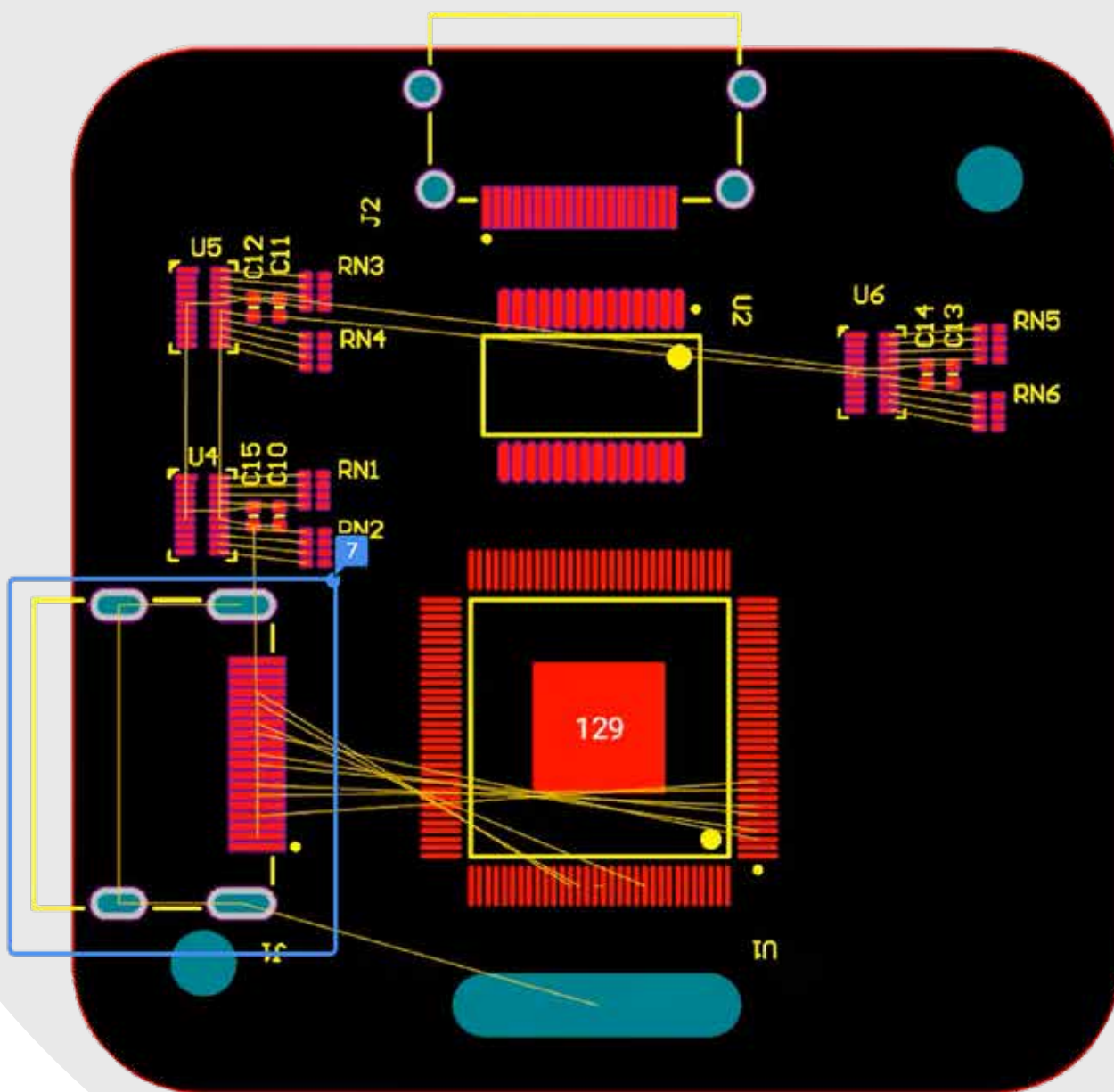
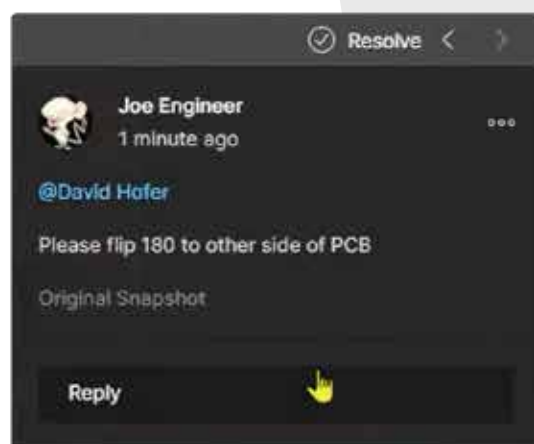




Team Collaboration & PCB Design

The Ultimate User Guide

Altium®

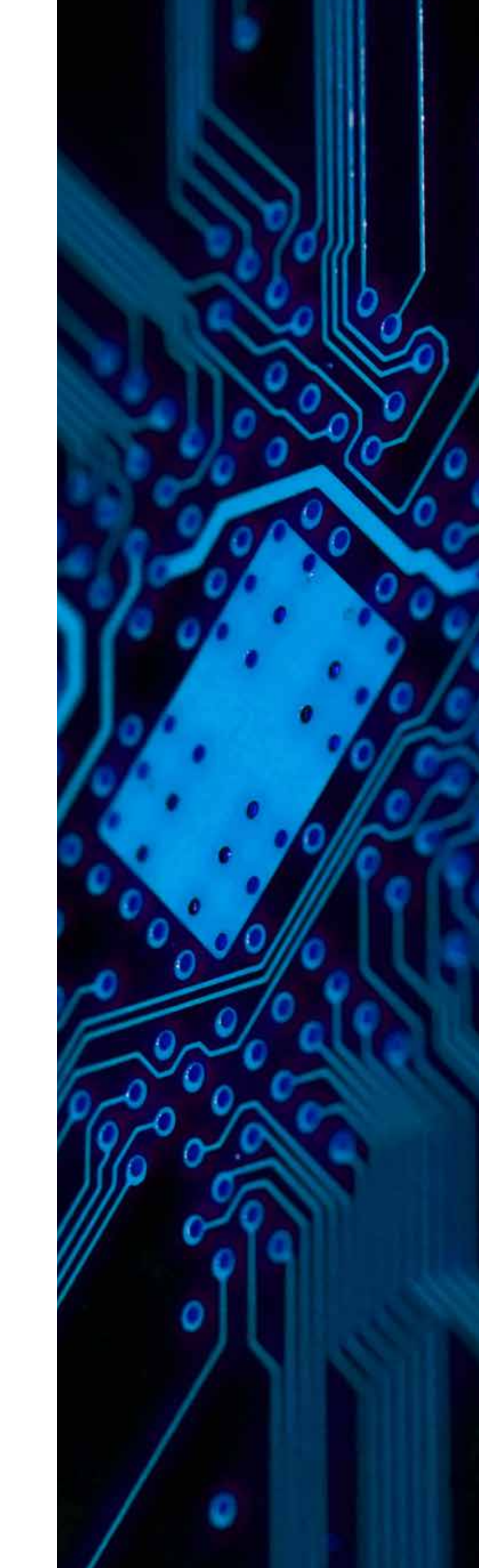


Table of Contents

Top 4 Time Wasters in PCB
Design Collaboration *p3*

Mechanical and Electrical
Design Come Together With
Altium's MCAD CoDesigner *p8*

Sharing PCB Data with Manufacturers
in the Cloud *p14*

How to Speed Up Your DFM
Report Process in the Cloud *p20*



Top 4 Time Wasters in PCB Design Collaboration

Don't let your time get wasted during PCB design collaboration. Try these strategies and tools instead.

An essential aspect of project management is time management, especially when your design team is working remotely. Your time management strategy is team-based and individual, but time can easily get spent on important tasks when working as part of a team. So how can you streamline important collaboration tasks for your design team to increase productivity?

The first step is realizing where time is being wasted during design, followed by finding the right tools to streamline communication, sharing, and collaboration on complex PCB designs. After managing a remote team of PCB designers, I've taken some pains to reduce time spent on tracking design progress, communicating with stakeholders, and getting designs completed on-time. If any of these resonate with you, there are some simple cloud collaboration tools that can save your design time.

Four Time Wasting PCB Design Collaboration Problems

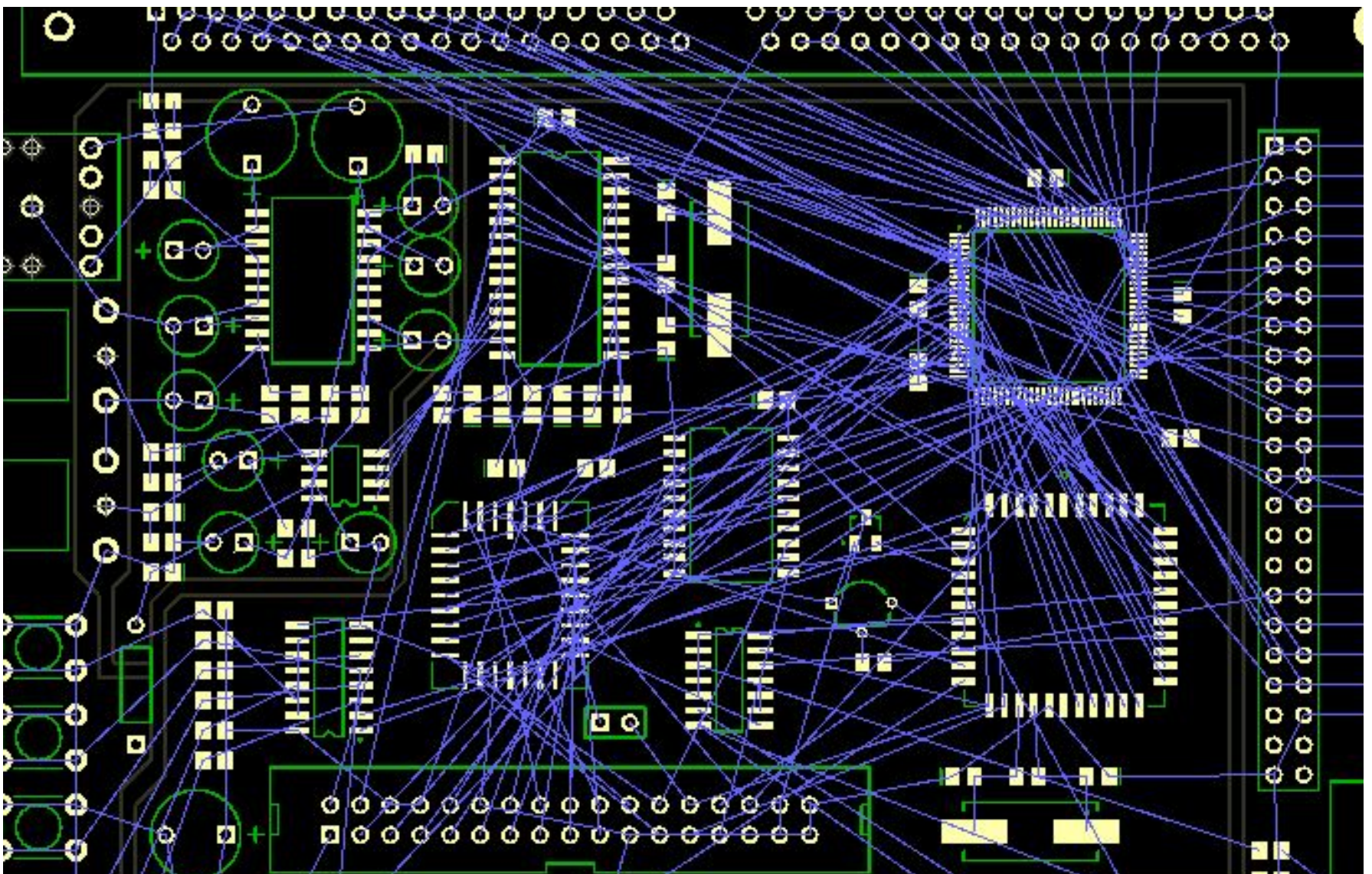
Even if you're a PCB design expert, working with customers and other team members in a remote environment requires communication and sharing tasks that can delay any project. Here are some of the top time-wasters I've noticed while working with team members.

