# Table of Contents

List of Figures and Tables..............................................................................................................4

1.0 – Introduction ..........................................................................................................................5

1.1 – Definition of DFA..................................................................................................................5

1.2 – Scope.......................................................................................................................................5

2.0 – Principles of DFA for PCBs..................................................................................................6

2.1 – Reduction & Combination ...................................................................................................6

2.1.1 – Reduction (Total Part Count) ..........................................................................................6

2.1.2 – Combination (Total BOM Lines) ....................................................................................6

2.2 – Standardization ....................................................................................................................7

2.2.1 – Parts & Vendors ..............................................................................................................7

2.2.2 – Component Packages .....................................................................................................7

2.3 – Minimization of Risk ...........................................................................................................7

2.3.1 – Reliability .......................................................................................................................8

2.3.2 – Fragility ...........................................................................................................................8

2.3.2 – Heavy and Unwieldy Parts .............................................................................................8

2.4 – Simplification .......................................................................................................................8

2.4.1 – Complex Parts .................................................................................................................9

2.4.2 – Complex Processes .........................................................................................................9

2.5 – Clarity ....................................................................................................................................11

2.5.1 – File Clarity & Files Required for PCB Assembly ...............................................................11

2.5.2 – Design Clarity ..................................................................................................................14

3.0 – Understanding Bittele’s Assembly Capabilities .................................................................15

3.1 – Orientation ..........................................................................................................................15

3.1.1 – Preferred Markings .........................................................................................................15

3.1.2 – Consistency ......................................................................................................................16

3.1.3 – Zero Orientation ..............................................................................................................17

3.1.4 – Wave Soldering ................................................................................................................17

3.2 – Spacing ................................................................................................................................17

3.2.1 – Part-to-Part Spacing .......................................................................................................17

3.2.2 – Part-to-Edge Spacing .....................................................................................................18

3.2.3 – Part-to-Hole Spacing .....................................................................................................19
3.3 – Sizing ......................................................................................................................... 21
  3.3.1 – Pad and Hole Sizes ................................................................................................. 21
  3.3.2 – Package Sizes ......................................................................................................... 22
  3.3.3 – Pitch ....................................................................................................................... 23
3.4 – Standards & Certifications .......................................................................................... 24
  3.4.1 - IPC-A-610 ............................................................................................................... 24
  3.4.2 – SMTA ..................................................................................................................... 24
  3.4.3 – ISO-9001:2008 ........................................................................................................ 25
3.5 – Reflow Soldering ........................................................................................................... 25
  3.5.1 – When to Reflow ...................................................................................................... 25
  3.5.2 – Heat Profiles ......................................................................................................... 25
3.6 – Wave Soldering ............................................................................................................ 26
  3.6.1 – When to Wave Solder ............................................................................................. 26
  3.6.2 – Heat Profiles ......................................................................................................... 27
  3.6.3 – Orientation & Shadowing ...................................................................................... 27
  3.6.4 – Pad Shapes ............................................................................................................ 28
3.7 – Manual Soldering ......................................................................................................... 28
  3.7.1 – When to Manually Solder ...................................................................................... 28
  3.7.2 – Manual Soldering Restrictions ............................................................................... 29
3.8 – Special Assembly Requirements ............................................................................... 29
  3.8.1 – Mechanical Component Assembly ...................................................................... 29
  3.8.2 – Adhesives ............................................................................................................. 30
  3.8.3 – Pressfit Parts ......................................................................................................... 31
  3.8.4 – Wire Bonding ....................................................................................................... 31
  3.8.5 – Wire Harness ....................................................................................................... 31
  3.8.6 – Enclosure Assembly ............................................................................................. 32
  3.8.7 – Conformal Coating ............................................................................................... 32
  3.8.8 – IC Programming .................................................................................................... 32
  3.8.9 – Serialization ........................................................................................................ 33
3.9 – Common Assembly Issues .......................................................................................... 34
  3.9.1 – Tombstones & Open Circuits ............................................................................... 34
  3.9.2 – Tin Whiskers ........................................................................................................ 35
  3.9.3 – Solder Bridges ..................................................................................................... 35
List of Figures and Tables

Figure 1 - RF Filter in a Single 0402 Package..................................................................................6
Figure 2 – Via-In-Pad Fanout (Left) and Dog-Bone Fanout (Right) .....................................................10
Figure 3 - Bittele's Sample BOM ........................................................................................................11
Figure 4 - Sample Centroid File .........................................................................................................12
Figure 5 - Mechanical Gerber Layer ..................................................................................................13
Figure 6 - Assembly Drawing Showing Component Placements ..........................................................13
Figure 7 - Clear (Recommended) Orientation Markings ......................................................................15
Figure 8 - Unclear (NOT Recommended) Orientation Markings ..........................................................16
Figure 9 - Example of Consistent Orientation ....................................................................................16
Figure 10 - Part-to-Board Outline Spacing ..........................................................................................18
Figure 11 - Part-to-Hole Wall and Part-to-Annular Ring Spacing ..........................................................19
Figure 12 - Bad Idea - Via Placed Between SMT Pads ......................................................................20
Figure 13 - Via Routing For QFN & QFP Packages - Preferred (left) & Acceptable (right) ..............20
Figure 14 - Minimum Ratio (C1 - Left) & Larger Ratio (C2 - Right) – Both Acceptable ..................21
Figure 15 - THT Hole-to-Lead Sizing – IPC-2222A ..........................................................................22
Figure 16 - Annular Ring Illustration ..................................................................................................22
Figure 17 - Bow & Twist on a Centrally-Placed BGA (Exaggerated for Clarity) ..............................23
Figure 18 - BGA Packages Placed "On Top Of" One Another ..............................................................23
Figure 19 - Reflow Heat Cycle ............................................................................................................26
Figure 20 - Poor Wave Soldering Orientation (Left) & Good Wave Soldering Orientation (Right) ......27
Figure 21 - Thieving Pads on a SOIC-14 Package ..............................................................................28
Figure 22 - Some of Bittele's Direct-Chip Programming Equipment ..................................................33
Figure 23 - Example of a Tombstone Defect (left) and an Open Circuit Defect (Right) .................34
Figure 24 - Reactions on Solder Joint for Bent Lead or Flat Pack ......................................................35
Figure 25 - Solder Bridge on a SO-Style IC Package .........................................................................36
Figure 26 - Solder Ball on a Passive Package .....................................................................................36
Figure 27 - X-Ray Image of Solder Joint Voids ..................................................................................37
Figure 28 - X-Ray Image of an Assembled BGA Package .................................................................38
Figure 29 - Bittele Test Engineer Performing FCT on an Assembled Board .....................................39

