Altium

PCB Design Applications

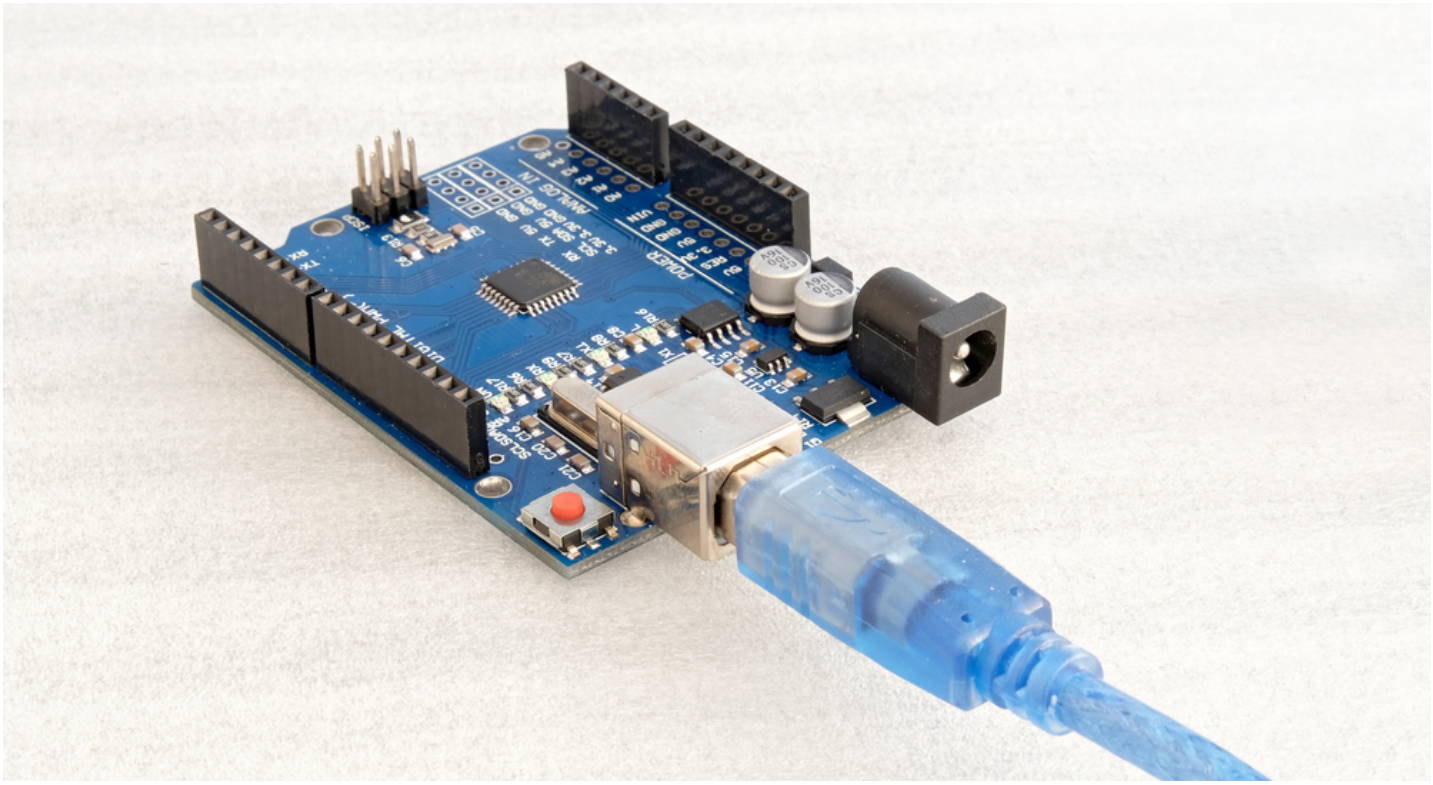
PCB DESIGN APPLICATIONS

Thanks to advanced PCB design tool technology, designers are able to save both time and money. Benefits to the Unified Data Model Approach include not just time savings, but the potential of an evolving suite of PCB design tools. Use that time to explore new tips and techniques.

