

Contents

Introduction
Chapter 1
Chapter 2
Chapter 3
Chapter 4
Chapter 5
Appendix

The Little Book of Altium

Getting to Know Us

The Little Book of Altium was created for Altium employees by Altium employees to describe who we are, what we do, and who we serve.

Over the next pages, you'll learn more about **Altium's** unique history, as well as our products and services, our customers and competitors, and about all the different ways **Altium** empowers the electronics industry.

Many thanks to everyone who helped make this possible, especially those who shared incredible stories about who we've been, who we are, and what we're becoming.

Like our industry, **Altium** is always evolving—and so, *The Little Book of Altium* will be updated routinely.

If you have any questions, comments, suggestions, or corrections, please drop us a line at brand@altium.com.

The Altium Brand Marketing Team Fall 2020



A Different Kind of Company

You can't work at **Altium** without noticing how different we are, especially compared to companies of our size and stature.

We're a global enterprise with the heart and soul of a startup—and that's something we've had since the very beginning. Changing the face of electronic design is in **Altium's** DNA; it's who we are and what we've stood for, for over 35 years: transforming the electronic industry.

Our history has shaped us and defined us and, unlike other companies in our sphere, we've never lost our upstart spirit.

This chapter walks you through the official **Altium** story, from the birth of our industry to the nascence of our company. And no history of any company in the electronic design industry can begin without the printed circuit board.







How the World Got Wired

Electronic devices are so central to everyday life that it's easy to assume they've always been there. And none of them would have been possible without printed circuit boards (PCBs).

PCBs can be found in every electronic device today, but they've only been around for the past century or so. The earliest boards date back to the early 1900s; in 1925, they were officially patented as insulated surfaces containing integrated electrical pathways.

Early boards were strange, bulky designs made out of anything, be it Bakelite or wood, with large sockets and components that always needed replacing. Fully operational boards weren't produced until 1943, followed in 1947 by the first double-sided PCBs featuring plated holes. Over the ensuing decades, materials became more standardized, boards got much smaller and production increased to meet the demands of an increasingly wired public.

Oddly enough, it took until the 1980s for the PCB assembly process to move from a complex manual system to one that was fully automated, harnessing a new software technology called **Electronic Design Automation (EDA)**.

It took until 1985 for **EDA software** to become accessible to everyone—thanks to a company that would eventually be called **Altium**.



The Dawn of Electronic Design Automation

Before **EDA**, PCBs had to be manually designed. Each layer of a circuit board had to be meticulously assembled; integrated circuits were designed by hand, and schematics were hand-drawn and delivered to component manufacturers as blueprints.

By the mid-1970s, automation started to become more commonplace, with multiple tools for different functions, like place and route (P&R). Larger companies began developing their own automation software internally, especially bigger semiconductor manufacturers.

Engineers and executives from these larger firms moved on in the early 1980s to focus on creating EDA software. By 1981, EDA was a burgeoning industry, featuring new companies like Mentor Graphics, Daisy Systems, and Valid Logic Systems. Mentor Graphics eventually acquired Daisy Systems; Cadence Design Systems acquired Valid Logic Systems in the early 1990s.

In 1982, a new PCB design software Computer Assisted Design (CAD) tool called **P-CAD** was released by a small Silicon Valley startup called **Personal CAD Systems**. By 1985, **P-CAD** was the most installed PCB tool of its kind; its importance to electronic design—and eventually **Altium**—cannot be understated.

CAD tools and EDA software made the process of making PCBs much easier, but programs were extremely expensive and hard to come by unless you happened to be a big company with high-end engineering workstations. The high cost of hardware and software effectively shut out entrepreneurs and individual designers; without software that could be used on PCs, they had no choice but to continue to create printed circuit boards the old-fashioned way.

An alternative that was accessible to everyone was sorely needed.

Enter the Upstart

As EDA software became more commonplace, the need for affordable, reliable tools grew exponentially. **HST Technologies**, a startup based in the unlikely locale of Hobart, Tasmania, emerged in 1985 to make PCB design software accessible to everyone.

The company began inauspiciously enough, without any venture capital or outside investment, and one bold idea: bringing electronic design to the PC platform.

Within just a few years, the bootstrapped business became world-renowned for its affordable PC-friendly PCB software, as well as a few major firsts that effectively changed the course of the electronic design industry forever.

EDA for Everyone

The first major release from HST was a DOS-based PCB layout and design software tool called **Protel**. The response to the vanguard software was nothing short of explosive, causing a near seismic shift in the electronic design industry.

News of the DOS-based tool spread quickly beyond the Australian continent; within less than a year, **Protel** formed its first distribution partnership with a San Diego, CA-based company called **ACCEL Technologies**.

The ACCEL partnership brought Protel's flagship product to North America. Christened in the US, Canada, and Mexico as Tango PCB, the software's popularity spread like wildfire.

Back home in Tasmania, the company changed their name from HST Technologies to Protel. Within just a few years, official Protel-branded software was available worldwide.

The Protel Years

By 1991, **Protel** was so successful, with footprints in North America, Europe, Asia, and Australia, that they relocated their headquarters to San Jose, CA, keeping their main R&D center in Shanghai.

The first US office was just as inauspicious as **Protel's** origins, occupying a nondescript strip mall, staffed by just seven people. Halfway across the globe, at **Protel's** R&D offices in Shanghai, the team was hard at work on three new products: **Protel Schematic**, **Autotrax**, and **Easytrax**.

Protel Schematic was a DOS-based circuit diagram editor; Autotrax was one of the first-ever professional PCB computer-assisted design (CAD) programs available for the PC platform. Easytrax was a stripped-down version of Autotrax, ideally suited for the more novice designer.

In 1991, Autotrax became the foundation for Protel's Advanced Schematic/PCB 1.0 for Windows—the first-ever Microsoft Windowsbased PCB design software tool. This was an exceptionally bold move for the industry, as the fledgling Microsoft program was in limited use, particularly in the electronic design space. As Microsoft Windows became the most widely-used operating system in the world, Protel's value skyrocketed. The tool was eventually renamed Protel 95, designed to work seamlessly with Windows 95.

For the remainder of the decade, the tool went through several iterations, released alongside each major Microsoft Windows release, including **Protel 98**, **Protel 99**, and **Protel 99 SE** in 2000.

In 2000, Protel also formally acquired ACCEL Technologies, setting the stage for a whole new decade—and a new company called **Altium**.



Enter Altium

At the dawn of the new millennium, **Protel** was poised for greatness. With multiple successful releases under their belts, Protel successfully completed an IPO in 1999 and went public, earning a home on the Australian Stock Exchange, where it remains to this day (ASX:ALU).