Table 1 - Part-to-Part Spacing Matrix (All Dimensions in mil / thou) .................................................18
Table 2 - Annular Ring Size by Copper Weight ..................................................................................22
Table 3 - AOI Thresholds ..................................................................................................................38
1.0 – Introduction

The purpose of this Design for Assembly (DFA) guide is to assist Bittele’s customers in designing printed circuit boards (PCBs) that can be assembled in an efficient and cost-effective manner. The following sections will define the various tolerances, rules, and procedures to which Bittele adheres during the PCB assembly process.

1.1 – Definition of DFA

Design for Assembly refers to a specific method of design that takes into account not only the functionality of a finished product, but also the cost and efficiency of the assembly process for that product. Many design decisions can have an immense impact on these factors of efficiency and cost during the assembly process, which will in turn affect the overall quality and reliability of the finished product.

For PCB projects in particular, a clear understanding of DFA will help to reduce the costs of parts procurement and component placement, which normally account for a majority of the total price in full turnkey service. These techniques can act to minimize a project’s overall lead time and protect against assembly issues that would otherwise necessitate design revision or time-consuming rework.

The principles of DFA are closely related to those of DFM (Design for Manufacturing), and both of these methods should be carefully considered in order to create the best possible product. Bittele Electronics also provides a DFM Guidelines document in an effort to provide clients with a complete picture of their options in PCB Fabrication as well as PCB Assembly.

1.2 – Scope

These DFA guidelines will proceed with a clear and comprehensive discussion around the strategies of DFA as they relate specifically to PCB projects. Though many of these strategies apply in a general sense to all areas of design, this discussion will offer definitive examples and illustrations in the realm of PCB assembly specifically. A brief outline of these strategies is given below:

- Reduction & Combination
- Standardization
- Minimization of Risk
- Simplification
- Clarity
- Understanding Assembler Capabilities*

Each of these six points are extremely important in the overall picture of DFA, but the final strategy of understanding assembler capabilities is the most explicit and multi-faceted; this point also tends to affect a designer’s ability to achieve the remaining five strategies. With this in mind, a longer section of this document has been dedicated to the specific PCB assembly capabilities of Bittele Electronics in the following areas:

- Orientation
- Spacing
- Sizing
- Standards & Certifications
- Reflow Soldering
- Wave Soldering
- Manual Soldering
- Special Assembly Requirements
- Common Assembly Issues
- Testing Methods
2.0 – Principles of DFA for PCBs

2.1 – Reduction & Combination

2.1.1 – Reduction (Total Part Count)

One of the most intuitive methods for reducing cost and improving ease of assembly for a PCB project is to decrease the total number of parts to be populated on a board. Not only will this decrease the total cost for the project’s Bill of Materials (BOM), it will also reduce the number of passes required for Bittele’s Pick & Place machines to assemble the board during production.

Of course, each piece of a particular design has its own specific function, but as a design moves into high levels of production it becomes increasingly important to balance the necessity of those functions against DFA concerns. It might be worth compromising on a few ancillary features for a more stable and cost-effective product, but even in the case where every component is essential, it is still possible to reduce the total part count through part combination.

2.1.2 – Combination (Total BOM Lines)

When a PCB design moves from prototype into production, it is crucial to look for part combination opportunities. Passive components are often available in array form, with multiple resistors or capacitors combined into a single package. In some cases, a certain circuit element can be replaced by a single part; for example, the image below shows a low-pass RF filter in a single 0402 package.

![Figure 1 - RF Filter in a Single 0402 Package](image)

Combination can also be achieved through generalization of passive parts with non-critical values; for example, parts such as decoupling capacitors can often be matched with other common parts in the design to reduce the total number of BOM lines for the project. This strategy might not reduce the total part count for the project, but it will reduce the necessary number of reel changes during assembly, and thereby decrease both the cost and time requirements of the process. Bittele’s PCB Assembly quotes take this factor into consideration, so these reductions in cost and lead time will be passed along to the client.
2.2 – Standardization

Any new PCB design brings with it a degree of uncertainty; even the most experienced engineer will have a hard time predicting every possible complication that could occur along the way. The key in standardization is to keep the level of uncertainty to a minimum by utilizing parts and processes that have worked in the past.

