and Blue indicators in the Assembly View screen to indicate the counts on each side.

If the user is zoomed in and a single component is selected, that component will be centered in the Assembly View screen. If more than one component is selected the midpoint is centered.

When the Main Screen is selected, the arrow keys will move down tree line items. This movement is done in a manner that speeds component checking. At Index Designs this scheme is used to verify that each part is reasonably placed and the orientation of polarized parts is correct.

If the user clicks on a component in the Assembly View screen, information about that component is placed in the upper left and that component is identified in the Main Screen Tree.

## 4.8 XYRS Summary

The key to successful automatic assembly is good data. XYRS data and Component Reference Point data are typically a "Throw It Over the Wall" issue for most PCB Designers. But operating in this mode requires manufacturing people to sometimes guess about what is required and this leads to errors. As components become smaller there is less room for

reference designators and incomplete data results in project delays.

BOM Builder contains the tools to automatically "Get it Right the First Time" but PCB designers must be aware of the required data.

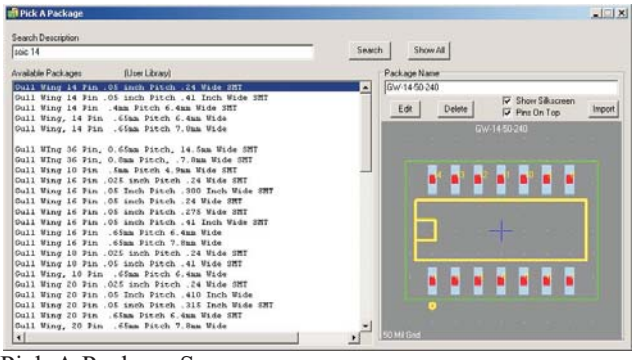
# 5 Package Select & Editor

Index Designs supplies a library of physical package models which are called Packages. All data is contained in a single text file named PACKAGEDATA.TXT. BOM Builder uses this data to render package graphics over top a user Gerber image. The resulting composite image is used to drive Index Designs manufacturing equipment.

Two screens are used to select, edit or generate new package models. The "Pick A Package" screen allows a single package to be selected from the library. A search engine is included to assist with locating a specific package. The "Package Editor" screen allows existing packages to be changed or new packages created.

## 5.1 Pick A Package Screen

All components that are to be mechanically placed must have a assigned Package. Pick and Place machines can require hundreds of parameters to pick up, transport, rotate and place a component. Index Designs uses "Package Names" to identify specific physical package styles. Each BOM line item that is to be machine placed must have a "Package Name" in the line item "Package Field". The "Pick A Package" screen is shown below and it allows users to select a package from the existing library.




Pick A Package Screen

The top field is the "Search Description" and it is this text that drives the search engine. In the above example "soic 14" was used to locate all existing "Gull Wing" type packages which had 14 pins. Search results are shown in the "Available Packages" and the top item is a .24 inch wide package . (Standard SOIC 14 package.) Clicking on a line in the "Available Package" area show both the Package Name (GW-14-50-240) and the graphic. Grid dots in the graphic display give an indication of the actual size.

The Show All button provides a listing by Package Name which is sometimes easier for viewing and selecting a package. Two buttons, Edit and Delete, allow the user to Edit or Delete the currently selected package. The Package Editor is a very powerful tool and is described in the following section.

## 5.2 Package Editor

The “Package Editor” allows users to define body and lead shapes for new electronic components. A series of “Tab Controls” select various fields for package generation while a graphic view is provided for data display and graphic editing. Tab labels and their function are:

- *Package - General shape, pin counts and default lead shapes.*
- *Pins - Controls numbering, connection offsets and side control*
- *PCB Pads - Controls generation of PCB Decals*
- *Paste - Control generation of solder paste apertures*

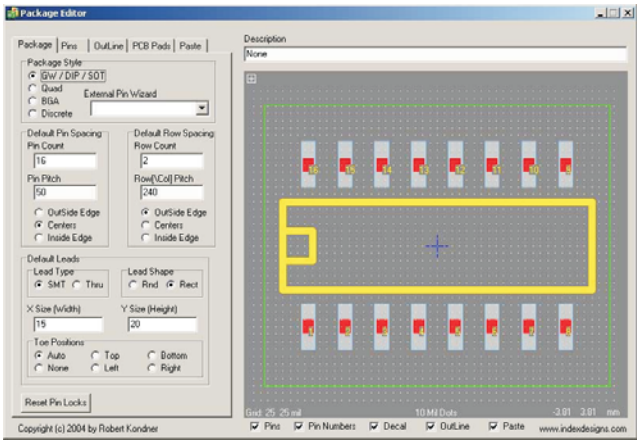
Various fields in the Package Editor accept dimensions for lengths or spacings. Units for these fields can be entered in either mil or mm formats. The logic is quite simple. If the parameter is entered with a decimal point it is considered as mm. If there is no decimal point then it is considered as mils. The user can force a particular unit by including mm, mil or a “. Entering mil forces the number as mils, entering mm forces millimeter and a “ forces inch units. For example 5 and .127 and .005” all refer to 5/1000 of an inch. Entering 2.5mil or 5mm overrides the decimal point rule.

A simple expression (very simple) can also be entered. A length parameter can be followed by a * / + or - operator sign to indicate multiplication, division, addition or subtraction. For example 200/2 is decodes as 100 mils. The units for the length are defined by the characters BEFORE the operator sign. For example 10+1.5 is decoded as 11.5 mils while 10.+1.5 is decoded as 11.5mm.

## 5.3 Package Tab

General package shapes are controlled using the top set of radio buttons. BGA, Gull Wing (includes

SOIC, TSOP, SOT and others) and Quad packages are the basic styles while a drop down list is used to select external “Pin Wizards”.



Package Editor

A Pin Wizard is an external program which is executed whenever it’s name is selected from the drop down list box. This list box is populated with all the file names of all the .EXE files found in the c:\BOM_Bldr\PinWizards directory. These programs can be DOS or windows programs. Each program will typically accept user input and will generate a text file that defines pin placements, shapes and types. Documentation can be found in the PINWIZARDS directory in file PINDATA_Sample.txt. A sample program can be found in file SampleWizard.dpr.

Once a package shape is selected using the Package Style radio buttons the pin counts, pin pitch, row count and pin pitch fields can be used to further define a package. If a Pin Wizard was used to generate the package these controls are greyed out and disabled. If a GW/DIP/SOT package was selected and a pin count of 3 and row count of 2 are entered a classic SOT-23 style package will be displayed.

The Default Leads group is used to initialize leads as through hole (THT), surface mount (SMT) size and shape. Toe Position is used by IPC 7351 rule sets when forming PCB decals. All leads default to these values when the package data structure is initialized. Users can edit these on a pin by pin bases by double clicking on a graphic lead when the Package Tab is selected.

While the Package Tab is selected a double click on a graphic pin will allow editing of pin data items. Several of these items are found in the default groups but several are new. Circuit Type allows pins to be defined as Signal, Mounting or Shield pins. Signal pins are the most common as they are the pins used to pass electrical signals. Mounting and Shield pins were defined to support higher level functions associate with drill size selection and net list error checking. Side is typically used for card edge connectors where copper shapes need to be defined on specific sides.

Pin locks are flags are used to identify that a user has modified specific pin parameters and the automatic generation of pin data or positions must be altered for that pin. When a pin lock is set the color of that pin changes which indicates that pin locks are active for that pin.

A "Reset Pin Locks" button is provided to unlock all pins. Clicking this button will reset pin locations, shapes, outlines, decal shapes and solder paste shapes.

Pin Positions can be altered by clicking on a pin while the Package Tab is selected. Pins can be dragged by their top, bottom, left, right or center. The drag mode can be selected using a Right Mouse Click in the graphic area or by using Modeless Commands DT, DB, DL, DR and DC for Drag Top, Drag Bottom, Drag Left, Drag Right and Drag Center. Modeless commands are commands typed on the keyboard. After pressing the ESC key to remove focus from an other control, press the D key. A modeless command window will appear which shows various commands which start with D.

The Modeless P command can also be used to select a pin for moving. To use the P command first make sure no field is being edited by hitting the ESC key. Enter a P followed by a pin number, for example enter P1 to move pin 1. The pin will highlight and the cursor will change shape. Arrow keys must be used to move the pin by GRID units as the mouse is locked out. Grid units can be changed using the Modeless G command. For example G25 sets both X and Y grids to 25 mils. Entering G25 100 will set the X grid to 25 mils and the Y grid to 100 mils.

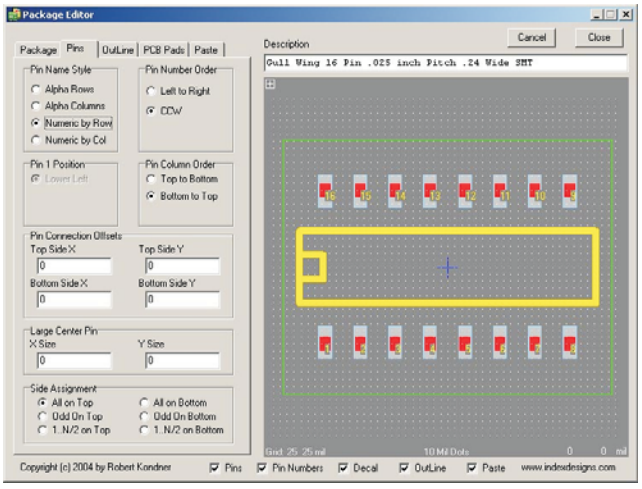
The origin for grid movement depends on how the pin was selected. When a pin is selected using a Left Mouse Click the pin is moved on a grid referenced to the package origin. The package origin is shown graphically as a BLUE cross. If a pin is selected using the Modeless P command the move-

ment is referenced to the starting pin position. This allows the user to move a pin by a specific distance using the grid settings even when the starting position of the pin is "Off Grid".

The above editing scheme is similar for both PCB Pads and Paste definitions. A click moves the graphic and a double click edits the nature of the graphic. If the PCB Pads Tab is selected PCB Pads are edited. If the Paste Tab is selected then Paste graphics are edited.