Join us as we discuss a variety of PCB Design Applications including:

- Untangle Yourself with an SD Card Bootloader
- Design Tips for Incorporating GSM Modules Into an Embedded System
- Take the Fear out of PCB Form Factors with These Four Considerations
- Circuit Board Sizes and Shapes, How to Make a PCB

UNTANGLE YOURSELF WITH AN SD CARD BOOTLOADER



Like Ray LaMontagne, I've got trouble. Smartphone trouble, trouble, trouble, trouble. I can figure out the apps I want, and I've got the games I need (wait, I might have those priorities backward), but transferring all my images and data is a bit more of a pain. There always seem to be music that gets left behind or images that, for whatever reason, refuse to back-up. I consider myself pretty savvy, but when I got a new smartphone I was starting to worry, worry, worry, worry. I couldn't even imagine if I was trying to transfer data from a larger device—getting all of the files from a hard drive or something larger seems like it would throw me for even more of a loop.

When I purchased my new smartphone, the sales team helped me out by using a utility app that migrated my data from the old to the new smartphone with ease. I didn't have to flounder with thirty cords and cables and hunt through all of my collections to determine what was being transferred over anymore. Now take this experience, and apply it to your work: how often do you have to replace the firmware of a microcontroller? And how often do you wish you could avoid that clumsy process of plugging in cables? One way to simplify this process is by using an SD card bootloader.

FLEXIBLE UPDATING WITH AN SD CARD BOOTLOADER

Conventionally, a microcontroller is programmed using a programming cable. This can become cumbersome and tedious if you are working in a space-restricted environment or if you're like me and you tend to leave your unfinished tasks hanging out all over the place. Furthermore, it lends itself to outside interference. What if you trip on a cable and knock everything off a table? Or spend

several hours trying to figure out why data is not transferring only to realize that the cable simply was not plugged in tightly enough?

In a microcontroller, a bootloader is a special program that receives the application firmware from an external source and writes it to the internal program memory. It tends to have a small footprint and resides in a reserved section of the program memory. When the microcontroller is powered on and attempts to read the latest firmware present in external media or resources, the bootloader program will erase the existing firmware in the program memory and write the new firmware onto it.

You could customize your bootloader to obtain its new firmware from serial interfaces, LAN, external media, and more. But my personal preference is for the SD card: it does not need to be a complicated setup on a computer, and it is practically wireless. Using an SD card for your bootloader program turns updating firmware into a matter of inserting the SD card and resetting the microcontroller.

With a bootloader, updating firmware can be done on the spot and with minimum interruption to the system operation. This feature makes bootloaders, and especially SD card bootloaders, incredibly helpful in embedded systems that may require updating or troubleshooting in the course of their deployment. If the specifications of your firmware may need to evolve with time or update regularly then an SD card bootloader would make the process that much easier.

Make sure the microcontroller has space for both the bootloader and the main application.

DON'T KNOCK YOURSELF DOWN AGAIN

The main idea behind an SD card bootloader is that the bootloader program is going to read the binary or hex file of the main application from an SD card and write this file into the program memory of the microcontroller. Keeping this central process in mind, here are some other issues to consider:

Program Memory Allocation: While there are clear benefits of having an SD card bootloader, not all microcontrollers can be loaded with a bootloader program. A bootloader is an application by itself. As such, it does require a certain amount of program memory space that may not be available if you're using a microcontroller with smaller memory space or your application firmware is too large. Allocate adequate program memory space for both the bootloader and firmware application if you're planning to incorporate a bootloader.

Implementing an SD Card Interface: An SD card bootloader will require additional components to be built on your hardware to accommodate the SD card storage. It will incur additional cost if your main application does not require the SD card interface. Besides that, you'll need to implement the File Allocation Table (FAT32) stack on your bootloader program to ensure that your bootloader program can read the binary or hex file of the new firmware.

Program Memory Flash Write Sequence: A bootloader basically rewrites a range of address in its internal flash with bytes of information of the application firmware to be loaded. This will require executing the right sequence of command on specific registers to erase and write on the internal flash. Ensure that the right address is being written onto and that the bootloader does not actually erase itself mistakenly.

