

EnviroSense Project

Create EnviroSense WiFi Weather Station Multi-board Design

Altium®



Table of Contents

EnviroSense Project *p3*

Why Create a Surface
Mount Module? *p4*

Module Surface Mounting
Strategies *p6*

Castellated Pad Design *p8*

Designing the Host Board *p11*

Switching to a Multi-Board
Assembly *p14*

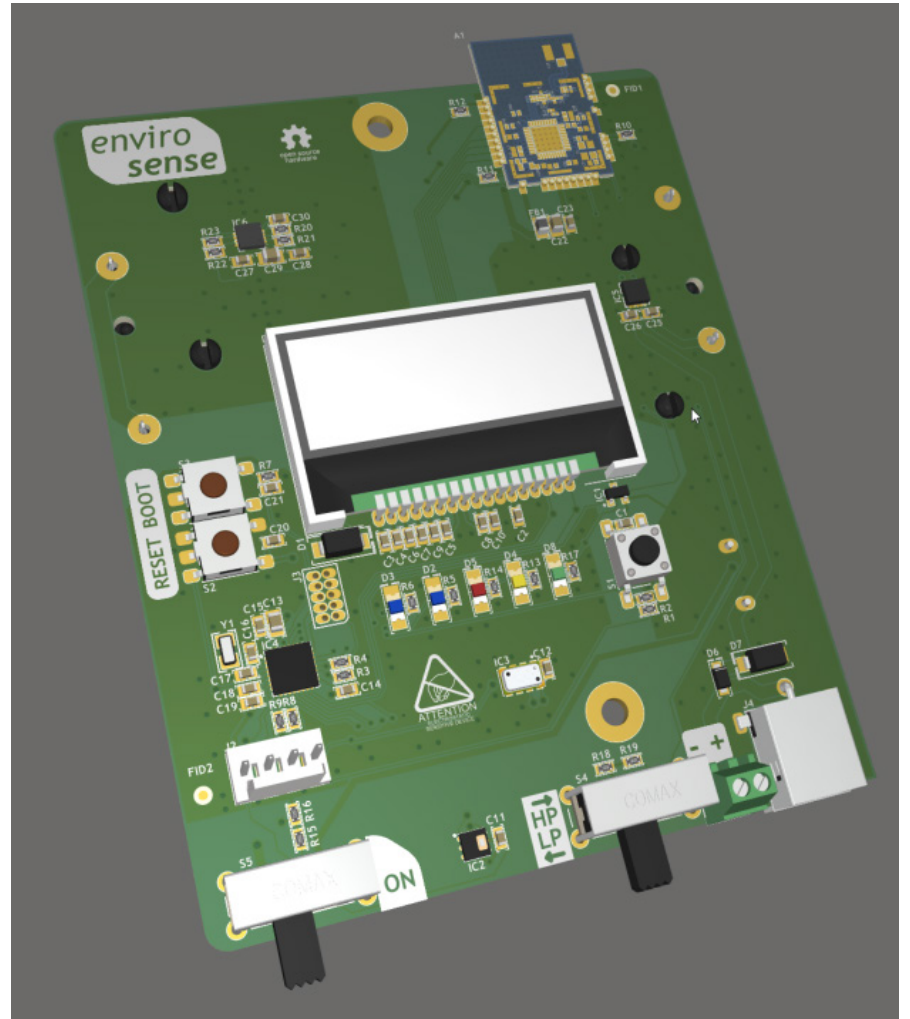
Creating the Multi-Board Project *p16*

Outcome *p20*

EnviroSense Project

Our EnviroShield project is getting a big upgrade in this article - we're taking it out of the prototyping phase where it was a Nucleo shield and building a standalone product. Ideally, this will be fully integrated into a mechanical housing featuring a passive radiation and moisture shield - but that's beyond the scope of this article.

NOTE: EMBED VIEWER IF IT SUPPORTS MULTI-BOARD ASSEMBLY



As with all my projects, this project is open source and free for you to use as you wish. You can find all the design files on [GitHub](#). Of course, there is no warranty, guarantee or liability associated with the project files as they are licensed under the MIT license.

As this is now going to be intended for indoor and outdoor uses, we also need a way to get our weather data back to a database, so we'll be adding a Microchip ATWINC1500-MR210PB WiFi module.

In this project, we're building the host board - but we'll mostly be talking about how and why you would build your own surface mount module. The Microchip WiFi module is a perfect example of why you would create your own surface mounted board.



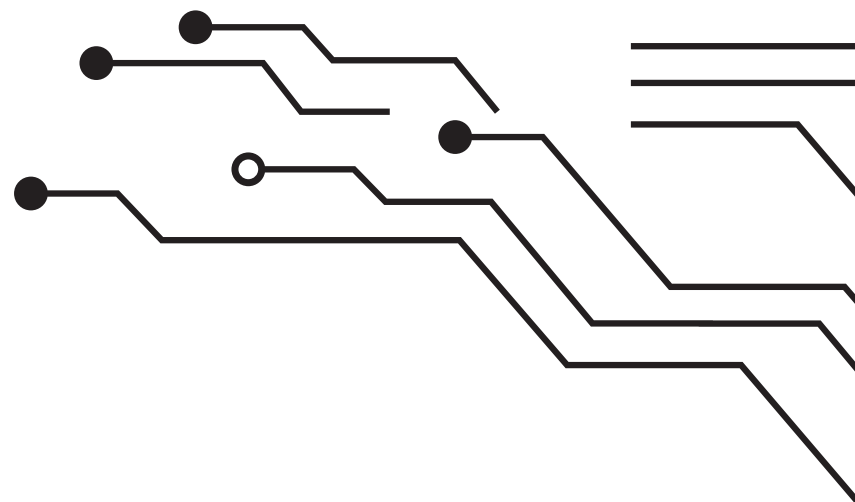
Why Create a Surface Mount Module?

Certification

Intentional radiator certification is expensive. If you're building a custom wireless solution into your products, it may make sense to move the radio hardware to its own sub-assembly you can separately certify. If you have multiple products using this custom radio module, you may be able to certify them under the much cheaper Unintentional Radiator class, as they integrate a pre-approved intentional radiator module.

You may also find this highly applicable to voltage regulators, motor drivers, or LED drivers that require certification such as for railway, automotive or medical applications.

The Microchip ATWINC1500 module we're using in this project is a perfect example of this approach. I could easily integrate the RF chip directly onto the board, however certification costs would jump considerably.



Cost Reduction

If a small portion of your hardware requires a specialised **PCB substrate**, specialised PCB features like plugged or blind vias, or a higher layer count - you may find some savings by moving that portion of the project to it's own sub assembly. As the non-specialised area of your circuit board grows, you are paying a premium price for circuit board features you are not necessarily taking full advantage of.

By separating out the specialised portion of your project, it also makes re-use of that schematic block much cheaper and easier in the future. You can simply place that existing module onto a non-specialised board and take full advantage of its features.

Standardization

If you find you are **reusing schematics** across multiple projects that have substantial engineering cost or risk laying out the board, it can be a good time to make a sub-assembly. A common example of this is a high performance or high reliability switched mode voltage regulator. Isolated regulators are also commonly split into sub assemblies.

By separating a portion of the design which has significant engineering investment, it allows for rapid reuse without a major engineering cost to qualify the design integrated into a new circuit board.

Module Surface Mounting Strategies

When creating your own surface mount module, you need to consider how your new module will attach to the host board. If you are going to be making large volumes of modules, you will likely need to evaluate both of the most common methods with your assembly process to determine the highest reliability and ease of assembly

