Paving the Way
Imagineering’s Unique Approach to PCB Prototyping and Assembly

Littelfuse
iDesign Software
PCB Amplifier Overview

Interview with Khurrum Dhanji
CEO of Imagineering
“There’s never an order too small, too large, or too complex that we can’t handle.”

...pg. 26

“READY TO LAUNCH

For the launch of the Tiva C Series Connected LaunchPad, TI has partnered with Exosite, mentioned briefly above, to provide easy access to the LaunchPad from the Internet. The LaunchPad takes about 10 minutes to set up and you can immediately interact with it across the Internet and do things like turn an LED on and off remotely from the website and see the reported temperature as well. It can also display approximate geographic location based on the assigned IP address and display a map of all other connected LaunchPad owners if they are active and plugged-in to Exosite. “In addition, it supports a basic game by enabling someone to interface to the Connected LaunchPad through a serial port from a terminal while someone else is playing with them through their browser. It is basically showing how you can interact remotely with this product and a user even if you are across the globe,” Folkens explained.

“START DEVELOPING

The Tiva C Series Connected LaunchPad is shipping now and the price is right; at $19.99 USD, it is less than half the price of other Ethernet-ready kits. The LaunchPad comes complete with quick start and user guides, and ample online support to ensure developers of all backgrounds are well equipped to begin creating cloud-based applications. “We have assembled an online support team to monitor the Engineering-to-Engineering (or E2E) Community,” Folkens said. “Along with this, you also got a free Code Composer Studio Integrated Development Environment, which allows developers to use the full capability. We also support other tool chains like Keil, IAR and Mentor Embedded.

Affordable, versatile, and easy to use, the Tiva Series Connected LaunchPad is well suited for a broad audience and promises to facilitate the expansion of ingenious IoT applications in the cloud. As Folkens concluded, “The target audiences actually are the hobbyists, students and professional engineers. A better way of looking at it is that we are targeting people with innovative ideas and trying to help them get those ideas launched into the cloud.”
The PCB Amplifier: Delivering Superior Performance
Modern Printed Circuits

Power Amps & the Superior PCB Amp

Traditionally, tube power amps were all point-to-point (PTP) wired or of stripboard construction. Components were wired to a long tag board, which was then mounted under a metal chassis.

From the tag board, hook-up wire was used to connect components to tube sockets, speaker binding posts, etc. This method helped enable mass production of amplifiers, radios, and televisions. In fact, some of the most expensive power amps still use this method.

In recent years, the PCB amps in production have actually been worse than tube amps quality-wise; however, the PCB technology is not at fault. PCB amps were produced with low costs in mind, using other low-level components or poor assembly practices. After all, PCBs are used in many high-level industries including defense, aerospace, and medical, where they must perform flawlessly. PCBs perform equally as well in amplifiers as long as the quality of the amplifier is constructed at a high level along with a high-level circuit board.

In comparison, it is very difficult to build a cheap PTP amp on turret, eyelet, and flying leads. High-quality PCB amps are ones that use PCBs for design reasons, using top-shelf components, and not cost-saving reasons.

“Most PCB amps that have been produced ‘have’ been poorer than tube amps, for reasons having nothing to do with the PCBs. Like plastic bobbins in transformers, this does nothing to change the tone. The ‘other’ poor practices that go with a cost-cutting attitude that were introduced at the same time may, but PCBs are unfairly indicted.”

—R.G. Keen, Geofex

A Greater Audience

With the advent of semiconductors and more compact components, the printed circuit board has quickly become the modus operandi for audio gear.

The biggest names in tube amps are moving more and more towards using PCBs in their amps. The tag board mentality has given way to PCB common sense. The need to speed up the operation with fewer faults must include PCBs. If you want to produce audio gear reliably and on a larger scale, it is hard to beat using a PCB.
PCB Amplifier: Advantages & Disadvantages

The PCB amp has several key advantages over traditional stripboard or point-to-point amps. While more care and planning needs to go into the design and fabrication of the PCB to avoid problems like crosstalk and induced capacitance, a high-quality professional PCB designer and PCB manufacturer will be able to make this worry-free. A poorly done PCB amp will also be more costly to repair. Thick boards with heavy copper layers are required for heat reduction and longevity, yet poorly designed boards will eventually degrade over time.