2.2.1 – Parts & Vendors

In the realm of PCB assembly, the components themselves are perhaps the most easily controlled sources of uncertainty in a project. Certainly this is not to say that a PCB designer should never list new, cutting-edge parts in their BOM, but such parts surely deserve extra attention both during the design phase and during PCB assembly. As such, Bittele would recommend that any new components in a design be balanced with tested and stable parts from previous designs.

The source of each component should also be carefully considered, since an unreliable source introduces greater potential for delay, misinformation, and even fake parts. Bittele provides two effective solutions in this regard: a professional parts procurement service available at zero component price markup, and a stock of reliable passives offered entirely free of charge.

2.2.2 – Component Packages

Simply put: the greater the number of unique component packages in a given BOM, the greater the time requirement for Bittele’s DFA check prior to PCB assembly. The DFA check includes a comparison of each land pattern in the PCB layout against the datasheet footprint for each part in the BOM. This comparison necessarily becomes more onerous as the number of unique packages is increased. If the DFA check process does discover footprint-to-land pattern mismatches, the necessary design revisions will also be more quickly completed when fewer unique land patterns exist across the design.

During the assembly process itself, different types of component packages often come with their own unique demands and process controls. As such, a reduction in the number of unique package types will simplify the PCB assembly process as a whole, thereby reducing both lead time and potential for error. Specific process requirements for certain component packages are discussed in Section 2.3.1 and Section 2.3.2, below.

2.3 – Minimization of Risk

Similar to the uncertainty mentioned in the previous section (Section 2.2), each PCB project carries with it some necessary risk; in fact, these two concepts are closely related to one another. A primary strategy in the minimization of risk is the minimization of uncertainty through standardization, but even well-known processes involve some degree of risk. In order to produce the best possible product, it is essential for both the PCB designer and the PCB assembler to understand and account for the specific risks involved in each individual project.
2.3.1 – Reliability

Parts chosen for PCB assembly should come from time-tested reliable manufacturers to ensure their accuracy and stability, as well as reliable delivery. Special attention should be paid to the current market status of each part as well; if a part is marked End-of-Life or Obsolete by the manufacturer, it is normally best to look for a replacement right away. Such deprecated parts may still be available for some time, but keeping them in the design any longer than absolutely necessary will likely cause delays for future runs.

Bittele Electronics boasts a professional parts procurement team that will work with clients to ensure all parts come from reputable sources, and that any stock or market status issues are resolved quickly and easily.

2.3.2 – Fragility

The physical properties of the parts in a particular design will impact the overall risk of production loss for the assembly process itself. Bittele Electronics always takes this into account, and ensures that each client’s order is filled in its entirety, regardless of potential complications during assembly. Still though, a clear awareness of the more risky parts will increase the overall efficiency of the process by minimizing the need for re-work or re-make after the fact.

Bittele can handle assembly for passive packages down to 0201 imperial, but production loss will likely be relatively high for these types of parts. This results mainly in additional assembly cost where 0201 passives are involved, to cover for the careful handling required as well as the additional stock necessary to protect against production loss. It is recommended to use 0603 or larger passive packages as often as possible to minimize both the price and production loss for each PCB assembly project.

2.3.2 – Heavy and Unwieldy Parts

The concern with particularly large or heavy parts is mainly in shipping and handling costs. Parts such as transformers or power resistors will incur much higher shipping costs for their initial transportation to Bittele’s assembly facilities, as well as the final shipment of the assembled PCBs to the client. It is not always possible to eliminate such parts from a design, but it is important to be aware of these potential sources of extra cost well in advance.

Parts requiring manual assembly will decrease the efficiency of a mainly-automated PCB assembly process. Through-hole parts on a majority surface-mount design or mounting hardware for final assembly will require additional attention from Bittele’s staff, which drives up both the cost and the lead time of a project.

2.4 – Simplification

The concept of simplification in PCB Assembly is tied in closely with minimization of risk (see Section 2.3). The difference for this section is that the impact on the assembly process comes from the complexity of the parts or processes involved, rather than their physical properties or market status. Some level of risk is unavoidable in any PCB project, but complexity can more often be reduced or even eliminated through savvy design techniques. This is not always the case, and sometimes complex parts or processes are indeed required, but simplification wherever possible will help to bring down the overall cost and lead time of any PCB project.
2.4.1 – Complex Parts

Complex parts are sometimes required for a complex product, but it is important to understand the additional considerations that must be made for these components in order to better plan and budget for a product during the design phase. As such, this subsection will proceed with a list of the more complex parts that Bittele can install, followed by a brief description of their requisite assembly considerations.

*BGA, LGA, QFN, CSP, & FLIP-CHIP*

The above four terms all describe parts with no external leads, meaning Bittele must use X-Ray inspection during the final quality check of the assembled PCBs. This verification process incurs some extra PCB Assembly cost compared to QFP-style packages. Rework is also much more difficult for these types of parts, which makes the entire assembly process much less forgiving. The very fine-pitch versions of these packages will sometimes require complex processes as well, such as via-in-pad filling described below in Section 2.4.2.

*POP (Part-on-Part)*

POP technology allows certain components to be stacked on top of one another during the PCB assembly process; for example, some advanced processors will have their memory modules mounted directly on top of the main processor package. This technology offers key advantages in miniaturization for many high-density designs, but also requires special consideration during PCB assembly, including multiple placement and reflow cycles. These additional considerations can drive up both cost and lead time for a PCB assembly project.