Getting Questions Answered by Customers and Stakeholders

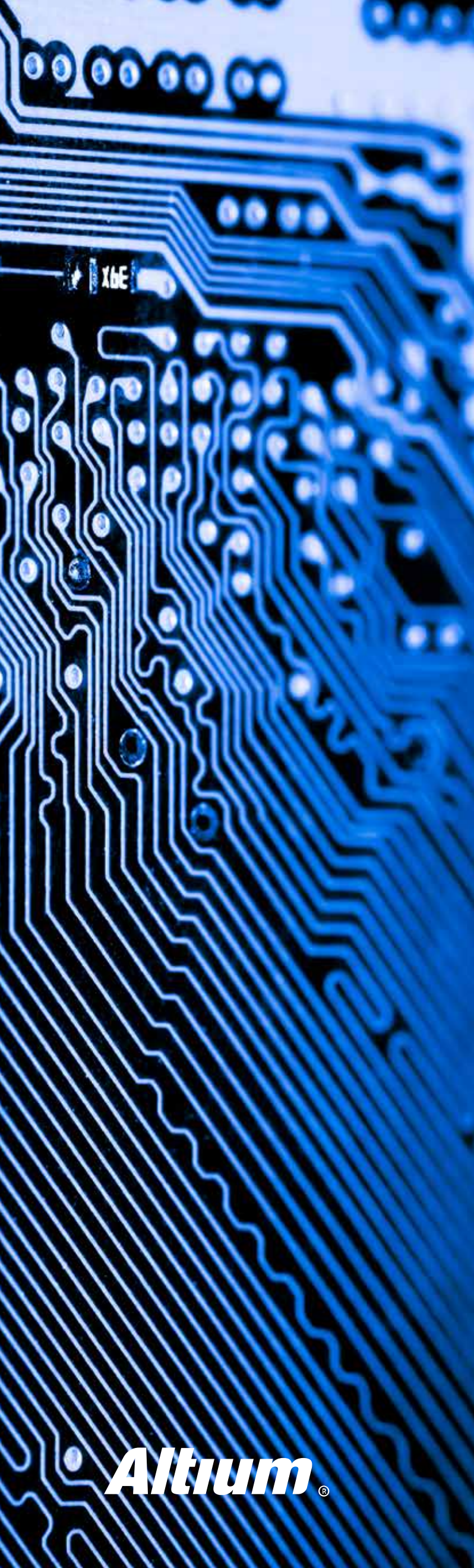
For me, this is probably the top time waster during a project. Design cycles are extremely short, and anytime a question arises for the customer it needs to get resolved quickly. Some design questions can stall an entire project, not to mention hold up moving to manufacturing. Even worse, some customers might want to take a hands-off approach; they assume you can read their mind, so they don't make themselves available for questions.

Whenever there is a question on a design, or you need to flag an error for the customer, you can't go further until you get their input. If questions go unanswered, the design gets held up and the schedule gets pushed back; no one is happy when this happens. Some common questions for customers and project stakeholders relate to:

- **Errors in a design document.** Our customers will sometimes provide schematics or a layout with some component placement. Errors aren't always obvious when you start a complex PCB layout, and you might be partially through a layout before something serious is noticed. If there is a problem in either of these, the customer needs to resolve this quickly before a PCB layout can get finished.
- **Unavailable or obsolete components.** When **ideal components are unavailable** or obsolete, you need to notify the customer and provide suitable alternatives. It helps to have **supply chain visibility tools** to quickly find alternative components, ensure they're in-stock, and get prices.
- **Must-haves and nice-to-haves.** Some design requirements cannot be violated (must-haves), such as mechanical requirements for an enclosure (this is one of the most common for me). Other requirements can be compromised (nice-to-haves), but there needs to be a protocol for implementing changes in case a design change might violate one of these requirements.



Certain ICs and connectors can have very specific placement requirements, and you'll have to discuss with customers or other team members when these need to be changed or moved.



Implementing Design Changes as Specified by Customers

Once you do get clear responses from the customer or stakeholders, it can spawn another round of questions around reviewing and accepting the design changes. This back-and-forth process can involve screenshots of a design, email chains, and copious video chats. If you're working on a regulated or sensitive product, such as an ITAR-regulated product, you need to use an FTP portal or other secure server to share design data. All of this takes up time, with most of the time spent waiting for responses to questions and reviewing design changes.

Communicating with Multiple Designers

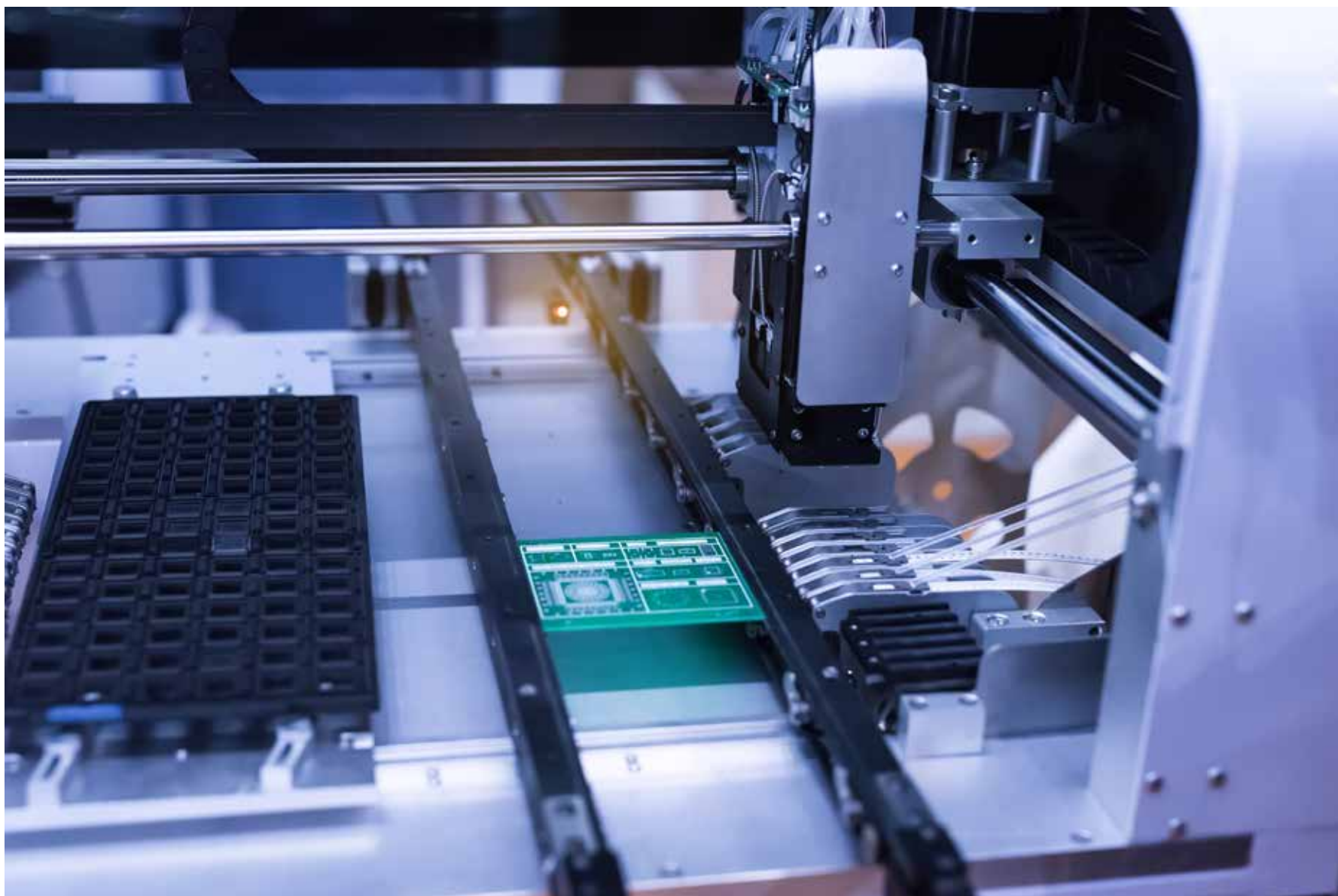
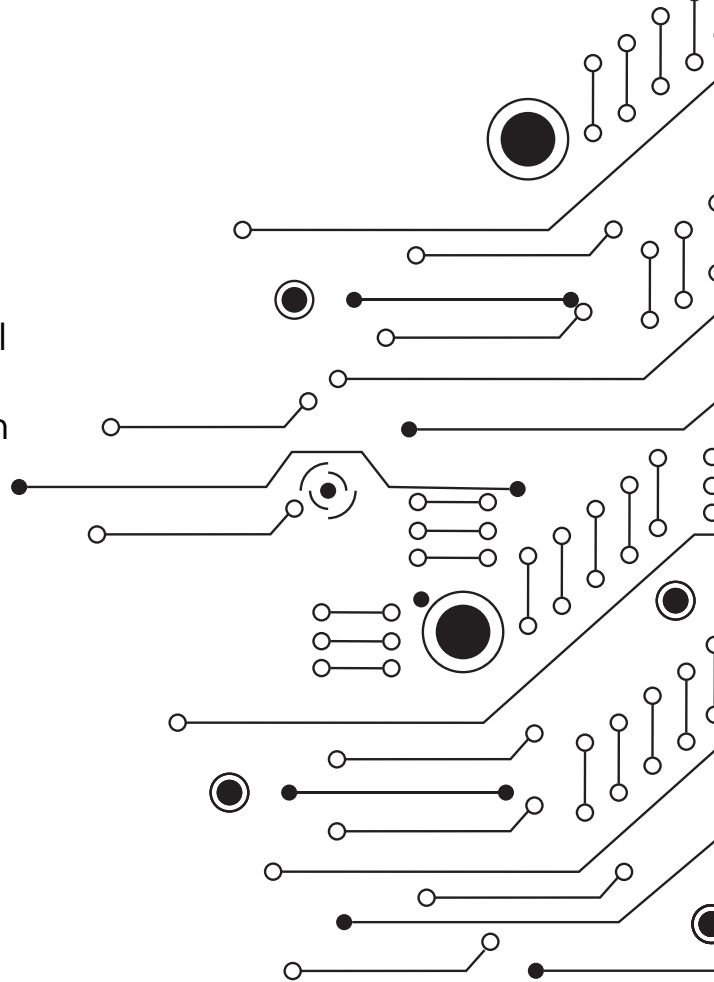
Remote PCB design teams need a single set of tools for communication between team members. Slack and Skype are great for this, but quickly sharing and reviewing design data takes time as well; you might as well send around emails with screenshots and design files as attachments. Going back-and-forth between different communication channels, especially when reviewing and editing design data with team members, also takes up significant time.