However, there are numerous advantages in using PCB amps, like sound quality. The amps are more tonally consistent and reliable due to the static nature of the placement of the board and components. They have absolutely fixed parasitics, while PTP amps can vary greatly from amp to amp due to the wiring positioning.

For mass production, the ability to produce many PCB amps is easy and consistently reproducible. The PCBs used for amps are quite simple relative to other applications and, therefore, are quite affordable. Be aware, however, that not all PCB amps are created equal. There are high-quality ones and then there are cheap throwaway amps that will eventually fail after extensive use.

The Modern PCB Amp

There is no faster way to build a power amp or preamp than by using a printed circuit board. The modern PCB is double or triple layered, plated through, solder masked, and screen-printed. Populating with components and soldering them in can be a snap. Effective PCB layouts can save time and money and can help reduce PCB assembly errors, cross-talk, and can maximize space.

Seeking out a reliable PCB fabricator with a solid track record and dedication to service and quality will go a long way.

This article was written by San Francisco Circuits, a PCB service provider that specializes in providing quick delivery of high-quality precision PCB fabrication and PCB assembly.
PCB manufacturers and assembly houses have the important responsibility of taking someone’s idea and turning it into a reality. CAD files, including Gerbers, are important in conveying exactly what is wanted, but it’s the personal communication which ensures that what is envisioned is what is made. Different phases of the manufacturing process require different files and information, but you can expedite the overall turn-time by having the necessary information prepared prior to approaching the respective vendor. Not only will this will make it easier and faster to obtain accurate quotes, but complete and clear documentation will also greatly reduce the chance of misunderstandings and errors.
Board manufacturers require, at a bare minimum, the Gerber and drill files for your board. With this information, they can typically fabricate a board that looks like the intended design. However, the manufacturer will have to make assumptions on several things, including the laminate and finish to be used. If there are any additional requirements, such as impedance control or testing, this will not, and cannot, be done without more information. Board thickness and copper weight are also not included in Gerber files and, if a deviation from the standard thickness and weight are desired, they must specifically be requested. There is also a great deal of information that, while included in the Gerbers, can be used to verify that the manufacturer has received all of the necessary information. Number of layers, board dimensions, requested turnaround time, minimum trace width and spacing, and quantity are all items that should be included in either the Gerbers or simple order forms. However, verifying them in a readme file will help catch omissions and errors.

Assembly houses require a few of the same items as board manufacturers. A select few of the Gerber files, specifically the copper, solder paste, and silkscreen files need to be provided to give the assembly house a reference to work from, as well as show solder paste placement. In addition to the Gerber files, assembly houses need a bill of materials (BOM) and the X, Y, rotation, side (XYRS) data. The BOM needs to contain quite a bit of information to be useful. While not all assembly houses require all of this information, it would be helpful for each part to have a reference designator, value, description, package type, part number (manufacturer or retailer), and even a link to the distributor webpage. If you know how parts will be packaged, such as tape and reels, cut tape, or in separate bags, including it will allow the assembly house to determine whether or not they can use a pick- and-place machine or if assembly will be done by hand.

A readme file is a common way for a designer to convey information to both PCB manufacturers and assembly houses. The readme file usually is a simple list of requested features but can also contain a description of the end product, which gives a better idea of the overall scope of the project. If a description is going to be provided, it is helpful to be clear and concise. Sometimes, instead of a readme file, designers will include text on one of the layers in the Gerber file. This is a good way of making sure that the readme information is not overlooked, however it does limit the amount of information that can be provided. It is easiest to have a standard readme file that can be updated with each new design, but this runs the risk of information not being updated.