## 5.4 Pins Tab

Control for pin number assignments is through buttons and check boxes located on the Pins Tab. If a package was defined using an external Pin Wizard then all these controls are grey and disabled. If the overall package shape was defined using a radio button on the Package Tab then these controls will be active.



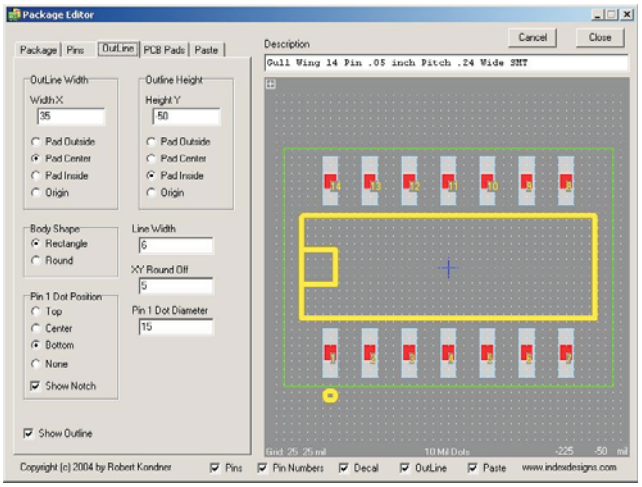
Package Editor Pins Tab

Most all BGA, connector and IC pin number formats can be defined using both numeric and alpha numeric pin numbers. DIN connectors with multiple rows, top and bottom mounted connectors and card edge connectors are automatically numbered. Pin connection offsets allow the connection points to be modified for parts like card edge connectors. Side assignments work with pin numbers which in turn are defined by other buttons. The number of combinations is considerable.

A large center pin can be constructed using two fields in the Large Center Pin Area. This allows thermal heat sink pads to be easily added for the construction of QFN and similar packages.

## 5.5 Outline Tab

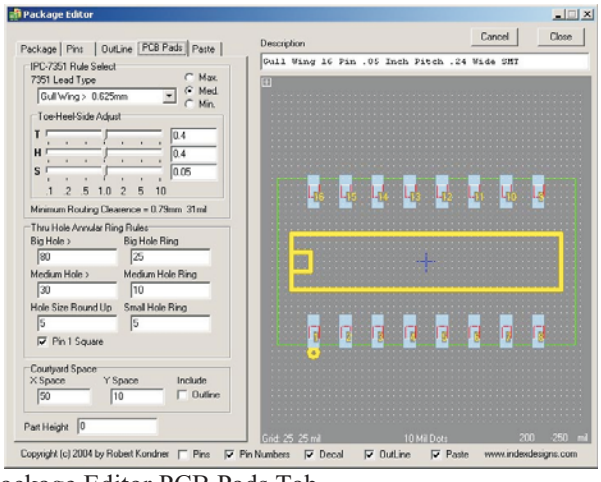
The Outline Tab allows the user to modify the size and shape of the package outline. Additional fields allow controls of line widths, placement grid and pin 1 indicators.



A left mouse click will allow a user to pick up and move line segments or corners. A right mouse click allows a user to insert additional corners or new outline sections. To add a new outline section right click, select Insert Outline, then left click where the shape is to begin. Click to add corners, double click to end.

## 5.6 PCB Pads Tab

A key function of the Package Editor is to apply various rule sets to lead data for the purpose of generating PCB land patterns (PCB Decals). Rules are defined using simple text files in the BOM_Bldr\Data directory. See files IPC7351Rules.txt and ThruRules.txt for details.



Package Editor PCB Pads Tab

A set of sliders and text boxes allow the user to adjust PCB land pattern size. These sliders provide logarithmically adjusted scale factors which are applied to the IPC-7351 rule setting. Component class rules are selected using the drop down box. Class rules are multiplied by the slider scale factor and the results or placed in the text box. A user can enter their own required TOE, HEEL or SIDE setting by entering values in the text box. Each pin includes a WHITE line which indicates the component TOE position.

Through hole rules are quite simple with values shown in the .txt file following the arrangement of fields in the Thru Hole Annular Ring Rules group. These 6 parameters force minimal annular rings for large, medium and small holes. The Round Up field forces computed annular rings to round up to specific intervals.

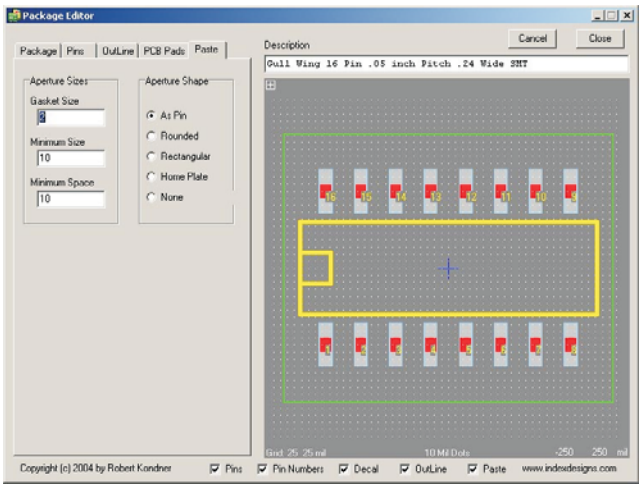
Annular rings are not computed directly from lead diameters but from required drill sizes. In the BOM_Bldr\Data directory is a file DrillRules.txt. This file is used to map through hole pins to a user defined set of drill sizes. The purpose of this logic is to control (reduce) the number of drill sizes called out in a complete job.

The component Courtyard is a definition of the space around a component while Part Height is an indication of component height. These fields are stored with the package data and are reserved for use by physical modeling software. The Courtyard can encompass space for just the pads or, if Include Outline is checked,the outline shape positions will be included in Courtyard calculations.

## 5.7 Paste Tab

When manufacturing surface mount assemblies the solder paste apertures are extremely important. For small and fine pitch components solder paste apertures can not be reliably extracted from PCB land patterns. Instead a completely independent data structure must be defined to control paste apertures.

BOM Builder includes logic for generating custom solder paste apertures. Fields are provided to define aperture shape and “Gasket” spacing. Gasket spacing is defined as an area of PCB Pad material that must gasket the area exposed by the paste aperture. The goal is to prevent paste leakage during the printing process which can result in solder shorts during reflow.




Home Base aperture shapes are often used to reduce solder ball formation on small chip components. Pick and Place forces tend to squeeze paste out from under small chip components with the result being the formation of solder balls during reflow. The “Home Base” design reduces squeeze out and reduces the chance of solder ball formation.

## 5.8 Color Control Check Boxes

Several check boxes are used to control the generation of graphic images. Pin numbers can be selected using the Pin Number check box. The remaining check boxes result in their associated items as being drawn as either solids shapes or as outlines.

## 5.9 Description Field

The description field is VERY important. The Index Designs search engine uses this description for storage and lookup. The user should uses existing descriptions as the guideline when generating new descriptions for new packages.

## 5.10 Toe Positions

As mentioned toe positions are used to control PCB Decal generation using IPC 7351 rules. Package Editor software displays Toe Position by drawing a white line on the Toe Side of each pins.

## 5.11 Pin Names

Each pin on a Package has an internal number and user number. Unless the user manually modifies the user pin number it will be the same as the internal number. While the automatic pin numbering logic in the Package Editor works for most new packages

there are times when it is required to modify default numbers. A right mouse click will allow the user to select "Re-Number Pin" mode which will start the renumbering process.

After selecting the "Re-Number Pin" selection from the right click menu the user can then left click on the first pin to be re-numbered, this pin will be numbered to 1 by default. As pins are left clicked they will receive sequential number. If the user wants to start number at a number greater than 1the keyboard can be used to change the number of the last key renumbered. Simply type on the keyboard to change the pin number of the last pin that was re-numbered with a left click.

Alpha numeric pin number such as A!, B27 for AA5 are allowed. Additional left click renumbering will advance the numeric part of the numeric pin numbers.

As the first pin is renumbered the "Unlock Names" button on the PACKAGES tab will be enabled. This button being active indicates that automatic pin numbering is disabled. Clicking this button will re-enable automatic pin numbering. Clicking this button while active will delete any user pin numbers.

Hit the ESC key when renumbering is finished.

# 6 Export & Reports

BOM Builder includes a number of export function in the FILE - EXPORT main menu. These export functions provide data in various forms to external applications. Included are outputs for OrCAD Capture UPDATE files, Excel .CSV files, PADS .ASC files, Parts & Vendors structured parts lists and embedded user files

## 6.1 OrCAD Update Export

OrCAD schematic capture tools (Capture) use a simple but powerful file format to “Back Annotate” just about anything back into a schematic. While BOM Builder maintains very detailed structures for part attributes many users still wish to duplicate the “Foot Print” information by back annotating their Capture schematics. BOM Builder will generate a .UPD file which Capture requires to back annotate footprint information into the source schematic.

Power Logic users are just plain out of luck. It is not possible to back annotate PART TYPE information into a Power Logic schematic.

## 6.2 Excel CSV Export

BOM Builder will format part information into a COMMA SEPARATED VARIABLES file format for reading by Microsoft Excel and many other spread sheet programs. The user is prompted for two inputs:

- *Destination File Name*
- *Count of Boards to Build*

After the file name is selected the Board Information screen is shown. The user should enter a number for "Number of Boards to Build" as this will be used as a multiplier for the output "Count" field. Each line item is output as a single row with the following output fields:

- *Find Number*
- *Part Count multiplied by Boards to Build Count*
- *Reference Designators*
- *Part Number*
- *PCB Decal*
- *Index Designs Package Name*
- *Description.*

## 6.3 Parts & Vendors Export

Export to Parts & Vendors is probably the most important function within BOM Builder. This function takes the current BOM and generates a structured parts list within the Parts & Vendors database. Note that only those parts with valid Parts & Vendors Part Numbers can be transferred. If a job contains new parts the Parts & Vendors database (or any MRP database) must have these new parts created first.