2000 was a very interesting and exciting year for **Protel**. The ACCEL acquisition did more for the company than just expand its footprint; due to a combination of interesting circumstances, it also made P-CAD a **Protel** proprietary technology.

Some years earlier, P-CAD was sold off, winding up eventually in the hands of IBM, one of the major companies who created their own EDA software tools back in the 1970s. IBM merged the P-CAD acquisition under a new imprint called Altium; IBM wound up selling it to ACCEL Technologies after reevaluating their space in the EDA sphere.

As **Protel** continued to expand its global footprint into Europe and Asia, the company encountered multiple businesses that were also called **Protel**, including hospitality software and telecommunications companies.

It became clear that a new name was needed—and thankfully, **Protel's** leadership team didn't have to look beyond their assets from the recent ACCEL acquisition. And so, in 2001, **Protel** officially became **Altium**.

As **Altium's** product line expanded, so did **Altium's** acquisitions and **Altium's** goals. **Altium** wanted to move beyond just EDA software to become more of an end-to-end solution, harnessing newer technologies at the time like field-programmable gate array (FGPA), a hybrid model that harnessed both software and hardware.

In 2001, **Altium** acquired **TASKING**, a global leader in embedded system compiler technology that, at the time, was a major leap into the then-booming FPGA market.

The acquisition was key to the creation of the NanoBoard, Altium's first and only foray into hardware development. The configurable development platform empowered FGPA, as well as a "soft design" process that used software to compile, debug, and update microcontrollers and microprocessors. The Altium NanoBoard went through three iterations from 2004-2009, but was ultimately shelved as the industry shifted away from FGPA to embedded software.

In 2001, **Altium** also bought the largest EDA software distributor in Europe, Hoschar AG in Karlsruhe, Germany that became home to **Altium's Customer Support** operations, as well as one of our major R&D centers.



The Birth of Altium Designer

Altium continued to make Protel-branded products, including Protel DXP, released in 2003. This comprehensive tool was a fully integrated suite for designing PCBs, with all of the functionality necessary to take boards from schematics into production using just one program.

The program became the foundation for Altium Designer, released as Altium Designer 6.0 in 2005. By version 6.8, Altium Designer was already changing the industry, featuring the first-ever Native 3D™ visualization and the ability to clearance check PCBs directly within the editor.

Soon, **Altium Designer** would achieve levels of popularity unseen anywhere in the EDA industry, becoming the #1 choice amongst PCB designers worldwide over just a few iterations. Yet, **Altium Designer's** price point remained within easy reach for PCB design professionals, staying true to **Protel's** goal of making EDA accessible to everyone.

Onward and Upward

Altium Designer catapulted the company's success as word of mouth grew about the program's many capabilities. Subsequent iterations fine-tuned Altium Designer's capabilities and expanded upon them, from the addition of Rigid-Flex design to library management features.

Altium's acquisitions helped guide the expansion of the company's product line, including the 2010 acquisition of Morfik Technology, an IoT development firm headed up by Altium's CEO, Aram Mirkazemi.

Other notable acquisitions which both diversified and expanded the **Altium** portfolio included **Octopart**, the leading search engine for electronic and industrial parts; **Ciiva**, a cloud-based BOM and electronic component management system, and **Gumstix**. We'll be examining these acquisitions in greater detail in the next chapter.

Major advancements in over dozens of iterations of **Altium Designer**, plus the introduction of **Altium 365**, the world's only cloud platform for printed circuit board design and realization, has **Altium** poised to completely transform the industry like never before.

And through it all, the **Altium** spirit remains, pushing the limits of electronic design into a new era, while keeping EDA within everyone's reach.

In the next chapter, we'll focus on Altium's core branded products and the Altium House of Brands—and how they work synergistically to ensure Altium's place as both a pioneer and a leader of the electronics industry.



The Many Faces of Altium: The Altium Branded House and the Altium House of Brands

Altium has earned its place at the top of the electronic design industry with Altium Designer, but there's much more to us than just one software program—and software in general.

The products and services we offer cover a wide range of needs, from the entrepreneur to the enterprise, as well as a wealth of resources for the benefit of the electronic design community. **Altium's** devotion to the community is reflected in our user-centric tools and open-access applications, as well as our extensive educational programs (which we'll explore at length in the last chapter).







The Branded House and the House of Brands

Altium's product lines encompass both a Branded House and a House of Brands.

A Branded House maintains the focus on a single, well-known and consistent brand. At **Altium**, our branded products are the core software tools and platforms we offer; these are distinguished as branded because they carry the Altium name (e.g., **Altium Designer**, **Altium Concord Pro**).

A House of Brands, on the other hand, is home to numerous brands, each independent of one another, and each with its own audience, marketing, look and feel. This is commonplace across multiple industries, most notably in the entertainment industry, where large, parent companies like Warner Brothers are home to autonomous studios like New Line Cinema and DC Films.

Within the Altium Branded House and House of Brands ecosystem, there are three discrete business units: Boards and Systems, Embedded Systems, and Modular Board Design. Each respective product and service exists to give professionals and aficionados all of the tools they need to make their electric dreams come true.



The Altium Branded House

Altium branded products include software systems, platforms, specialized search engines, and APIs. Most of the **Branded House** products can be used together.

These products fall under the **Boards and Systems** business unit; we'll revisit them again in the next chapter, just in a very different way.



Altium Designer® is our flagship software product. Representing decades of innovation and development, Altium Designer empowers a truly unified design environment, uniting teams worldwide with every aspect of the PCB design process. With the addition of Altium 365, users can choose between accessing the program online and on-premises.



Altium 365® is the world's only cloud platform for printed circuit board design and realization. It takes Altium Designer and Altium Concord Pro to the next dimension by creating seamless collaboration points across the PCB development process, making it the most connected design experience in the industry.



Altium Concord Pro® works together with Altium Designer to create a comprehensive system that gives designers ability to source parts, easily manage and share all of their design and parts data, and co-design with mechanical engineers. Altium Concord Pro provides organizations with complete, out-of-the-box infrastructure of component and design data management.



Altium CircuitMaker Pro® is a straightforward schematic capture and project management tool combined with a powerful PCB design engine that supports 3D editing. The program is an easier way to design PCBs for those just getting started, as well as ideal for more casual users.



A free version of this tool, with routing, hierarchical schematic entry, autorouting, and Native 3D™ called **Altium CircuitMaker®** is also available, ensuring EDA is accessible to all.