My opinion on the matter is: try to consolidate everything into as few communication channels as possible. Ideally, you should have one chat channel per project, phone calls/text messages (emergencies only), and a tool for sharing and annotating design data (NOT email). Keep these channels dedicated for certain tasks and topics, don't let people have a free-for-all in your chat channels. If you do use something like Skype or Slack for project communication, create a channel/room for each project to keep everyone on track.

Sharing Design Data

If you're **collaborating with a remote team**, you need a tool to quickly share design data while tracking revisions in a single location and controlling user access. No one should have to deal with separate chains of design files, and team member visibility should be controlled when needed. The best cloud collaboration tools will integrate with your PCB design software while also providing revision tracking and user access control.

Once you're ready to prepare for fabrication and assembly, these same sharing and communication tasks need to apply to your manufacturer. If you can (or if they're willing), loop them into your chat channels with your lead designer. Sometimes, you're at your manufacturer's mercy as far as which communicating and sharing channels they prefer to use, but bringing your manufacturer into your design process early can help you **get through a design review faster**.



The right sharing tools can help you get through a design review quickly and get your circuit board into fabrication.



Solving PCB Design Collaboration Problems in the Cloud

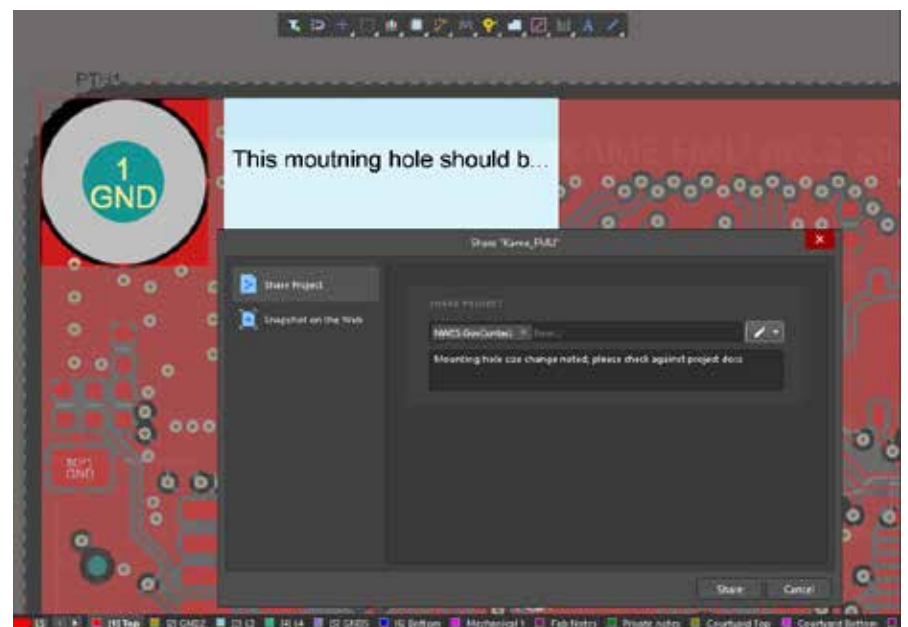
From the list I've shown above, three of the four time wasting problems in PCB design collaboration revolve around communication. The communication aspect is understandable; people go on vacation or take personal time, they are involved in other projects that require attention, or they need to get an answer from a 3rd party. Tools like Slack, Skype, text messages, and old-fashioned email have their place, but real collaboration and sharing takes an integrated approach.

When you use **Altium Designer** on the Altium 365 platform, you'll have access to supply chain data, commenting features, and data sharing tools to help you streamline PCB design collaboration. Your team can resolve questions and comments directly on design data rather than spending time using outdated communication tools for sharing data.

You'll have access to many other features to help your team stay organized and work efficiently. Version control, user access management, and revision tracking features are built into the **Altium 365™** platform. Your team can take advantage of a single tool for design data access, data sharing, and communication that integrates with Altium Designer.

Altium Designer® on Altium 365 is bringing an unprecedented amount of integration to the electronics industry until now relegated to the world of software development, allowing designers to work from home and reach unprecedented levels of efficiency.

We have only scratched the surface of what is possible to do with Altium Designer on Altium 365. You can check the [product page](#) for a more in-depth feature description or one of the [On-Demand Webinars](#).



Comments can be placed directly on design data with Altium Designer and Altium 365. This helps you quickly pin down needed design changes and gives everyone visibility into design progress.

Mechanical and Electrical Design Come Together With Altium's MCAD CoDesigner

Electronics designers working in small design houses or in large enterprises often encounter a common set of challenges when interfacing with mechanical designers. For many new products, the mechanical form factor requirements will constrain the electrical design in terms of physical layout, location of interfaces, and component selection.

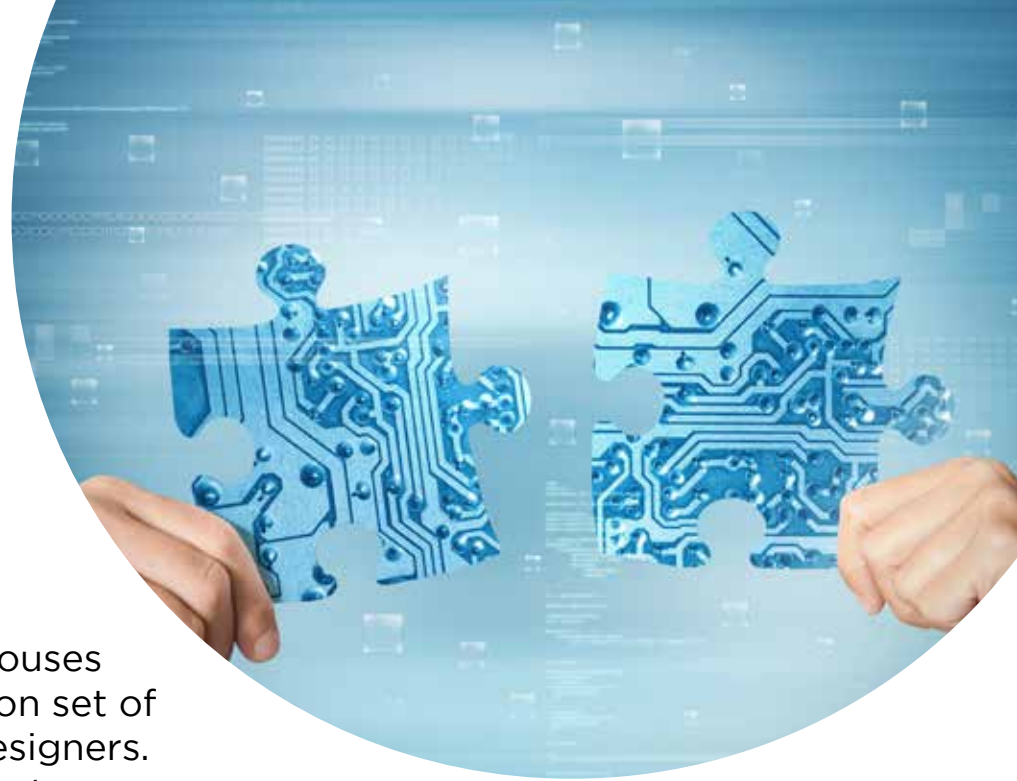
For many electronic products, flex and rigid-flex board designs require careful modeling of the mechanical behavior of the flex region to ensure form factor constraints are satisfied and the flex regions are reliable. These problems are compounded by faster product release cycles, requiring electrical and mechanical engineers to work together more than ever.

Success in these areas requires close collaboration between MCAD and ECAD domains to ensure mechanical constraints are not violated and the finalized design can be produced at scale. Because today's design and data management systems still lack basic integration features, collaboration between these domains still relies on file exchange between each side.

Bringing ECAD and MCAD Together

Altium has stepped up to help design teams overcome these productivity challenges with the MCAD CoDesigner extension. This simple utility connects Altium Designer projects to popular MCAD applications via the Altium 365 platform. By unifying the traditional ECAD/MCAD workflow, multifunctional design teams can eliminate many of the manual file exchange processes normally required as part of enclosure design, defining constraints, and interference back-checking.

Innovative companies like Quantel Laser use the MCAD CoDesigner to collaborate on all aspects of product development. Mixed design teams can have visibility into ECAD and MCAD data via the Altium 365 platform, which helps streamline product development and eliminates many inefficiencies in ECAD/MCAD collaboration.



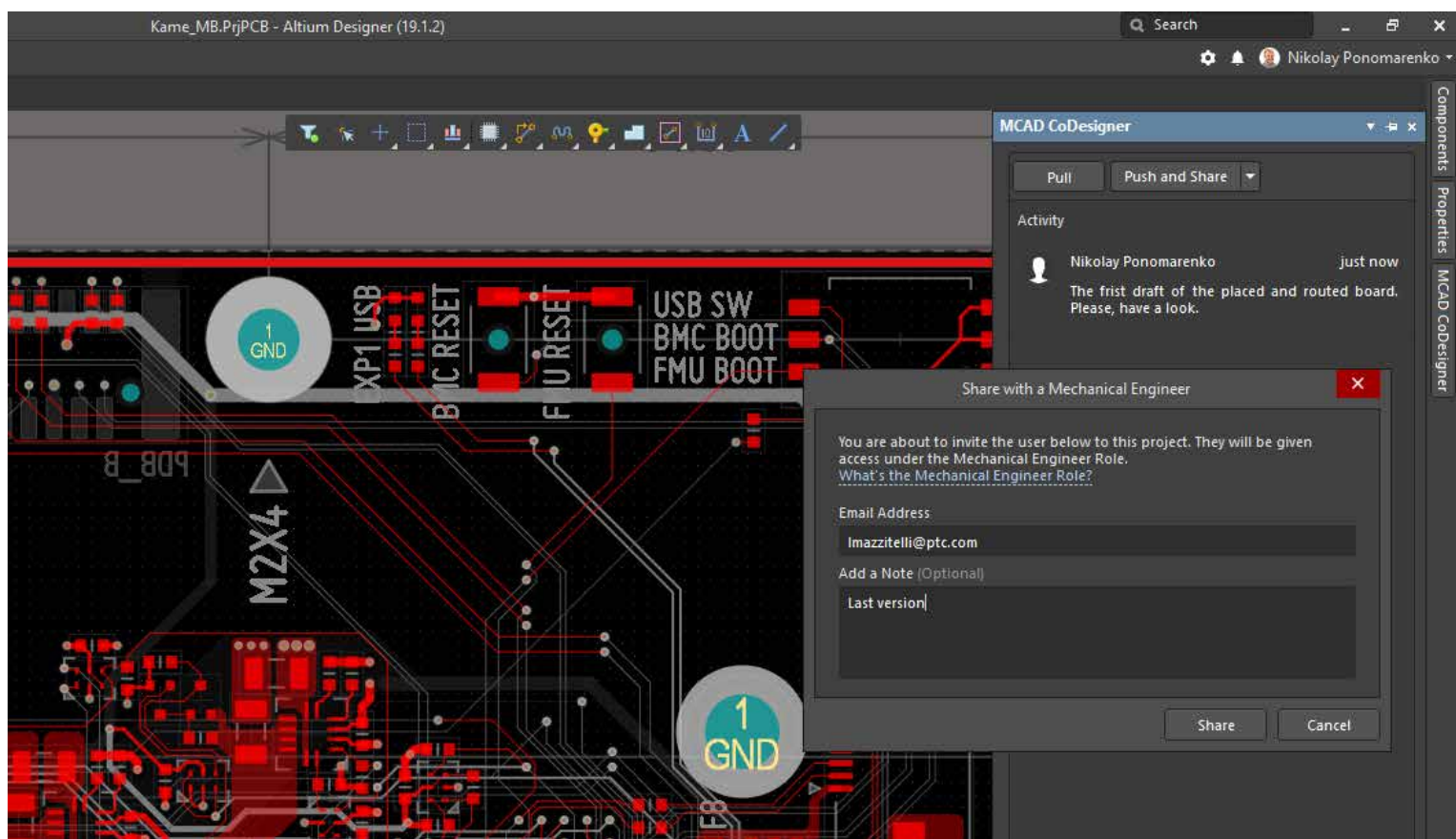
Empowering More Efficient Workflows

The CoDesigner panel in Altium Designer and its corresponding add-on panel in MCAD software allows electrical and mechanical designers to interact seamlessly, enabling an efficient collaborative workflow. Behind the scenes, this workflow is facilitated by robust server-side support allowing design changes to be transferred, reviewed, and accepted or rejected without the need for manual file transfers in lossy file formats. The CoDesigner extension provides best-in-class model support by preserving references to board features, ensuring mechanical updates in one application are accurately reflected in the corresponding application.