Mapping the Binary Files to the Right Address: A normal application is usually written on the starting address of 0x00000000. However, with a bootloader, the starting address of the main application or the bootloader may differ, depending on how they are allocated in the program memory space. Modify the memory link file to ensure that hex files of both programs are compiled for the right memory address.

Rigorous Testing: The purpose of an SD card bootloader is to simplify firmware updating in the field, not to add unnecessary errors or difficulties to the process. It is important that you test your bootloader thoroughly and ensure that both the bootloader and the main application runs as expected. If the main application is being written improperly or mapped incorrectly you may find that the system will behave erratically.

You'll want to ensure the bootloader is writing the right code at the right space.

There are other benefits, too, to using a bootloader program aside from quality firmware development. A good hardware design, particularly for the SD card interface, will rule out crosstalk and electrical interference. Altium Designer's signal integrity analysis tool may be handy for that.

Simplify your workspace and make necessary firmware updates flexible and capable of on-the-go processing with a bootloader program. Unlike with smartphones, proper care and set-up for your SD card bootloader program can last you much longer than the few months before another phone upgrade is released. And the best part is you won't need to call customer assistance each time you need to use it.

Need more help in designing an SD card bootloader? Get in touch with the experts at Altium now.

DESIGN TIPS FOR INCORPORATING GSM MODULES INTO AN EMBEDDED SYSTEM

As loving and rewarding as a family is, sometimes they can be a pain. My aunt, for example, refuses to communicate with my grandfather. They live less than thirty minutes apart; however, refuse to see each other. This makes coordinating visiting my family unnecessarily difficult as, often, I need to make separate day plans in order to see each so-to-speak faction. But what's most stressful, is that often it leaves the middleman communication role to me: if there is something going on with either my aunt or my grandfather, then I am the one who has to inform either of them. This process takes a lot of mental energy and requires a lot of keeping track.

Serving as a middle link for communication can be exhausting. This is especially true for mobile devices such as tablets and phones when you integrate a Global System for Mobile Communications (GSM) module with an embedded system. While I don't often empathize with machinery and modules, the difficulties with maintaining power integrity and machinic exhaustion were feelings that I often felt when trying to plan out alternating holiday parties.

MOBILE DEVICES AND GLOBAL SYSTEMS FOR MOBILE COMMUNICATIONS

A GSM module is used to set up communications between an embedded system and the GSM cellular network. The GSM operates at different frequencies worldwide. Frequencies of 900Mhz and 1800 MHz are commonly used in Europe, Asia, Oceania, and the Middle East, while the United States uses 950 Mhz and 1850 Mhz.

A GSM module enables the embedded system to send and receive text messages, send data on the General Packet Radio Service (GPRS) network, and make or receive voice calls. As with a regular mobile phone, the GSM module requires an activated SIM card to operate.

The age-old GSM technology is also widely used in other kinds of applications, including vending machines and energy systems. GSM technology enables an embedded system to transfer operational data to a central server without the need for human intervention. A GSM module could be used for anything as convenient as vending machines capable of alerting suppliers when ingredients need to be restocked, or as particularly helpful as energy efficiency systems that monitor electrical parameters and enable building management teams to control the system remotely.

Some vending machines use GSM modules to keep track of inventory.

HOW A MICROCONTROLLER INTEGRATES WITH A GSM MODULE

GSM modules are commonly available in a ready-to-mount PCB format, together with a SIM card socket and an antenna jack. Modules are also available in an integrated circuit (IC) package, but that requires you to design the complete circuitry for the GSM IC.

The SIM900A is a popular GSM module that I've used in my designs. The SIM900A IC operates at a range of 3.4V to 4.4V. However, it has a peak current that can go beyond 2A, and this can affect the way you design your PCB. The SIM900A includes some critical communication circuitry for the GSM module— with the microcontroller using the Universal Asynchronous Receiver-Transmitter(UART) and a connection to the SIM card.

The microcontroller uses the standard AT&T protocol to communicate with the GSM module. Operations like sending and receiving text messages are completed by sending the correct command AT&T sequence to the GSM module. This shouldn't be an issue for an experienced firmware developer unless the hardware design is at fault to start with.