Once a BOM has been exported into Parts & Vendors automatic purchasing and inventory control is possible. Kitting, generating RFQs, generating POs and a number of functions are available. Also available are ECO controls which help to maintain a consistent product configuration.

Please see the Import Section for information about reading MRP data from Parts & Vendors and how that data is distributed to multiple designers and how it is synchronized using the Index Designs indexing software.

## 6.4 User Files

BOM Builder allows a user to embed various file types within a .BMB file. Embedded files can be of any type and used for any purpose. Examples of such files include:

- *Device Programming Data*
- *Test Instructions*
- *Assembly Instructions*
- *Documentation*

User files are attached using the FILE - IMPORT - USER FILE and they are extracted using FILE - EXPORT - USER FILE. To remove a copy of the file the file must first be selected in the Main Screen tree structure. Using the FILE - EXPORT - USER FILE will extract the file from the BOM and place a copy in the C:\BOM_Bldr\AttachedFiles directory. If this file type has an associated application then that application will be invoked to open the file. To import a file the user is presented with an open file dialog. The selected file will be read and embedded into the BOM Builder database.

User files can also be deleted. Select the file in the Main Screen tree structure and use the EDIT - DE-

LETE USER FILE function. The file will be removed from the BOM Builder database.

## 6.5 Export Netlist to PCB

BOM Builder exports design data to PCB Design systems through the Netlist to PCBe export function. The resulting PADs .ASC file format is not only very common among PCB design tools but it merges netlist, attributes and component decal selections into a single file. The export process for this file starts with a read of an existing file PADs file and then writing a new version of the file to a second directory.

After selecting FILE - EXPORT - NETLIST TO PCB DESIGN the user is prompted for an input netlist file. This file is read and the user is prompted for an output net list file. The user will be directed toward a different directory with the file name for writing the same as the input file name. The input file is read and written to the output netlist file. During the transfer input file *PARTS* data is re-written with the PCB Decal information contained in the BOM Builder database. This allows all components to be assigned PCB decal data without annotation data being placed in the source schematic. This simplifies the use of schematic generated by:

- *Outside Consultants*
- *Different schematic capture programs*
- *Same capture tools but different libraries*
- *Sample schematic from evaluation kits*

Both input and output file names are saved in the BOM Options screen as they are used for the netlist transfer. file. Once valid file names exist in these BOM Options fields the user is no longer prompted for file names, the transfer takes place immediately. As the decal names are transferred they can be translated as described in the next section.

## 6.6 Decal Map Table

A Decal Map Table (EDIT - DECAL MAP) is included which allows PCB Decal names to be translated as they are written to the output PADS .ASC file. This logic allows a single MRP system with a single set of PCB Decal names to drive multiple PCB design tools where each tool uses different libraries. PCB Design service organizations can use a single “In House” set of PCB Decal mappings and still prepare input files for PCB Design using customer libraries.

This map table is driven by simple text files which can be read or written using any text editor. Two columns exist in the table, “Local Decal/RefDes” and “Target Decal”. As BOM parts are processed their Reference Designator and PCB Decal are compared to the first column and if a match is found the Target Decal in the second column is written to the PADS .ASC file. The result is a PCB decal name translation.

The syntax of the first column allows a single reference designator or a series of reference designators to be renamed. It is also possible to rename on a PCB decal bases or a combination of a reference designator and PCB Decal name. Examples of this syntax are:

- *R21 :Component R21 will be output using table Target Decal.*
- *R* :All components with "R" as their initial set of reference designator letters will be output using the table Target Decal. (RN2 would not be changed as it's initial set of reference designator letters is "RN" and not "R".*
- *1206 :All components currently assigned PCB Decal 1206 will be output using the table Target Decal.*
- *C*0603 :All components with "C" as their initial set of reference designator letters AND a current PCB Decal of 0603 will be output as the Target Decal.*
- *If no match is made the PCB Decal is output directly.*

No changes are made to the current BOM during this process. The changes to the decal names applies only to the decal names written in the *PART* section in the output netlist. Multiple sets of mapping tables can be maintained and selected in the EDIT - DECAL MAP screen.

## 6.7 Purchasing BOM Report

Main Menu sequence REPORT - PURCHASING BOM will generate a list of material required for board assembly. The Board Information screen will be shown where the user can specify the number of

boards to be built. The count of component in each line item will be multiplied by this board count.

A report preview screen will allow the user to review and print the report. In addition to printing an Adobe Acrobat file (.pdf) can be directly generated. If a .pdf is generated it will be opened where the Acrobat functions makes it easy to send email or print copies.

Only parts marked as IN BOM and STUFFED will be included. This allows product variations to be easily constructed within BOM Builder.

## 6.8 Assembly BOM

The Assembly BOM is very similar to a Purchasing BOM except that the parts are sorted by assembly process requirements. The sorting categories are:

- *Machine Parts - Top and Bottom Side*
- *Hand SMT - Top Side*
- *Hand SMT - Bottom Side*
- *Through Hole - Top Side*
- *Through Hole - Bottom Side*
- *Not Stuffed*

Color coding is used to define side information. Top side components are marked using alternating colors of red and white. Bottom side components are marked using alternating colors of blue and white. The first part in a side group is always in red or blue so there is no ambiguity about which side a parts is located. Machine parts are all marked with alternating colors of green and white. No distinction is made for top and bottom side machine parts. No stuffed parts are shown in alternating gray and white.

Dividing parts into these categories speeds production as it allows various component types to be routed to different work locations. Hand SMT parts must be mounted before reflow of their associated side. Through hole parts are mounted after SMT is complete. The not stuffed list is important during final inspection.

Including all none stuffed components in the BOM and marking them as "Not Stuffed" is very important. Final inspection must verify if any required parts are missing. The only way required parts can be verified is if "No Stuff" parts are identified.

## 6.9 Kitting Labels

BOM builder contain a "Kitting Labels" report which will generate a set of packing labels to be used when materials are collected prior to assembly. As with other reports the Board Information Screen is opened and the user should set the "Number of Board to Build" as required. The quantities for each line item will be multiplied by this number as the labels are generated.

Kitting labels are generated 10 per page using a standard 4 x 2 inch format. Either adhesive labels can be printed or standard 8.5 x 11 letter paper can be printed. Cutting the printed pages with scissors or a paper cutter produces a single label for each required line item. Description, part numbers and reference designators are included on each label which simplifies component identification.. A standard bar code is also printed on each label in a "2 of 5" format.

## 6.10 Board Image

A complete image of the PCB can be printed using the REPORT - BOARD IMAGE menu sequence. Gerber files will be printed first followed by Package graphics. How the image is scaled depends on whether or not a board outline has been defined.

If the board outline is defined then the board image will be scaled such that the board outline is fitted onto the printed page. Landscape vs Portrait printing will be automatically adjusted to fill the printed page. If the board outline is not defined then the board will be sized to include the entire Gerber file image along with all components.

The main use of the Board Image is component verification. Package libraries from Index Designs contain orientation marks for polarized components. Users MUST verify the rotations of these components. Thin white body lines and orientation marks must be compared with silkscreen or copper patterns which indicate orientation. If the user failed to include such information in the Gerber data they will probably get it right the next time.

The Gerber files for the BOM Builder silk screen images can be setup differently than the actual PCB silkscreen image. A significant amount of fine

dense text that would not print on a real PCB can be included in the BOM builder image. Further BOM Builder will show any silkscreen over top of the copper image. Unlike a real PCB the silk screen image is not broken by pads or vias.

## 6.11 Smart .PDF

The REPORT - SMART .PDF report is similar to the Board Image report except for two major functions:

- *Output is to an Adobe Acrobat .pdf file.*
- *BOM Information is placed into the .pdf file.*

The result is a .pdf file that can be opened in Acrobat Reader and the Acrobat HAND cursor can be used to inspect component information. Moving the HAND cursor over a component will result in a text flag being displayed with Part Number, value and Description data.

The resolution of this file is 1000 DPI and all data is saved in vector format. Even at significant zoom factors the PCB image is quite accurate yet the file size is reasonably small.

## 6.12 Set Default Printer

The REPORT - SET DEFAULT PRINTER function allows a user to set a default printer for various reports. When a printer is selected the name of this printer is saved to the .INI file. When BOM Builder is started it uses the printer name from the .INI file to select a printer.

Assembly BOM, Production BOM and Printing Label reports check if a default printer has been specified. If no printer has been specified, or if the printer name found in the .INI file is not found, these reports will prompt the user to select a printer.

# 7 Import Function

The usefulness of BOM Builder depends on its ability to import and arrange data from various sources. By reading data from MRP systems, PCB CAD tools and its own Package Editor, custom CAD libraries can be eliminated and up-to-date information is always available to design engineers. All data can be transferred as simple text files since BOM Builder contains all the required "Data Server" logic and all required data tables are stored within simple file structures.

Several of these import functions are based on dynamically loaded .DLL files which are easily constructed for various data sources. Each .DLL is only 50 or 100 lines of code and the .DLL used for a specific function is defined though fields in the System Options screen.

The BOM Bulder import functions will process data from:

- *MRP Database Systems*
- *Parts & Vendors & Other Databases*
- *PADs .ASC Files (See section 2.1)*
- *Input Parts List .dll (See section 2.4)*
- *X/Y Rotation Data from PCB Design Tools (See section 4.1)*
- *Excel .CSV File Import*
- *Text files from CAD tools.*

## 7.1 MRP Data

The feeding of MRP data into BOM Builder is very important as BOM Builder does not replace an external part database. At a minimum BOM Builder needs the user to have a component database with a Part Number, Description and a single field for PCB Decal/Package associations. Parts & Vendors is an excellent small database tool that includes many purchasing features and ECO functions all at a very low cost. Many companies have existing database systems and the MRP Data Import function is how data is moved from these external systems to BOM Builder.

The MRPData.txt file can be distributed to all design engineers who wish to have access to such data. Each time BOM Builder is started the time and date stamp is read from MRPData.txt and it is compared to the time and date of the last update. When a new file is found it is processes at startup which leaves all engineers using fresh consistent data. The MRPData.txt file can be transferred via a local network, written to all engineers using simple batch files, or even downloaded over the Internet using a web browser.