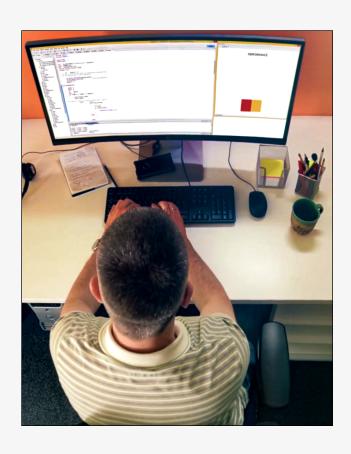
Altium NEXUS® is a unique solution for larger firms that acts as a hub for the entire PCB design process with full ECAD/MCAD and PLM integration. The comprehensive system connects entire teams, from PCB designers and mechanical engineers to Project Managers and Master Librarians, engaging everyone in the design and release process.

Octopart

Octopart® is a powerful search engine that makes it fast and easy to compare millions of components, find technical information, quickly make component selection decisions, and purchase parts from thousands of distributors. Data is normalized to ensure that searches are easy and efficient; the system is also EDA-agnostic, so almost anyone can use the tool, regardless of whether or not they use Altium Designer, Altium 365, or any other Altium-branded product.



Octopart can be used as a search engine directly or via an API token that customers can add to their own ecommerce websites. The API token powers searches on third-party platforms; customers using the API token display a "Powered by Octopart" badge on their websites.



The Altium House of Brands

Altium has always been more than just our products—since the early days as **Protel**, we've been on a mission to advance the field of electronic design.

In the first chapter, some of **Altium's** acquisitions over the years were highlighted, including **Morfik** and **Ciiva**, a 2015 addition to the **Altium** family. We'll explore **Ciiva** a little more at the end of this chapter.

Other acquisitions have become central to the Altium House of Brands, which encompasses both the Embedded Systems and Modular Board Design business units.







Modular Board Design

Altium's Modular Board Design business unit is the youngest part of the company, encompassing Upverter, Gumstix, and Gepetto. The business unit positions the company as a real one-stop-shop for the electronic design community, empowering aspiring and established designers alike to create and produce their own custom boards.

Upverter, a 2017 acquisition, was originally incubated via Y Combinator's startup program, the platform that launched some of the most successful startups of the 2010s, including Stripe, Airbnb, Twitch, and Dropbox.

The open-access tool employs a web interface that makes it easy to design, share, edit, and review schematics and PCB designs, as well as generate a bill of materials, Gerber files, and 3D renderings. Upverter seamlessly supports Altium Designer file formats, as well as Autodesk Eagle and Allegro OrCad.

Altium acquired Upverter in August 2017, originally planning to integrate the tool into Altium CircuitMaker. An early 2019 acquisition, Gumstix, changed the tool's course; Upverter became more of an educational tool, supporting high school STEM programs.

Gumstix expanded the Altium universe into manufacturing far beyond the existing capabilities of the Altium Branded House and House of Brands. Gumstix offers inventors, creators, and innovators the fastest way to go from concept to creation.

Getting PCBs produced can be a lengthy and costly process for small companies and startups. By providing users with easy and intermediate free web-based tools and streamlining the manufacturing process, **Gumstix** makes it easy for people to get custom boards made.

The **Gumstix** process relies on a proprietary online application called **Gepetto** that can be used by anyone, regardless of experience, to design PCBs. Users then submit the designs they make in Gepetto to **Gumstix** for production.

The **Gepetto** platform can be used in combination with **Altium Designer** as well as **Upverter**; users can also harness the **Altium 365** platform to view their designs alongside co-designers and team members.

After users submit their files, their designs are manufactured according to their exact specifications. The boards can then be used to produce prototypes and limited product runs.

To further democratize electronic design, in 2018, **Altium** acquired a full turnkey PCB assembly company called **PCB:NG** that allows designers to purchase custommade boards for as little as \$3 USD per unit. Designers can submit Gerber or dril files for production from any EDA software application, including all **Altium** tools and platforms.

Ciiva: Altium's Connective Tissue



Ciiva occupies a unique space in the Altium ecosystem. Ciiva technology is interwoven throughout Altium's most high-profile platforms, including Altium 365 and Octopart, as well as Altium Designer.

Interestingly, before its acquisition, **Ciiva** was an integrated technology for another company, **Toradex**, a computing solution provider serving the embedded technologies industry.

Before Ciiva was founded in 2010, BOM-management and part change data was a complicated process that relied largely upon the use of Microsoft Excel® spreadsheets. Programs that streamlined the process were priced prohibitively high; Ciiva—just like Altium decades earlier—was founded to make solutions affordable and easy to implement.

Ciiva was created by two of Toradex's employees as a weekend project to address reuse, part-change, and end-of-life issues. Their efforts resulted in a robust, cloudbased electronic component search and BOM management system.

And **Toradex** was **Ciiva's** first client. The company was also their chief financial backer, and integrated their technology into their internal systems accordingly until **Ciiva** branched out on their own in early 2015.

By Fall 2015, Ciiva was officially part of Altium. Before long, it had its own dedicated research and development team operating out of Ho Chi Minh City, Vietnam. The small team has been vital to the integration of Ciiva technology, adding power to Altium Designer's ActiveBOM™ capabilities, Octopart's price and availability functionality and IHS Markit integration, accessible via Altium Concord Pro inside the Altium 365 platform.



Altium Customers: Who They Are and How to Reach Them

Altium doesn't have just one kind of customer, nor do we make just one kind of product.

Altium serves a niche customer base—including PCB designers, electrical engineers, mechanical engineers, design teams, managers, hobbyists, and students—generally, this is a more discerning and conservative audience than the average consumer.

In this chapter, we'll explore some customer archetypes and **Altium's** core products.









Altium Customer Archetypes

Dave the PCB Designer

Dave is an electrical engineer who works for a startup. He works mostly with contractors, so he needs software that helps him work alongside them throughout project development. He also needs software that integrates easily with other ECAD/MCAD tools and highly specialized thermal and electrical testing software the contractors use, including **Cadence Allegro**.

As Dave works for a small company, he doesn't have his own budget or any real purchasing power, but since he's the firm's main PCB designer, he can influence the decision makers. The software's capabilities are also important to his firm's investors; they want to ensure that whatever Dave and his team creates can go from prototype into production without errors, delays, or added costs.

Dave also has a quiet dream of opening his own contracting firm, so finding PCB software he can actually afford, if and when he breaks out on his own, is a serious value add.