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
<th>G</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Project BOM</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Name</td>
<td>Reference Designator</td>
<td>Notes</td>
<td>Package</td>
<td>Value</td>
<td>Quantity</td>
</tr>
<tr>
<td>3</td>
<td>Header</td>
<td>U55</td>
<td>12pin</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>USB Port</td>
<td>JP1</td>
<td>Micro-B</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>Power Regulation</td>
<td>U51</td>
<td>3.3V</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>Changing Circuit</td>
<td>IC1</td>
<td>Li-Po specific</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>LED</td>
<td>U53</td>
<td>RGB</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>Switch</td>
<td>U54</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>Resistor</td>
<td>R4, R5</td>
<td>RES 100 OHM 1/8W 5% 0603 (x2)</td>
<td>0603</td>
<td>100 ohms</td>
<td>1</td>
</tr>
<tr>
<td>10</td>
<td>Resistor</td>
<td>R2, R6, R8, R10</td>
<td>RES 10K OHM 1/16W 1% 0402 SMD (x4)</td>
<td>0402</td>
<td>10K ohms</td>
<td>4</td>
</tr>
<tr>
<td>11</td>
<td>Resistor</td>
<td>R7, R9</td>
<td>RES 20K OHM 1/16W 1% 0402 SMD (x2)</td>
<td>0402</td>
<td>20K ohms</td>
<td>2</td>
</tr>
<tr>
<td>12</td>
<td>Resistor</td>
<td>R3</td>
<td>RES 560 OHM 1/8W 5% 0603 SMD (x1)</td>
<td>0603</td>
<td>560 ohms</td>
<td>1</td>
</tr>
<tr>
<td>13</td>
<td>Capacitor</td>
<td>C6</td>
<td>CAP CER 4.7UF 6.3V 20X X5R 0402 (x1)</td>
<td>0402</td>
<td>4.7uF</td>
<td>1</td>
</tr>
<tr>
<td>14</td>
<td>Capacitor</td>
<td>C3, C7</td>
<td>CAP CER 2.2UF 10V 20% X5R 0402 (x2)</td>
<td>0402</td>
<td>2.2uF</td>
<td>2</td>
</tr>
<tr>
<td>15</td>
<td>Capacitor</td>
<td>C1, C2</td>
<td>CAP CER 1UF 6.3V 10% X5R 0402 (x2)</td>
<td>0402</td>
<td>1uF</td>
<td>2</td>
</tr>
<tr>
<td>16</td>
<td>Capacitor</td>
<td>C6</td>
<td>CAP CER 47UF 6.3V 20X X5R 0402 (x1)</td>
<td>0402</td>
<td>47uF</td>
<td>1</td>
</tr>
</tbody>
</table>
Modern Printed Circuits

Advanced Assembly was founded to help engineers assemble their prototype and low-volume PCB orders. Based on years of experience within the printed circuit board industry, Advanced Assembly developed a proprietary system to deliver consistent, machine surface mount technology (SMT) assembly in 1-5 days. It’s our only focus. We take the hassle out of PCB assembly and make it easy, so you can spend time on other aspects of your design.

"If possible, have a third party within your organization review the documentation to see if it is easy to understand. What seems clear and straightforward to you may not be to others."

20100 E. 32nd Pkwy #225 | Aurora, CO 80011 | www.aapcb.com | 1-800-838-5650

You need 35 PCBs assembled, all SMTs machine-placed, with 0402s, leadless QFNs and a 484 pin BGA, with parts and boards too, AND you want them in 3 days? Sure.

(There’s a cheetah in our logo for a reason.)

Engineers save time and money by turning to us for their low-volume PCB assembly needs. Working with us is just like having an assembly expert sitting beside you. We will work with you throughout the assembly process to deliver top-quality boards you can trust.

To get a same-day quote with pricing you won’t believe, call 1-800-390-5542 or visit www.aapcbassembly.com.