After the initial component placement is completed by the PCB layout engineer, it becomes the mechanical engineer's job to check that everything fits the enclosure and communicate required changes as necessary. In many cases, mechanical designers need to perform detailed mechanical checks, finite element analysis (FEA), and modify placement to ensure mechanical constraints are satisfied. The CoDesigner extension enables these tasks and many more for PCB assemblies with synchronization and modeling features.

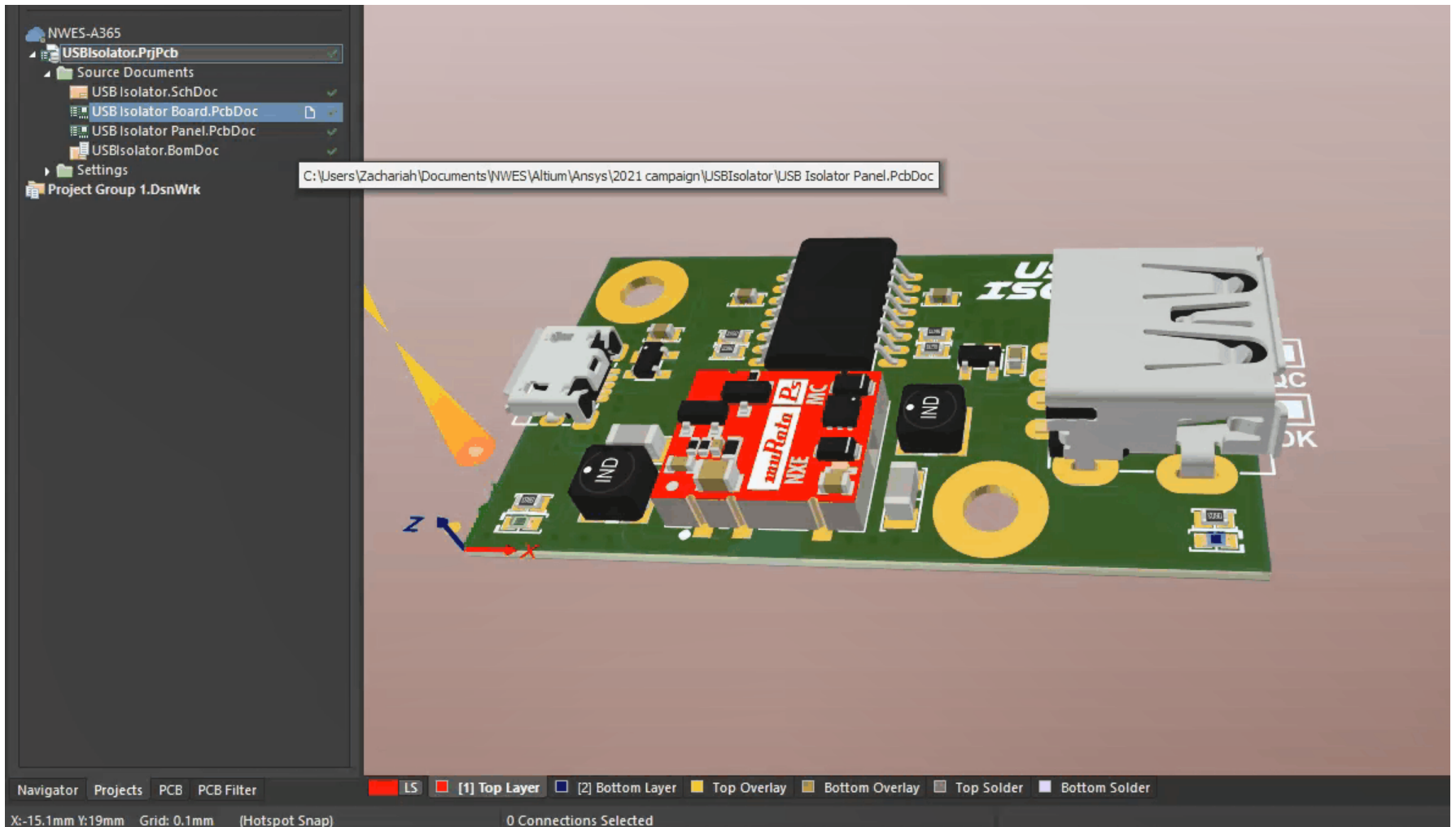
Design Data Synchronization

The MCAD CoDesigner capability implements a simple Push-Pull process, where updates are exchanged between each side with a button in the CoDesigner panel. Updates on one side can be transferred to a colleague in seconds, and this triggers a notification in the collaborator's CoDesigner panel. Each side can keep track of changes by adding comments to each Push operation. This instant exchange of critical ECAD and MCAD data is accomplished without the need for manual IDF/IDX/STEP/DXF file imports and exports. Engineers in both domains can focus on design instead of creating and transferring files.



The push-pull functionality in the MCAD CoDesigner makes an Altium project accessible to a mechanical engineer from within Altium Designer.

Once the design is imported into an MCAD application, any changes made in the MCAD tool are synchronized back to the ECAD side with the Push-Pull functionality. Once the design is pulled back into Altium Designer, the PCB layout data will immediately update to reflect changes in the board outline, copper, hole placement, or component locations. Similarly, subsequent changes in Altium Designer can be pushed back into the MCAD side, causing the board model to update once the design is pulled back into the MCAD application. This back-and-forth synchronization is also applied to the enclosure, which can be pushed into Altium Designer.

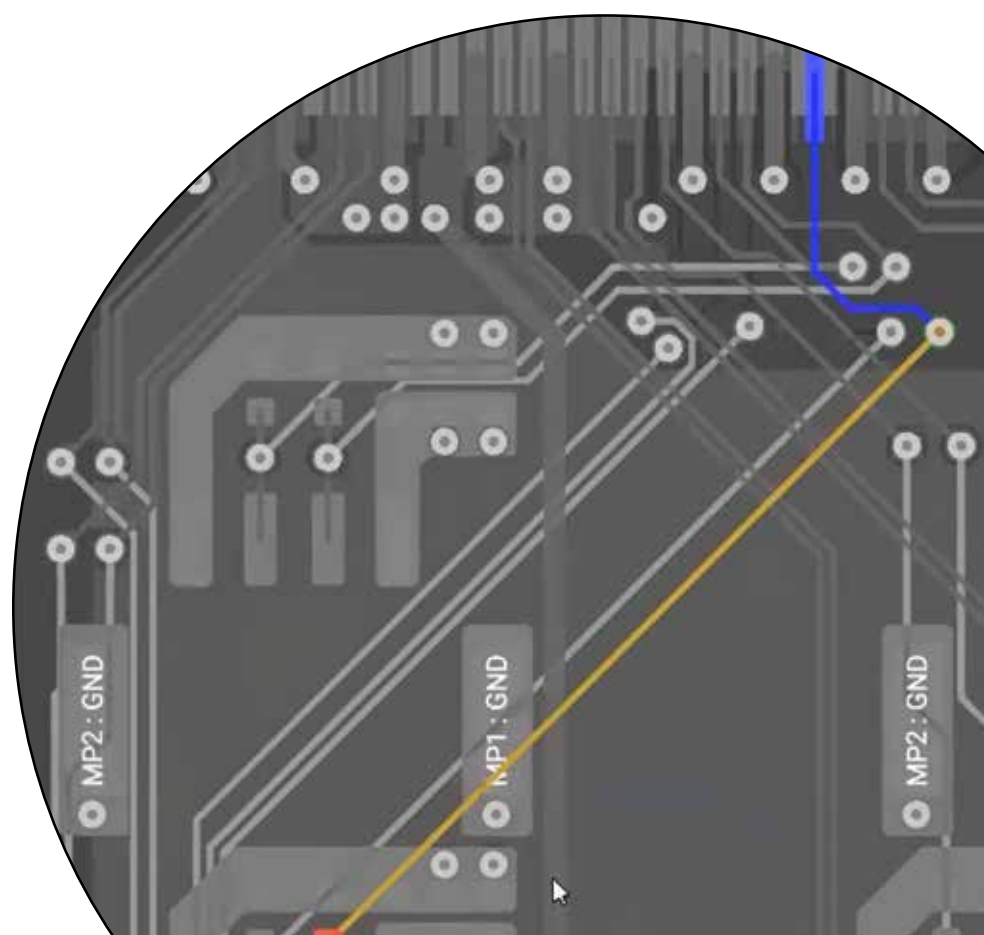


Altium Designer users can view their enclosure in Altium Designer with the push-pull functionality. The MCAD CoDesigner plugin quickly synchronizes ECAD data to Altium 365, and designs are immediately available in MCAD applications.

Precise Model Representation

Once a PCB assembly is imported into an MCAD application, the mechanical engineer needs to have precise board geometry, and, in many cases, a precise definition of copper and solder mask. That precise model can be used for performing detailed mechanical checks and for finite element analysis (FEA) simulations, such as thermal analysis or vibration analysis.