Mike the Mechanical Engineer

Mike works for a larger aerospace company alongside a team of mechanical and electrical engineers. The company has been using bigbudget desktop software programs, and while they've served their needs for years, a more streamlined approach is needed

Co-designer capabilities are essential for Mike, who works alongside a team of electrical engineers, PCB designers and even manufacturers every single day. Even the slightest change can derail entire designs. Mike is in a senior role, so he has some influence over the real decision makers—he'll have the best success if he can present the easiest, fastest solution that requires no additional expenses or reliance on the company's internal IT department.

Pam the Project Manager

Pam manages an internal PCB design team at a large consumer electronics company. Pam works with electrical and mechanical engineers as well as third-party manufacturing firms.

She wants to ensure that her team has the best software to work with, while she leads many projects. Due to one too many eleventh-hour crises, she also needs to ensure that component parts are available early on in the project stages. To help with this process, her team recently started using **Windchill**, a product lifecycle management (PLM) system.

She reports directly to the company's CTO, so she can easily discuss her needs with decision makers. While budget is less of an issue to Pam, if she can suggest a PCB design suite to her boss that integrates with PLM and other CAD systems, she'll be able to get the buy-in necessary to get software that works for every member of her team.

Sam the Student

Sam is a Southern California high school junior who grew up watching TV shows like **BattleBots** and **Punkin' Chunkin'**. These shows fueled a love of electronic design in Sam early on. She has been building little robots and mechanized slingshots, starting in junior high.

Sam has been designing her robots and mechanical slingshots using GitHub and open-source design tools, but she wants a portfolio featuring complex schematics and impressive prototypes to help her get into MIT, Stanford, or UC Berkeley.

Her mom wants to buy her daughter a respectable, reputable PCB design software tool, but considering how expensive the schools Sam wants to attend are, she's trying to keep costs down. Her ideal solution is a reasonably priced system that Sam can use now and when she's in college.

Altium Branded Products

In the last chapter, we looked at **Altium's** three major business units: **Board and Systems**, **Embedded Software** and **Modular Board Design**.

The **Boards and Systems** products carry the **Altium** brand name and comprise our core products, including:

Altium Designer:

Our flagship product and the choice of more PCB designers than any other tool worldwide

Altium 365:

The world's only cloud platform for printed circuit board design and realization, seamlessly accessible from Altium Designer

Altium Concord Pro:

Streamlines data, libraries and supply chain management, accessible with Altium Designer and Altium 365

Altium CircuitMaker Pro:

Entry-level, professional PCB design tool ideal for students, hobbyists and light users

Altium NEXUS:

Unique solution for larger firms that integrates the entire PCB design process with project management tools, MCAD and PLM software

Octopart:

Powerful search engine that provides normalized part, supply chain and ECAD symbol and footprint data

These core products provide comprehensive solutions for every aspect of the PCB design process.

Altium Designer + Altium 365: The Ultimate Game Changer

Representing decades of innovation and development, **Altium 365** takes **Altium Designer** to the next dimension of PCB design. **Altium 365** is available on a subscription basis, in **Standard** or **Pro** packages.



The Right Product for Each Customer Archetype

Now that you've met some of our different customer archetypes and our products, let's see which **Altium** solution is right for each customer.

Dave the PCB Designer

For Dave, Altium Designer + Altium 365 is the perfect solution, from the tool's capabilities to its price point, as well as robust updates in each Altium Designer release plus seamless integration with MCAD tools.

With the addition of **Altium 365**, Dave can easily work with offsite contractors and manufacturing partners. All Dave has to do is share a link to his designs for easy, realtime collaboration and review, at no cost to reviewers. Dave can share schematics and completed PCB designs with anyone who needs to see them, including contractors, clients, and suppliers who don't have **Altium Designer**, while keeping his company's IP secure and his designs under control.

Without a prohibitively high cost, **Altium Designer** plus **Altium 365** also helps fuel Dave's quiet dreams of becoming his own boss.

Mike the Mechanical Engineer

Mike is a mechanical engineer who works alongside a PCB design team on a daily basis. **Altium Designer's** co-designer capabilities provide a perfect solution for Mike and his PCB design colleagues

With Altium Designer + Altium 365, Mike can co-design alongside his PCB design and electrical engineer colleagues with full transparency from anywhere, so there are never any surprises when it comes to their designs. The system can be set up in seconds without the need for IT support.

Pam the Project Manager

Pam's needs are a bit more complex than Dave's or Mike's. Pam manages people like Dave and Mike, product and component engineers, procurement specialists and librarians. Many of her company's designs also have to go through rigorous quality and regulatory controls.

For Pam, while she wants to ensure her team uses a powerful PCB design tool, she also has to keep the needs of all of the other key stakeholders she manages in mind. Her ideal solution streamlines product lifecycle management, markup and review functionality plus project management software so she can ensure everything is organized and completed on time with all the necessary documentation at her fingertips.

Altium NEXUS is the ideal solution for Pam, because it includes everything she needs and integrates with all of the platforms her team relies on, from seamless integration with eCAD/mCAD tools to PLM software, including Windchill, Arena, Oracle Agile PLM, SAP PLM, and Teamcenter.

Pam can also assure her boss that dedicated **Altium NEXUS** specialists will help with the entire onboarding and rollout process, as well as provide ongoing support whenever needed.

Sam the Student

Sam the high school junior doesn't have the same needs as Dave, Mike or Pam, but that doesn't mean she doesn't deserve real, professional PCB design software. Her mother—Sam's key stakeholder—has concerns about costs, but also wants to help prepare her daughter for a great career in electronics.

Altium CircuitMaker Pro gives Sam a professional, reliable and respected platform to work with that makes it easier for her to master Altium Designer and other platforms. Altium tools are also part of the Stanford and UC Berkeley and Intro to PCB Design course syllabuses, to set her up for success in college and beyond.



Everyday Interactions with Altium Customers

The archetypes and product solutions explored above are based on real conversations with **Altium** customers. You may not encounter people exactly like Dave, Mike, Pam, or Sam but your efforts will impact customers who share similar wants, needs, and concerns.

As **Altium** expands our product offerings and our users grow, remembering who they are understanding their concerns is paramount, from the humble high school student to the key decision makers at major corporations.

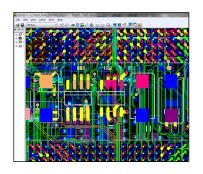


Altium's Competitors and Customer Pain Points

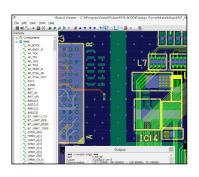
Since our early days as **Protel**, **Altium** has cornered the market with comprehensive, affordable electronic design solutions.

This chapter focuses on **Altium's** most prominent competitors, including

Siemens Mentor, Cadence, Autodesk, and Zuken software tools, with feedback about each system from former and current users.