Go Ahead. Test Us.
Functionality is usually the main focus the first time someone designs a circuit board. With time and experience, designers are able to focus on improving their designs to not simply be functional but also to increase performance. A seasoned designer can make the board robust and cheap while still easy to manufacture and assemble. Particularly, those only vaguely aware of the requirements for mass methodologies of soldering, such as reflow or wave soldering, will create boards that can only be hand soldered or will require extensive manual rework. While hand soldering can be more forgiving of poor layout, this is at the expense of increased time and frustration. Every board designed is unique, will have different requirements, and will require different trade-offs on the design, but there are basic guidelines of placement that will go a long way toward creating a professional board that can be easily moved from the design table to reality.
An overall strategy that will help considerably, no matter the soldering method, is grouping components into functional blocks. There are typically different portions of a circuit on the board, a portion dedicated to power, another portion to communications, another portion to digital logic, and so on. When first approaching the board, it is important to group these together as much as possible. Not only does it help keep things organized and easier to track, but it also decreases the average track length and provides a good place from which to start. Once all of the parts have been placed on the board and grouped together, you can get an idea of about how much space you will need and can make a good approximation of the board size. Of course, the board size can change at any time, but it is helpful to have some loose constraints as it is natural to expand to fill the space provided.

If you anticipate using wave soldering, keep in mind the direction the board will move through the wave. The largest concern with wave soldering is to eliminate shadowing. As the solder flows across a board, shadows are created when a small part of the board behind the component doesn’t receive solder. While this shadow is small for small items such as the leads of through-hole resistors, larger, surface-mounted devices in particular create a significant shadow. To avoid the shadowing effect, you use a combination of staggering pads so that one pad is not directly behind another. Another option is to put enough space between the components so that the solder comes back together in time for the next part. Spacing between components needs to be a minimum of 25 mil but could be up to 100 mil if it is a larger pad. For reflow spacing, a general rule of thumb is that the parts can be approximately one half the distance required for wave soldering. Reflow soldering has a different type of shadowing that may be a concern, and that is the shadow cast by taller components. If the infrared source of the reflow oven is not directly over a part, but at an angle, a taller part can leave the smaller part in a shadow. Most reflow ovens provide convection heating to even out the temperature profile, but there are still warm and cool pockets. Even in conveyors where the entire board is moving, thus changing the angle of the infrared source and the flow of air, the time in the shadow could be sufficient to ensure that the solder paste of the smaller components do not actually reflow. If two tall components are placed next to each other with connections between them, those connections are highly unlikely to receive the energy they need for the solder to flow. PCB assembly shops with vapor reflow capabilities avoid this issue by providing uniform, consistent heat across the entire board.

There are certain size restrictions that need to be considered for assembly. If a main consideration for the design is size or if the board deals with high frequency signals, there may be the temptation, or even the requirement, to use passives in the English 0201 or 01005 package sizes. However, not all assembly houses can mount components that small which can result in a premium for those services. If the board is going to be mixed technology and will be wave soldered, the restrictions are even more acute. 0805 and smaller surface-mount devices should not be wave soldered as the surface area underneath the component is not sufficiently large for placing the glue properly.

Aside size restrictions, care must be taken in terms of the thermal mass of adjoining components. If one portion of the board contains many large components while another portion contains only small passives, there will be a thermal mass imbalance across the board. For the solder and joints to reach the necessary temperature to ensure a good bond, the side of the board with a larger thermal mass requires a higher temperature than the side of the board with the smaller thermal mass. However, both sides are subjected to the same conditions. Either the larger components won’t be well soldered or the smaller components will be subjected to damaging levels of heat. By intermixing the two sizes as much as possible while still maintaining the functional blocks and avoiding shadowing, thermal load is spread more evenly across the board.