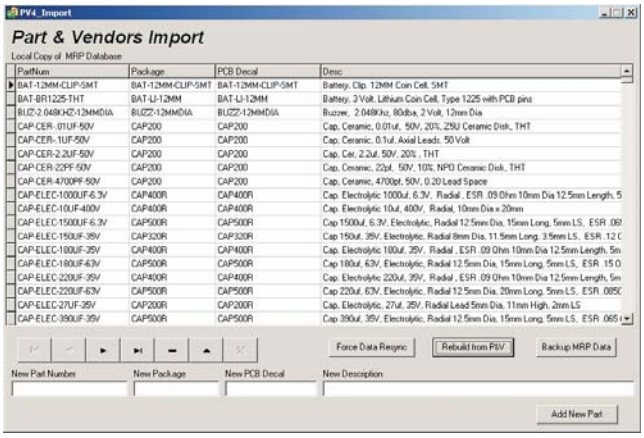
## 7.2 Parts & Vendors Import

The Parts & Vendors Import screen allows control over the local copy of the MRP Database within BOM Builder. There are three different methods of setting up this data.

- *Prefered: Reading from Parts & Vendors*
- *Using MRPData.txt file from a MRP system.*
- *Dumb: Building it by hand.*

The ideal process is to automate the collection of data that will be used in forming BOMs and the driving of the manufacturing process. A single point source of data is preferred, one where a minimum of Part Number, PCB Decal and Description are available. The saving of Package definitions in this database is advantageous but BOM Builder AOT logic can learn and provide package names if required.

Selecting (FILE - IMPORT - PARTS&VENDORS DIRECT) displays the PV4 Import screen which is shown below. In the top section is a database grid that shows the contents of the local BOM Builder MRP image. Several buttons allow synchronization and backup and several fields are provided for adding new parts or attributes.




Parts & Vendors Import Screen

A grid showing all the existing MRP components is displayed along with their Package and PCB Decal assignments. In the screen lower right corner is a “Rebuild from P&V” button. Clicking this button will delete the existing MRP database and will start a rebuild from the Parts & Vendors database through an ODBC connection. Do not click the rebuild button since chances are you have neither the P&V database or the proper ODBC connection. If you click "Rebuild" without the proper setup so all you will get is an empty database.

Once P&V is setup, and its first user field (User1) contains PCB decal and package data, then pushing the “Rebuild Button” will extract data from the P&V database. This extracted data is written to internal tables, external file MRPData.txt and is used to synchronize the Index Designs search engine. Be sure to verify the path for the P&V database file (.MDB file) which is shown in the System Options screen.

Data is extracted from the P&V Title and Detail fields to form a description. The User1 field is used to provide a PCB Decal and Package name.

The format of data in the P&V User1 field is quite simple. It is a text string which includes two names separated by one or more spaces. The first name in the string is the PCB Decal associated with the current part and the second name is the Package. For Example:

CR1206 1206

If the User1 filed for a part contains the above string the PCBDecal for that parts will be CR1206 and the Package as 1206. If the User1 field contain only a single name that single name will be used for both the PCB Decal and Package. If User1 is blank the part will not be included in the BOM Builder database or the MRPData.txt file.

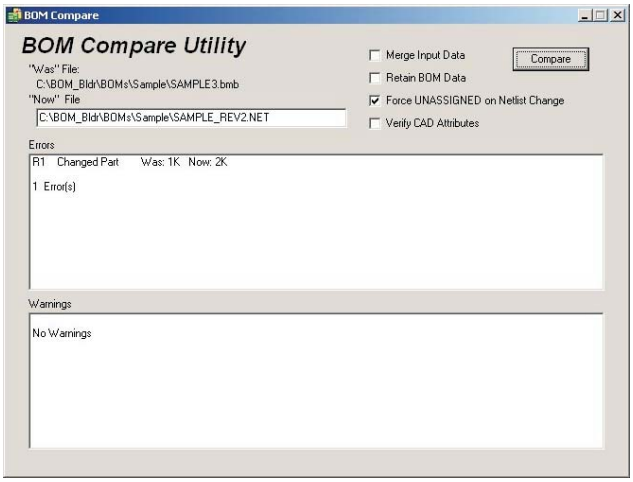
## 7.3 BOM Compare

A compare function has been included that detects and reports difference between two BOMs. This compare function will also except a Pads netlist as input though the number of possible detectable differences is reduced. Several check boxes provide processing options when comparing a netlist.

The bottom portion of the BOM Compare screen contains text fields were errors and warnings are reported. Clicking on strings in these text areas will open copies of these areas in a text editor. Several check boxes provide additional control.

- *Merge Input Data will result in differences being applied to the BOM.*
- *Retain BOM Data will maintain BOM Data for fields not read during netlist compares.*
- *Force Unassigned will force Package and PCB Decals to UNASSIGNED during netlist compares.*
- *Verify CAD Attributes will test input PADs attributes during netlist compares.*

The reason for the above flags is a completed BOM with full details might be compared to a PADs netlist which have missing, incomplete or simply bad data. These flags control how netlist data is treated and how the BOM is optionally updated.



BOM Compare Screen

An example has been provided in the SAMPLE directory which provides a net list to compare with the SAMPLE3.BMB file. A new netlist has been created which has a different value (2K) for R1. With SAMPLE3.BMB loaded use the Main Menu FILE - COMPARE sequence to select the file SAMPLE_REV2.NET . When the File Compare Utility screen is shown, click on the COMPARE button.

Parts will be compared and R1 will be shown as changed. Click on the “Merge Input Data” check box so that it becomes checked then click on the “Compare” button again. This time the error will be gone as the difference have been merged into the BOM. Close the Compare Utility window and observe that R1 is now RED. When the changed part

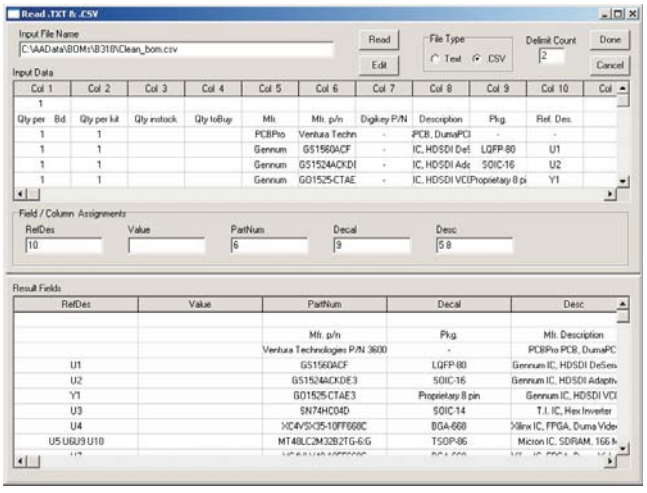
was merged to the BOM database prior package and PCB selections are removed and selections must be made.

## 7.4 Excel .CSV and .TXT Import

In many cases data about input parts data is provided as a spread sheet. This is unfortunate as hand generated spread sheets are typically incomplete, inconsistent, out of date or just wrong. Despite these problems many engineers continue to use spread sheets so BOM Builder includes several .dll functions for reading spread sheet data.

The TXT_CSV read function is also used read many types of text reports that are generated directly by many CAD tools. When reading these .TXT files the key issue is what determines field boundaries. When reading .TXT files space characters are used to delimit fields. The number of spaces required to delimit a field is controlled by a "Delimit" count. Tab characters will also serve as a delimit of input fields.

In all cases the spread sheet or text file must be inspected for invalid or inconsistent fields. Spread sheet files must be exported as a .CSV file. A generic .CSV / .TXT import engine has been provided that will read the .CSV or .TXT file and allow input fields to be converted into a set of output fields. A sample screen is shown below.




CSV and TXT Import Screen

The top portion of the screen contains the input data and the bottom portion the results. In the center is a set of "Input to Results" mapping entries, one for each result field. These center entries select input fields as data sources for result fields. Data from the .CSV or .TXT file is read and the input data columns are formed. Result fields are then formed from the input columns as determined by the input field number in the center "Fields / Columns" section. Any input column may be used any number of time or not at all. The number and name of the results fields varies depending of the code in the .dll routine. Result fields are provided for the required and options IPL or XYRS data field requirements.

Each Field / Column entry defines which input columns are used as data for the results section. The name of the Field / Column sections are the same as the results columns. The numbers in those center fields select the .CSV or .TXT input columns by number.

Several XYRS_TXT_CVS_xxxxx.dll files are provided. These routines process the X and Y values as either Inch, mm, mils (1/1000 Inch) or as microns (10**-6 meter). Be sure to setup the CW and CCW flages in BOM Options before using these routines to read XYRS data. Proper application of this data requires the "Sense" of rotation input.

At the screen top are Read and Edit buttons. The Edit button will invoke the appropriate editor for the selected file type. The Read button will read the input file and regenerate the results fields. If the Field / Column number in the screen center are changed the Read button must be clicked to regenerate the results columns.

Finally, when the result data "Looks Good", the Done button will return the result array to BOM Builder.

# 8 Misc Function

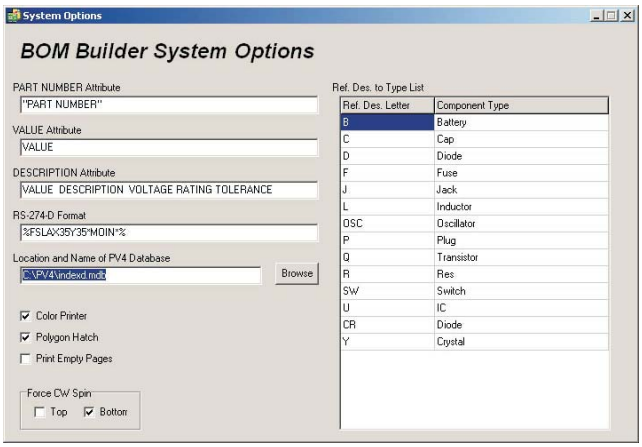
This chapter describes a number of remaining BOM Builder functions. These functions provide control for:

- *Schematic Back Annotation*
- *Gerber Options*
- *Part and Gerber Deletion*
- *Package and AOT Data Control*
- *Board Information*

These subjects and others are covered in the following sections.