Mentor, MentorGraphics, Xpedition, and PADS

Mentor General Information and Customer Pain Points

Mentor software has been widely used throughout the electronic design industry for over 40 years. Mentor, also known as MentorGraphics, creates a series of electronic design automation (EDA) and PCB design tools, including the Xpedition and the PADS system, a direct competitor of Altium Designer Mentor is a business unit of the Siemens Corporation.

Mentor/PADS customer pain points include:

- Price and program complexity, lack of intuitiveness, clunky UI, poor integration with other tools
- Customers have to buy multiple modules to get one fully-functional PCB design software suite. The add-on packages don't necessarily work together, so the program is hard to scale. Most of these various module features are included in Altium Designer, including Rigid-Flex support, FGPA-PCB optimization and DC drop. Term license with 1 year maintenance is approximately \$10,500 USD without options, or \$3000 USD for a 1 year lease. Too many add-on packages can make the product cost even more; depending on the number of add-on packages, the costs can skyrocket to as high as \$100K for multiple licenses.
- PADS is equally expensive and clunky but with less capability; PADS Professional costs around \$18K, PADS Standard Plus costs \$10K, and PADS Basic costs \$5K, before license and maintenance costs

Altium Customer Feedback about Mentor Products

"I was using Mentor before switching over to Altium because electrical engineers didn't have access to a comprehensive PCB design software system. Mentor was a mess, full of footprint errors, library mismanagement and exporting errors. It was also incredibly expensive; back in 2009, it was costing \$81,000/year for inefficient, piecemeal software. Even exporting .pdfs cost \$3500...switching to Altium Designer was less than just the annual maintenance costs of Mentor. Altium Designer is an all-in-one program and affordable."

Founder/President, PCB Design Consulting Firm

"It was really a pain in PADS to switch between units. The 3D view in Altium Designer is so much better than in PADS, which just gives you blocks."

Mechanical Engineer, Telecommunications Firm

"We started working initially with MentorGraphics and our engineers absolutely hated it. Constantly having to switch between different PADS applications was an endless source of frustration; I even attended a PADS user group to learn workarounds, but I was the only one who even showed up!"

Senior PCB Designer, Cable Testing Technology Company

"When I started my own consulting, I priced out a seat of PADS Professional/Xpedition/
DXDesigner (schematic capture; it was \$150K/
seat). It was crazy—everything was an add-on!
I think they got away with charging so much because it was the only tool that tied into PLM systems and 15 engineers can work on the same file at a time. So Mentor got ingrained in these big companies. Buying Altium Designer was a no-brainer for my company; I've not had one customer ask me about using Mentor tools!"

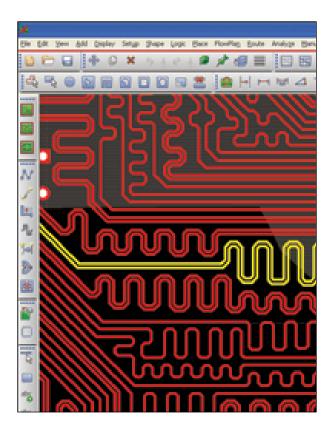
Principal Hardware Engineer and Consultant, Embedded Electrical Specialist Firm

Cadence Allegro/OrCAD

Cadence Allegro and OrCAD General Information and Customer Pain Points

Cadence makes two PCB software systems, **Allegro** and **OrCAD**, a PCB design tool that's been in use since the 1980s.

Cadence Allegro is a design and verification flow software suite; its "Fit for Purpose - Tool Confidence Level 1 (TCL1)" certification ensures users can meet stringent ISO 26262 automotive safety requirements. While Allegro is a direct competitor, it's often used by Altium customers for its proprietary PSpice analog simulation and thermal testing capabilities, and can be used in conjunction with Altium Designer.



Cadence Allegro customer pain points include:

- Too many add-on features make the program more expensive than it should be, many of which are included in Altium Designer and other electronic design automation software systems
- UI is disjointed and looks dated
- Not as tightly integrated as Altium Designer (which has a unified environment)
- Very hard to learn, even if you're just transitioning from OrCAD
- Experienced Allegro users find that it's too similar to OrCAD, so it doesn't justify the higher price point

OrCAD pain points include:

- Ignored by Cadence for over a decade, so bugs haven't been fixed and features haven't been updated
- Requires multiple add-on features for full functionality
- Convoluted back annotation process compromises schematic fidelity
- Different syntax requirements than in Allegro, so it's challenging to use both programs concurrently
- Overpriced for such a clunky, poorly updated tool; OrCAD Professional costs around \$10K while OrCAD Standard has a base of \$5K
- OrCAD Standard users frequently have to purchase add-on features, which can increase the cost to nearly \$10K



Altium Customer Feedback about Cadence Allegro

"Altium's process is much more integrated than Allegro's when you're trying to look at simulations. Allegro also causes issues with re-pinning and adjusting footprints. With Altium Designer, we don't even need to generate drawings for markups; eliminating all of the paper trails we had with Allegro has made the entire design process much easier."

AUV Supervisor,
Division of Marine Operations at
a Major Research Center

"I was using Allegro and really didn't like the program, other than the component management, but with Altium, everything is easier and faster."

Founder/President, A/V Firm

"Altium is 1000 times better than Cadence on doing a simple DXF. With Allegro, if there were complex curves; Allegro could not close the outline shape. I'd have to go back to a very old version of the tool in order to create the outline. I spent a long time talking to their engineers and they just said it's broken and we will not fix it. Their documentation was abysmal compared to Altium Designer. It was a collection of release notes. User groups were ad hoc and had questionable quality. There were less and less people supporting one another. The ability for me to bring in a step model and use it as a board outline has been fantastic. Altium Designer made up 10 times over for things that I missed in Cadence."

Senior Electrical Engineer, Thermal Management Solutions Company

Altium Customer Feedback about Cadence OrCAD

"The problem with OrCAD is that it looks cheap when you first evaluate it, but you have to keep paying for additional capability. With OrCAD everything's an add-on. Each piece feels like its own separate program!"

Hardware Engineer,
Managed Cloud Computing Company

"I started off working with OrCAD in my career. I hated that it felt like 45 tools held together with scripts. OrCAD has horrid support and horrid tools."

Principal Hardware Engineer and Consultant, Embedded Electrical Specialist Firm

"OrCAD doesn't put constraints into schematics very well, like clearances, maximum and minimum thickness. OrCAD doesn't allow users to go through the interface from schematic to layout, so I had to do this manually. This will be much easier with Altium Designer."

Electrical Engineer,
Computer Hardware Manufacturer

"I honestly did not see any difference between OrCAD and Allegro."