**Consider Component Size**

"For reflow spacing, a general rule of thumb is that the parts can be approximately one half the distance required for wave soldering."
Align Polarized Components in the Same Direction

To reduce the opportunity for mistakes, it is also a good practice to align all polarized components in the same direction. All polarized capacitors, diodes, or LEDs should have their anode in one direction, cathode in the other. This information can even be silk screened or placed in the copper layer to help the assembler, whether it be you or someone else, know how to place the parts. If a general direction is not provided, then properly labeling every polarized component is important for the same reasons. If the boards will be hand soldered, it will reduce the risk of human error on every board. Even for machine populated boards, if the pick-and-place is programmed incorrectly due to unclear polarization, it will be a very costly mistake.

Traces and Pads

The setup of the traces and pads can significantly affect the ease of assembly and rework required for your board. From skewed to tombstoned components, careful trace and pad design can practically eliminate these problems.

For reflow soldering surface mount devices, the key ideas to consider when laying these out are knowing what pads will come to temperature faster than others and knowing what direction the solder will go when reflowed. These factors become more important as the component becomes smaller, and therefore lighter. Very small passives, such as 0805 components and below, are much more susceptible to becoming skewed or tombstoning than their larger counterparts.

Avoid Tombstoning By Ensuring Consistent Thermal Loads

Tombstoning is a result of one side of a two-terminal device reflowing faster than the other. As the solder reflows and moves up to cover the terminal, it exerts force, pulling down and toward the pad. If that same force is not being exerted on the other side, as it has not reflowed yet, then the component will be pulled out of the solder paste on the other side and stand straight up, dipped in the refloved solder. This vertical rectangle is reminiscent of a tombstone, which is why it is called tombstoning. To reduce the chance of this happening, both sides of the device need to have similar thermal loads. This not only includes the size of the pads, but also the traces leading up to them. Even under the solder mask, a large trace requires more energy to be heated than a smaller trace. The easiest solution is to reduce the size of the entire trace. However, if that’s not ideal or possible, then simply necking down the trace for the last 10 mil of the run works as well. If it is a high-current trace and necking is not possible, then providing multiple smaller traces to the pad is preferable to one large trace. Vias absorb a considerable amount of energy as well, so if there is a via directly next to a small passive component, then the via should either be moved farther away or the load should be balanced on the other side of the passive. Particularly in high-frequency designs, the closer the via is to the device, the better.

Avoid Tombstoning

Control Solder Flow with Traces

For very low-cost boards without a solder mask, the direction a trace leaves the pad is very important. The solder mask typically stops the solder from flowing anywhere except on the pad, however, without that protection, the solder can and will flow as far down a trace as possible. This flow down the trace is generally not a problem in terms of solder deficiency, however, the way it pulls components may skew them off the pad. If the forces on the component are not controlled, it can twist one end of the component so much that the other end comes off the other pad. This can also be controlled with careful design. In essence, the traces leading to the pad need to work in tandem. The simplest method is to have the traces leave the pad directly outward or inward. If this isn’t possible, then have the traces

“If one portion of the board contains many large components while another portion contains only small passives, there will be a thermal mass imbalance across the board.”
leave the pad in similar directions, which may pull the component in a certain direction, but should be straight and less likely to pull one end off a pad.

Create Thieving Pads to Reduce Bridging

For wave soldering surface-mount components, these two issues are not a concern as the components are glued to the board. The bigger concern for wave soldering besides shadowing is reducing bridging on high pin-count devices. This is done by creating thieving pads for your devices. As a long row of leads have the solder wave flow over them, the leads pull the wave from one lead to the next, depositing the correct amount of solder on each lead. However, at the end of the leads, the solder wave does not have any further leads to pull the wave along. The solder then creates a ball on the last lead, frequently creating a solder bridge between the last two leads. This is fixed by creating a larger pad on the trailing edge of the device. This larger pad has the surface area needed to wick the excess solder off the lead, pulling it away from other leads.

The Board Itself

To ease assembly, the board itself needs to be taken into consideration. While boards can be made in practically any shape imaginable, odd shapes make automation very difficult.