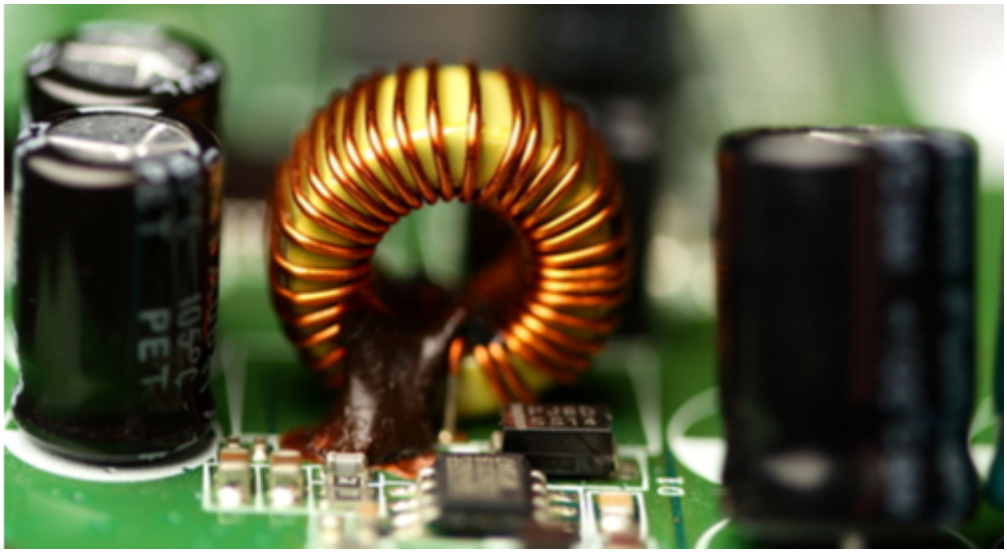
BEST PRACTICES WHEN DESIGNING A PCB WITH A GSM MODULE

In my first ever prototype, I spent hours trying to discover the reason why the microcontroller reset itself each time it tried to send text messages. After ruling out runaway codes, bad pointers, and stack overflow, I finally realized that the voltage regulator was insufficient to withstand the current drained by the GSM module during data transmission.

One of the common issues that plague GSM module design is the limited power supply capacity. It isn't difficult to ensure that the GSM module receives the correct voltage level, but the trick is in ensuring that the power is adequate when transmitting data. A typical GSM module may draw more than 2A when transmitting.

You need to ensure that the voltage regulator that's supplying the GSM module is able to handle the sudden spike in current. Not only that, the power supply copper connection has to be wide enough and thick enough to handle the high current. Otherwise, you'll risk damaging the copper track itself. It is also important to use proper heat dissipation techniques for the power management circuit, as it can produce a great amount of heat.

Electromagnetic interference (EMI) can also be a problem that affects the stability of your embedded system. An antenna is usually connected to the GSM module to boost the radio wave signal strength. The whole system needs to undergo stringent testing to ensure the microcontroller is not affected by the EMI, particularly during transmission and reception. Common best practices, such as ensuring adequate clearance between GSM and other onboard modules, helps in reducing EMI problems.



Ensure that the power management circuitry can handle current draw from the GSM module.

The GSM module introduces new challenges and uncertainties into a design. While you can't design around your family, you can design around power integrity. This is where using tools like Altium Designer's PDN Analyzer will ensure that the current density in the power connections is at the appropriate level.

Still worried about the potential problems for a GSM module? Learn more by talking to the experts at Altium.

TAKE THE FEAR OUT OF PCB FORM FACTORS WITH THESE FOUR CONSIDERATIONS

Throughout the early 2000's, we were blessed with an array of new and exciting products, shows, music, and technology. Myspace was just coming into view as one of the top social networking sites of its time, Livestrong bracelets were the most fashionable accessories around, and who could forget the launch of the very first iPod. However cool and hip all these things were, there was one that clearly stood out among the rest: the NBC show Fear Factor. OK, maybe it didn't quite steal the spotlight entirely, but for me and my family Tuesday night Fear Factor was the thing that kept us going. Live bugs, adrenaline pumping stunts, and hair-raising situations; this is what TV was meant to be.