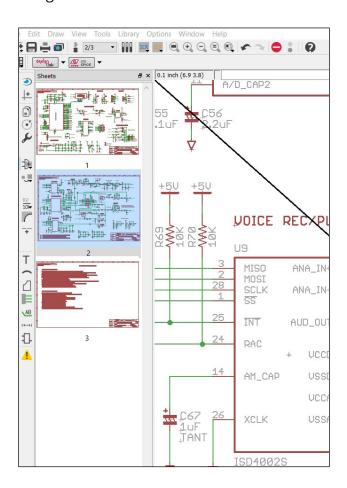
Senior Electrical Engineer, Thermal Management Solutions Company

Autodesk Eagle

Autodesk Eagle General Information and Customer Pain Points

Autodesk Eagle is a lower-priced PCB design tool from Autodesk, the makers of the world's leading architecture design software suite AutoCAD. Autodesk Eagle has less functionality than its major competitors, including Altium Designer, as well as its open-source counterparts.

As it has fewer features, **Autodesk Eagle** is more analogous to **Altium CircuitMaker** than **Altium Designer**, from its capabilities to its price point. Some users have likened **Eagle** more to **KiCAD**, an open-source electronic design software tool.



Autodesk Eagle cutomer pain points include:

- It's geared to hobbyists and students more than professional PCB designers, so it's not taken as seriously as Altium Designer and other major PCB software tools
- No perpetual ownership and no regular updates; ownership is terms-based, so you never own the software outright—if you stop paying for it, you lose access to it. This time-based license model makes a lowercost product more expensive over time.
- Rudimentary routing and 3D capabilities, no library sharing
- Creating documentation is challenging, poor template support and tool pre-adjustment

Recently, **Autodesk** introduced an online platform called **Fusion 360**. **Autodesk Eagle** is now only available only with a **Fusion 360** subscription, which further impacts the cost factor.

Altium Customer Feedback about Autodesk Eagle

"Even before I graduated, Altium was THE software to use. Unfortunately, we didn't have access to it—we had Eagle and KiCAD, mostly—but the industry takes you more seriously when you know how to use Altium because it's an enabler of getting your skills applied. And, unlike Eagle, it removes handicaps and doesn't limit you. My team and I use Altium Designer and Altium CircuitStudio to design everything. Altium CircuitStudio's common design projects make migration to Altium Designer easy."

Principal Designer and Team Leader, Next-Generation Combat Robot Designer

"Eagle just looks and feels like a hobbyist's tool to me. It's not the kind of thing you want to use as a consultant, especially if you're hoping to get bigger clients."

Founder/President, PCB Design Consulting Firm

"We used Eagle for eight years, and while the tool was pretty straight-forward, there are so many features in Altium Designer we needed. With Eagle, we couldn't manage components or design libraries. With Altium Designer, we can move from schematics into prototypes much easier and show clients 3D models instead of just flat board designs."

Lead Hardware Engineer, IoT Development House

"There's nothing I really miss about Eagle that's not in Altium Designer. The help menu in Altium Designer is my best friend, and the hierarchy in the Help Document is really helpful. When I was using Eagle, I had to Google things!"

Associate Electrical Engineer, Medical Technology Company

"I don't miss anything about Eagle! Eagle is like a go-kart that someone made in their backyard."

Senior Electrical Engineer, Thermal Management Solutions Company

"Eagle and KiCAD are hobbyist tools. I don't mess with that."

Principal Hardware Engineer and Consultant, Thermal Management Solutions Company

"I could never really do a complicated board in Eagle, but I can with Altium Designer."

Electrical Design Engineer, Aerospace Engineering and Manufacturing Firm

Zuken General Information and Customer Pain Points

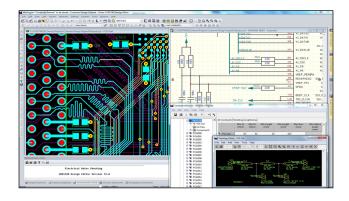
Zuken is a Japanese electronic design automation company who control over 95% of the Japanese PCB design market. They make several PCB software programs as well as the E3 wire harness design software suite. **CADSTAR** is **Zuken's** best-selling desktop PCB software tool, used by approximately one-third of all PCB designers worldwide.

Zuken CADSTAR

CADSTAR is offered at multiple price points to meet most budgets, including CADSTAR Lite (the lowest cost option), CADSTAR Dynamic, and CADSTAR Kynetic. Each option includes increasingly more features; the cost increases proportionately with overall functionality.

CADSTAR software suites can be enhanced with add-ons, including schematic and variants, constraint browsers, placement planners, board modelers (allowing for eCAD/mCAD bi-directional design translation), place and route editors and signal testing tools. CADSTAR also integrates easily with the E3 series.

Recently, **Zuken** added 3D functionality to the **CADSTAR** platform, plus a web-portal for easier remote access. The online version, eCADSTAR, is essentially the same program, with a few new features that **Altium Designer** already has.



CADSTAR pain points include:

- High price tag (~\$200K per seat), even for CADSTAR Lite
- All versions require at least one add-on for full functionality
- Most of the add-on features are available in Altium Designer already at no additional cost
- Hard to find, from the product itself to the technical documentation
- Updates are sporadic, not routine
- eCADSTAR isn't much of an improvement; it's just another way to use CADSTAR

CR-5000 and CR-8000 pain points include:

- High cost; CR-8000 is at least 5x more expensive than Altium Designer
- The main package averages around \$90K for CR-5000 and \$100K for CR-8000
- CR-8000 has to be purchased new; it's not included in the subscription and it doesn't matter if you already have CR-5000
- Weak data management capabilities
- Add-on features may be necessary for full-functionality and also have a high cost

CADSTAR User Feedback

"This software is an extreme waste of money. It has so many shortcomings. The productivity is low and you routinely lose all your work. The support line is terrible—all they try to do is sell you on attending their seminars."

Electrical engineer and longtime CADSTAR user, Zuken CR-5000/CR-8000

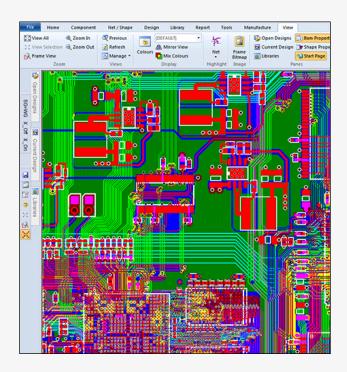
The **CR** series are **Zuken's** higher-end PCB design tools, offering more flexibility and overall functionality than **CADSTAR** and eCADSTAR.