Boards Should Have At Least Two Parallel Edges

To make sure that your boards can be gripped properly, you will need to provide at least two parallel edges. If the board needs to be circular, or some other odd shape, then the printed circuit board manufacturer can leave the boards connected to their larger panel while providing either V-grooves or rat bites so that the boards can be separated after assembly is complete. Do not forget to include overhanging parts, such as connectors, when deciding where the boards will be gripped. If there is an overhanging USB port or audio connector, it will interfere with the grip, and special grips will be required, adding another potentially expensive step to the assembly process.

Include Tooling Holes in Design

Tooling holes can be used throughout the manufacturing and assembly process. By providing tooling holes, boards can be stacked on top of each other, to reduce the amount of drills a machine must make. During assembly, these tooling holes can be used as either mounting holes while in the pick and place for both placing components and applying solder paste. Tooling holes can even be used as fiducials. Well-designed tooling holes can multitask to the point of assisting in electrical testing and in the mounting of a product’s enclosure. These tooling holes can be placed in the supporting portions of a panel or in each board individually. Tooling holes are standardized in size and it would be beneficial to speak to your board assembly house as well as the engineers designing the enclosure to ensure compatibility with all stages of the assembly. Due to variances in the plating process, tooling holes are never plated. Minor variations in the size of the tooling holes could cause a myriad of issues, so in the read-me file you send your circuit board manufacturer, it is best to give explicit instructions to not plate certain holes.

The difference between an adequately designed board and a well designed board can be subtle yet substantial. While there is no strict method that will yield perfect results every time, knowing the tools at your disposal as well as the benefits and drawbacks of different methodologies will allow you to make well-informed decisions that will boost your yield, increase repeatability, decrease costs, and create an efficient design. With time, the tips and tricks introduced here will become a natural part of the process, and you will find particular methods suited to your design style, methods that will result in higher quality circuit boards.

“Well-designed tooling holes can multitask to the point of assisting in electrical testing and in the mounting of a product’s enclosure.”
HARDWARE DESIGN MADE EASY.

PCBWeb Designer is a free CAD desktop application for designing and manufacturing electronics hardware. The tool supports schematic capture and board layout, including integrated "click-to-order" manufacturing.

www.PCBWeb.com
Imagineering, an award-winning, family-owned PCB manufacturing company, has over 30 years of experience in the industry, with a customer list that includes technology leaders such as Whirlpool, Siemens, Motorola, Honeywell, and GE. The company is known for their same-day prototype and aggressive quick-turn offering, and is proud to be one of the industry pioneers in creating a truly Web-based approach to quoting and ordering PCBs, components, and assembly on a global scale.

IEEE recently spoke with Khurrum Dhanji about the company’s origins and some of the complex new technologies it is targeting.
What is your position at Imagineering?
I am currently the acting CEO of the company as of March of this year. I am now in charge of spearheading the company in terms of market strategy, new product development, future growth, and sustainability. It is a family-owned business and was started in 1985 by my father, Ramzan Dhanji, and in 2000, my mother, Parvin Dhanji, took over and really brought the company to where it is at in terms of management, marketing and vision. Now it is my time to show what I am capable of and take the company to the next level.

What inspired your parents to start Imagineering?
In 1985, my parents founded Imagineering with the goal of providing employment and the opportunity for their extended family to share in the American dream. Imagineering began as a photo-plotting business, providing photo plots to a number of companies in the printed circuit board industry. My father was one of the most successful photo plotters, and Imagineering was one of the first companies to have a McDonnell Douglas Fire 9000 to test. In 1998, we acquired a second plotter, the 9001.

Why would engineers choose your services as opposed to your competitor’s?
Imagineering is a value-added printed circuit board supplier as well as an electronics contract manufacturer. We do all of the legwork for the customer in terms of developing relationships with suppliers and finding the best suppliers for what the customer needs. We have customers that come in and ask for simple, two-layer jobs, and some customers that come in and ask for 18 to 24 layers. As a result, we have developed unique relationships with each of our supplier partners and we provide the customer with the best PCB for their price point. For the customer, it is difficult to source the best supplier for any set job, whereas Imagineering encompasses the whole gambit of PCB manufacturing in terms of multiple layer counts and assembly.