Altium®



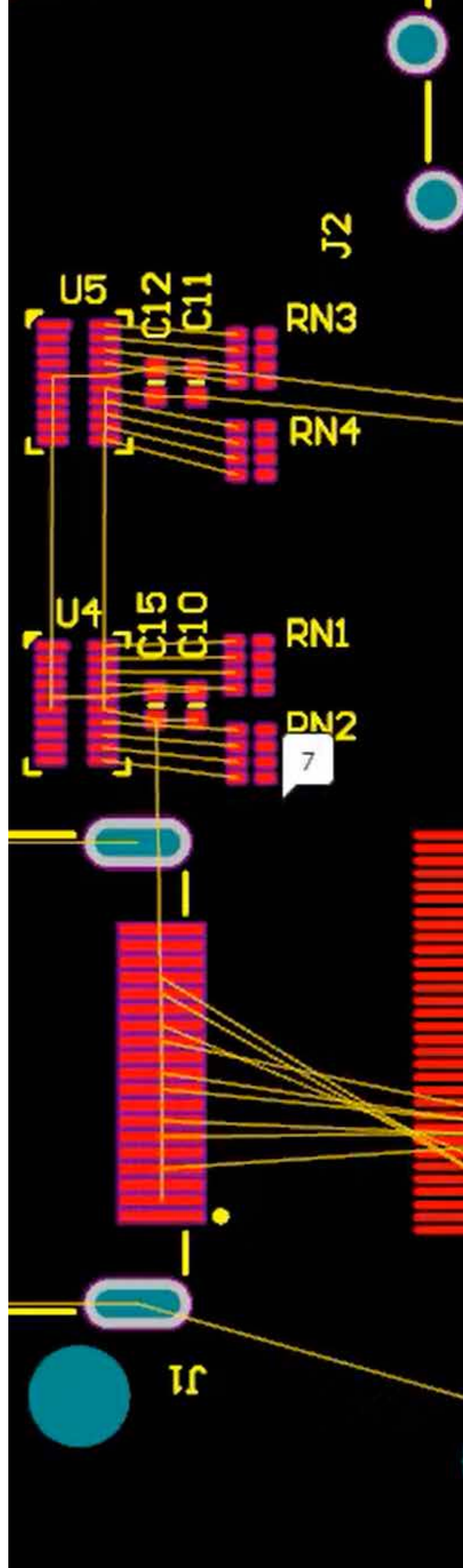
Synchronized ECAD and MCAD Environments

The CoDesigner extension synchronizes many of the important aspects of an ECAD environment in an MCAD application, and vice versa.

1. The mechanical designer can work with a model of the PCBA directly in an MCAD application.
2. The mechanical designer can select and share elements of an enclosure back to the ECAD application, allowing the EE to see a model of the housing and any relevant constraints.
3. Because the MCAD CoDesigner works with native MCAD data directly (and carefully!), any mechanical constraints and dimensions defined among elements on the bare board, PCB components, or enclosure are preserved during each sync. Additionally, since ECAD allows to lock components on the PCB, CoDesigner synchronizes ECAD's "locked" state with MCAD's "constrained" state.
4. A standard method for enforcing mechanical constraints in an electrical design is to define a keepout region in the MCAD model. Because model references are preserved on each side, keepouts can be defined in the MCAD tool and synchronized back to the ECAD side.

Synchronized Constraints

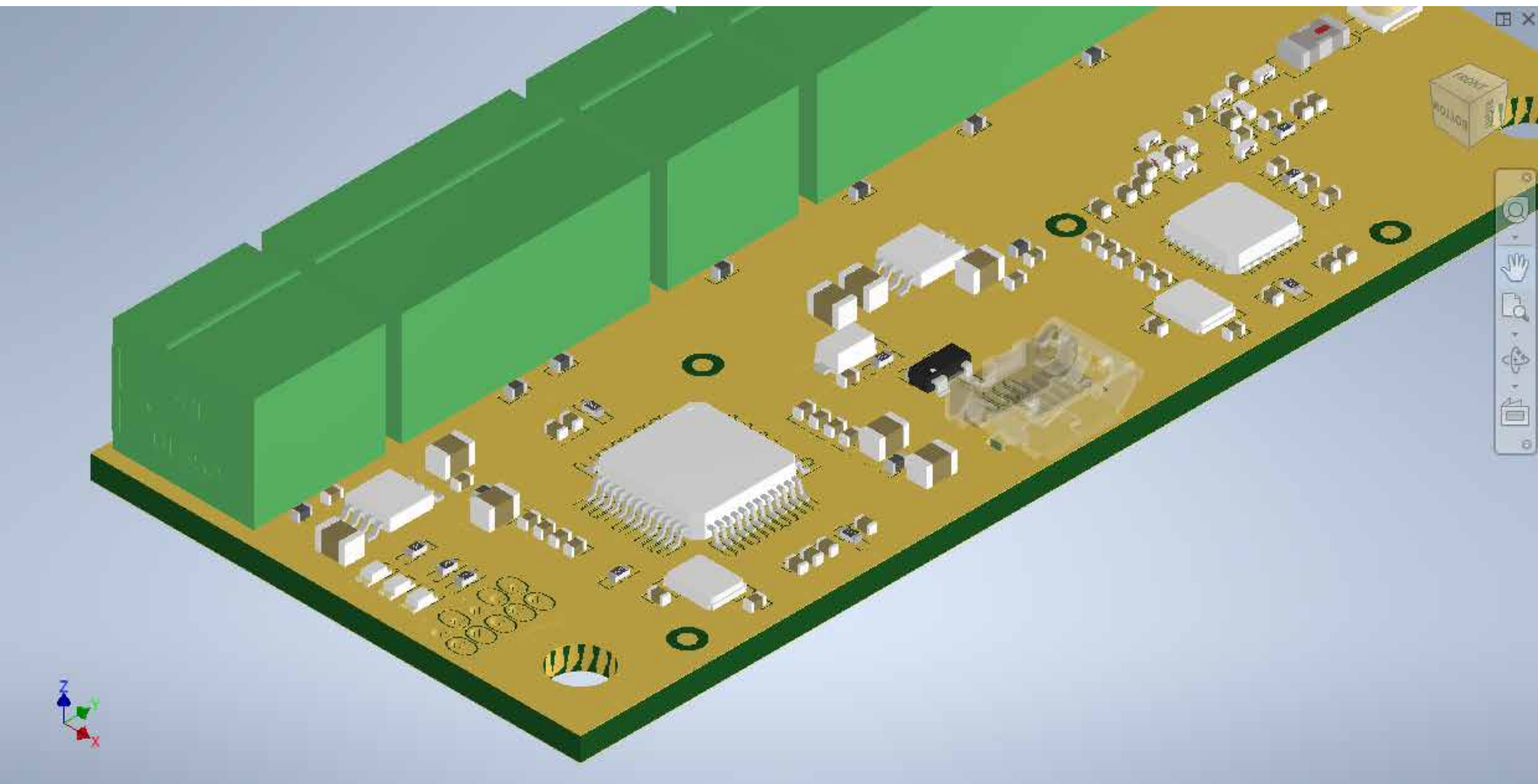
Altium Designer's native 3D design tools provide constraint and clearance checking in 3D, and the same types of clearance definitions and constraints can be pushed into an MCAD application for use in mechanical design tasks. Similarly, these constraints and clearances can be defined in the MCAD application, and they can be pushed back into Altium Designer. This level of synchronization helps expedite interference checking on both sides and allows interferences to be caught early.



ECAD and MCAD Component Linkage

The CoDesigner extension automatically synchronizes components placed in the MCAD tool with corresponding components in the ECAD tool, and vice versa. This saves time in design collaboration as the mechanical designer can immediately determine the appropriate location for mechanically constrained components, such as connectors and mounting holes. Placement in the MCAD application helps ensure enclosure constraints are satisfied as the mechanical designer will have full visibility of enclosure constraints, and the component definitions in the ECAD side are synchronized automatically.

Keepouts can be defined as top or bottom layers in the MCAD tool, and these keepout assignments will be reflected in the PCB layout and checked for interference against the design rules in the ECAD side. If required, the keepout shape can be adjusted in Altium Designer and pushed back to the MCAD side.



Component placement and copper definitions as defined in Altium Designer are reflected in your MCAD tool.

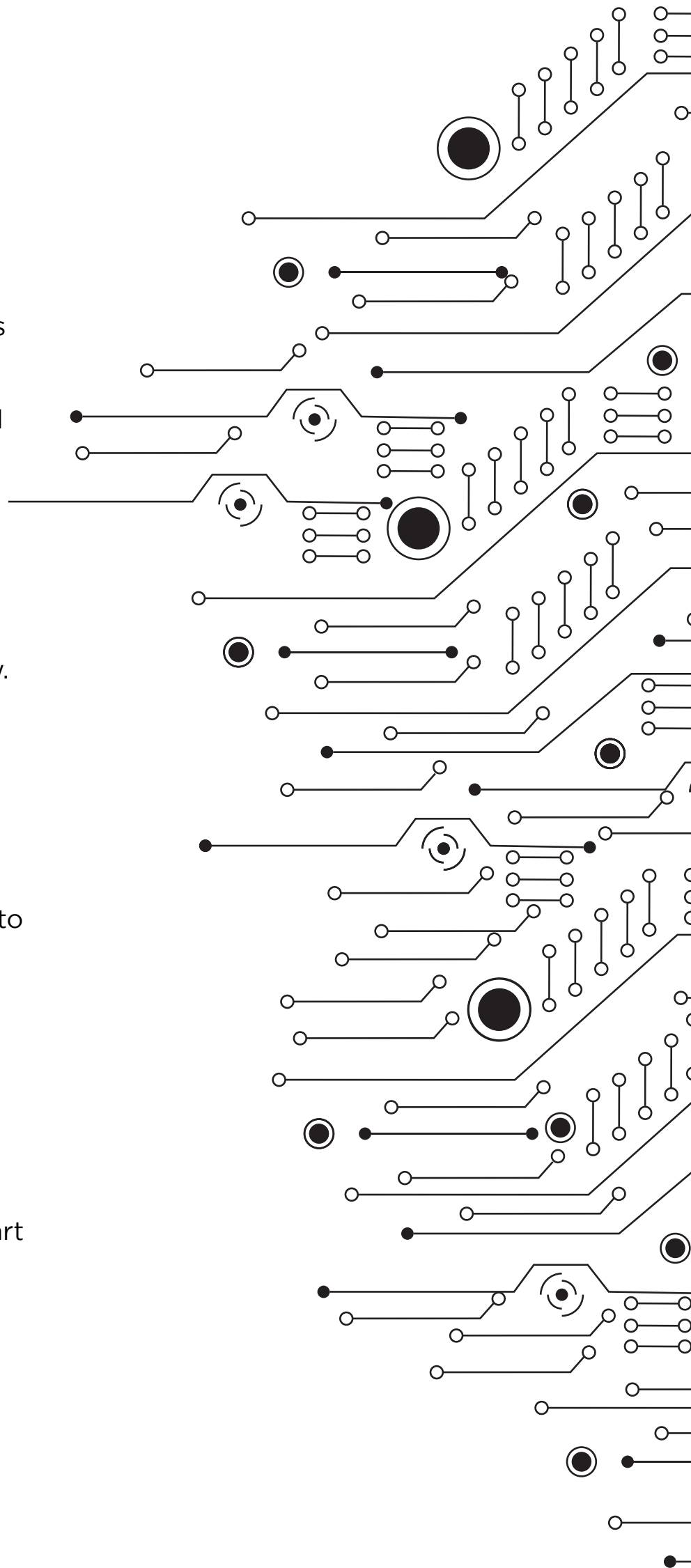
Bi-directional Support for Rigid-flex Boards

Flex and rigid-flex assemblies require additional modeling and evaluation beyond what is applied in rigid PCB assemblies. The MCAD CoDesigner plugin provides support for flex and rigid-flex boards in both directions. Rigid and flex regions, bending lines, and board shapes can be defined in the MCAD application and pushed back into Altium Designer. Once the layout engineer places components and completes initial routing, the design can be evaluated and previewed in the MCAD side alongside an enclosure. Bi-directional support for rigid-flex designs is seamless and follows the same workflow used for rigid PCBs.