Land Grid Array

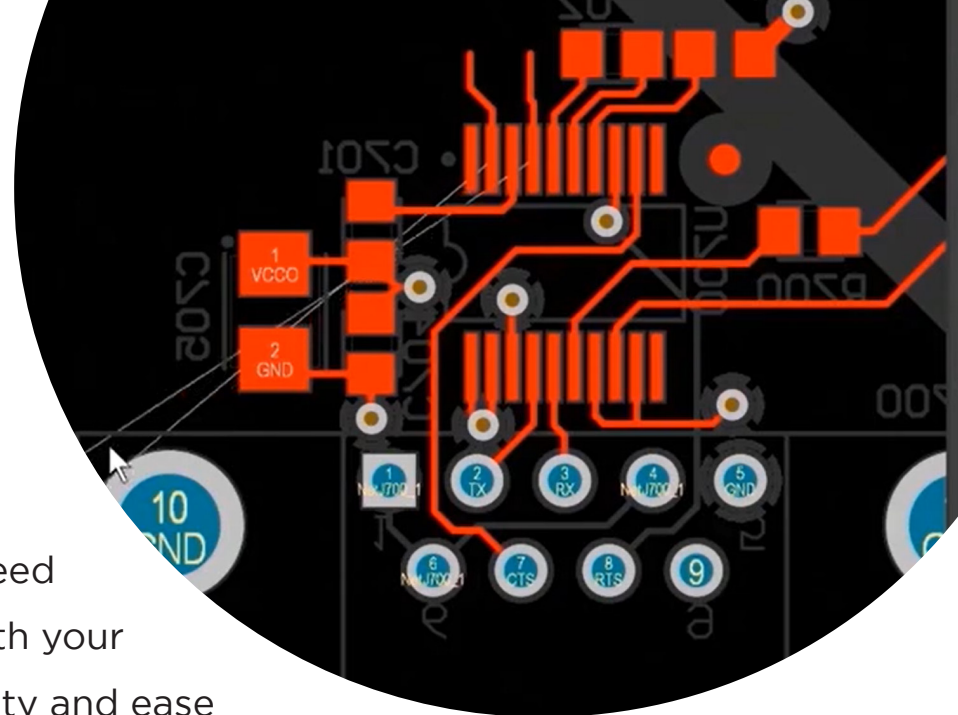
When selecting components you've almost certainly come across Land Grid Arrays (LGA) - bare pads under the IC or assembly. If you have a large number of connections to make in a small space, LGA can be an ideal approach. The major disadvantages are the same as any other LGA component - difficulty inspecting the joint and challenges with prototyping. You may also find additional costs with your board assembly partner due to the extra machine setup time - involving your board and assembly partners early in your design process can greatly improve your reliability and reduce your cost.



Source: *U-Blox NOR-B106 on Digi-Key*

You should avoid via in pad on both the host and module boards to ensure you don't create air pockets in the solder, or thief the solder through capillary action.

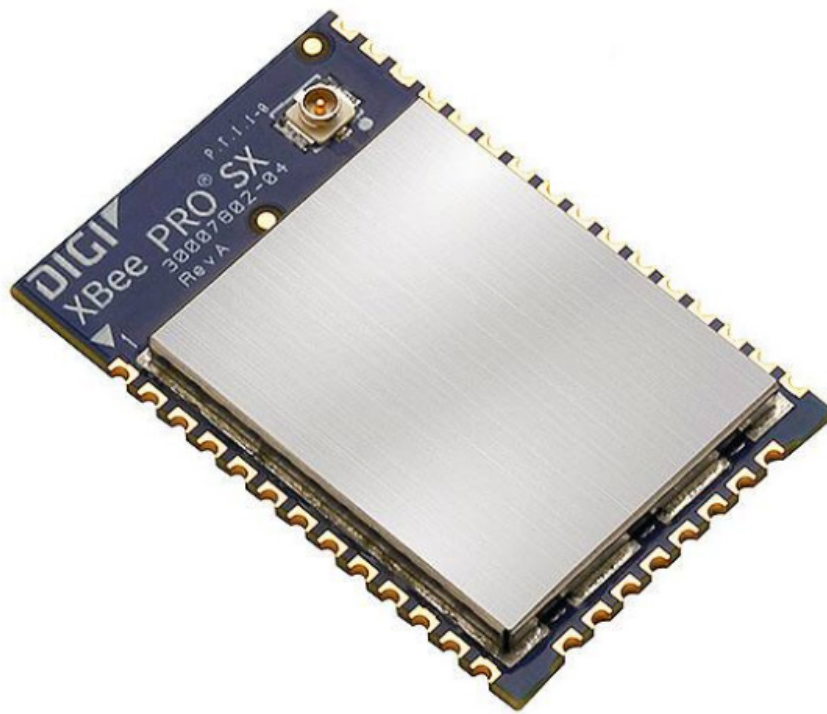
LGA is also highly cost effective for the module manufacturing. Some board manufacturers may charge more for castellated pads.



Castellated Pads

Many commercial modules utilize castellated pads for their modules, and with good reason. Castellated pads are highly reliable and very easy to hand solder and inspect. Due to solder wicking up the side of the board you can also achieve good current density.

A castellated pad is effectively a plated through hole that is routed in half during the board manufacturing process. The routing toolpath in your panel will pass through the centre of the hole, leaving a plated scallop on the board edge that has a pad on the top and bottom of the board.



Source: Digi Xbee Pro XB8X on Digi-Key

Only having pads on the board edge also makes routing the host board easy, and will not interfere with via or track placement on the module board. The edge pads can be directly connected to a track or plane on any layer, which may make routing to the interconnect pads easier on the module.

Castellated Pad Design

It's hard to go wrong with castellated pads, however there are some things to consider which will set you up for success. If you're new to creating castellated pads, it's worth downloading Altium files for commercial modules where they are available from the manufacturer, as I have done for the ATWINC1500 module I'm using in this multi-board design. Looking at the pad stackup and layout of commercially successful modules in a similar class to the module you are creating can give you some ideas on what works well before you begin your own experimentation.

Asymmetric Pad

When creating a castellated pad, the stackup will almost always be asymmetric. The top and inner pads are typically going to be round and sized just as any through hole pad or via would be. The bottom pad however is typically going to be enlarged to provide greater copper area and solder contact. This makes the castellated pad able to be used reliably with solder paste stenciled boards as well as hand soldering.

Edge Plating

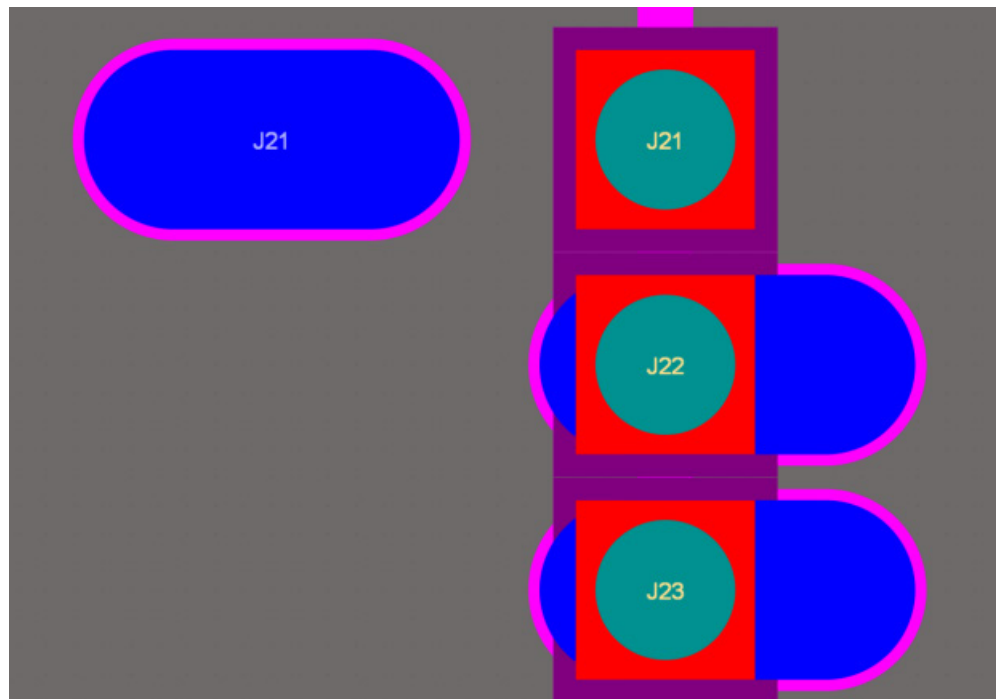
If you require higher current density, or even greater reliability you can also have the edges of the board plated beyond the through hole. This plating is typically added to the width of the bottom pad, providing more contact area on the side of the module. Most board manufacturers will charge more for this service and low cost high mix board manufacturers may simply ignore this if they even notice the fab note.

A Good Starting Point

If you're not short on edge space, start with larger pads for your initial prototypes. As you begin to feel more comfortable with the process and see the results of your initial modules you can begin to optimize the size downwards if necessary.

Microchip ATWINC1500 Pads

As an example of this, the Microchip pads on the ATWINC1500 module have a 0.635mm plated hole and a square 0.8mm pad for all layers. They have an additional 1.7 x 0.8mm round pad on the bottom layer (shown below moved to the left, to show they are separate).