2.4.2 – Complex Processes

Complex processes are sometimes required for certain complex parts, as mentioned in the previous section, but might also come into play even for standard PCB designs. A PCB designer must observe certain physical requirements depending upon the environmental stresses that will be placed upon their finished product, and some of these requirements can be satisfied through complex PCB assembly processes. This subsection aims to give designers an idea of the PCB assembly considerations required by some of the more common complex processes offered by Bittele Electronics.

*Via In Pad*

This process is sometimes required for extremely fine-pitch parts such as μBGA packages when there may not be enough room between pads for a standard “dog-bone” style fan-out – pictures are included below for both of these fan-out methods.

Vias in component pads will create a risk of leaking tin during the reflow process, which can cause shorts or corrupt pads. In order to stabilize the assembly process, vias in surface mount pads must be filled with epoxy, which requires additional cost and lead time. Bittele can confidently handle via-in-pad designs, but would recommend avoiding this requirement if possible to simplify the PCB assembly process.
Wire Bonding

Bittele offers a selective soft gold surface finish option, which can be applied to specific areas or specific pads on a board during the PCB Fabrication process to allow for wire bonding. This allows for a stronger welded joint on the pad in question, which offers increased durability for leads soldered directly to the PCB. This process requires a mask drawing to specify the regions for soft gold finish, as well as additional documentation to describe the wire bonding procedure, and might not be viable for volume orders. Generally, it is more cost- and time-effective to use connectors and terminated cables for interconnection, but wire bonding might be necessary for size-restricted designs that are unable to incorporate these larger parts.

Wire Harness Assembly

Bittele is focused on board-level PCB Assembly services, and will only offer wire assembly on a case-by-case basis. This requirement should be mentioned during the quoting stage of the project so that the associated design drawings can be analyzed in advance by Bittele’s engineering team. Wire or cable assembly will most often require manual attention, which can cause a significant increase in both cost and lead time for a project. Rather than deal with wire harness assembly, some clients choose to combine modular design components onto a single PCB or to use special features in PCB fabrication such as gold fingers or rigid-flex interconnection.

Conformal Coating

For PCBs that will be used in damp, humid, dusty, or other harsh environments, a special process known as conformal coating is often required. Through this process, Bittele will cover the assembled PCB in a layer of non-conductive, protective material such as silicon, acrylic, urethane, or paraxylene. Once this coating is cured onto the board, it will increase the overall durability of the product while also protecting it from outside contaminants.

Clients should submit a masking drawing along with standard design files to stipulate any areas of the board that should not be subjected to the coating, such as connectors that will need to be accessed at a later time. The conformal coating process does require additional time and cost after PCB assembly, and Bittele also requires that functional testing (see Section 3.10) be performed on assembled PCBs prior to coating. Once the coating has been applied, the boards are fully sealed, and rework can no longer be performed.
2.5 – Clarity

2.5.1 – File Clarity & Files Required for PCB Assembly

Inconsistent or unclear data in design files will prompt additional questions and introduce confusion prior to the PCB assembly process, potentially increasing the overall project lead time. Most commonly, these types of issues arise when information in one document disagrees with information in another, so a thorough check that all design revisions are reflected on all documentation can pre-empt many questions during the assembly process.

Aside from maintaining consistency between design files, there are a few general strategies for file clarity in each of the four main types of design files Bittele requires:

**BOM Clarity**

Many CAD programs for PCB layout offer the option to automatically generate a BOM based on reference designators, footprints, and component values specified during the schematic design phase. These options can be used to save on time and minimize common errors such as mis-matches between quantity and number of reference designators. That being said, a manual review of the software-generated BOM is still essential to ensure maximum clarity.

Some of the most common BOM issues that Bittele has noticed in the past are listed below:

- Incomplete, ambiguous, inaccurate, or missing part numbers
- Quantity to reference designator mis-matches
- Part number to description mis-matches

Some manual work might be required on the software generated BOM in order to bring it into a clear and concise format, but this will save time and pre-empt mistakes later in the process. At the very least, a BOM should include complete information for Quantity, Reference Designators, and Part Number on each line; including Manufacturer, Description, and additional instructions will further protect against potential delays or mistakes. Find Bittele’s sample BOM here.