The CR-5000 has an integrated schematics and PCB design platform, with checking and verification functionality. The CR-8000 is more sophisticated than the CR-5000, allowing for multiple-board design. The software system features including thermal design support, pin swapping to optimize routing, and system-level circuit design. It's capable of multi-board design, featuring new technology including a native 3D layout and 64-bit architecture.

CR-5000 User Feedback

"I've used Zuken CR-5000 for a few years. It's very expensive and getting outdated."

CR-5000 user





Connecting to the Community

Altium's users are the lifeblood of our business. We depend on user feedback to make our products and services better with each iteration.

Our commitment to our users has helped **Altium** become the leading EDA solution for more PCB designers than any other tool worldwide. And so, we're big on giving back, from free tools and comprehensive solutions to a wealth of educational resources. This chapter will explore some of the many ways **Altium** helps empower our community.





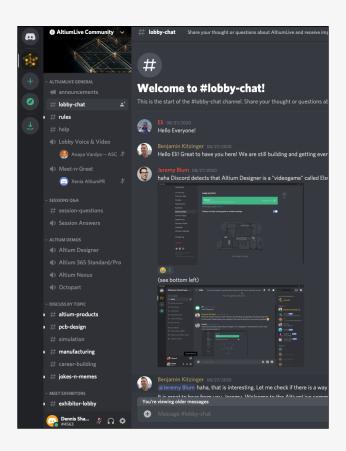


Altium User Forums: Engaging with Each Other

Altium maintains one of the most active forums in our industry. Altium forums are the best way to participate in discussions, share tips, ask/answer questions, and locate resources. The online community forums also give users an easy way to share ideas and designs.

Altium forums include everything from product support, training videos, and bug reporting tools to free electronic design intellectual property, including Unified Components, Reference Designs, and Board Templates. Members can suggest new software enhancements and features, influence which bugs get fixed the fastest, and bounce ideas off of each other.

Altium forums are always free to join and are constantly updated in real time. Click **here** to join in on the fun!



The AltiumLive Conference: Learning, Connecting, and Inspiring

Probably the most high-profile way we connect with the community is through **AltiumLive**, a unique conference that highlights electronic design like none other.

Many corporate-sponsored conferences are thinly veiled promotional events, featuring select speakers and activities, geared mainly towards selling products to the audience. AltiumLive is the antithesis of the traditional conference because it has no ulterior motive—the event is a true labor of love that's devoted to the enrichment of the community. The one-of-a-kind summit unites designers, engineers, hobbyists and design teams, providing ample opportunities to learn and network over the span of several days.

Two **AltiumLive** events are held annually to accommodate **Altium's** global user base, one in the US and one in Europe. The US events are held at one of the most scenic resort hotels in San Diego, so attendees experience the royal treatment, from gourmet meals to panoramic views of America's Finest City. The presentations are recorded and archived online to ensure everyone can benefit.

AltiumLive's keynote speakers, presenters, and educators are among the industry's most notable and accomplished, but the event is also full of hands-on fun, including mini-BattleBots-inspired contests where attendees form teams, creating robots and pitting them against each other in lighthearted combat.

The event has grown exponentially since its inception in 2017, drawing hundreds of attendees in both locations each year. In 2020, due to the global pandemic, **AltiumLive** was held virtually, guided by the event's first-ever official steering committee.

Presentations and podcasts from **AltiumLive** events are archived online here.



Altium Stories: Showcasing Inspiration

Another high-profile way **Altium** connects with the community is via a special **YouTube** channel called **Altium Stories**.

The channel, launched in late 2019, celebrates the electronic design community through short video documentaries that capture the creativity and enthusiasm of each featured guest. Within just a few months of launching, the channel grew to over 8000 subscribers, with some videos receiving as many as 35,000 views.

The videos are professionally produced and filmed largely on location. After they're edited and posted, additional footage is used to create **Behind the Stories** videos, featuring even more original content about each subject.

Featured guests have included **Arduino**, the world's leading open-source hardware and software ecosystem, the legendary hardware hacker **Joe "Kingpin" Grand**, and the lead engineer behind the **PianoArc**, the most astonishingly creative keyboard ever produced.

In 2020, **Altium** launched the first **#MyAltiumStories** contest, expanding the reach beyond industry luminaries to everyday innovators and enthusiasts. This community-based contest garnered national attention, attracting entrants worldwide.

Altium Academy: Empowering Through Education

Altium's YouTube presence also includes a purely educational channel called **Altium Academy**, officially launched in Spring 2019.

Altium Academy brings a modern approach to educating the electronic design community, featuring industry experts and professionals sharing their knowledge via live webinars, in-depth Whiteboard sessions, and exclusive video presentations, as well as a wealth of "how-to" videos designed for Altium users.

With over 300 different videos on the **Altium Academy** YouTube channel, the electronic design community has a wealth of resources to learn not only how to optimize **Altium** tools, but how to get a deeper understanding of their craft.

To further empower the community, **Altium Academy** is expanding from online education into official certification programs, making **Altium** the only place electronic designers can get a real practical application education, encompassing the ins-and-outs of PCB design as well as how to master **Altium** software.

Explore **Altium Academy's** online presence **here**.



Altium Academic Programs: Stewarding the Next Generation

It may come as a surprise, but PCB design isn't taught in school. It's not taught formally anywhere; rather, it's learned on the job, like many practical application skills.

To help ensure aspiring electrical and mechanical engineers can get the skills they need to get jobs in the electronic design field, **Altium** offers students and educators a wealth of resources, from low-to-no cost software licenses to sponsoring student design teams at the high school and college level.

Over 250 universities have benefitted from Altium's efforts, as well as dozens of different design teams throughout the US, Canada, and Europe. Sponsorship teams have included multiple Hyperloop, robotics, rocket, and satellite design teams, giving students the tools and hands-on experience they need to flourish as the next generation of electronic design.

Students and teachers can also take advantage of **Upverter**; as mentioned in Chapter 2, the online tool is an educational resource for high school STEM programs.

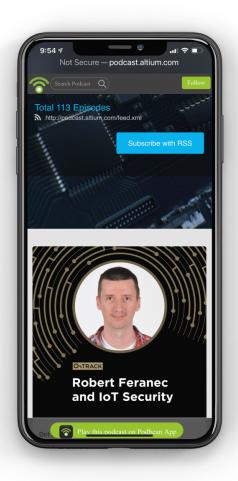
You can watch some of these incredible student teams we've sponsored and more here.

The OnTrack Podcast Series

Podcasts have become omnipresent in today's world, covering just about every topic under the sun. Altium ventured into the podcast arena in 2017 with **OnTrack**, a program devoted to all things electronic design.