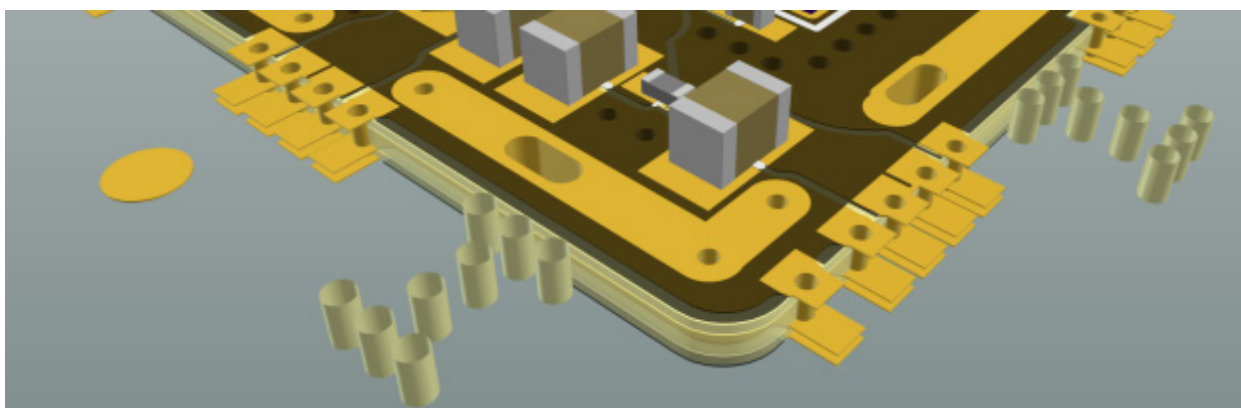


Microchip recommends a 0.8 x 1.9mm pad on the host board, with the pad centred on the module outline.

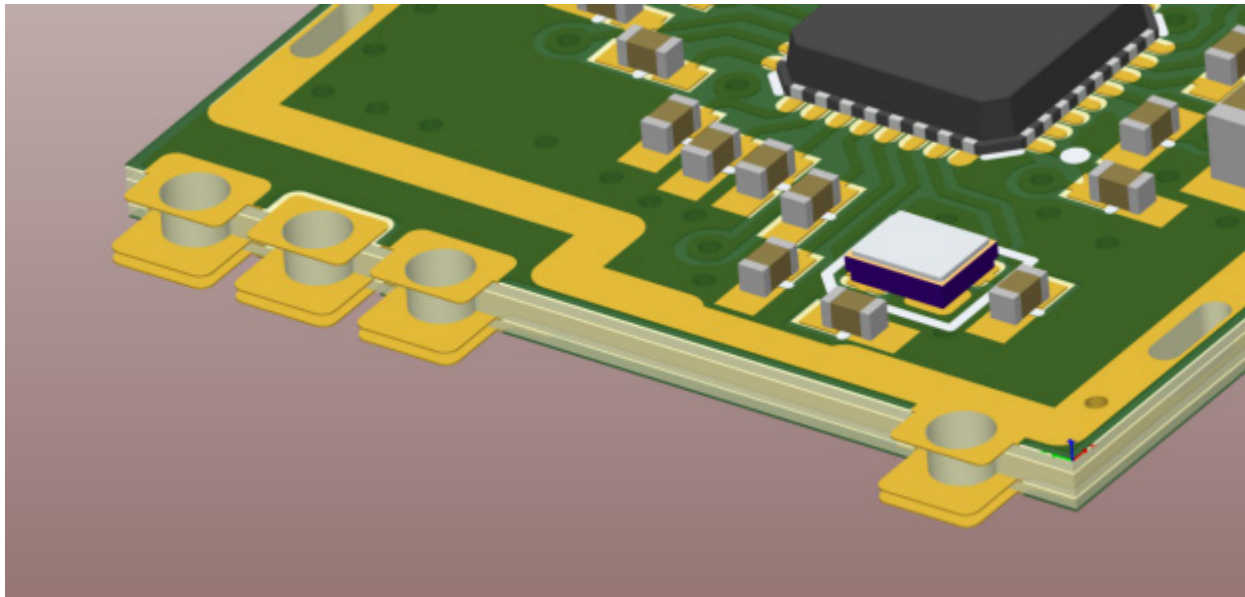
My Castellated Pads

I've always done things a little differently to the Microchip example above, preferring to specify the full pad stack in a single pad.

My pads for RF modules typically have a 0.4mm plated hole, with a 0.8 x 1mm rectangular top pad. The bottom pad is 0.8 x 2mm centred on the hole, providing a 0.8 x 1mm exposed copper area under the board. The surface mount pad on the host board for these pads is 0.8mm wide, and 1.8mm long centred under the module outline.

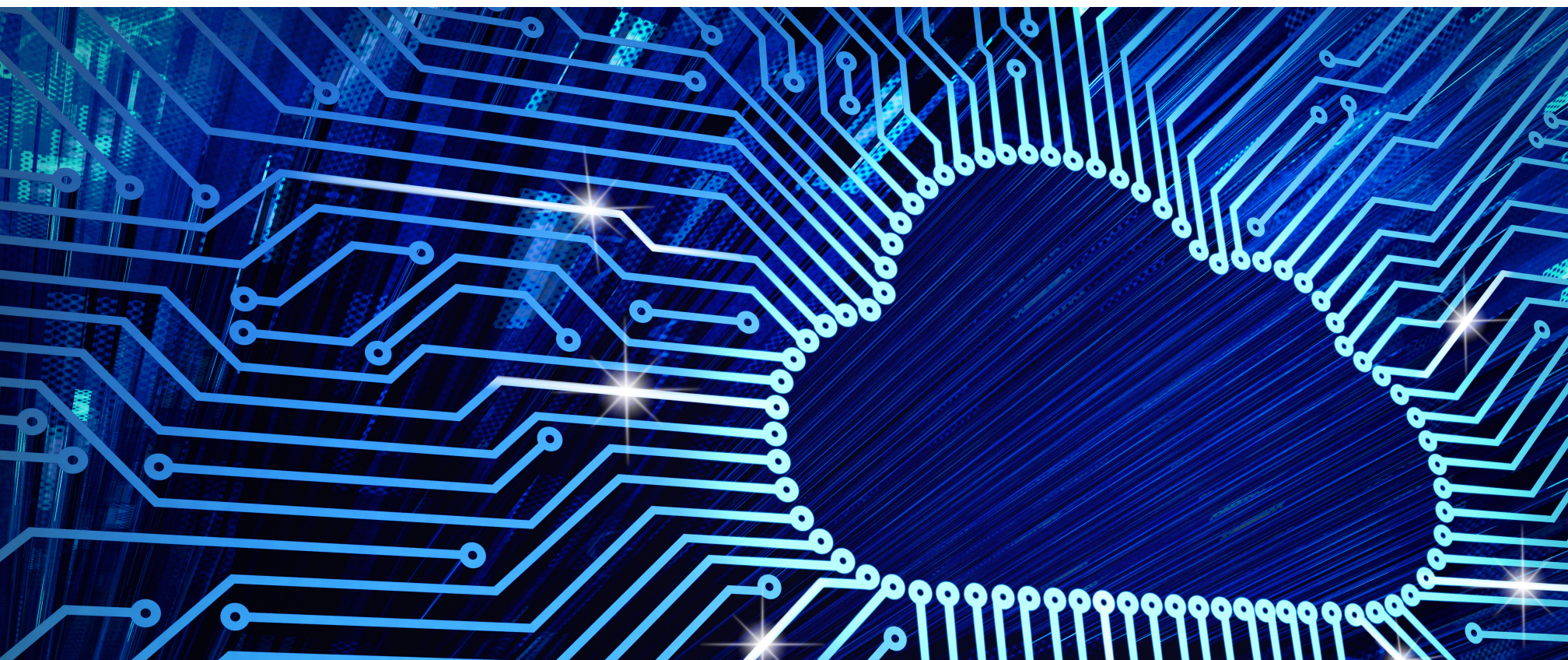


For lower volume boards that are almost certainly going to be hand soldered, I'll use larger holes where pin density isn't critical. These have a 1.2mm plated hole, and a 1.5 x 2mm top pad with a 2.5 x 2mm bottom pad. These have less exposed copper under the board, as the larger plated hole on the side will provide most of the solder contact when hand soldering them to the host board.

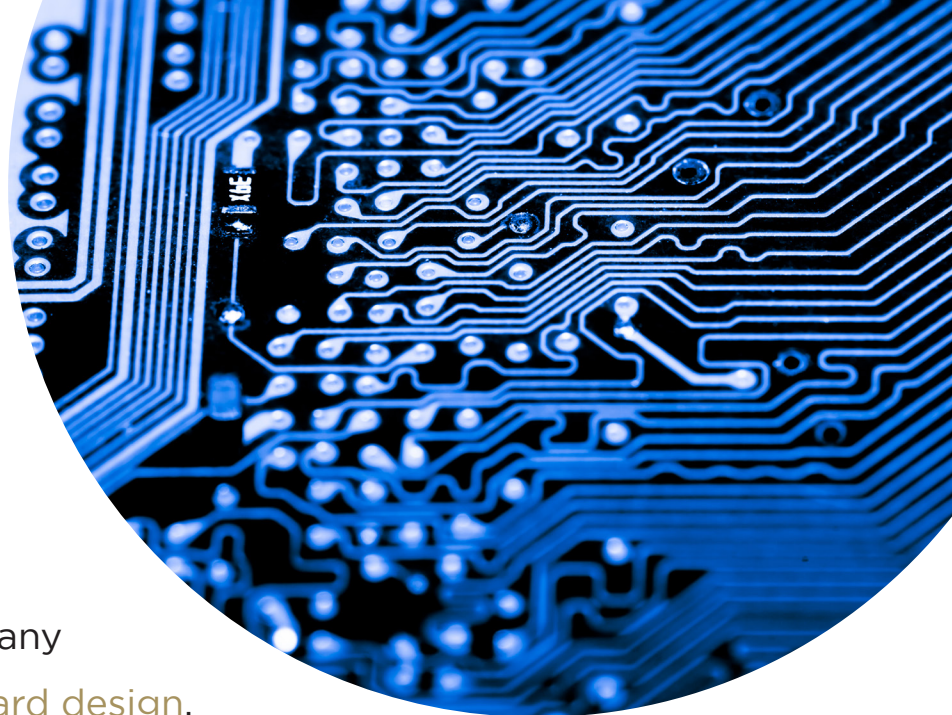


Your Own Module

When you create your own module, you should create your castellated pads as a single footprint/symbol, as this will make creating a multi-board project easier, and also ensure no single pad or cluster of pads can be accidentally moved causing alignment issues in the future.