## 8.1 System Options

The System Options Screen, shown below, contains a number of fields and check boxes that control various BOM Builder features.




System Options Screen

The first three fields contain default attribute selection for BOM Builder fields when importing PADs Netlist and .ASC files. When a new BOM is created (FILE - NEW) these fields are copied into the identical fields in the BOM Options which become part of the saved BOM. When saving and reading BOMs it is the BOM option fields that exchange data. System Options are only used for new BOM generation.

Notice that PART NUMBER is inclosed in double quotes. If a attribute name contains a space character then enclosing that field in double quotes preserves that space in the name. Not using these double quotes in the above example would result in attribute PART and attribute NUMBER being extracted and merged.

The next field specifies the Gerber input format when reading RS-274-D type Gerber files. D-Code aperture data for RS-274-D files must be read in using a .dll routine. If no apertures are read, or any are missing, BOM Builder will render these items using 10 mil round apertures. See file Gerb_Apertures.txt in the c:\BOM_Bldr\IntfTools directory. A simple solution is to convert -D files to -X using Gerber reader programs. RS-274-X is suggested. In all but a few cases -D reading is no possible.

Three check boxes allow for control of printing options. While color printing can be disabled the use of a color printer is highly recommended. BOM Builder does not render "Clear Layers" in RS-274-X polygons which can result in some "GND Planes" hiding tracks and pads. Using a cross hatch display for "Dark Layer" polygons solves this problem. The printing of pages for board sides which do not contain parts can be enabled with the third check box.

To properly adjust component rotations using AOT data BOM Builder needs to know the rotation sense of the PCB Design tool used to generate the gerbers. This would be the same program that generates the XYRS information and BOM Builder must rotate parts in the correct direction.

It is very possible that a user might deal with multiple CAD systems each with different spin direction. While there are direction controls in the System Options Screen there are also a set in the BOM Options Screen. When a new BOM is generated the System Option check boxes are copied to the BOM Options check boxes. When BOMs are read or written check box status is saved and restored with the BOM Option check boxes. This allows the spin direction control data to follow the BOM allowing Group Moves to operate correctly at all times.

The right side of the System Options screen contains a table used to form associations between Reference Designator letters and electrical functions. When BOM Builder forms line items the intimal letters of each component's reference designator are used to associate that component with a function. Components are grouped into identical "Functional Areas" and the Main Scree Tree is sorted by functions in an alphabetic manner. This is how BOM builder knows that R's are resistors and C's are ca-

pacitors. Unknown reference designator letters are mapped into he "Other" function.

Users should populate the right side table with their own definitions. Having parts sorted by function is a great way to locate parts and assists with manufacturing and test. This table is saved to and restored from disk as BOM builder is closed and reopened.

## 8.2 Delete Functions

The Main Menu EDIT menu contains delete functions for Parts, All Parts and All Gerbers. Delete Part will delete the parts defined in the Line Item Editor located in the lower portion of the Main Screen.

Delete All Parts will remove all parts but will maintain Gerber Images and BOM parameters. This different from the FILE - NEW function which deletes parts Gerber images and resets all BOM parameters. Delete All Gerbers simply deletes all Gerber images. These functions are very useful as new revs of a design are made since a significant amount of data can be reused.

## 8.3 Renumber

It is a common practice for a PCB designer to renumber all reference designators on a PCB. This process results in a design where it is easier to located components during assembly or test. When renumbering a PCB a WAS IS list is generate which is used to identify old and new reference designators. BOM Builder will read this list and update all reference designators accordingly. A sample of this list would be:

C1 C27
Q7 Q1

In the above example C1 is renamed to C7 and Q7 is named to Q1. When reading the WAS IS list every BOM component is checked for receiving a new reference designator. Components that do not receive new reference designators are checked for deletion. If another component has been renamed to the old WAS reference designator value then the component that did not receive a new name is deleted. If two or more components are mapped to the same new reference designator then only the first will be renamed. In all cases errors are generated.

Error conditions are checked during the process and if detected an error report is generate. The user will be given the count of errors and the errors report will be opened in the default text editor.

When EDIT - RENUMBER is selected using the Main Menu the user will be prompted for the WAS IS text file. Any file contents that can not be used to update a BOM component are reported in the error report. Extra or missing references are also reported. If errors are encountered the user should resolve the error BEFORE the modified file is saved. It is not possible to recover an old version once a renumbered BOM is saved. Do not SAVE the BOM until processing completes with no errors. Always spot check the conversions.

## 8.4 Package Utilities

Several functions are provided in the Main Menu PACKAGE function for the control of Package Libraries.

- *Update Cache: Update all BOM graphics from the Package Library.*
- *Edit Packages: Open Package Select Screen where packages can be seleted and edited.*
- *Backup: Save all Packages to PackageBakup.txt file.*
- *Restore: Delete Package Library and restore from the PackageBackup.txt file.*

When packages are backed any existing backup file is renamed down through an _Old, _Older and _Oldest sequence. Restore should only be used if the BOM Builder database has become corrupted. When the library is restored any unsaved packages will be lost.

The format for the PackageBackup.txt file is identical to the PackageData.txt file found in the c:\BOM_Bldr directory. Each time BOM Builder is started it checks the date of PackageData.txt. When a new date is detected PackageData is read and the local package database is updated. All packages contain time stamps which prevents old data from over writing new data.

PackageData.txt is the vehicle by which Index Designs distributes current package information. Placing a new copy of PackageData.txt in the c:\ BOM_Bldr directory is all that is required to update

BOM Builder with the latest package information. Index Designs libraries are specially marked and they will ALWAYS be restored from PackageData.txt. Users who wish to generate their own packages should use a $ character as the first character of the package name. This will place user packages at the beginning of the "Show All" list and keep them separate from Index Designs packages.

## 8.5 AOT Files

AOT files are used to store data about how a user's PCB Decals are converted to Index Designs Package names. Included is information about how the XYRS data associated with those decals is positioned relative to the Index Designs package.

The construction of AOT files is semi-automatic as new user PCB Decals are encountered during the reading XYRS or IPL data. As a user assigns a Package Name to a line items that Package Names becomes associated with the PCB Decal for that line item. During checking an operator might position and/or rotate the package graphics in preparation for machine placement. All these Angle, Offset and Translation parameters are saved in a local database and this database is eventually saved to an .AOT file. Whenever a new part with an known PCB Decal is accepted these AOT translations can be applied via a manual or automatic means.

The "Save Package Map to AOT" and "Apply AOT File to BOM" functions in the Main Menu PACKAGE menu are the manual means for controlling AOT data. In each case the user is prompted for an AOT file to used for transfer.

Automatic AOT logic is controlled by two .ini file variables and the .AOT filename specified in the BOM Options Alignment Table field. The two .ini variables are AOTEnabled and AutoAOT. Both are boolean variables (0 or 1) and they function as follows.

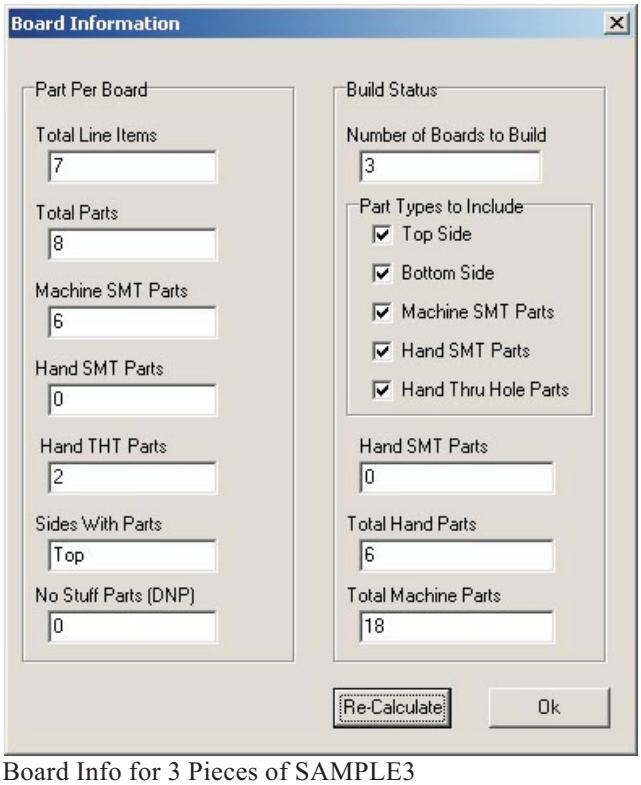
If AOTEnabled is 0 then all automatic AOT logic is disabled. This is very useful for a service organization where a number of different .AOT file must be maintained. When disabled operators will never receive messages about missing AOT files or prompts about saving AOT data. This setting simplifies BOM Builder operation put requires manual processing of AOT data.

If AutoAOT = 1 then a single AOT filename will be used and most AOT file prompts are eliminated. When a BOM is closed and new AOT data exists, the user will be prompted about saving AOT data. This is the default mode for BOM Builder. These settings are appropriate for individual engineers who always work with a single PCB design tool and set of PCB decal libraries.

When AOTEnabled = 1 and AutoAOT = 0 then full scale AOT logic is active. AOT Files are read as BOMs are opened and the user is prompted if files are missing or as AOT data must be saved. This is an Expert mode designed for service organizations who must deal with many PCB designers and libraries and who generate all the AOT files in house.

## 8.6 Board Information

The Main Menu BOARD - BOARD INFO function gives the user a quick overview of line item counts, part counts and the relative difficulty of board assembly. A multiplier is included that allows total part counts to be evaluated.The left side is a display of various counts while the right side allows selection of quantities of boards and part types to mount.