---

**Sample Bill of Materials (BOM)_Bittele Electronics Inc.**

<table>
<thead>
<tr>
<th>Item</th>
<th>Reference Designator</th>
<th>Quantity</th>
<th>Manufacturer</th>
<th>Mfg Part #</th>
<th>Description</th>
<th>Package</th>
<th>Type</th>
<th>Your Instructions</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>U1L</td>
<td>1</td>
<td>Linear</td>
<td>LTC3108E568F</td>
<td>C:BUCK-BOOST CONVR 24-SOIC</td>
<td>24-SOIC</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>U6L4</td>
<td>2</td>
<td>Fairchild</td>
<td>LM2934AM</td>
<td>C:OPAMP EIAL JO/PA C:8-SOIC</td>
<td>8-SOIC</td>
<td>DNS</td>
<td>DNS (Do Not Source)</td>
</tr>
<tr>
<td>3</td>
<td>U3L</td>
<td>1</td>
<td>ON</td>
<td>LM35105H070S</td>
<td>LMP1710 RREG 10K ADJ 50MA BPAK</td>
<td>BPAK</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>L5L</td>
<td>1</td>
<td>Microchip</td>
<td>PIC16F628-I/S</td>
<td>C:PIC MCU FLASH 40X14 28SSOP</td>
<td>28SSOP</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>Q1L</td>
<td>1</td>
<td>Diodes Inc</td>
<td>GBU1005F</td>
<td>RECT BRIDGE GPP 50V 10A SBU</td>
<td>GBU</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>RM.R10</td>
<td>2</td>
<td>Panasonic</td>
<td>ER435P-1470K</td>
<td>RES 2.2K OHM 1/8W 1% 0805 SMD</td>
<td>0805</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>R16</td>
<td>1</td>
<td>Samsung</td>
<td>KL9700WT-0033</td>
<td>RES 100 047 1/8% 0805 SMD</td>
<td>0805</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>R16, R21</td>
<td>2</td>
<td>Panasonic</td>
<td>ER460F470V</td>
<td>RES 4.7K OHM 1/8W 1% 0805 SMD</td>
<td>0805</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>C5L</td>
<td>1</td>
<td>Murata</td>
<td>GRM316071C225K472L</td>
<td>CAP 0.1UF 10V 0805</td>
<td>0805</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>C7L</td>
<td>1</td>
<td>Murata</td>
<td>GRM316071C105K472L</td>
<td>CAP 0.1UF 10V 0805</td>
<td>0805</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>CS.c22</td>
<td>2</td>
<td>Murata</td>
<td>GRM316071C105K472L</td>
<td>CAP 0.1UF 10V 0805</td>
<td>0805</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>CN6L</td>
<td>1</td>
<td>Molex</td>
<td>346803016-01T</td>
<td>2.5mm Connector Vertical pin SMT 6P</td>
<td>SMT 6P</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>L1L2</td>
<td>2</td>
<td>Cortronics</td>
<td>SOT23-100-R</td>
<td>INDUCTOR SHIELD 10UH SMD</td>
<td>SMD</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>L3L</td>
<td>1</td>
<td>TDK</td>
<td>ACM1211-102PFL-TL</td>
<td>CHOKE COMM MODE 1000 OHM 5A SMD</td>
<td>SMD</td>
<td>SMD</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>C03,OM4</td>
<td>2</td>
<td>Molex</td>
<td>S3017-0510</td>
<td>CONNECTOR SPOS 1.25MM VERT TIN</td>
<td>SPOS</td>
<td>SMD</td>
<td></td>
</tr>
</tbody>
</table>

Figure 3 - Bittele’s Sample BOM
Centroid Clarity

Clarity in the Centroid file – also known as XY-Coordinate or Pick & Place file – is extremely important, since this data will be directly used to program Bittele’s Pick and Place machines for automatic part placement. Centroid data is generated directly by the CAD software used for PCB layout, but this does not make it immune to inaccuracy or ambiguity. The following issues are the most common for Centroid data:

- Disagreement with the BOM, usually due to project updates not reflected in one document
- Missing parts, usually due to errors in the CAD software
- Inaccurate coordinates, usually due to a Gerber file update without generating a new Centroid

A sample Centroid file is shown below:

<table>
<thead>
<tr>
<th>RefDes</th>
<th>Layer</th>
<th>LocationX</th>
<th>LocationY</th>
<th>Rotation</th>
</tr>
</thead>
<tbody>
<tr>
<td>C2</td>
<td>Bottom</td>
<td>3.03</td>
<td>1.08</td>
<td>0</td>
</tr>
<tr>
<td>IC2</td>
<td>Top</td>
<td>3.25</td>
<td>0.875</td>
<td>180</td>
</tr>
<tr>
<td>IC3</td>
<td>Bottom</td>
<td>3.25</td>
<td>0.9</td>
<td>0</td>
</tr>
<tr>
<td>R2</td>
<td>Top</td>
<td>3.425</td>
<td>0.8</td>
<td>90</td>
</tr>
<tr>
<td>R3</td>
<td>Top</td>
<td>3.545</td>
<td>0.105</td>
<td>270</td>
</tr>
<tr>
<td>R4</td>
<td>Bottom</td>
<td>1.475</td>
<td>0.5</td>
<td>270</td>
</tr>
</tbody>
</table>

Figure 4 - Sample Centroid File

Gerber File Clarity

Gerber files are used mainly in the PCB Fabrication process – see Bittele’s DFM Guidelines document for more detailed information about Gerber files – but they do also play a role in the PCB Assembly process.

The first thing to note about Gerber files and their role in PCB Assembly is the Silkscreen layer. This layer shows reference designators for specific components beside their associated land patterns. During Bittele’s initial DFA check sequence, these reference designators are compared with those in the BOM and Centroid files to ensure that any errors in one file are caught early and brought to the client’s attention. The silkscreen layer also contains markings for the orientation of polarized components, which can help to avoid questions and potential issues during assembly – more on part orientation in Section 3.1.

Some HDI boards do not have enough room between components for a silkscreen layer, and some clients simply do not want to include silkscreen markings on the finished board for aesthetic reasons. In this case, Bittele will require assembly drawings or mechanical Gerber layers to act as substitute for the silkscreen layers.

Mechanical layers in the Gerber set can include specific instructions for the mounting of certain components, and many designers include additional notes in these layers for standard and special requirements. For example: a particular part might require unconventional mounting orientation, or perhaps the PCB in question requires the use of water-soluble flux rather than standard no-clean flux. Including these notes in the Gerber files will ensure that Bittele’s engineering team makes note of them early in the manufacturing process and plans accordingly.