Designing the Host Board

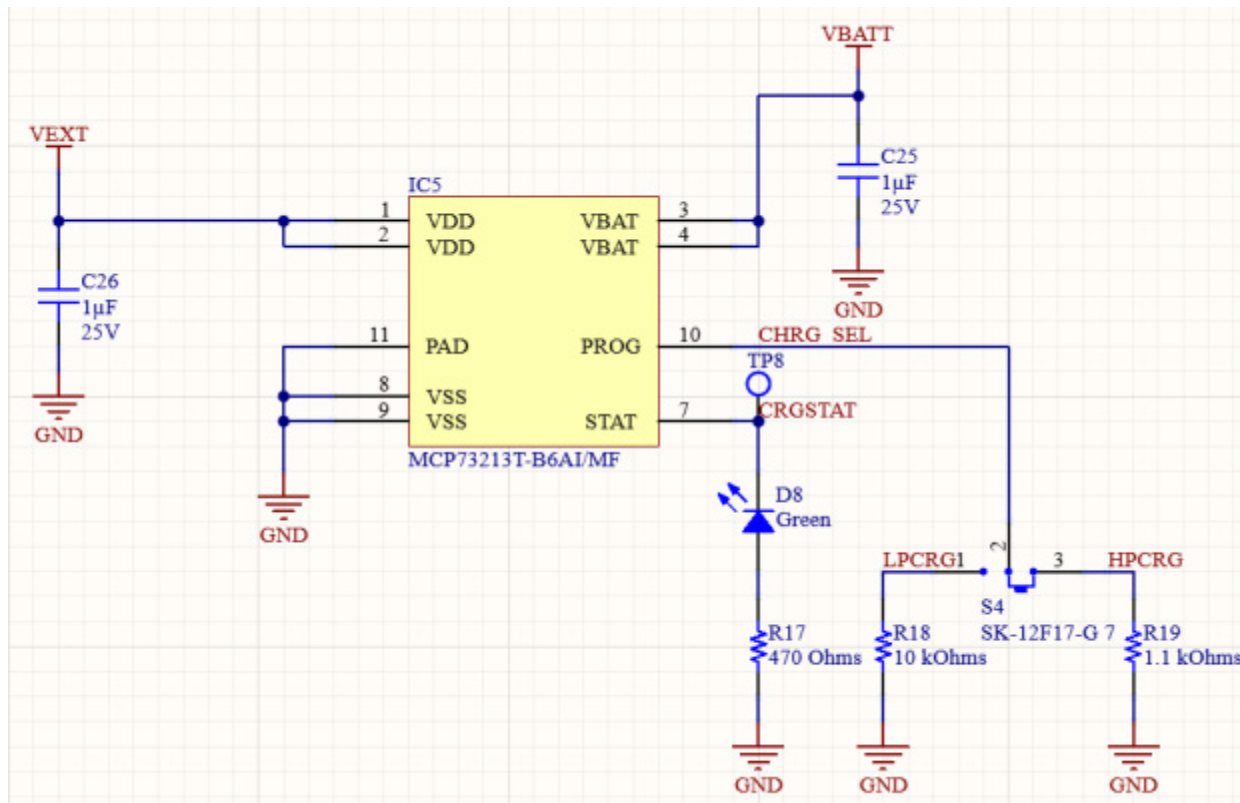


As mentioned earlier, we're going to be taking our previous sensor and LCD display schematic and making it a standalone board. Initially, this board is going to be developed using a library part that has the WiFi module as STEP model embedded in it. Many company libraries utilize this approach to **multi-board design**, however it is less than optimal. With **Altium's impressive MCAD-ECAD integration**, working directly with the board design makes more sense than exporting a STEP model of your module and re-importing it - especially early on in the design phase. As your module and host board requirements evolve, using a **multi-board assembly** allows you to ensure everything is kept in sync.

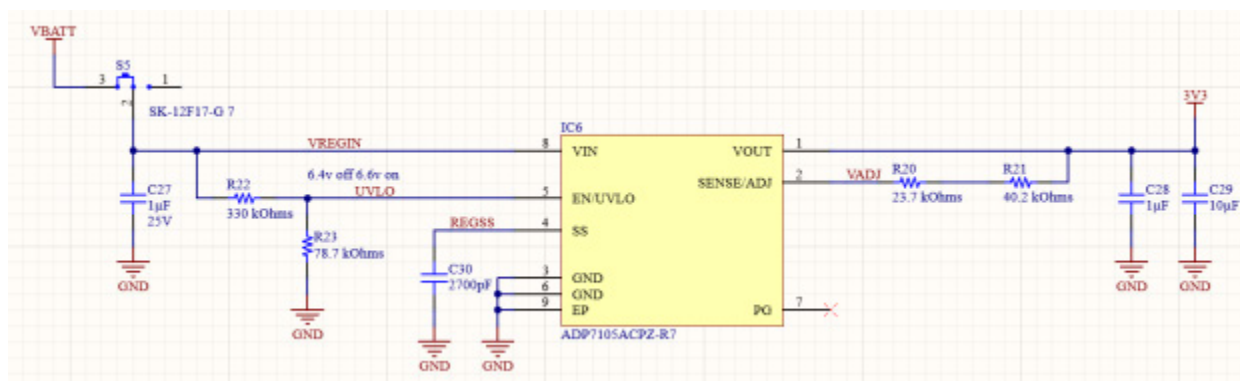
Once we have the board design completed using a library component, I'll show you how easy it is to switch your existing library based multi-board projects over to a footprint + multiboard assembly.

To make this hardware standalone and able to operate outside as a weather station we need a power source. I'm using the STM32L031K microcontroller which will be configured for **very low power consumption**, however I want to ensure the board can continue to operate without external power for some time if required. I'm using two 18650 batteries just like I did in my **12V UPS project**, however this time I'm using a different charger IC due to chip shortages and different requirements.

I would like my weather station to be **solar powered**, however I'm located in the very far north of Scotland where in winter we have 5 hour long days and can be trapped in a fog bank or dense cloud for days at a time - not exactly ideal conditions for solar powered devices. Using two 18650 batteries will allow me to run the weather station for around 6 weeks without external power if necessary. I also have a massive 6000uF capacitor on the input, to help deal with varying light conditions, keeping the charger running optimally.