Since we are strategically linked with our suppliers, we are able to supply our customers with virtually any type of PCB they want. We are very proactive about this—if there is a new technology on the verge of coming out, we will have already done the legwork in finding suppliers and manufacturers that can provide that particular part or service to our customers.

What types of customers comprise your customer base?
We cater to basically every industry that needs circuit boards. Because of our capacity and of our range of capabilities, we are not limiting ourselves to just one type of customer in a particular industry—we try and encompass the whole gambit in terms of our PCBs and the customers we serve. There is never an order too small, too large, or too complex that we can’t handle. We have linked ourselves so well with different suppliers that we truly can handle anything thrown at us, from two-layer, to complex HDI boards.

“If there is a new technology on the verge of coming out, we will have already done the legwork in finding suppliers and manufacturers that can provide that particular part or service to our customers.”
As new CEO of Imagineering, is it your goal to steer the company towards more complex technology?

Absolutely. I think the future of PCBs are in the high-density interconnects (HDI)—things become smaller and more complex and the real estate available on the board becomes smaller as the devices get smaller. Upcoming wearable technologies will create a much higher need for HDI technology, which we have already been working on. We have been working very closely with our suppliers on furthering this technology so that we can offer some of the most complex PCBs to our customers.

We have recently purchased some new machines that will allow us to carve out a niche in the assembly industry as well. We are providing our customers high-mix, low-volume assembly in a very accelerated turnaround time. We are able to do jobs in as little as one day, and most small productions in as quick as three to five days—whereas other assembly houses take one or two weeks to just send you a quote.

To further that niche, we also developed a proprietary online quoting engine that allows our customers to get not only boards, but also labor and parts, all in one place at one time. All the customer has to do is upload their materials, fill out some information about the Gerber specification, and within minutes, a quote is sent to them for parts boards and labor. It has really helped change the industry in terms of downtime that a customer has to wait to get their quote before they start the project. When we first starting doing this, one of the biggest pitfalls we found was that a lot of our suppliers that we wanted to work with would take three to five days to get their quote out. To us, that is unacceptable because the customer typically has a deadline to figure out if their project is feasible and within budget, and they cannot do that until they have figured out the cost of parts and the cost of labor and the cost of the board. A lot of the time, the customers themselves search on online components vendor sites and send the labor quotation out to another supplier and then send the board quotation out to another supplier, and that is too much to deal with, and is not an efficient way to determine product feasibility. That is why we developed the strategy of providing a one-stop solution for all of our customer’s needs—from design all the way down to assembly. [Imagineering] developed a proprietary online quoting engine that allows our customers to get not only boards, but also labor and parts, all in one place at one time.
Imagineering, Inc. was the winner of the 2014 Illinois Family Entrepreneurship Award. The company takes pride in accepting this award on behalf of its customers and vendor partners. For the last 29 years, Imagineering has treated its employees as part of its extended family and they have taken this philosophy to build strong relationship with its customers and vendor partners.

For 21 years, the Family Business Center at Loyola University Chicago Quinlan School of Business, has presented the Illinois Family Business of the Year Awards. The Awards honor and recognize family businesses who demonstrate the following qualities:

- A strong commitment to both business and family
- Positive family/business linkage
- Multi-generational family business involvement
- Contributions to industry and community
- Innovative business practices and strategies

Imagineering, Inc. was the winner of the 2014 Illinois Family Entrepreneurship Award. The company takes pride in accepting this award on behalf of its customers and vendor partners. For the last 29 years, Imagineering has treated its employees as part of its extended family and they have taken this philosophy to build strong relationship with its customers and vendor partners.