See the picture below for an example of a mechanical Gerber file.
Issues around Gerber file clarity in PCB Assembly are often a result of design revisions that are not reflected across all files, causing a discrepancy between the Gerbers and the BOM, Centroid, or Assembly Drawing. Notes are sometimes copied from a previous design to save time, but design differences are often overlooked in this case; it is recommended to use an empty notes template and fill it for each project individually to minimize these sorts of errors.

Assembly Drawing Clarity

Assembly drawings are a supplementary file, used to offer additional clarity on the required PCB assembly and to describe in detail any special requirements for the project in question. Most PCB Layout software suites include an option for generating assembly drawings, which will generally output to PDF both a top-down and a bottom-up view of the board, as below:
A basic assembly drawing, such as the one shown above, can help to clarify placements and DNI (Do Not Install) parts, but it is sometimes necessary to provide more information to clarify special requirements. Processes such as wire assembly, as well as specifications for temperature tolerances, flux and solder types, or manual assembly parts, should be included in the assembly drawing. These types of specifications can be added on a separate page of the PDF document or simply included in the margins of the drawings themselves.

Bittele Electronics recommends including at least a basic assembly drawing with any PCB Assembly project submission. As a general rule: the more information that is included on the drawing, the fewer the number of questions asked during assembly.

2.5.2 – Design Clarity

Clarity must be observed not only with respect to the contents of design files provided to Bittele, but also the PCB design itself. Bittele often needs to pause the assembly process to clarify ambiguities in the design, sometimes causing a delay to the overall project timelines when a clear design could have resolved these questions before they ever arose. Taking the extra time during the design phase of a project to ensure orientations are marked as clearly as possible will reduce the potential for delays during PCB Assembly.

Assembly side is another potential source of ambiguity; though it is mostly handled by the Centroid file for automatic assembly parts, the safest method is simply to stick with single-sided PCB designs as often as possible. For manual assembly parts, it is best to note orientation and assembly side clearly in both the BOM and the Assembly Drawing. In general, a detailed silkscreen and mechanical layers in the Gerber files will greatly improve efficiency during assembly through improved design clarity.
3.0 – Understanding Bittele’s Assembly Capabilities

3.1 – Orientation

Part orientation is a very common topic for pre-assembly engineering questions. As such, it is important to follow a clear and concise method for orientations in order to avoid questions and keep the PCB assembly process flowing smoothly.

3.1.1 – Preferred Markings

For diodes, it is recommended to use a clear schematic symbol that will still be visible after part placement on the finished PCB to ease the final inspection process. For through-hole parts, the symbol can be placed between the two pins, but for surface-mount parts, the symbol should be placed beside the device. These symbols can take up quite a bit of space, and shrinking them down too much will make them unclear, so on HDI boards a bar above the cathode pad or a simple marking of A (anode) or K (cathode) will be sufficient. Note that K rather than C should be used for the cathode marking, to avoid confusing the part with a capacitor.

For polarized capacitors, it is recommended to use a ‘+’ for the side of the device that should be connected to power. To make the polarity even more clear, the side that should be connected to ground can be filled in with silkscreen, as in the image shown below.

Finally, for ICs with many pins, Bittele would recommend that the clearest possible marking is simply the number 1 beside the intended Pin 1 on the footprint.

The images below are examples of clear orientation markings, recommended by Bittele, for various parts.

![Figure 7 - Clear (Recommended) Orientation Markings](image-url)
Some common examples of unclear markings are given below; most notably, the + or – notation for diode polarity is considered unclear and is not recommended. This is due to the fact that Zener diodes and many LEDs are operated in reverse bias, so the + notation could refer to either the anode of the part, or to the side of the part that should be connected to higher voltage.

For IC packages, a circle placed inside the land pattern, beside Pin 1, is a very common convention. Bittele can recognize this marking as meaning Pin 1, but it is still not recommended since the circle will not be visible during final inspection, once the part has been installed. A circle can still be used for the Pin 1 ID if preferred, but it should be located outside the land pattern.

3.1.2 – Consistency

To further reduce ambiguity, Bittele recommends that similar parts be grouped together and placed with the same orientation as often as possible. Such consistency will also facilitate a more efficient automatic assembly process. For example: all QFP packages could be placed in a row with pin 1 at the same corner for each IC.
3.1.3 – Zero Orientation

The centroid file that a client provides for PCB assembly will specify rotation values for each part along with its X-Y coordinates. These rotations are normally based on one of two standards, specified by IPC’s planned revision C to their 7351B standards and described below:

- Level A: Pin 1 in upper-left-hand corner of package when Rotation = 0
- Level B: Pin 1 in lower-left-hand corner of package when Rotation = 0

Bittele can handle either of these two standards, and is also able to determine the zero orientation used during design by examining the centroid file and comparing against Gerber files or assembly drawings. That being said, specifying this parameter clearly in the assembly drawing or Gerber notes for a project can help to speed up the pre-assembly engineering check process.

3.1.4 – Wave Soldering

When PCB Assembly is accomplished by wave soldering, part orientation becomes even more important. Issues such as solder bridging and shadowing can be prevented by specific orientation strategies, and a firm understanding of these strategies is of critical importance when designing a PCB that will undergo wave soldering. For more on these considerations, see Section 3.7.

3.2 – Spacing

The spacing and land pattern design for each component package will impact the overall reliability and timeline requirements for the PCB assembly process, and will also have a considerable impact on the repairability of a PCB. This section will discuss Bittele’s suggested minimums for component spacing to ensure quality assembly.