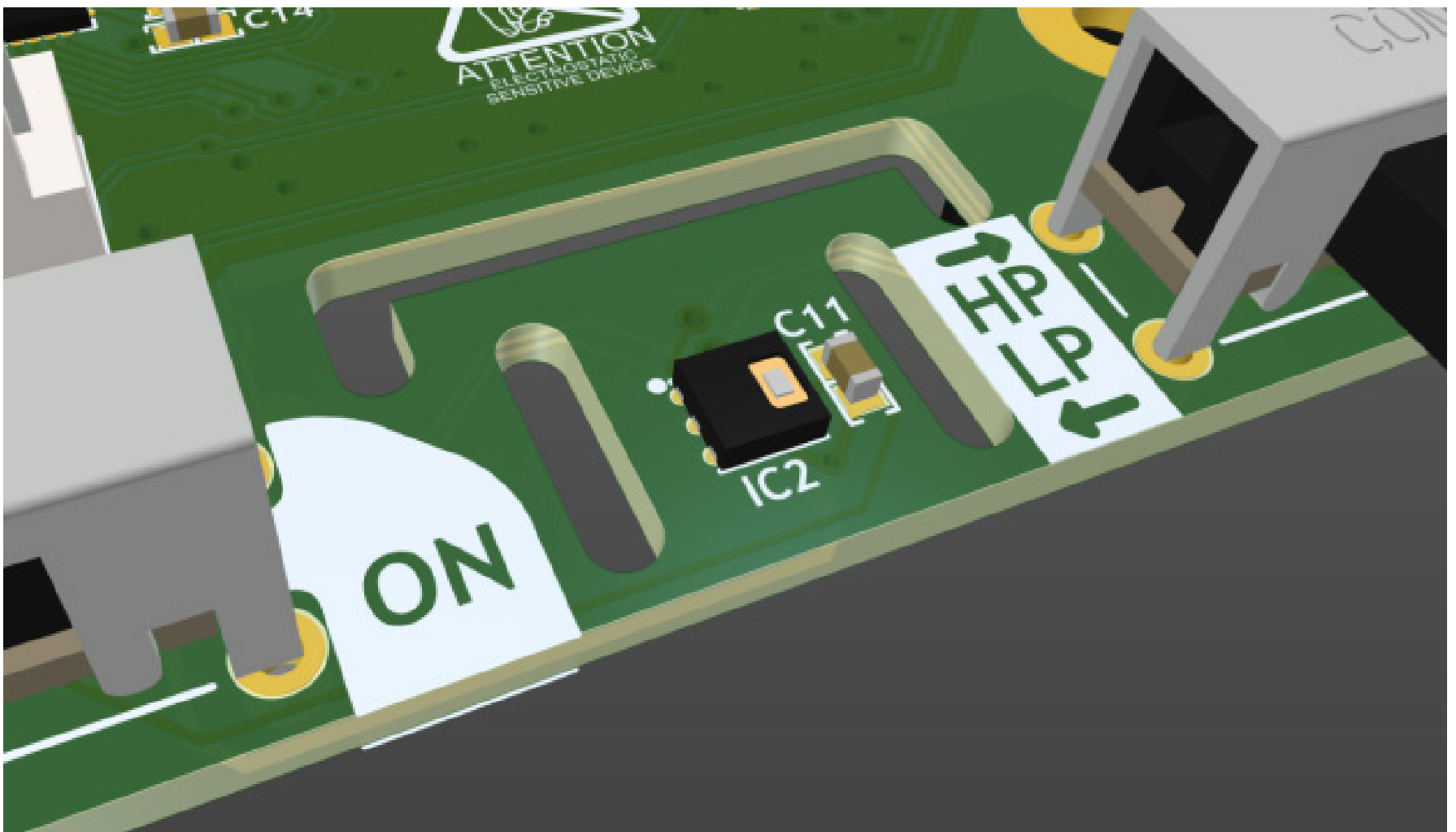
I'm using the Microchip MCP73213 two cell lithium battery charger IC for the project. It has a wide input voltage range which works well for either solar power or AC power if used indoors as well as programmable current. I'm specifically using the 8.2V charge model, despite my batteries being 8.4v maximum charge. As I expect these batteries to be at full charge most of the year, leaving them in a slightly less than full state should improve their lifespan, and with such high capacity the small hit to capacity shouldn't be noticed. I have the charge current selectable through a slide switch, so it can be set to a low current for solar powered applications, or fast charged by an AC adapter if needed.

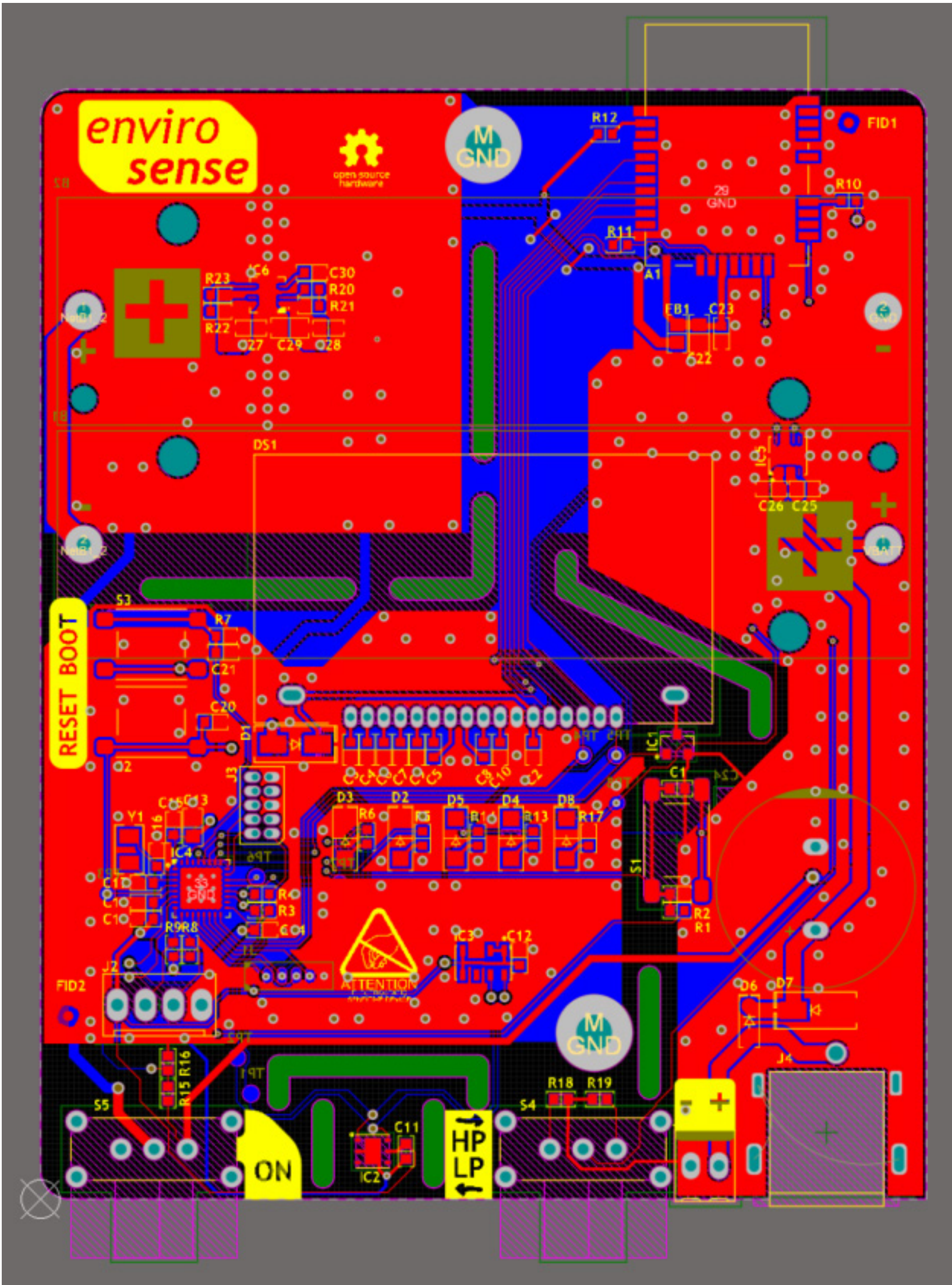


Rather than using a battery monitoring IC, I'm taking the simple voltage cutoff route in the event the batteries are fully depleted. I'm using a linear regulator for this design, the Analog Devices ADP7105, which has an Under Voltage Lock Out pin to switch the regulator off if the battery voltage goes too low. Typically running a 3.3v device on a linear regulator from a 8.4v supply would be considered very inefficient, and not fantastic for battery life. After a lot of searching for high efficiency light load switched mode regulators that are in stock, I found the linear regulator to be far more efficient overall.

The challenge with this device for a switched mode regulator is the WiFi module, with a transmit current draw of 290mA, however if we transmit data every 10 seconds then the remaining 98.5% of the time the current draw is likely to be in the low double digit microamps where a switched mode regulator really struggles. In this case, taking the efficiency hit on the transmit current is well worth it for the relative efficiency and reliability at low current.

Layout wise, I have separate ground pours for each of the potential temperature zones, keeping the regulator and charger at the top of the board with the WiFi module. I also have slots in the board around these potential heat generating components in an attempt to further break thermal conductivity to the temperature sensor. Just as with the shield, the temperature sensor is at the bottom of the board in it's own thermally isolated region.