OnTrack podcast is accessible to everyone—including people who don't use Altium technology. The podcast's companion monthly newsletter follows the same approach, ensuring that everyone can benefit, regardless of what tools they use. The podcast has developed a global following since its inception, garnering an average of 61,000 audio downloads per month. The podcast can be heard in 82 countries and is in the top 20-30% of podcasts worldwide.

Get caught up with **OnTrack here**.



Altium User Groups: For Altium Users, by Altium Users

One of the more unique ways **Altium** connects with the community is via **User Groups**, local meetups that are independently created by **Altium** users for other **Altium** users.

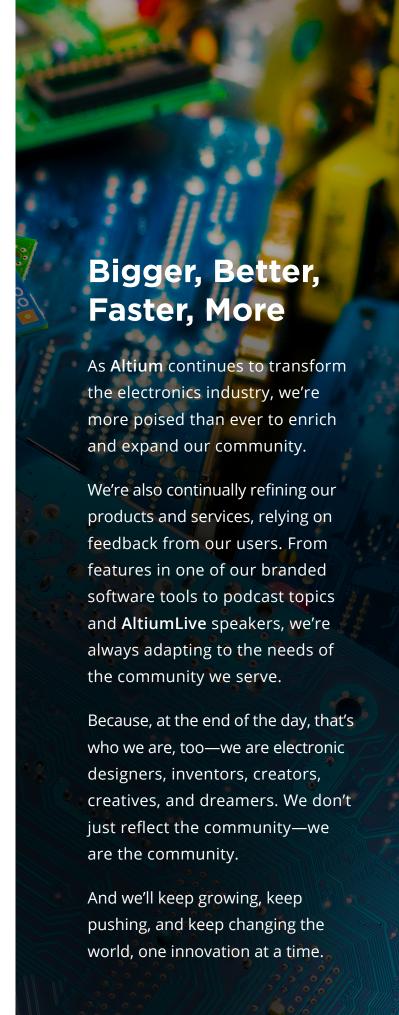
These groups are located all over the world, showcasing everything from different PCB companies and component manufacturers to individual electrical and mechanical engineers. The largest group in North America is in Salt Lake City, Utah.

Each individual group sets the number of meetings per year, as well as the locations, topics, speakers, and formats. The meetings can feature industry professionals, roundtable discussions, working sessions, and peer-topeer assistance.

Altium is often in attendance, with Field Application Engineers in the audience, ready to address concerns, questions and receive feedback to incorporate into product updates and new releases.

To learn more about **Altium's User Groups**, please go here.





Appendix

The ABCs of PCBs

Altium develops a special type of software that engineers and designers use to design printed circuit boards (PCBs). Circuit boards are found in just about every piece of electronic equipment in the world, from tiny USB devices to giant military aircraft carriers.

Electrical engineers, PCB designers, mechanical engineers, and manufacturers comprise Altium's customer base. These technical experts often use acronyms. To help you navigate your way through this technical jargon, here is a list of commonly used terms and abbreviations.

PCB Design Essential Terminology

Bill of Materials (BOM): A list of all the needed parts required to manufacture a specific printed circuit board. A BOM helps designers calculate costs and crosscheck different components before *printed circuit boards* are assembled.

Circuit Board: Thin, rigid boards infused with electrical circuits. Circuit boards can be found in every type of electronic device; they can be as small as a microchip and as large as a kitchen countertop. Altium's software tools help design printed circuit boards.

Computer Aided Design (CAD) software: A type of software program used by designers and engineers to create two-dimensional and three-dimensional models of physical components. CAD software has all but replaced the T-squares and protractors used

by the designers in the past in a process known as manual drafting, the traditional 'pencil on paper' approach to engineering and design. There are two basic kinds of CAD software—ECAD and MCAD. ECAD is short for *electronic computer aided design* software, which is used to design electronic components (Altium Designer is an ECAD software); MCAD is short for *mechanical computer aided design* software, used to design mechanical components (Dassault Systèmes SolidWorks is an MCAD software).

Computer Aided Manufacturing (CAM): This is the software used by designers and manufacturers to create engineering files used in the final manufacturing process.

Copper Traces: These are green, blue and red lines that connect board components.

Double-sided PCB: These are the most common types of circuit boards, featuring copper on the top and bottom of the board and plated holes.

Electronic Manufacturing Service (EMS): This refers to a company or contractor that manufacturers electronic components, including printed circuit boards and hardware.

Footprint: A pattern for an electronic component that will eventually be soldered there. Whether it is a throughhole connector, a surface mount capacitor, or a large ball grid array (BGA), every part that gets soldered onto or into a printed circuit board needs a footprint. PCB design software, like Altium Designer, comes with libraries that feature footprints of different components to make the designing process easier.

Gerber File: These are design files used to make printed circuit boards, including information about drill size, where to make holes, what components are named and other essential data required for manufacturing.

Impedance: The measure of a circuit's resistance to the passage of a current when voltage is applied.

Layers: These are the different surfaces that make up printed circuit boards; a single board typically features top, bottom and inner layers. The top layer is used to draw component layouts, while inner and bottom layers are used to connect different components.

Mounting Hole: An unplated, non-conductive hole used to secure a circuit board during the assembly process.

Pads: Small copper surfaces on printed circuit boards. They help connect components to the board surface using copper traces.

Placement: One of the major steps in printed circuit board design; this process determines where active elements of integrated circuits (ICs) components are located on the board. Routing typically follows placement in the design process.

Printed Circuit Boards (PCBs): Circuit boards with printed or etched conductive lines. Electronic components are mounted on the board, connected by traces that work together to form a working circuit.

Rigid-Flex PCB: A slim board that features both hardboard and flexible circuits. This hybrid design enables rigid-flex boards to fold, bend or continuously flex without breaking or losing its shape.

Respin: A common term which refers to the process of laying out, routing, fabricating, populating, and testing a PC board. A board *re-spin* refers to a design change and manufacturing turnaround.

Routing: Connecting components on a printed circuit board with copper traces to allow for electrons to flow between components and for the PCB to function. As this almost always follows placement, this process is often called *place and route*.

Silkscreen: This is how imagery and lettering is printed onto circuit boards. It's usually done using liquid photo-imageable ink (LPI) through photo-imaging techniques.

Solder Mask: This is the coating on the surface of printed circuit boards, which is typically green, red or blue. The coating is nonconductive to help ensure the process by which components are attached to the board is safe.

Via: These are the components of circuit boards that allow electrical currents to pass through the board; they are connected to pads using copper traces.