3.2.1 – Part-to-Part Spacing

Adequate spacing between components on the board protects against potential faults such as solder bridging, and allows for easier manual soldering and/or rework. The greater the part-to-part spacing, the better in terms of quality and ease for PCB assembly, but Bittele does understand that certain applications will require tight spacing to achieve an altogether small form factor.

The table provided on the following page shows Bittele’s minimum spacing requirements between various different surface mount parts. Measurements in this table are given in mil (thousandths of inches). Measurements should be either from the edge of the pad or the part body, whichever is the smaller distance.

A few special cases should also be noted on the topic of part-to-part spacing for sensitive packages, such as BGA, POP, or larger QFP/QFN:

1. IC sockets should be spaced as far as possible from sensitive packages. Frequent loading and unloading of the IC into the socket will place undue stress on nearby solder joints
2. Sensitive packages should not be placed in the center of a PCB, since this is where the maximum bow & twist tends to occur, which can result in broken connections
3. BGA and other lead-less packages should be placed only on one side of the PCB. If they must be placed on both sides, they should not be in the same x-y positions (“on top of” each other), since this will greatly complicate X-Ray inspection and rework.
### 3.2.2 – Part-to-Edge Spacing

Part-to-edge spacing refers to the distance from a given component a PCB to the board edge. This condition is important for the depanelization process, which is performed after PCB assembly. When PCBs are depanelized, either through V-Scoring or Tab Routing (see Bittele’s DFM Guidelines for more information on panelization), parts located near the board edge will be placed under stress that could threaten the integrity of their solder joints. Bittele’s requirement is 125 mil part-to-board edge spacing for the top side of the PCB.
Part-to-board edge spacing for any components on the secondary side of the PCB should be increased to 300 mil. The greater requirement in this case stems from the solder paste screening process, which requires that hold-downs be applied to the secondary side to prevent the PCB from moving while solder paste is applied. Any surface-mount components in this area could be damaged by the hold-downs, or simply blocked during the screening process and left with insufficient solder paste.

It should be specifically noted that the above specifications refer to component pads and bodies for those parts installed by automated assembly processes. Manually assembled parts (see Section 3.7) may be placed closer to the board edge since they can be installed after reflow and depanelization. Copper traces can also be run much closer to the board edge. Both copper traces and manually installed parts should still be kept at least 10 mil from the board edge to allow a solder mask gap and prevent encroachment of the pads.

Some designs require copper plating or castellated holes at the board edge itself. Bittele can handle these requirements, but clients should be advised that the controls involved to produce such PCBs reliably will introduce additional cost and lead time to a project.

3.2.3 – Part-to-Hole Spacing

Part-to-hole spacing applies to both through-hole components and PCB vias; it stipulates the minimum spacing required between a component pad or body and either of these types of holes. This spacing is actually divided into two specific parameters, and both of these must be satisfied in order to assure a quality assembly.

1. **Part-to-Hole Wall**: Measured from the edge of the actual hole in the PCB to the edge of a pad. Bittele’s minimum requirement for part-to-hole-wall spacing is 8 mil
2. **Part-to-Annular Ring**: Measured from the edge of the hole’s annular ring to the edge of a pad. Bittele’s minimum requirement for part-to-annular ring spacing is 7 mil

![Figure 11 - Part-to-Hole Wall and Part-to-Annular Ring Spacing](image)

This distinction is drawn because minimum annular ring size and minimum hole size are not proportional, so it is important to ensure that both of these conditions are met during the design phase. For more information on minimum annular ring and minimum hole sizes, see Bittele’s DFM Guidelines.
One last note on this topic is that it is generally a bad idea to place a via between two surface-mount pads, even if Bittele’s minimum part-to-hole spacing is satisfied. This sort of arrangement increases the likelihood of a solder bridge forming under the component during reflow.

Vias placed underneath a QFP or QFN package are less problematic, but should still be avoided if at all possible for maximum quality and yield.

![Figure 12 - Bad Idea - Via Placed Between SMT Pads](image1)

Vias placed underneath a QFP or QFN package are less problematic, but should still be avoided if at all possible for maximum quality and yield.

![Figure 13 - Via Routing For QFN & QFP Packages - Preferred (left) & Acceptable (right)](image2)
3.3 – Sizing

In the PCB industry, there are many different sizing requirements that can affect the overall cost, quality, and turnaround time for a project. This section will discuss those sizing requirements that relate to the PCB assembly process specifically and Bittele’s capabilities therein. For more information on sizing requirements for PCB Fabrication, see Bittele’s DFM Guidelines.

3.3.1 – Pad and Hole Sizes

It is generally most advisable to use the pad and hole sizes recommended by the datasheet for a given part, and Bittele is generally capable of handling any such designs. It does sometimes happen that a datasheet does not give pad or hole information, and sometimes a PCB designer wonders about pushing beyond these recommendations to economize board space. In the latter scenario, this design decision should be explicitly stated in the assembly drawing for the project to avoid pre-assembly questions from Bittele, and to ensure that Bittele employs the correct quality controls during PCB assembly.

Where a design must deviate from datasheet suggestions, Bittele’s minimum capabilities are as follows:

**BGA Pad Sizing**

Bittele’s minimum capability for BGA Pad Size depends to some extent upon the surface finish used on the PCB (for more on surface finishes, see Bittele’s DFM Guidelines).