Board Info for 3 Pieces of SAMPLE3

In the above example 3 pieces of SAMPLE3 are to be built including all SMT and Through Hole part

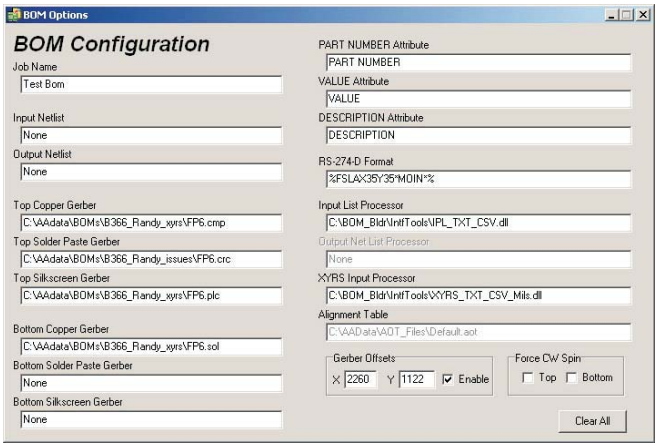
types. The total number of machine placed parts is 18 and the total number of hand placed parts is 6.

Several reports will cause the Board Information Screen to be displayed prior to running the report. This gives the user a chance to set the "Number of Boards to Build" field which is then used as a multiplier for part totals.

## 8.7 BOM Options

A BOM Options window is used to configure various parameters associated with a single BOM. This window is accessed from the main menu using EDIT - BOM OPTIONS.

Use a DBL Click on any of the file name fields to open a file selection dialog.



BOM Options Screen

The top line allows the user to enter a board name that will be included on all reports. This name is saved in the .BMB file with all other BOM parameters and is restored when a .BMB is read.

Input and output netlist file names are used to simplify the appending of PCB Decal names to PADs netlists. The BOM Builder EXPORT - NETLIST to PCB DESIGN function uses these fields as the source and destination file names. As the input file is read the input PCB Decal field for each part is replaced with the data from the BOM Builder PCB Decal field. The result is a output file with PCB Decals selected from a company MRP system. Using BOM Builder in this manner eliminates the need to duplicate the MRP database parts in CAD libraries. Double click to select a different input and file names.

Gerber file name fields are automatically filled in as gerber file are selected in the Read Gerber screen. The names are displayed here for convenience. Double click to select a different file name.

Various ATTRIBUTES fields allow a user to define the attribute names used to populate BOM Builder fields during PADs netlist and .ASC file read. Three BOM Builder fields, Value, Part Number and Description, can each contain a number of attribute names. If any single attribute name contains a space then that attribute name should be inclosed by " characters.

The RS-274-D field is used when reading RS-274-D gerber files. Note that aperture files are read using .dll libraries. See file Gerb_Apertures.txt in the c:\BOM_Bldr\IntfTools directory for more information.

Three .dll files provide control of several input and output processes. These files are used during Import and Export as their name suggests. See the Export and Import chapters for more detail. A DBL Click can be used to select different .dll processors.

AOT files are used to control part X/Y placement and rotation as required by a specific set of user CAD libraries. Assuming the Component Reference Points have been correctly set an AOT file adjusts the component placements to where they become synchronized with Index Designs libraries. AOT files are automatically generated as a user corrects part placements when GROUP MOVE is enabled. See the XYRS chapter.

As gerbers are read and aligned the last set of alignment coordinates are retained in the Gerber Offsets box. These offsets are used during subsequent gerber reads which simplifies gerber alignment. A check box is provided to disable the automatic use of these offsets. This is required when gerbers are "Centered" or otherwise ill-prepared.

A set of check boxes are provided to change the "Spin Direction" used in BOM Builder. These allow BOM Builder to operate properly with CAD tools which rotate in a CW direction. Individual controls are provided for both top and bottom sides.

# 9 CAD File Formats

BOM Builder supports several different CAD Tools:

## 9.1 Eagle

Index Designs provides a script that prepares a single file used for both IPL and XYRS input. See file:c:\BOM_Bldr\IntfTools\ Eagle_XYRS-IPL_Script.ulp

A Sample output of this script is shown below.

```
Layer = Top
Report Origin = (0.0, 0.0)
Units used = mil
RefDes, LocationX, LocationY, Rotation, Value, Package
C1, 26.92, 13.59,   0, 6.8pf, C0402
C2, 11.83, 13.34, 270, 10pf, C0402
C3, 11.98,  9.84, 270, 10pf, C0402
C4, 20.70,  3.34, 270, 0.01uf, C0402
```

## 9.2 OrCAD Layout

The "Component Insertion Report" from OrCAD layout drives both IPL and XYRS input functions. This is a standard report. A Sample is shown below

```
************************************************
*                                              *
* INSERTION LIST REPORT                         *
*                                              *
* E:\ORCAD\DESIGNS\myboard.MAX                  *
* Wed Jun 22 13:03:02 2005                      *
*                                              *
************************************************

REF DES  VALUE    FOOTPRINT    XCOORD     YCOORD     ROTATION BOARDSIDE
-------------------------------------------------------------------------
C8     0.1uF        SM/C_0805     1800.00 m    -858.00 m   270      BOTTOM
C9     22nF         SM/C_1206      865.00 m     -2400.00 m  0        TOP
C12    0.1uF        SM/C_0805     1950.00 m    -168.00 m   270      BOTTOM
C13    1nF          SM/C_0805     1810.30 m    1600.30 m   225      TOP
C15    1.5nF        SM/C_0805      909.68 m     -2109.70 m  0        TOP
C16    4.7nF        SM/C_1206     1364.03 m    1894.03 m   225      TOP
C17    0.1uF        SM/C_0805     -998.00 m    1960.00 m   0        BOTTOM
C18    22nF         SM/C_1206      600.00 m     -1855.00 m  90       TOP
C5     10nF         SM/C_1206      215.00 m     2620.00 m   0        TOP
```

## 9.3 PADs XY Location Report

A Pads XY Location report can be generated from within PADS using a supplied Basic script name "XY Locations". Click on "Tools - Basic Scripting - Basic Scripts" and you will see a list of scripts. If the"XY Locations" script is not seen click the "Load File" button and load from directory:

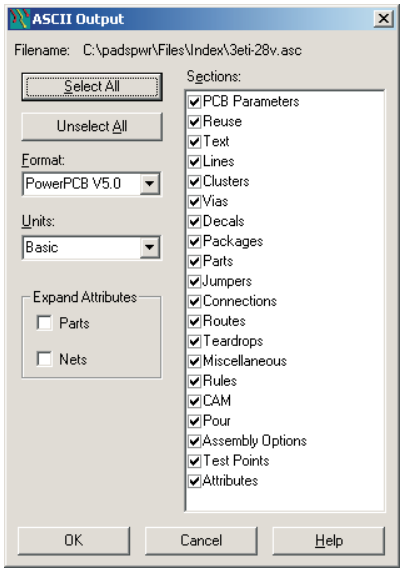
c:\padspwr\ole\unsupported_user_samples\XY Positions.bas

Running this script generates a "Positionsxxxx.txt" file where xxxs is some random set of digits.

Sample PADs XY Location Report:

```
RefDes      X           Y           Rot.       Exact      (FREE)     Side
C1     775.0000    1475.0000   270         EXACT      (FREE)     Top
C2     975.0000    1475.0000   270         EXACT      (FREE)     Top
JP1    325.0000    625.0000    180         EXACT      (FREE)     Top
P1     1625.0000   1000.0000   090         EXACT      (FREE)     Top
Q1     950.0000    650.0000    000         EXACT      (FREE)     Top
R1     600.0000    650.0000    000         EXACT      (FREE)     Top
R2     1275.0000   950.0000    000         EXACT      (FREE)     Top
R3     950.0000    1000.0000   090         EXACT      (FREE)     Top
```

## 9.4 PADs .ASC File Generation




Pads .ASC files contain XYRS, Input Part List and PCB Decal info. Also, depending on the design environment it might also contain company part numbers. The user should always generate a .ASC file when possible.

To Generate a .ASC File in PADs complete the PADs ASCII Output screen as shown to the left.

Files - Export - <filename> Save

Then select the output contents and format:

SELECT ALL Format Power PCB V5.0 Units BASIC

The .ASC file is generated and written to disk.

## 9.5 Protel

There are a large number of Protel products and each seems to have differences. The two common formats seem to be the XYRS format and the Netlist format.

XYRS Format:

Sample Protel XYRS Report:

```
Designator Pattern  Mid X  Mid Y  Ref X  Ref Y  Pad X  Pad Y TB  Rotation

U5  SM/SOT-23A 4766mil 1959mil 4808mil 1922mil  4808mil 1922mil  T  0.000
R12 1206       3418mil 1799mil 3418mil 1744mil  3418mil 1744mil  T  0.000
R4  1206       6310mil 2462mil 6310mil 2518mil  6310mil 2518mil  T  0.000
R3  1206       6490mil 2046mil 6546mil 2046mil  6546mil 2046mil  T  0.000
R2  1206       6151mil 2748mil 6096mil 2748mil  6096mil 2748mil  T  0.000
C4  CASE-F     7792mil 2590mil 7662mil 2590mil  7662mil 2590mil  T  0.000
```

Sample Protel Netlist Format for Input Parts List

```
[
C1
PCB_Decal
Value



]
```

## 9.6 PCAD

P-CAD Format

(They seem to have the same data in two formats. The second is a CSV format.)

```
================================================================================

Report Origin = (0.0, 0.0)
Units used = mil
RefDes          Layer           LocationX  LocationY  Rotation Fixed
--------------- --------------- ---------- ---------- -------- -----
C1              Bottom             5787.0     5473.0    180.0 No
C2              Bottom             5857.5     5194.5      0.0 No
C3              Top                5934.0     4595.0    180.0 No
C4              Top                5934.0     4938.0    180.0 No
C5              Bottom             6934.0     4821.0    270.0 No
C6              Bottom             7188.5     4726.0     90.0 No
C7              Bottom             5877.5     5567.5      0.0 No
```