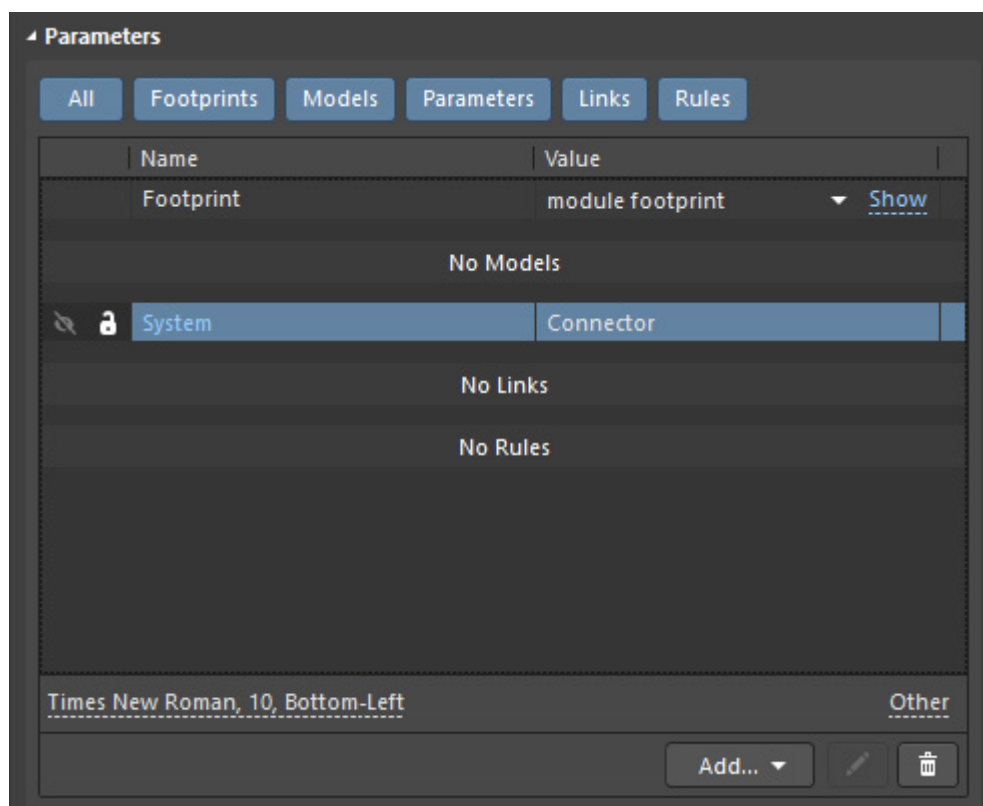




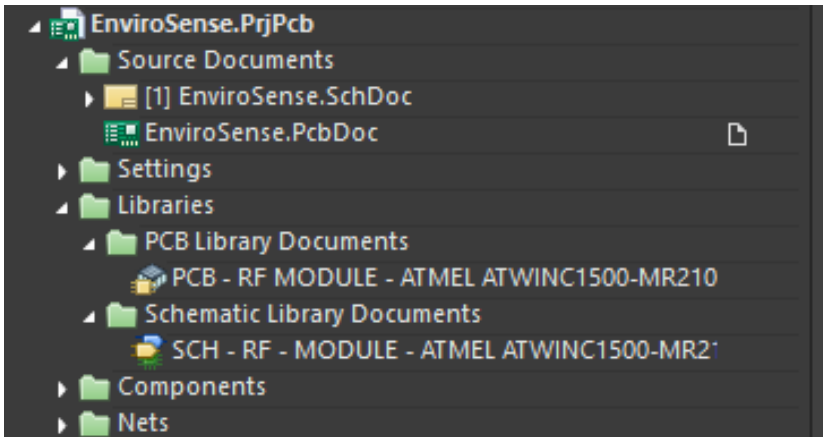


Switching to a Multi-Board Assembly

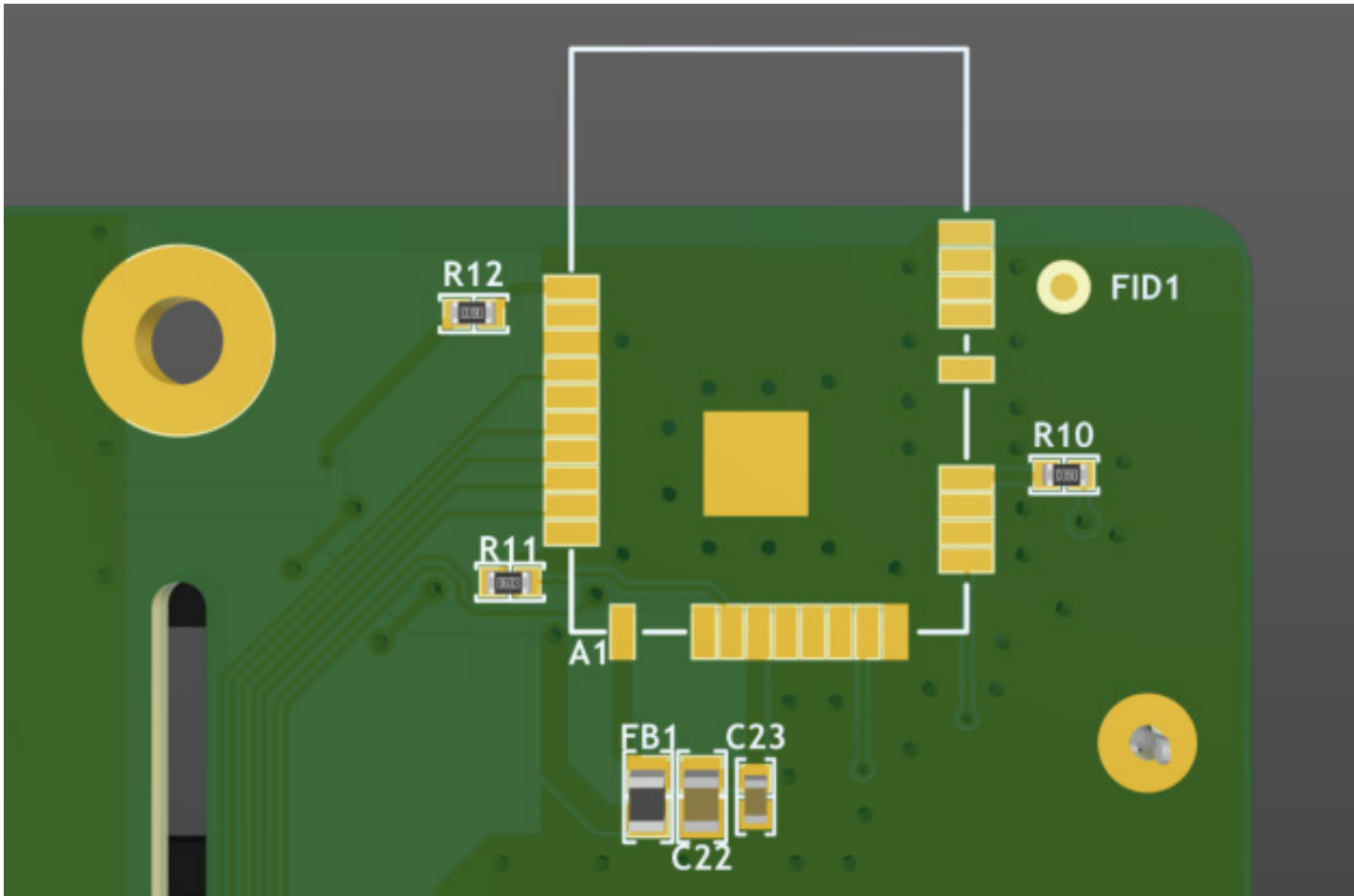
Up till now, this project has used a library item for the RF module, which is how your company might currently be dealing with internal sub assemblies. I downloaded the Altium files for the board from [Microchip's website](#). As the design files use a single footprint for the castellated pads all I needed to do was [tell Altium Designer](#) the connector is for a multi-board project by adding a parameter called "System" to the schematic symbol, and giving it the value "Connector". Preparing the sub assembly board for multi-board connection is that simple.



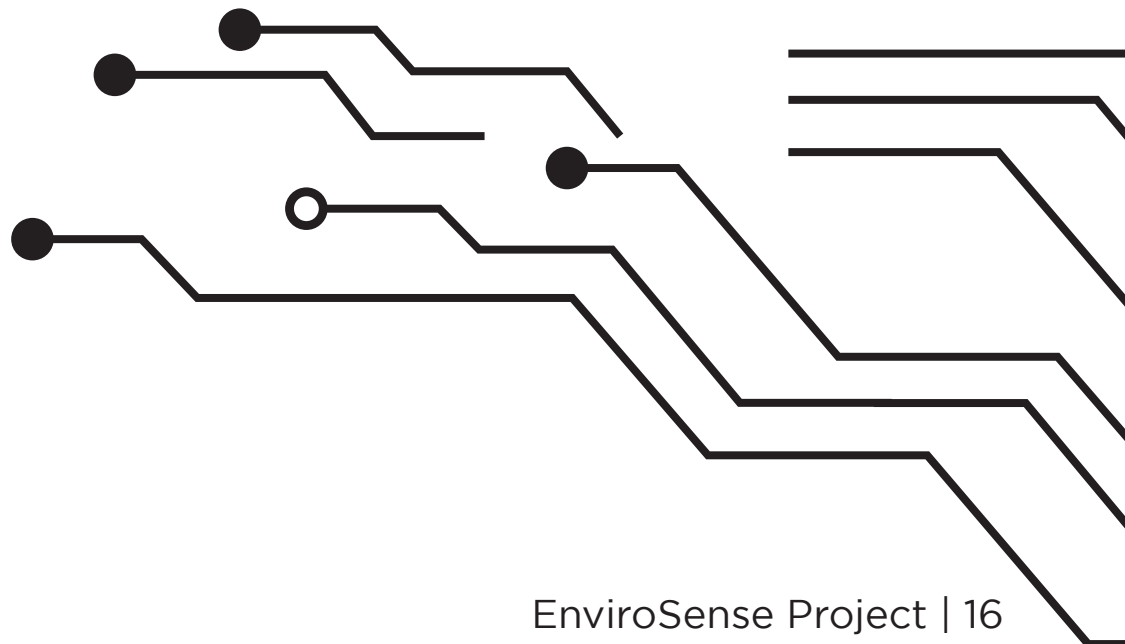
For the host board, I had to do a little more work, as I already had a footprint with a STEP model in it. I copied the symbol and footprint files from my library into the project folder so I would have a local copy to modify.