Now that I'm a bit older, the hair-raising situations that I'm finding come from unexpected demands or challenging errors within my PCB designs. Instead of having a fear factor, though, I worry about form factor; there won't be scorpions crawling across them, and I won't have to try soldering in a straight line while tarantulas are walking along my shoulders, but considering my PCB design shapes and sizes are a beast to tackle in their own right. But fear not! Form factor only requires a bit of forward-thinking, possible with some of these design considerations.

THE SMALLER THE FORM FACTOR, THE LARGER THE FEAR FACTOR

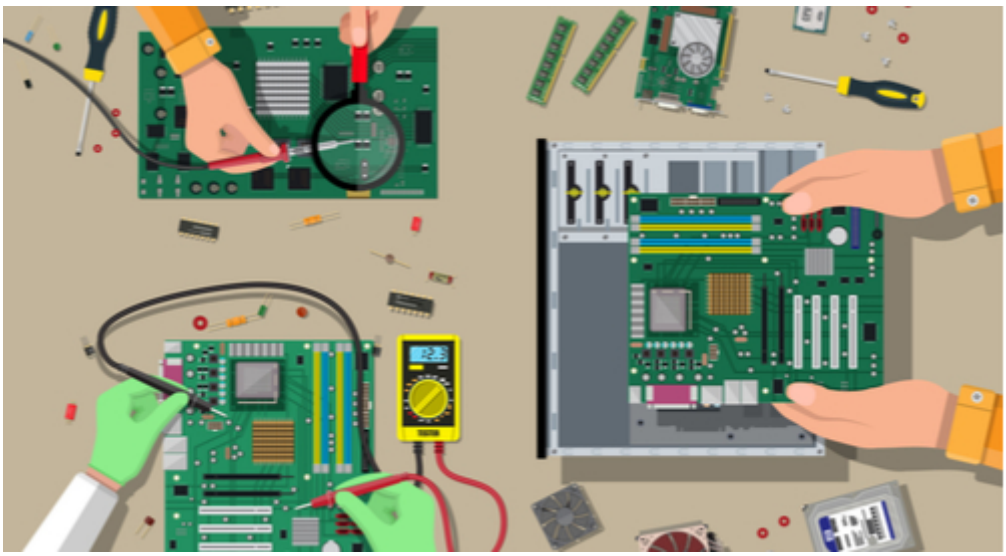
Moving well past the early 2000's, our iPods are getting smaller, demands are getting bigger, and form factor couldn't be more of an importance in PCB design. Fitting hundreds of components within smaller and smaller 'forms' is certainly cause for heightened fear. And even now, with demands for RF boards and innovations in electronics, a versatile board design has to keep in mind form factors.

To define: form factor is an aspect of hardware design which defines the shape, size and other physical properties of a PCB design as a whole. This includes the chassis, internal board configuration, mounting scheme, and so on. With the decrease in the size of PCB devices, and moving further into the small form factor (SFF) arena, there are a handful of considerations to keep in mind to ensure efficiency of your design throughout the design process.

DESIGN FOR ASSEMBLY

When in your initial stages of your PCB design, this step is understandably the most overlooked. Early prototyping effort is simply to ensure the design functions and not necessarily the fit and form of the board. Efforts within your design team should be taking all aspects of fit and form into consideration as early as possible so as to keep forward planning as viable of an option for you.

Not only should you require your design team to keep assembly in the back of their mind, but your manufacturing provider should as well. Obviously, you may not be to the point of even considering a manufacturer quite yet, but when you do, ensure that they heavily rely on multiple team interaction. In other words, ensure the design, fabrication, assembly, and procurement teams within the manufacturing plant are in perfect synchronicity with your projects specifications. This will ensure that all problematic areas within the 'form' or assembly aspect of the design will be identified early on, saving you many headaches further down the road.



Designing your board for assembly will ease manufacturing and assembly snags later on.

MONITOR, MONITOR, MONITOR

It wouldn't make sense to cook a Thanksgiving turkey without measuring the temperature at least once, would it? Perhaps you're feeling lucky though; this is understandable—we all have our bouts of luck. But would you feel the same luck when sending your initial design down the assembly line for the first time? How many times would you think you'd want to test, measure, compare? Placing as many test points along the line as possible under as many variables that could change will ensure segmentation of your form factor issues allowing you to catch mistakes as they happen. This, in turn, will substantially increase your first pass yields.