For 21 years, the Family Business Center at Loyola University Chicago Quinlan School of Business, has presented the Illinois Family Business of the Year Awards. The Awards honor and recognize family businesses who demonstrate the following qualities:

- A strong commitment to both business and family
- Positive family/business linkage
- Multi-generational family business involvement
- Contributions to industry and community
- Innovative business practices and strategies
In April of last year, Littelfuse introduced the iDesign™ online simulation and product selection tool. The tool was the first of its kind in the industry, allowing engineers to virtualize their fuse selections before buying them. Littelfuse designed the tool with flexibility in mind, which will allow them to add additional circuit protection devices in the future. Just last month, Littelfuse announced an addition to iDesign: an ESD selection tool that helps circuit designers choose from the company’s growing line of TVS Diode Arrays. Chad Marak, director of semiconductor business development at Littelfuse, gave EEWeb an exclusive look at the new capabilities iDesign.
For such an important aspect of circuit design, it is surprising that a circuit protection selection tool had not yet been designed. The iDesign platform was developed in response to requests from Littelfuse customers. It offers a fast, intuitive way to identify the best component for an application, find parts documentation, and order part samples for prototyping, all in one convenient package. The tool walks users through the component selection process, quickly winnowing the available options based on the application parameters input, including parameters of the IC the designer is trying to protect. This new addition to iDesign specifically targets ESD suppression and what devices (or devices) are the most optimal for protecting a given IC from very fast and often damaging voltage transients, such as electrostatic discharge (ESD) and lightning-induced surges.

Marak believes that this online simulator offers a huge advantage over other circuit protection providers: “We are not just quantifying our products with line items in a data sheet—we are actually modeling our devices in the lab against real-world transients to show their performance.” While this seems like a complicated process, the Littelfuse team made sure that iDesign is easy to use while accurately simulating the testing process before the engineer commits to purchasing components. The iDesign web-based tool is comprised of four intuitive pages that have entry fields for the parameters needed to determine the best circuit protection devices for the design.

**System Settings**

The first tab “System Settings,” allows the user to enter their design parameters such as required ESD level, maximum DC operating voltage, etc. Based on the entered information, the ESD tool narrows down the product selection to the TVS diodes that fit the application, filtering them in real-time. This section also displays the schematic of the full design. “Typically, most of the design time is spent on the selection of or functionality of the ASIC (Application Specific Integrated Circuit),” Marak explained “This makes circuit protection lower on the list of priorities, which provides more evidence as to why this tool was needed.”

**Device Settings**

After entering the system parameters, the tool then guides users to the “Device Settings” tab, which shows the appropriate Littelfuse TVS diode arrays based on the parameters. This section also allows the user to enter the application (e.g., USB 3.0, HDMI, etc.) they are designing for so that iDesign can provide a more accurate recommendation of circuit protection devices. There are also more advanced parameters that the user can select from to narrow in on specific circuit protection devices if the application feature is not used. These include channels of protection required, maximum capacitance, maximum leakage, etc.
Device Selection
The user then moves on to the “Device Selection” tab, where they select from the narrowed-down list of devices prior to simulation and analysis. The user can run a simulation for up to three different TVS diode arrays at any given time. This section also provides overall product specifications, device packaging, electrical schematics, and access to the datasheets for the selected devices.

Analysis
The “Analysis” tab displays the entire circuit design with the circuit protection components that the user has selected to test. From here, the user can click “Simulate” after which the tool provides a live simulation based upon the selected circuit protection devices. The graph shows the potential robustness of the components versus a view of how the design will likely perform without circuit protection.

After selecting the TVS diode based on the simulation, the user clicks the “Finish” button to get pricing or to order a free sample from Littelfuse. System printouts are also available. With just four simple pages of parameters and testing, the Littelfuse iDesign platform brings circuit protection to the forefront of the design process.

“We are not just quantifying our products with line items in a data sheet—we are actually modeling our devices in the lab against real-world transients to show their performance.”