In the footprint, I deleted the 3D body, then added the footprint to the schematic symbol. After changing the symbol in my schematic to the new local symbol, I added the System = Connector parameter to the symbol. Finally, I updated the PCB document from the schematic, and then on the PCB ran an Update From PCB Libraries to refresh the footprint.

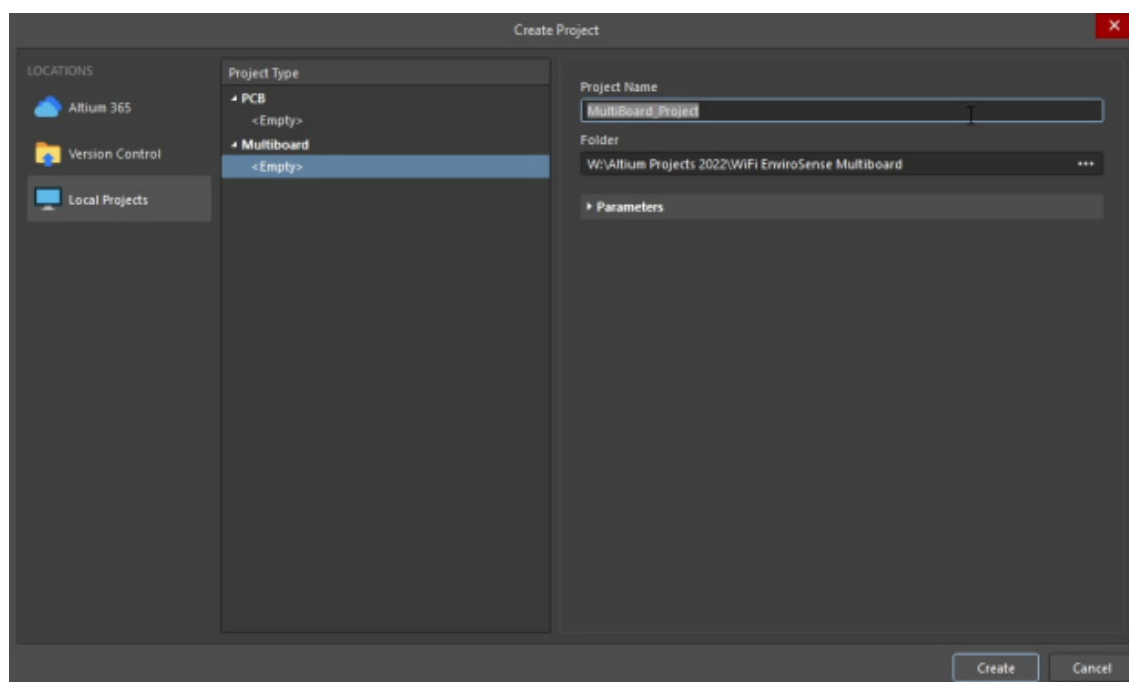


The PCB now has a footprint with no body and is marked as a multi-board connector ready for assembly.

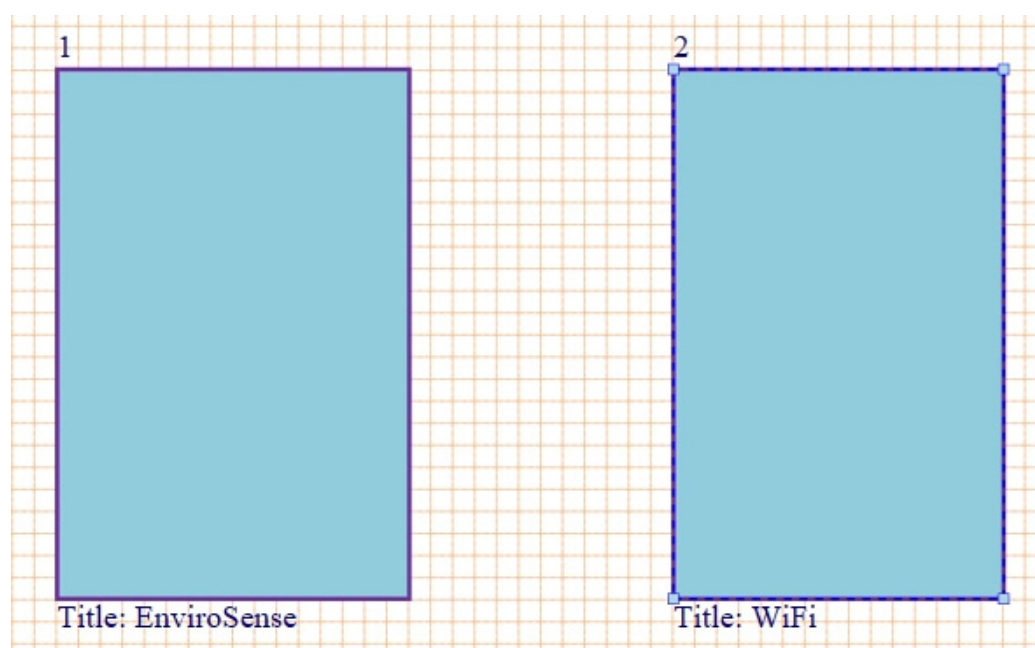


Creating the Multi-Board Project

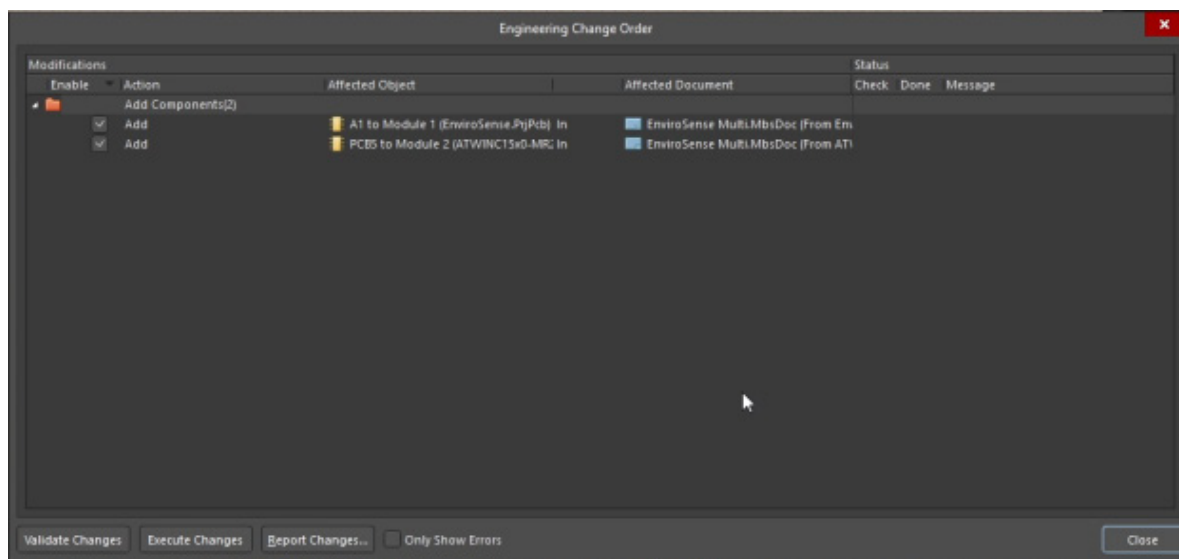
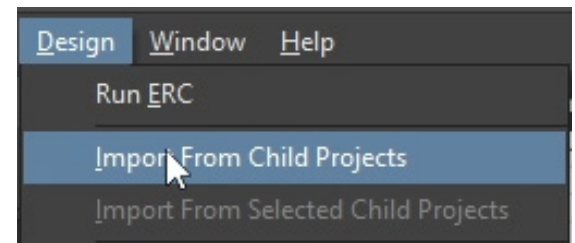
To add our RF module board to the host board, we need to create a new multi-board project. Creating a new multi-board project is almost the same as a typical PCB project, except we choose Multi-board rather than PCB under the project creation screen.