The days where PCB assemblies are “thrown over the wall” to mechanical engineers are over, and Altium is helping design teams overcome the productivity and collaboration challenges created by the traditional ECAD/MCAD workflow. A powerful collaboration tool like the MCAD CoDesigner from Altium helps bridge the gap between ECAD and MCAD domains. Hundreds of companies rely on the MCAD CoDesigner extension from Altium to stay productive and produce more advanced designs in less time.

Instead of going through repetitive file transfers to collaborate with MCAD users, use the **MCAD CoDesigner plugin** for Altium Designer. PCB designs can be imported into SolidWorks, Autodesk Inventor, Fusion 360, or PTC Creo through the Altium 365™ platform, giving you everything needed for product development in a streamlined workflow.

We have only scratched the surface of what's possible with Altium Designer on Altium 365. Start your free trial of Altium Designer + Altium 365 today.



Sharing PCB Data with Manufacturers in the Cloud

Once you've finished your new project and you're ready to push it to your manufacturer, you'll normally be stuck in an endless email chain with an engineer, or you'll have to share cloud links with each other. Personally, I find the long email chains and long list of links overwhelming, especially when I have plenty of other tasks that require my attention. Sharing PCB data with manufacturers shouldn't be a headache, it should be a seamless process.

The cloud sharing and design release tools in Altium Designer and Altium 365 are a huge help in this area. This integrated set of design and collaboration features is ideal for creating assembly data and pushing your fabrication/assembly drawings to your manufacturer. In this tutorial, I'm going to take an existing project I've worked with in a number of recent blogs, create some fabrication and assembly documentation, and finally push this data to a manufacturer using Altium 365.

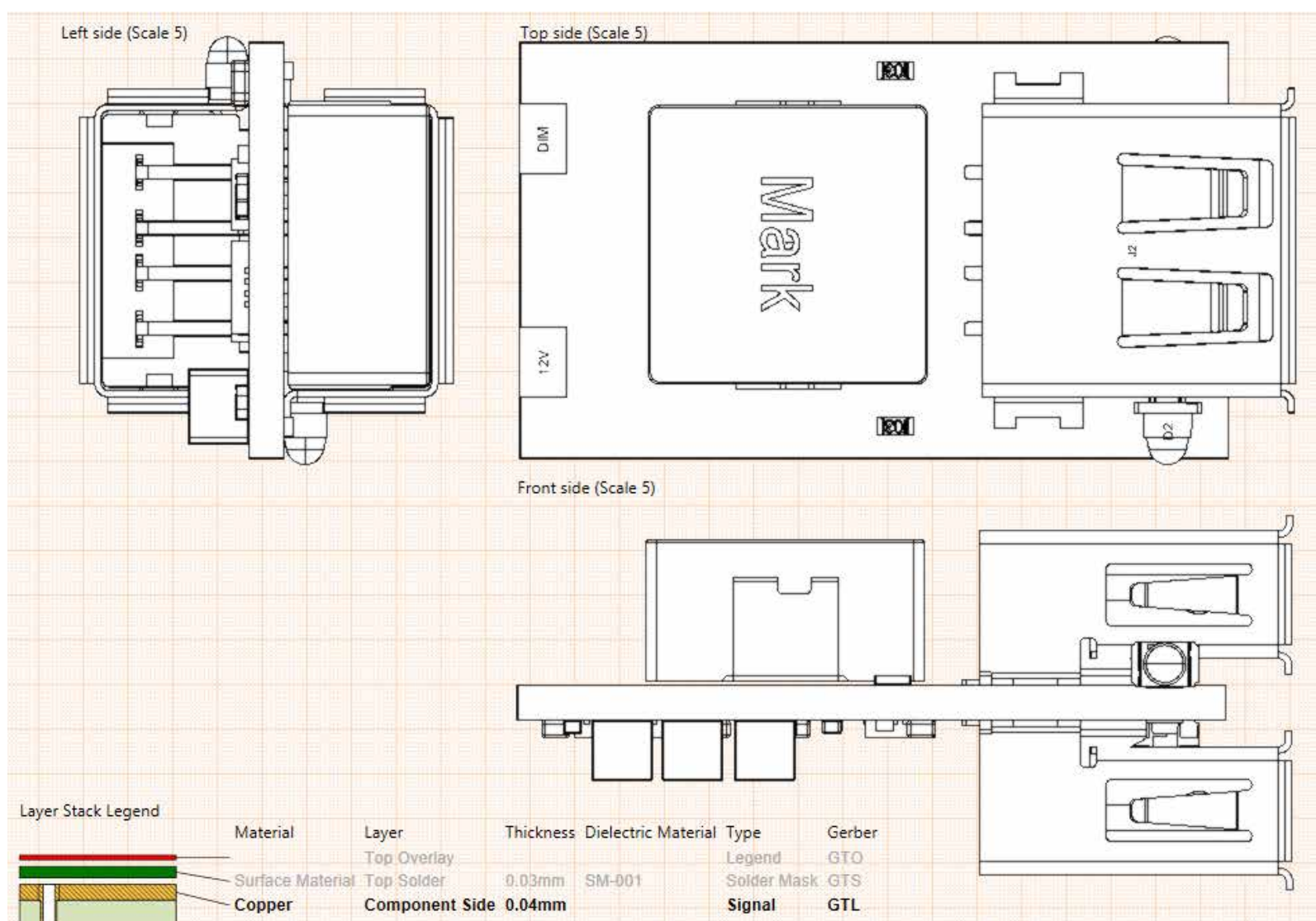


Creating and Sharing Your Manufacturing Documentation

In a [previous blog post](#), I looked at how to deploy and comment on a new project in Altium 365. At this point, let's assume you and your team have made all the required modifications to your project, and you're ready to go through a design review with your manufacturer.

If you've never created manufacturing documentation, such as Draftsman drawings, bills of materials, pick-and-place outputs, or Gerber files, you can easily create these documents in Altium Designer. [Mark Harris recently created a blog](#) that shows how to create some of these important documents. Here, I'm going to create some of these documents in my Altium Designer project for a USB charger module I've been showing in some articles. Then I'll deploy these into my Altium 365 workspace on the web.

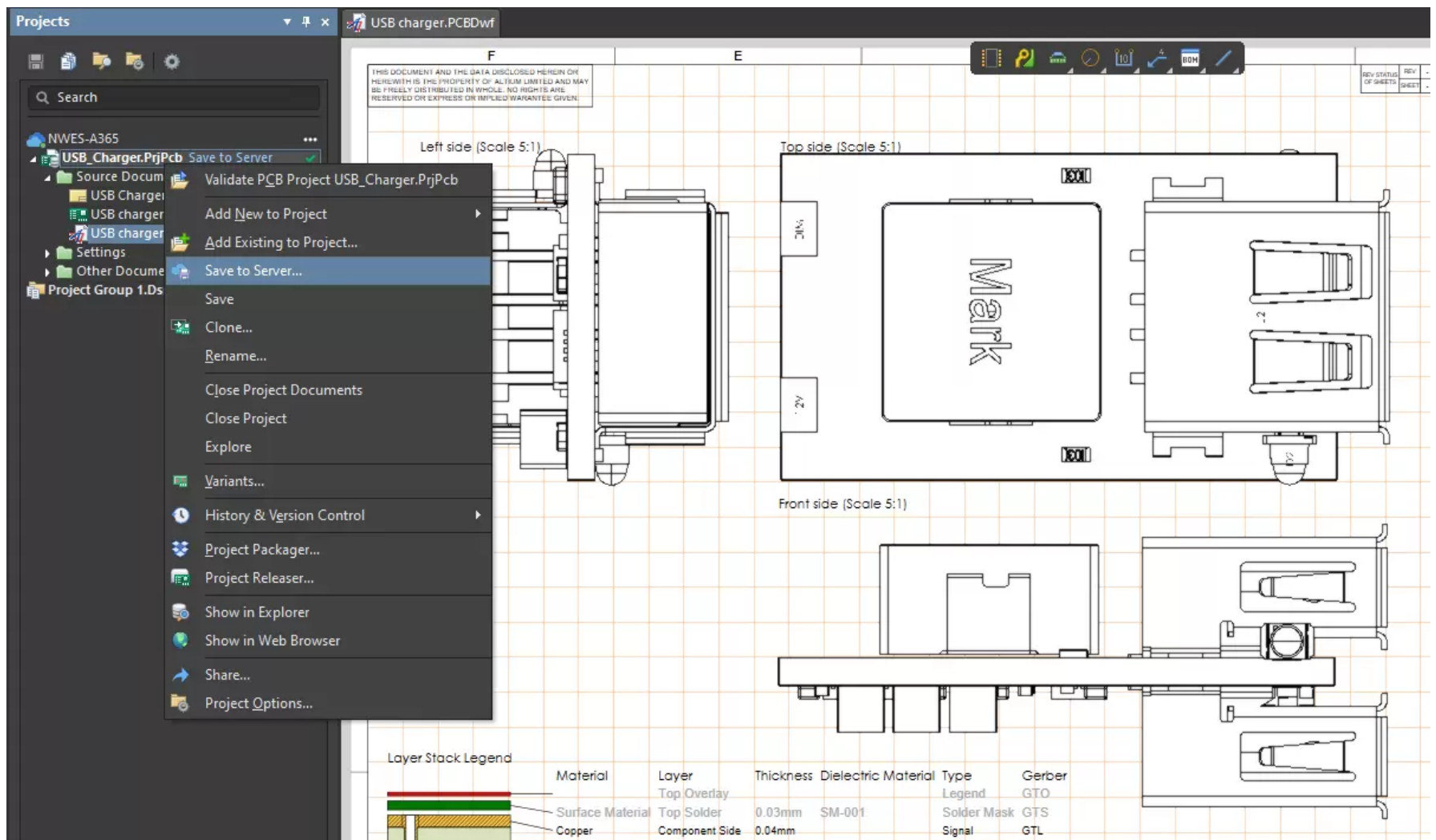
To get started, make sure you've [logged into your Concord web instance](#) in Altium Designer. You should be able to access your project from the Explorer window, and everything will be downloaded to your local machine. You can then modify the project, add new files to the project, make deletions, or create your manufacturing documentation. My Draftsman document (.PCBDwf file) for my USB charger module is shown below. Here, I just went to the Project menu and added a new Draftsman Document to my existing Altium 365 project (this project can be found in the Projects panel).



Draftsman file output in Altium Designer

Creating and Pushing Your Documentation to Altium 365

I also added a layer stack legend to this document, and I moved the different portions of the drawing around so that everything is easier to see. Once you're happy with your documentation, you can add a new Output Job File to the project and create any other outputs your manufacturer requires. Read through [Mark Harris' blog](#) if you're unsure of how to do this. Once you've created your manufacturing documentation, you can go through the Commit and Push process to move everything over to your Concord web instance.