- For LF-HASL, 12mil diameter is the absolute minimum
- For any other surface finish, 10 mil diameter is the preferred minimum & 7 mil is absolute minimum

Bittele is able to control pad side tolerance for BGAs to within +/- 1.5 mil preferred, or +/- 1.2 mil minimum

**SMT Pad-to-Lead Aspect Ratio**

For surface-mount parts, the pads in a land pattern must be somewhat larger than the actual leads of the part to allow for solder to flow and form proper connections. When in doubt, designing the pads slightly larger necessary is always safer than designing them too small. The necessary pad-to-lead aspect ratio differs not only between package types, but also between the specific technology incorporated in given devices of the same package. As such, Bittele would strongly recommend following datasheet recommendations for surface mount pad sizes. If no such information is listed in a given datasheet, then it is advisable to follow the sizing listed for a similar part.

Figure 14 - Minimum Ratio (C1 - Left) & Larger Ratio (C2 - Right) – Both Acceptable
**THT Hole-to-Lead Aspect Ratio**

For through-hole parts, the diameter of the holes for assembly should be the diagonal of the cross-section for the lead to be installed.

![THT Hole-to-Lead Diagram](image)

Figure 15 - THT Hole-to-Lead Sizing – IPC-2222A

**THT Annular Ring**

Bittele

<table>
<thead>
<tr>
<th>Copper Weight (oz.)</th>
<th>Minimum Annular Ring Requirement</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;= 1.0 oz.</td>
<td>4.0 mil (3.5mil in a few instances is acceptable)</td>
</tr>
<tr>
<td>1.5 oz. to 2.0 oz.</td>
<td>6.0 mil (5.0mil in a few instances is acceptable)</td>
</tr>
<tr>
<td>&gt;= 3.0 oz</td>
<td>8.0 mil (6.0mil in a few instances is acceptable)</td>
</tr>
</tbody>
</table>

Table 2 - Annular Ring Size by Copper Weight

![Annular Ring Diagram](image)

Figure 16 - Annular Ring Illustration

**3.3.2 – Package Sizes**

Bittele can handle assembly of small passive packages down to 0201 resistors and capacitors to meet the electronics industry’s growing demand for smaller products. Accurate automated assembly and meticulous quality assurance checks allow these delicate components to be flawlessly installed, but it is still recommended to stick with larger sizes whenever possible for DFA purposes. The additional process which must be observed for the assembly of very small parts, coupled with the inherent production loss involved in handling them, means extra cost for the project that could be avoided by up-sizing passives to a more comfortable 0603 sizing.
On the other side of the package size equation, larger QFN, QFP, and BGA modules are sometimes required to perform very complex or demanding processes. Bittele can handle the assembly of these parts as well, but some specific placement notes are strongly recommended for such components:

1. Large packages should not be placed in the center of a PCB, since this is where the maximum bow & twist tends to occur, which can result in broken connections.

2. Large packages should be placed only on one side of the PCB. If they must be placed on both sides, they should not be in the same x-y positions ("on top of" each other), since this will greatly complicate X-Ray inspection and rework.

3.3.3 – Pitch

Bittele can handle assembly for fine pitch parts as small as 0.4mm using regular processes; 0.35mm pitch is feasible, but may require some additional cost for specialized quality control processes.

Certain BGA packages, known as “dual-pitch BGAs”, specify a different pitch for row-axis pads than for column-axis pads; that is, vertical spacing between pads might be different than horizontal spacing between pads. Bittele can handle dual-pitch BGAs as long as the above minimum pitch requirements are met.
3.4 – Standards & Certifications

Though Bittele can handle many specific and unique PCB Assembly requirements, it is often more efficient to simply specify that the PCB must meet certain industry standards. Trade organizations such as IPC, SMTA, and ISO exist for exactly that reason: their standards allow for a straightforward understanding between a PCB designer and their PCB assembler of choice.

These standards and certifications can act to assure that PCBs will meet or exceed expected qualifications, and can also help to streamline communication between Bittele and PCB designers. Rather than specifying each individual requirement for a particular PCB, a client can just let Bittele know that the board must follow IPC-A-610 Class 3 standards; if a few key aspects can or should differ, then these can be mentioned separately.

This section will offer basic information about Bittele’s qualifications around industry standards and certifications as they relate to Design for PCB Assembly principles. For more complete information on each specific standard, clients should visit the individual trade organizations’ websites.

3.4.1 - IPC-A-610

IPC-A-610 is the most widely-recognized quality standard in PCB Assembly, and Bittele is certified to offer both Class 2 and Class 3 assembly under this standard. The majority of commercial and industrial products require no more than Class 2 assembly, and so this is Bittele’s default standard of quality, but Class 3 is also available for some additional cost. From a PCB designer’s point of view, it might be necessary to require Class 3 assembly if the end-product will be placed under very demanding conditions; such demands might stem from harsh environmental conditions or extremely long-term durability requirements.

It is important to keep in mind that PCBs must be designed specifically for Class 3 assembly in order to fully meet the standard, as per IPC-2221. Bittele’s assembly quality can only go so far toward the finished product’s overall quality rating, so PCB designers are advised to consult IPC documentation carefully before beginning the PCB layout process.

3.4.2 – SMTA

The Surface Mount Technology Association (SMTA) is a trade organization whose members include PCB assembly service providers from across the globe. Through this organization, Bittele maintains strong ties with peers in the industry, which ensures a continued awareness of the latest industry trends and practices.

Clients of Bittele Electronics can rest assured that their PCB assembler of choice will always offer the latest in PCB assembly technology and quality assurance standards.
Please Sign up [here](#) to read the full version of DFA guide.