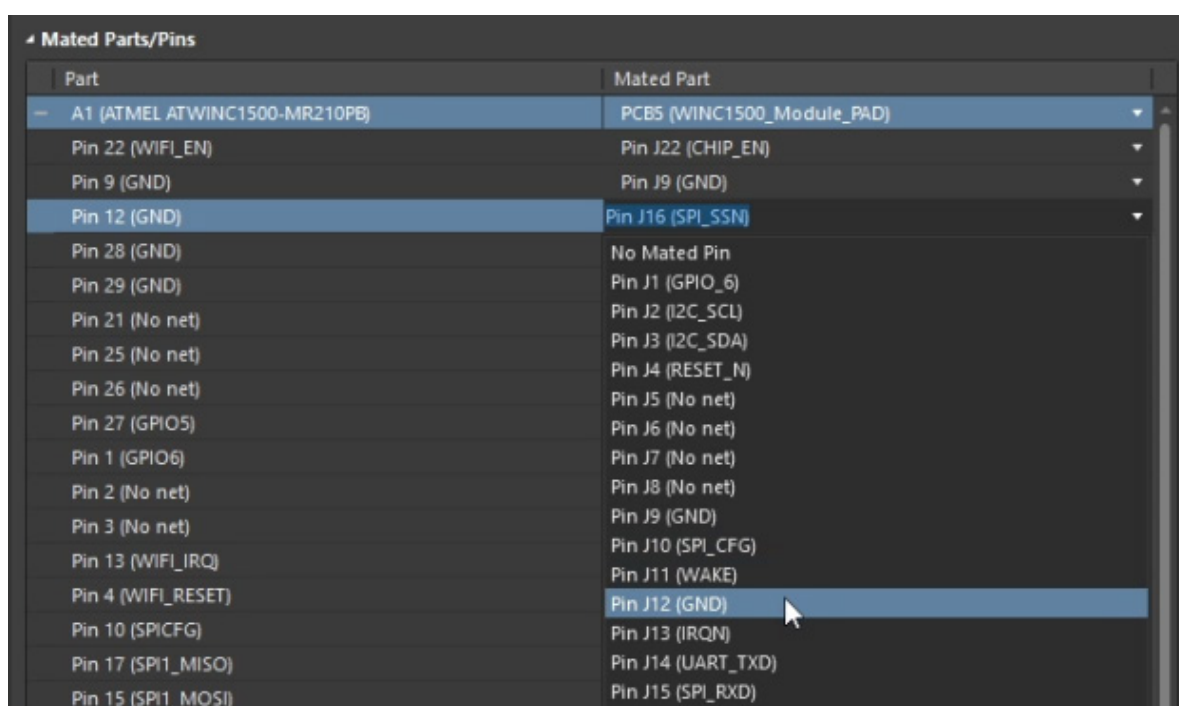
Next, we add a multi-board schematic and two modules. Each module is given a title and a project/board is selected for the module.



To add the connectors we set up earlier, we go to Design -> Import Child Projects which will add the connectors to the modules.



Then we can add a direct connection between the two modules. Altium Designer wasn't able to automatically match up the connections for the two modules this time, so we can click on one of the connections and in the properties window manually assign the mating pin.



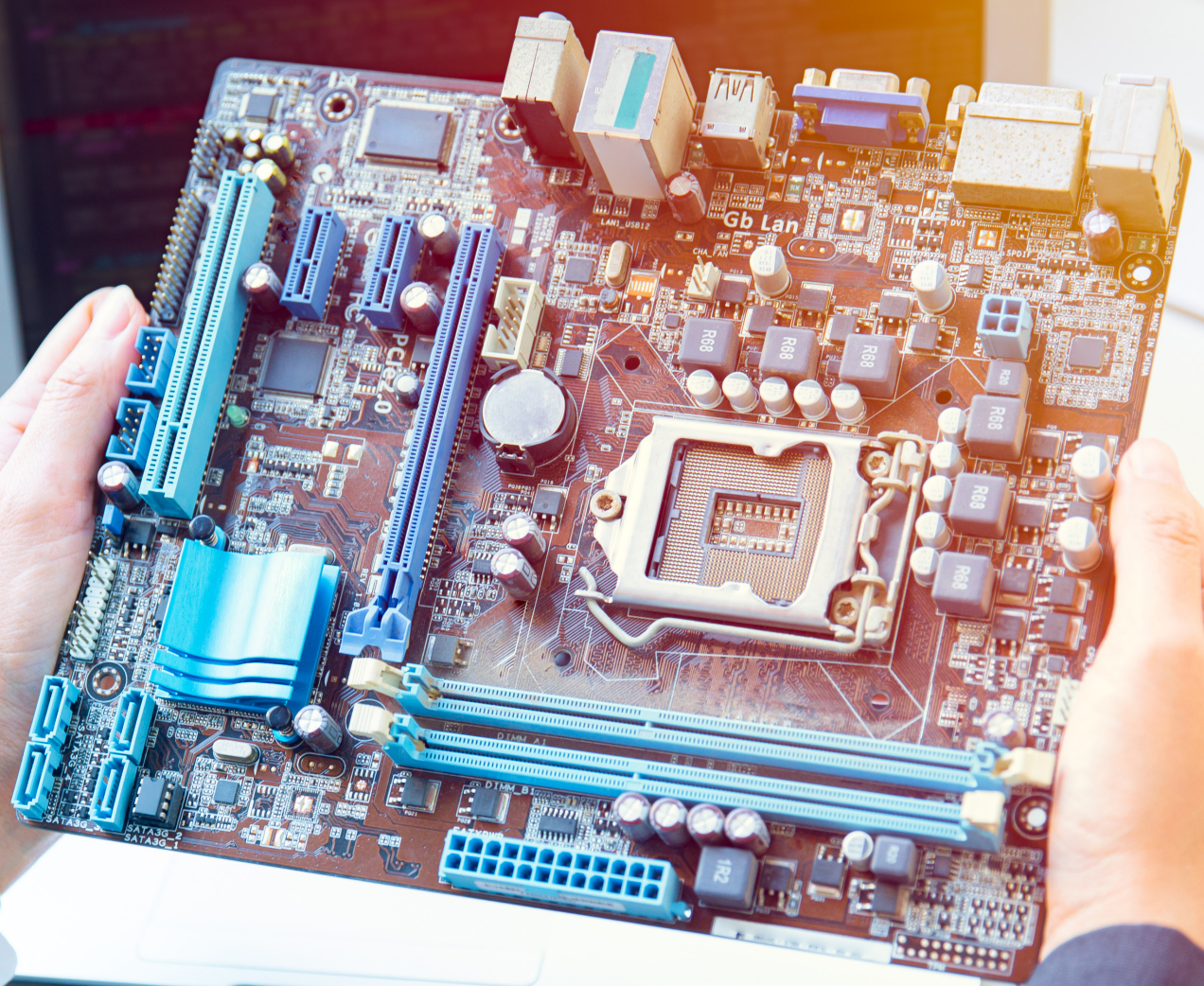
With the pin mated correctly, we can add the multi-board PCB to the project, saving it as soon as it has been added. Then, from the schematic, go to Design -> Update Assembly to import the modules to the multi-board PCB. Depending on the complexity of your hardware, and the specs of this machine this might take some time. However, for this relatively simple project, it took under a second on my computer.

Outcome

In this article we have built an environmental sensor board that could act as a simple weather station for indoor or outdoor use, demonstrating the usefulness of a pre-certified RF module sub assembly. We have discussed how and why you would create a sub assembly module for your own projects. Using a multi-board assembly rather than a static 3D model in a library for custom surface mount modules ensures your boards stay in sync throughout the design process, and allows superior mechanical integration between the boards.

I also have a [Davis Instruments Vantage Pro2](#) weather station, so it will be interesting to compare the long-term data off this weather station project to a NIST calibrated product.

How will multi-board assemblies save you time, and increase your productivity? [Talk to an expert at Altium](#) now to find out.



ABOUT ALTIUM

Altium LLC (ASX: ALU) is a multinational software corporation headquartered in San Diego, California, that focuses on electronics design systems for 3D PCB design and embedded system development. Altium products are found everywhere from world leading electronic design teams to the grassroots electronic design community.

With a unique range of technologies Altium helps organisations and design communities to innovate, collaborate and create connected products while remaining on-time and on-budget. Products provided are Altium Designer®, Altium Vault®, CircuitStudio®, PCBWorks®, CircuitMaker®, Octopart®, Ciiva® and the TASKING® range of embedded software compilers.

Founded in 1985, Altium has offices worldwide, with US locations in San Diego, Boston and New York City, European locations in Karlsruhe, Amersfoort, Kiev and Zug and Asia-Pacific locations in Shanghai, Tokyo and Sydney. For more information, visit www.altium.com. You can also follow and engage with Altium via [Facebook](#), [Twitter](#) and [YouTube](#).