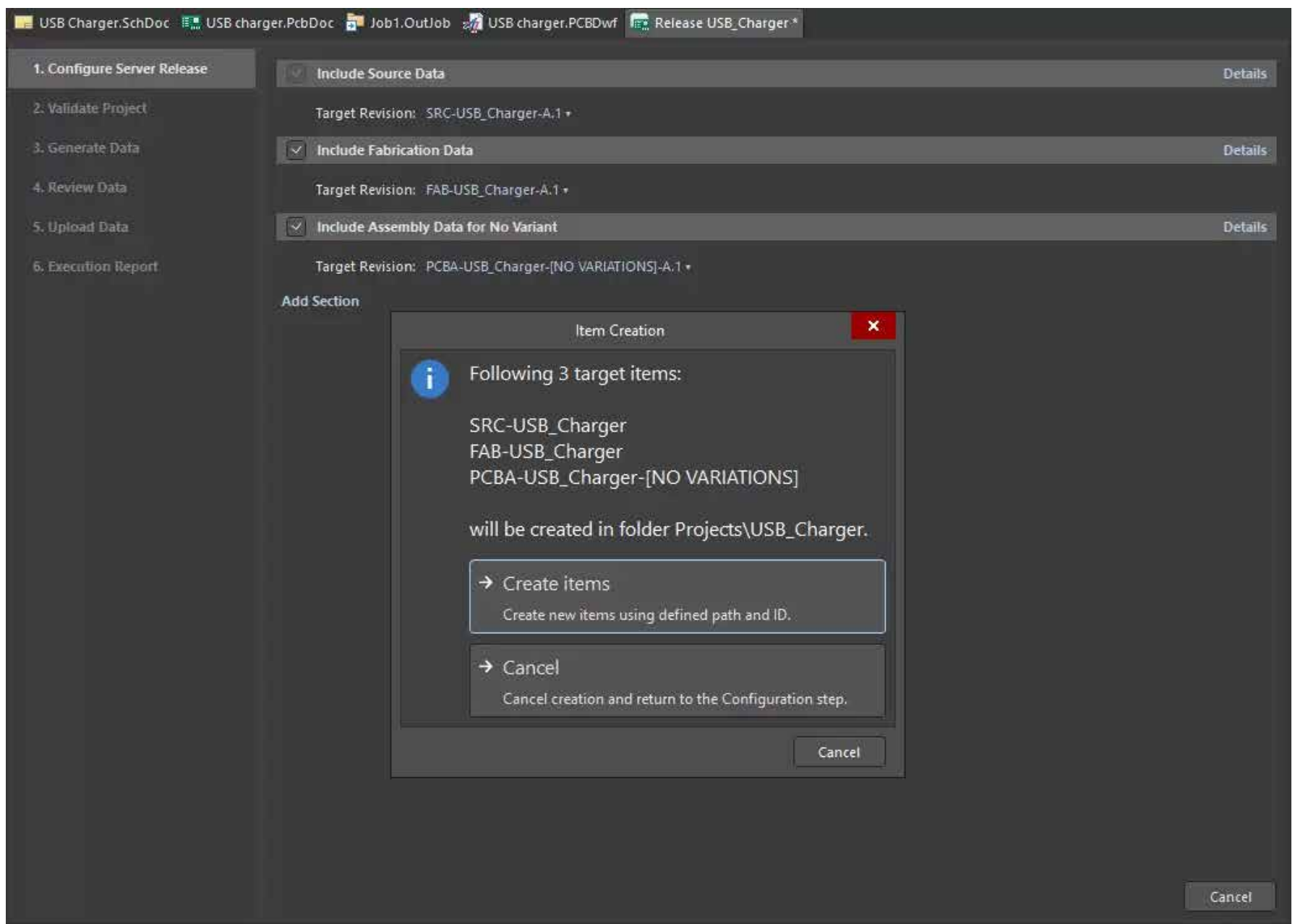
Pushing my Draftsman document onto my Altium 365 web instance

By deploying these project files into my Altium 365 web instance, one of my engineers can access these files, run a design review within Altium Designer, and then push any modifications back to the Altium 365 workspace.

Releasing Your Manufacturing Documentation

Simply pushing your files to your Altium 365 workspace will not make them available to a manufacturer as a released project. To do this, you'll need to release the project with Altium Designer. In the Projects panel, right-click on your project file, and click on Project Releaser. This will bring up the window shown below, allowing you to configure your project release.

I'm going to take an existing project and deploy it in my Concord workspace so that it can be accessed by a collaborator. To do this, I'll create a new project directly in Altium Designer and set Altium 365 as the project location. You'll see a window that allows you to create your project release documentation and deploy it to your Concord workspace.

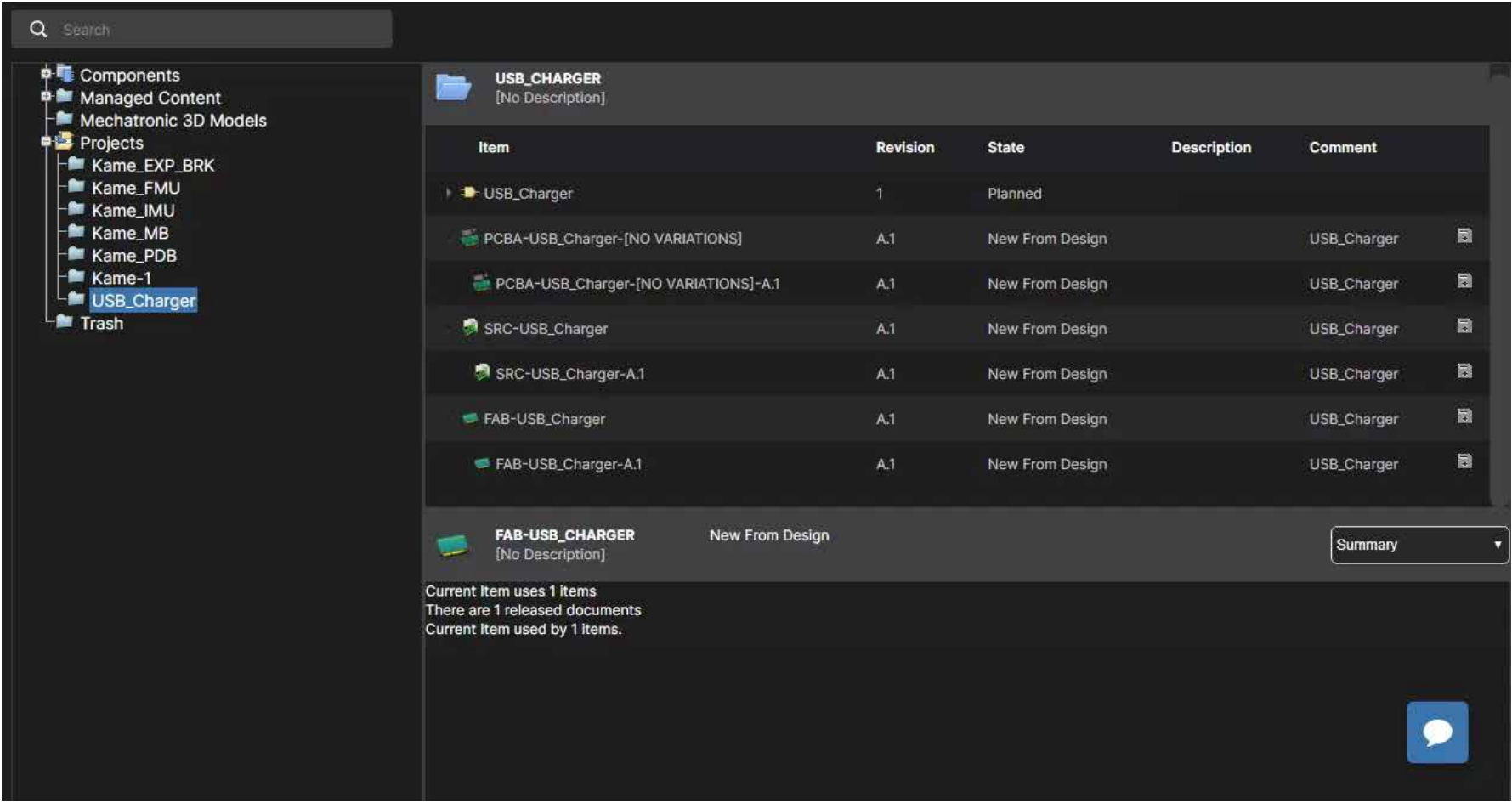


Releasing a project to your Altium 365 workspace

Once you've gone through the steps on the screen, everything can be deployed to your Concord workspace. You can now log in through your browser and access your design data.

Where are the Project Release Documents?

If you log into the Concord workspace, it may not be obvious where the project release files are located. If you click on the “Uploads” tab, you won’t be able to see any of the released project files. To see all the files associated with a project, click on the Explore tab in the Concord workspace, then expand the project folder. Here, you’ll be able to see all the project release files. To see the schematic and PCB layout files, simply open up the project from your dashboard. The image below shows the list of project release files in my Concord workspace.

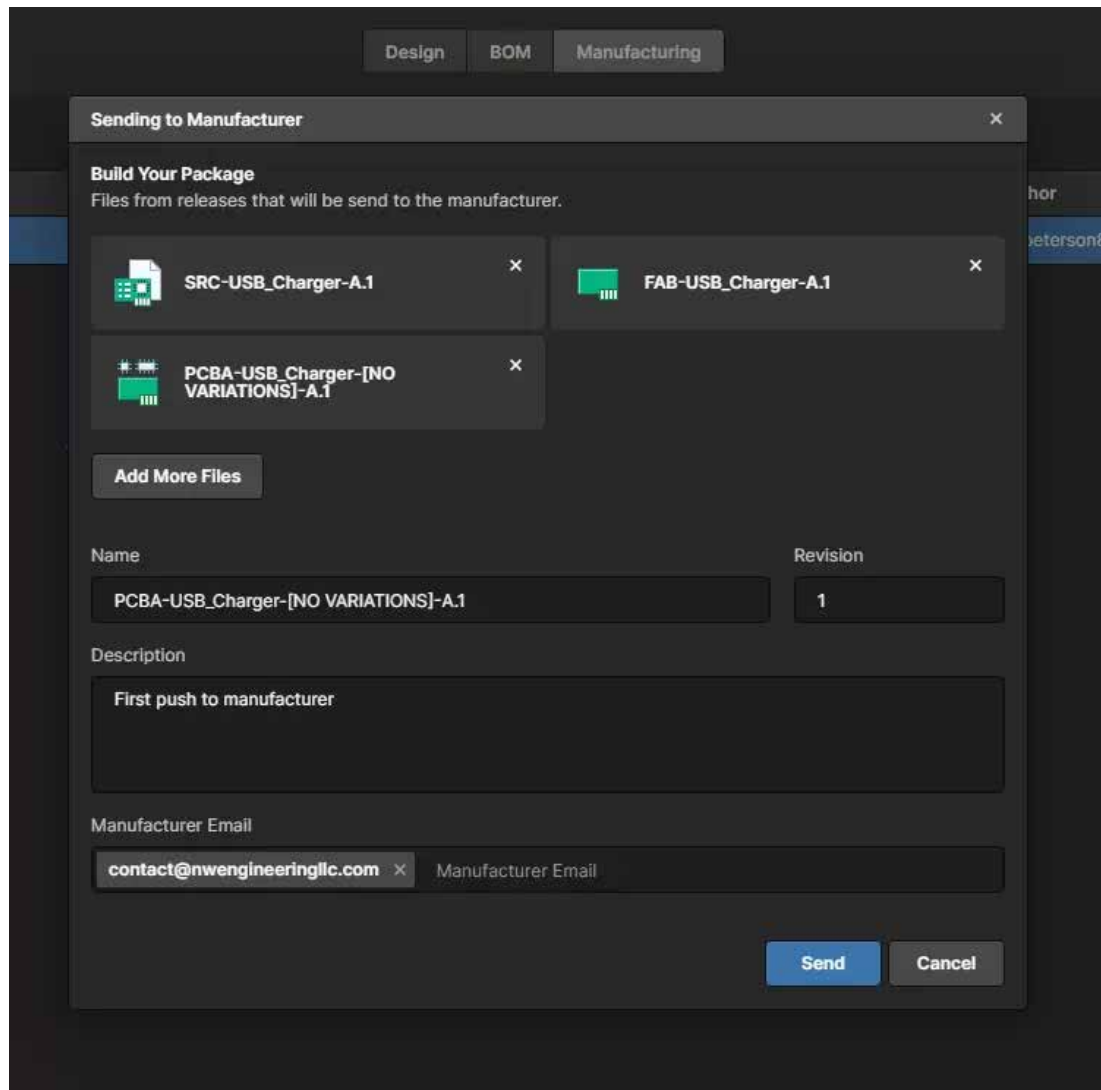


Project release files are accessible through the web in the Altium 365 Workspace.