In addition to keeping an eye on the probe testing throughout the assembly, equipment related tolerances must also be kept in mind. Production machinery inclusive of paste printers, pick and place machines, selective and wave-soldering machines, and the router in charge of the physical cutting (used for board breakout) all have tolerances that may or may not wander. Obviously, these tolerances could wreak havoc on your design when dealing with small form factors. Keeping adequate testing to ensure tolerances are kept in check is never a bad idea.

PANELIZATION CONSIDERATIONS

Arguably the most important aspect when dealing with form factor along your PCB design road is the aspect of panelization. Panelization essentially constructs multiple PCB boards from a single larger board. This saves space, improves the efficiency of assembly, and saves loads of time and money compared to assembling one board at a time.

Keeping tolerances in mind like whether you choose a V-groove or perforated breakout design may make or literally break your design. Edge clearance comes into play here more than ever, and when dealing with smaller and smaller PCB form factors, you will need to ensure proper clearance values to avoid breakout flaws.

DUCKS AND DOCUMENTATION: KEEPING THEM BOTH IN A LINE

Keeping your levels of documentation in sync should be extremely high on the priority list when closing in on your production goals. Losing sight of various design changes along the way will leave you disoriented and at a loss for time when trying to find your way back around. Hierarchical design, component and data management, and a unified platform for tracking changes will all be essential when entering into the later stages of design.

Keeping your documents in a line and in working order will save you time and money late in the game.

As amazing as the early 2000's may have been for some, we are certainly far past the days of jamming out to NSync, rolling down the sidewalk in our Heelys, and the unfortunate and inevitable burn out from Fear Factor. However, now that we can confidently go about mitigating form factor issues within the PCB process, our factor of fear will burn out just the same leaving us free and clear to focus on what we as PCB designers do best; design awesome PCBs!

There are a plethora of helpful products and features within Altium Designer that will assist in nearly every form factor capacity. These include schematic capture capabilities, component management, board layout, and the list goes on.

If you would like to discuss your specific PCB's form factor related issues, talk to an Altium expert today!

CIRCUIT BOARD SIZES AND SHAPES, HOW TO MAKE A PCB

For many years my career took me to a lot of different locations as I traveled to conduct business. One of the wonderful things about travel is how it opens you up to new experiences and individuals. You only truly begin understanding how diverse our world is after witnessing the differences throughout various parts of it. I just wish I brought a dog with me on those trips, so I could name it "Toto," because I sure wasn't in Kansas anymore.

Just as diverse people and cultures can open up our minds to new ideas, working with different boards will also open up our minds to new technologies and design practices. When you who work for one corporation and design the same shape and sized boards over again, you probably won't get to see much variety in your boards. After working at many different companies and/or service bureaus, we have probably have seen all kinds of different board shapes and sizes.

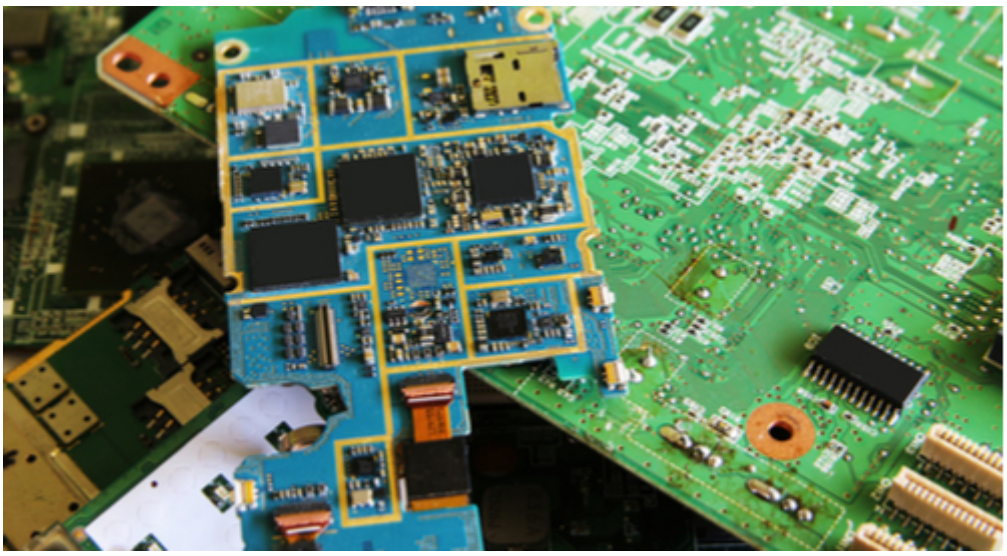
Circuit boards are sized and shaped to fit the needs of what they are used for, and those uses drive the design technology of the board. Here are some of the different size and shapes of PCBs that I have worked with over the years, what they were used for, and what it takes to create them. There's a lot more to the world of design than what we see in front of us on a regular work-day. Fortunately, we don't need a wizard to see it, even if we do live in Kansas.