The second format is shown below. The VALUE field has been generated such that it will contain the schematic VALUE attribute or Part Number. Many designers will use the VALUE attribute for passives and the Part Number attribute for

IC type devices. IC type devices often have blanks VALUE attributes. A script that substitutes the Part number if the VALUE is blank helps to provide BOM Builder with consistent data about parts.

```
Report Origin = (0.0, 0.0)
Units used = "mil"
"RefDes","PatternName","Type","Value","Layer","LocationX","LocationY","Ro-
tation"

"C1","RC0805","CAP0805","0.33uF","Top","8500.0","3545.0","90.0"
"C2","CAPTANTC","CAPTANTC","15 uF 10V","Top","7946.0","3424.0","270.0"
"C3","RC0603","CAP0603","0.1uF","Top","5430.0","3800.0","180.0"
"C4","RC0603","CAP0603","0.1uF","Top","5375.0","3905.0","270.0"
"C5","RC0603","CAP0603","0.1uF","Top","8065.0","2695.0","90.0"
"R1","RC0805","RES0805","DNI","Top","7345.0","3405.0","0.0"
"R2","RC0805","RES0805","50","Top","5705.0","3350.0","180.0"
"R13","RC0805","RES0805","331","Top","3495.0","1525.0","0.0"
"U1","PWP-20","TPS767XX","TPS76733","Top","8243.0","3465.0","180.0"
"U2","SOT223-5","REG103G","REG103GA-5","Top","5600.0","3975.0","270.0"
"U3","SOT223-5","REG103G","REG103GA-5","Top","8250.0","2525.0","180.0"
"U4","TQ144","XC95288XL-TQ144","XC95288XL-TQ144","Top","6494.7","3205.5","
0.0"
```

## 9.7 Ultiboard

"Ultiboard Information Export File"

```
"Design Name  : LEDModule(LEDModule) - Parts Centroids"
"Report Date  : 12 September 2005"
"Report Time  : 10:40"
"-------------------------------------------------------------------"
"REFDES","VALUE","SHAPE","X_CENTRE[nm]","Y_CENTRE[nm]","ROT","SIDE","TYPE"
"C1","100pF","C0603","53238400","36118800","0","TOP","SMD"
"C2","100nF","C0603","32052000","108534200","90","TOP","SMD"
"C3","100nF","C0603","42516800","108534200","90","TOP","SMD"
"C4","100nF","C0603","48793400","6096000","270","TOP","SMD"
"C5","100nF","C0603","42824400","9067800","0","BOTTOM","SMD"
"C6","100nF","C0603","63601600","23114000","270","BOTTOM","SMD"
```

## 9.8 Graffy

In the PCB layout module - You go - File -> Export -> Archive This generates the pcb archive file .

```
$$ GRAFFY Technical Graphics Editor
$$ Revision            : W.11.06
$$ Archival Date (M/D/YYYY)  : 9/18/2006
```

```
$$ Archival Time       : 8:36 AM
UNITS INCH,10000;
LOCK 45.0000;
GRID 0.0250,2 0.0000,0.0000;
```

## 9.9 Allegro

The Allegro export functions produce files using a semicolon as a field delimit characters. There is one set of fields for the Input Parts List and another for XYRS information. XYRS info is in mils (1/1000 Inch).

Parts List:

```
PartNumber;Description;Package;RefDes
10MQ100N-ND;International Rectifier 100V 1.5A Schottky SMA;SMA;D3
160-1435-1-ND;Lite-On Inc Green LED 0603 80mCandela LED;0603_LED;D2,D1
240-1018-1-ND;FB 0805 500mA 600 Ohm Ferrite Bead;0805; L18,L17, L16,L15
296-14174-5-ND;TI OPA569 Power Amplifier, R2R 2A;TSSOP20;U6,U8
296-17549-ND;TI Dual 14bit D/A Converter;TQFP48G;U18,U17,U16,U15
296-17730-1-ND;TI THS4509 Diff. Amp.;QFN16;U30,U29,U28,U27
```

XYRS Information

```
RefDes;PartNumber;X;Y;Rot;Side
U22;296-17999-ND;1173.57;4236.72;180.000;Back
U21;296-17999-ND;2270.00;4780.00;180.000;Back
U20;296-17999-ND;4690.00;4780.00;180.000;Back
U19;296-17999-ND;5615.00;4245.00;180.000;Back
C49;399-3098-1-ND;840.00;1420.00;180.000;Top
C40;399-3691-1-ND;6150.00;2755.00;0.000;Back
C216;495-1509-1-ND;780.00;4540.00;270.000;Back
```

## 9.10 Proteus

Proteus XYRS files have the following form:

```
LABCENTER PROTEUS PICK AND PLACE FILE
=====================================

Component positions for G:\mydir\myfile.LYT

Fields: Part ID, Value, Package, Layer, Rotation, X, Y
Units:  Rotation - degrees, X, Y  - thou
Notes:  The X, Y value is the centre of package as drawn in ARES.
        The origin for these values is the Output Origin.
        The values are a guide only and must be checked manually when
        setting up automatic insertion equipment.
```

```
"U1","LT1366","SOP8.225",TOP,270,-3962.5,2025
"U8","LT1366","SOP8.225",TOP,270,-3962.5,1625
"U13","LM4120","SOP_23_5",TOP,0,-4055,542.5
"C18","1.0uf","0805",TOP,0,-3907,595
"C19","0.1uf","0805",BOT,0,-3842,735
"R25","54.9K","0805",TOP,270,-3750,1992
```

# 10 Package Names

A key component in the BOM Builder software environment is a library of electronic component Packages. There are two major purposes for these packages:

- *Present the user with a graphic image of how the packages will fit on the final PCB.*
- *Setup programming data for Index Designs pick and place equipment.*

When constructing a new a BOM, previously used parts can be assigned a Package Name automatically while new parts must be assigned a Package Name manually. In order to locate and select the proper Package Name for a new part, a description has been assigned to each package as a search aid. In addition to the description, the Package Name is formed in a manner which helps to identify package style and key dimensions.

A search engine is provided to assist with locating packages. Package descriptions include information about overall shape, important dimensions and appropriate functional information. The search engine takes user inputs and finds packages with matching descriptions.

Before creating new packages carefully study existing packages. Observe how packages are named and study their descriptions. Careful checking might identify an existing package that matches a new part or a similar package can be used as a starting point.

## 10.1 Package Descriptions

Every Package in the Index Designs library includes a description. These descriptions feed the "Search Indexer" which allows these packages to be "Found" as users search using known package parameters. A key point to remember is that "Packages" are not "Components". A SOIC-14 package could be a resistor network, analog IC or a transistor array. A pick and place does not care what is inside a package, only the dimensions and pin locations are important.

The first portion of every package description is the "Style" and pin count. Style must be VERY CAREFULLY defined. Industry standard names (if there are any) for many packages use ambiguous names. Packages like SOIC, TSOP, SSOP are all GULL WING style devices. All packages with leads on 2 sides which come out and then down are named Gull Wing. The search engine will resolve most common names for these devices to Gull Wing packages.

After the initial style characters is a pin count, at least for most components. Some package styles imply a fixed number of pin so pin counts are not included, for example an 0603 package. If a common package does have a different number of leads then include the pin count in the name and description. For example some filter components come in 0805 packages but with 3 leads. This description should include "3 Pins" and the name will include a -3L

At the end of the description include a SMT or THT to help identify the assembly style of the package. A package with a single SMT type lead is SMT. Only packages with all THT leads is a THT package.

## 10.2 Package Name

While BOM Builder includes a search engine to assist with locating packages, most CAD library tools only provide a alphanumeric sort on decal name. BOM Builder also provides a similar sort based on package names. For this reason package names should be VERY CAREFULLY chosen.

The most important step in defining a package name is knowing the package "Style" and reviewing how existing packages of that style are named. To help with this process the following sections describe the existing package styles in the BOM Builder library.

## 10.3 Dimension Convention

The BOM Builder Package Editor uses a simple but powerful scheme to define units. Any length dimension without a decimal point is considered as in mils (1/1000 Inch). Any dimension with a decimal point is considered as mm.

When naming a package use this convention. If a package is primarily defined in metric use mm other wise use mils. Be consistent.

## 10.4 Gull Wing & QFP

Packages with leads which stick out from the body on two sides are called Gull Wing (GW). If leads

stick out on 4 sides then the package is a QFP. Industry names like TQFP and VQFP are all considered as QFP style packages. SOIC, TSOP, SSOP, SOP, MSOP are all Gull Wing packages.

Gull Wing packages are named with a "GW-" followed by the number of pins. For pin counts less than 10 add a leading zero. Following the pin count is the pin pitch followed by "Width Across Leads". For example a standard 8 pin SOIC is a **GW-08-50-240**. This device has has 8 pins, 50 mil Pitch and is .24 inches from lead tip to lead tip. The description for this part is:

**Gull Wing 8 Pin .05 inch Pitch .24 Wide SMT**

QFP packages use an aaaaXbbbb notation to indicate the X Size (aaaa) and Y Size (bbbb). For example a 144 pin QFP with .5mm pin pitch is named **QFP-144-.5-22.0X22.0** and the describtion is:

**QFP 144 Pin, .5mm Pitch, 22mm x 22mm SMT**

## 10.5 Industry Standard Styles

Deciding on which package names are "Industry Standard" is very difficult as no two people in the industry seem to have the same standards. Still there are a few standard names. Style names like SOT, BGA, SOJ are quite common though additional parameters are required. Diodes names like SMA, SMB and SMC are probably the most well defined names while names like SOIC and TTSOP could mean lots of different packages.

Another set of "Industry Standard" names are the numeric chip type components. Examples are 0603, 0805 and 1206. The descriptions for these contain both the inch and metric names. Several of these have 3 or 4 leads, their names include a -3L or -4L at the end.

## 10.6 BGA, Connector, SOJ and PLCC

In general these are straight forward in naming. All parts start with the style, pin count, pitch then size. SOJ are like Gull Wings in that they have leads on 2 sides except they bend in and under the part. PLCC are like QFP parts in that they have leads on 4 sides. PLCC parts are like SOJ in that leads bend down and under forming a "J" lead.

Sockets are connectors. A socket for a PLCC type has totally different dimensions than the PLCC component it accepts. When placing a socket at a location be sure to select a socket for the device as sockets take more room and this could offer a surprise during assembly.

## 10.7 LCC, LCQ and QFN

All these packages are leadless where their leads are flush with the bottom of the package. These packages they offer a unique set of assembly challenges. LCC packages have leads on two sides or have 4 pins where the 4 pins are in the corners. LCQ parts have pins on 4 sides. QFN packages have leads on 4 sides and they include a thermal tab. QFN parts are singled out as they require special assembly processing.

## 10.8 Inductors

Inductors come in a wide variety of strange shapes and sizes. Inductors use IND as the style followed by pin count if the pin count is greater than 2. Pin Pitch is included if it is obvious and finally the X and Y size is included. See the current package library for examples of inductor names.

## 10.9 Functional Names

While the idea of naming packages is to avoid including functionality, sometime it is the best solution. Examples of such names include:

- *LED Led packages are all over the spectrum.*
- *SW for Switch. Highly unique. Many SW devices an not be machine mounted.*
- *CAP Some capacitors have unique and industry standard packages.*
- *RN Chip resistor networks. Many sizes that are unique to this electrical function.*
- *TP Test points*
- *XFORM Transformers have unique size, pin styles and thickness.*
- *XTAL Crystals and Ceramic Resonators*

Try to avoid functional data in names. Place functional information in descriptions which is general in nature. Component part numbers like LM555 or 4N35 should never be contained in package names of descriptions. Never.

# 11 Keyboard Shortcuts

BOM Builder incorporates a number of "Keyboard Shortcuts" in several different screens. These key codes are used to reduce BOM coding effort and to control special functions.

## 11.1 Main Tree Screen

The Main Tree Screen must be the window with focus for the following key to function. The function is applied to the part or parts identified in the Line Item Editor area.

- *F5 Toggle STUFFED Status*
- *F6 Set Machine SMT Status*
- *F7 Set Hand SMT Status*
- *F8 Set Hand THT Status*
- *ESC Clear Line Item Editor Area*
- *^F Open Part Search Screen*
- *^Z Toggle Auto Zoom Mode*
- *^E Launch Windows Explorer*

The Part Search Screen allows the user to search on either a reference designator or general text. Text search will display a number of Line Items in lower screen area. A user mouse click on these items will adjust the Main Tree cursor position to the selected item.

Auto Zoom is a feature which allows rapid inspection of part placements. When ^Z is pressed Auto Zoom will toggle ON and Off. When it toggles ON a Beep is generated as confirmation. When the Assembly View is displayed Auto Zoom will force the Assembly View to the proper Zoom and Center as to display the details of part placement. As subsequent parts are selected in the Main Tree the Zoom and Center will be automatically adjusted to show these parts. Auto Zoom is automatically disabled if the user does a manual zoom (Page Up or Page Down) in the Assembly View window. Another ^Z will turn Auto Zoom back ON.

Pressing Ctrl-E will launch Windows Explorer. The directory opened will either be the default BOM directory or, if a BMB file is loaded, the directory where that .BMB file is located..

## 11.2 Read Gerbers Screen

Several key are used to modify the gerber image view and data.

- *Page Up Zoom In*
- *Page Down Zoom out*
- *Home Zoom to show all data.*
- *Insert Center to current Mouse Position*

The Delete key will delete the currently selected gerber shape. Shapes are selected using mouse clicks and they are highlighted when selected. The remaining keys allow the user to zoom and pan in the graphic display.

## 11.3 Pick A Package Screen

Several keys and mouse buttons allow the user to preview how a component will "Fit" on PCB gerber images before a final package is selected.

- *ALT [SHIFT] Arrow Up Down Left Right*
- *ESC Clear GREEN Preview*
- *CTRL Right Click*

Whenever a Package is selected in the Available Packages area, and the Assembly View screen is open, the currently selected Main Tree item is displayed using the Pick A Package selection. This display is only a temporary display so it is shown in GREEN to indicate the difference from a normal package display. Normal Package Assignments are always shown in Red of Blue and never GREEN. This display allows a user to see now alternate packages might fit on the user PCB image.

ALT Arrow keys can be used to adjust positions. Positions are adjusted my 1 mil (25.4 um) for each arrow key hit. If the SHIFT key is pressed with the ALT key movements are in 5 mil (127 um) steps. These adjustments are only temporary, they are discarded when the GREEN package is removed.

A Right Click with the mouse on a Available Package will cause the displayed GREEN package to rotate by 90 Degrees. Again this rotation is only temporary. Final rotations require XYRS data and/or adjustments in the Assembly View screen.

## 11.4 Package Editor Screen

Several keys are used to control the generation of new packages.

- *Page Up Zoom In*
- *Page Down Zoom Out*
- *Home Zoom to show all data*
- *Insert Pan and Insert Pin Control*

Insert will center the graphic screen at the current mouse pointer unless the user is adding a pin. When Insert Pin mode has been selected the Insert Key will cause a new pin to be inserted at the current mouse point. The size and shape of this new pin is copied from the pin selected when the Insert Pin mode was initiated.

The Page Up, Page Down and Home keys provide control over the Zoom and Pan of the displayed graphics. These function as in other graphic windows.

## 11.5 Assembly View

Assembly View has the most complicated set of keyboard command as compared to other BOM Builder screens.

- *Page Up Zoom In*
- *Page Down Zoom out*
- *Home Zoom to show all data*
- *ESC Clear current special modes*
- *Various Mouse Clicks*
- *ALT [SHIFT] Arrow Keys*
- *^B Set Board Outline*
- *^P Set list of Ref Des for manual placement*
- *^R Adjust Component Reference Point*
- *^C Copy screen graphic to Clipboard*
- *F5 Toggle STUFFED Status*
- *F6 Set Machine SMT Status*
- *F7 Set Hand SMT Status*
- *F8 Set Hand THT Status*

The Page Up, Page Down and Home keys are similar to the functions in other screens. These keys allow the user to Zoom in and out and control the graphic display.

The ALT Arrow keys (with the option of SHIFT) serve two functions. The normal mode is used to adjust the position of the currently selected part. Parts are selected using a mouse click and the selection data in shown on the screen upper left. Using ALT Arrow keys this parts can be repositioned in 1mil (25.4 um) steps. Including the SHIFT key changes this to 5 mil (127 um) steps. While most placement is done using XYRS data these keys allow fine tuning of placement data.

During Adjust Component Reference Point mode, which is entered using ^R, the arrow keys adjust the XY Placement reference point. The final effect is that all parts move relative to the gerber image. Normal movements are 1 mil while the SHIFT key provides 5 mil (25.4 um and 127 um) moves.

The ^B key is used to start Board Outline mode. In this mode the user can click on gerber items to select the XY points of the board outline. The BACKSPACE key can be used to remove previous points.

To support documentation requirements the ^C key will copy the current graphic image to the Windows Clipboard which allows the image to be incorporated in emails, reports or other desktop applications.

While most components will be placed using XYRS data there are occasions where parts are placed by hand. To simplify this task the ^P key allows the user to specify a sequence of parts, by reference designator, to manually place. This allows the user to concentrate on a single area of the board and complete all the components in that section. Parts from the list will be "Cycled" to the graphic cursor using the ESC key.

The F5-F8 keys provide the same function as they do in the Main Tree Screen. The currently selected Assembly View part is changed according to the key pressed. The ESC key will cancel any current command and deselect and any current modes. If there are additional parts in the Placement List (^P Function) then the next reference designator in that list will be used and that part's graphic will be attached to the cursor for placement. Various mouse clicks are also used to adjust component positions. Please see the XYRS chapter for these functions.

# 12 INI File Variables

BOM Builder maintains a number of configuration setting in file BOM_Bldr.ini located in the c:\BOM_Bldr\Data directory. There are three major sections:

- *Forms: Size and locations of forms.*
- *Files: List of previously opened files.*
- *Defaults: Many system wide parameters.*

When BOM Builder is opened it reads the INI file and saves setting in memory. When BOM Builder is closed it re-writes the .INI file. If sections of the INI file are deleted before it is read default value will be used and these default value will be written. This is especially useful if a form becomes positioned off screen and can not be manually repositioned. Simply delete the forms section from the INI file and defaults for all form positions will be re-established.

## 12.1 Parameter Definitions

| | |
|---|---|
| PV_File | Location of Parts & vendors database |
| PCBPath | Dir of default PADs directory for PCB Designs. |
| AOTEnable<br>AutoAOT<br>DefaultAOT | Used to enable AOT processing and, if so, if automatic AOT processing is used the DefaultAOT names the file. |
| GerberFormat<br>GerberRotate | Initial gerber format in 274-X format. Default Rotation. |
| PartTypeFile | File used to relate Ref des letters to a part type name. |
| IPL_FileName<br>XYRS_Filename<br>MRP_FileName<br>ONL_Filename | Defult file names for .dll processors and MRP input data. |
| DescAttributes<br>ValueAttributes<br>PartNumAttribute | Each is a list of PADs "Field Names" used to form Desc., Value and Part Number. |
| ColorPrinter | Used to alter printing colors |
| MRPDataDate<br>DecalDataDate | Time stamps used to detect changing Package Data and MRP Data. |
| LibUpdateDir | Default directory used to distribute Package and MRP Data. These are "Sniffed" for new data each time BOM Builder is started. |
| StartupCount | Tracks count of program starts. |
| KitLabelYOffset | Used to center printing of assembly kit labels. |
| PrintEmptyPages | Used to control printing of Top or Bottom when no parts are detected on that side. |
| ForceCWTop<br>ForceCWBot | Set default spin direction for XYRS imports. |
| DefaultPinter | Last Printer Selected |
| IgnoreAbsentParts | Disables warning messages about non-stuffed parts. |
| AutoAsmView | Auto. Open of Assembly View |
| AutoAsmNotes | Auto. Open of Note Files |