BOARD SHAPES OF ALL SIZES

Having worked for many different corporations both small and large, as well as different service bureaus, I have seen lots of different boards. There's no way that I'll ever remember them all, but here's a sampling of those that do stand out in my memory.

- Standard computer type boards including both plugin boards and motherboards.
- Skinny little boards that had to tuck around the sides of a CRT.
- "L" shaped boards that wrapped around a glass screen.
- Huge industrial power boards for some kind of circuit box.
- Tiny little boards that fit inside of a watch.
- Round boards.
- And lastly, a skull-shaped board. It was the general shape of a skull, and where the eyes should be there were two giant PGA sockets. Where the mouth should be there were two horizontal connectors. This one was kind of spooky as it really did look like a skull.

As I said, there have been much more than this short list, these are just some of those that are most prominent in my mind. I can honestly say that designing all of these different board shapes in my board design career never left me bored.



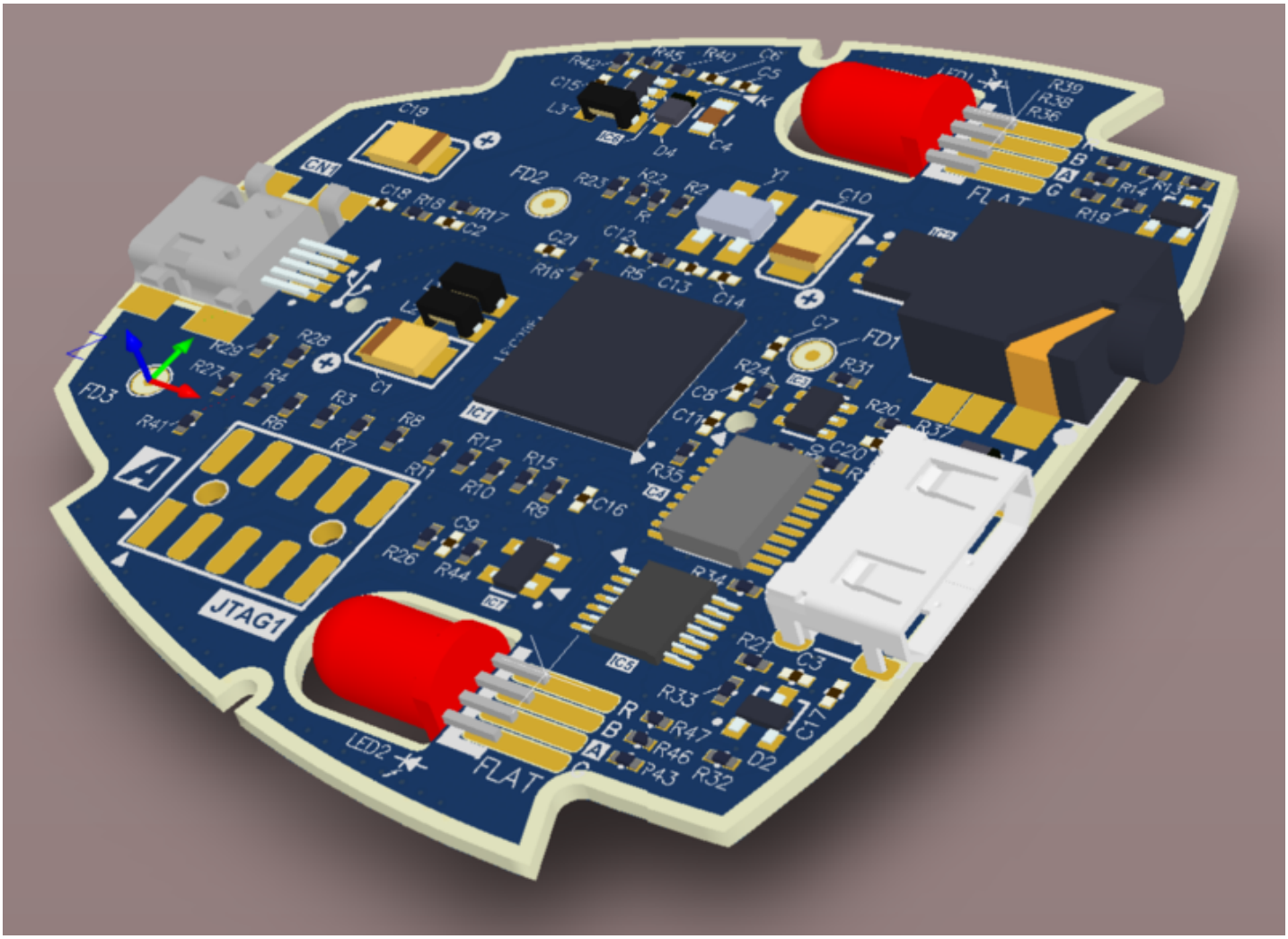
Circuit boards come in all kinds of shapes and sizes

DIFFERENT BOARD SHAPES FOR DIFFERENT PURPOSES AND TECHNOLOGIES

These different board shapes all served different purposes and had to fit in different enclosures. Those skinny little boards were a

prototype for one of the first touch screens back in the days of CRT monitors. The “L” shaped board wrapped around a display, while those huge power boards were to able to handle a lightning strike and still function. The small board that went into a watch was years before IoT was a reality, and the round boards were for huge IC testing machines.

I honestly have no idea what that skull board was for, it was just pretty funny to come into work and have it peering back at me from my CAD system. But all of these designs used very different design technologies. The computer boards required high speed design and HDI techniques while the large power board used primarily huge areas of copper pours. And the small board that fit into a watch took some of the smallest space and trace sizes at the time.



3D design environments can help you to handle even the most unusual board shapes

VERSATILE PCB CAD TOOLS IS HOW TO MAKE A PCB WITH AN UNUSUAL

SHAPE OR SIZE

Designing all these different boards at several different companies has given me an opportunity to work with an assortment of PCB design tools. A lot of these tools are no longer around, and some of them you may never have heard of before. The first CAD tool that I ever worked on was a Calma GDS system, and before you even ask, yes I am that old. My best friend growing up was the Triceratops down the street until the mean T-Rex on the next block ate him for lunch.

I had to use a variety of PCB design tools for those different shapes, but today there are some great tools out there making jobs significantly easier. I wouldn't have to trick the CAD system into doing something that it wasn't designed to do, or manipulate the netlist or the output files in order to get what I needed. The answer to how to make a PCB that will guarantee your success is to arm yourself with the best PCB design tools that you can get.

I have found that PCB design software, like Altium Designer 18, has the versatility, power, and flexibility to handle anything that I can throw at it, including working in a 3D design environment. I obviously can't go back in time to revisit those old designs. Going forward though, I'm very happy to have a design tool like this to help me handle whatever the next oddly shaped and sized PCB that is coming my way.
Would you like to find out more about how Altium can help you to design whatever the next PCB design that is coming your way? Talk to an expert at Altium.

ADDITIONAL RESOURCES

Thank you for reading our guide on PCB Design Applications for Altium Designer. To read more Altium resources, visit the Altium resource center here or join the discussion at the bottom of each original blog post:

- Untangle Yourself with an SD Card Bootloader
- Design Tips for Incorporating GSM Modules Into an Embedded System
- Take the Fear out of PCB Form Factors with These Four Considerations
- Circuit Board Sizes and Shapes, How to Make a PCB