Releasing a Project to Your Manufacturer

This is the final step in getting fabrication documents over to your manufacturer. Rather than sending a full data package (e.g., as an email attachment), you'll send a link to your Concord workspace directly to your manufacturer's email address. To do this, simply open up your project, and click on the Manufacturing tab at the top of the screen. Once you get to this window, you'll see your list of current releases and an option to send the data to your manufacturer's email.

The image below shows the release screen for my USB charger module. In this example, I'm going to send the data to one of my company's email addresses.



Sharing PCB data with manufacturers in Altium 365

Once you click Send, you've sent a link to your manufacturer, which they can use to access the manufacturing data and download it directly. The manufacturer can then go back and add comments to the design directly in the Concord workspace if anything needs to be changed before fabrication. You can also invite other designers to collaborate by sending them an email invite.



How to Speed Up Your DFM Report Process in the Cloud

Experienced fabricators will tell you: any design could have some hidden DFM problem that will interfere with manufacturability, quality, or yield. Making use of your design rules is just the start of preventing DFM problems, you'll want to collaborate with your manufacturer throughout the design process if you want to spot and correct DFM problems.

Within Altium Designer, there are multiple reports you can generate for your projects that will help you summarize important information on your board for a client or a manufacturer. When my team is working on backplane projects or networking equipment designs, we often use the Net Status report to summarize lengths for critical nets, and these can be cross-checked against important signal integrity rules.

If you're preparing a new design for manufacturing, the Altium 365 platform allows your team to share your validation reports as part of your project releases. This is a great way to easily share important design data with your manufacturer, customers, or other designers as you prepare your board for manufacturing. The version control and Project Release features in Altium 365 let you send your reports directly to the shop floor while also tracking changes to your design, documentation, and project releases over time.

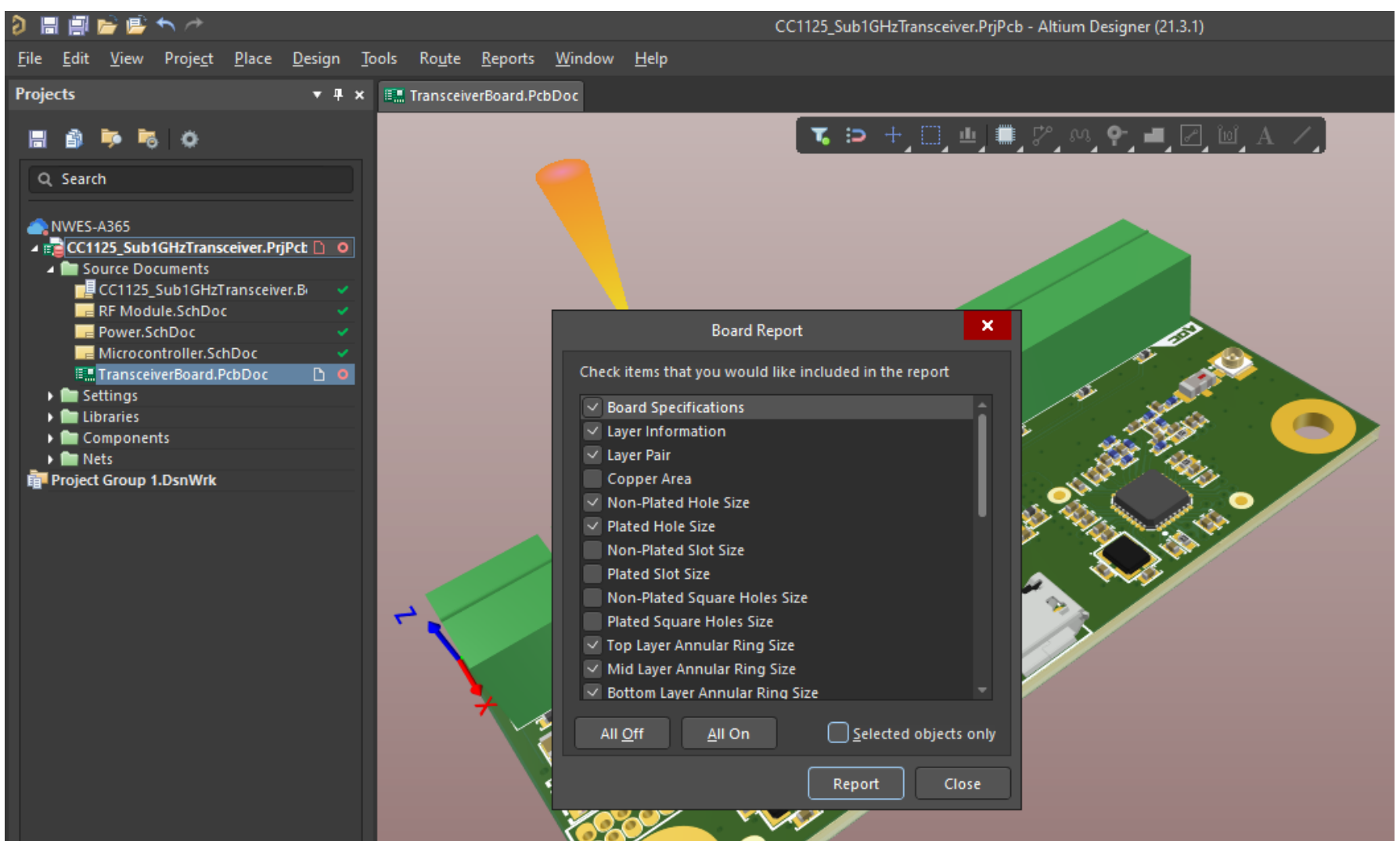


Altium®

Using Validation and DFM Reports in Altium 365

The **Board Report feature** in Altium Designer is an excellent way for a designer or a manufacturer to summarize information on a PCB layout and check this against their capabilities. Some manufacturers prefer to develop their own reports, sometimes with external software packages that scan through Gerber files. With this feature, you or your manufacturer can quickly summarize important board information before finalizing a design for manufacturing.

To access this feature, click on Board Information in the Reports menu with the PCB editor open. When this window opens, you'll see a list of options that can be automatically generated from your PCB layout. In the example below, I'm using a modified version of **Mark Harris' sub-1 GHz radio transceiver project** as an example.



You can use the Board Report feature to generate DFM reports.

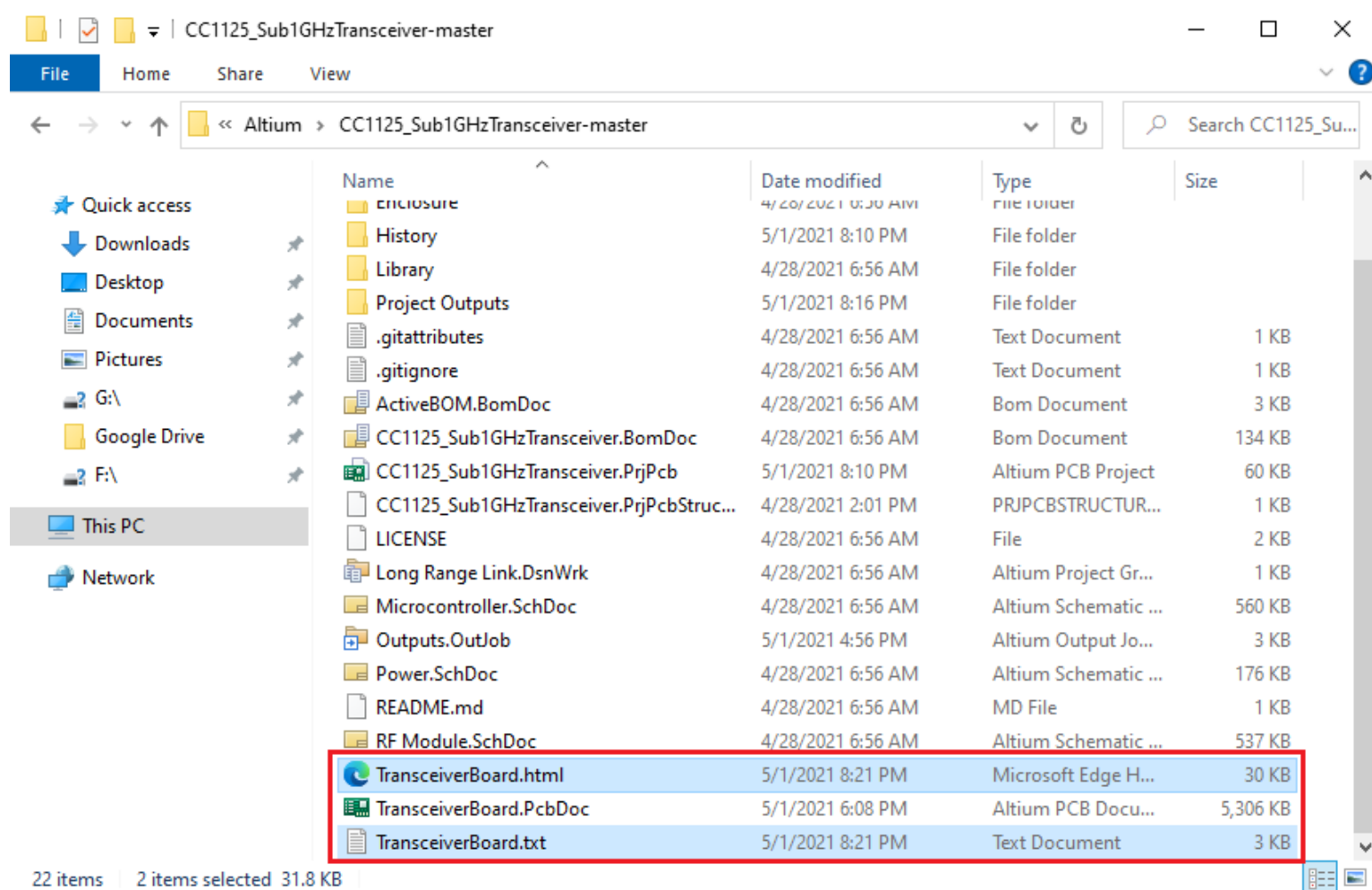
There's a lot you can generate from this feature, but your fabricator won't need all of it. There are some major points here that any fabricator will look at to ensure manufacturability:

- Annular ring sizes, particularly if you're aiming for **IPC Class 2 or Class 3 compliance**
- Minimum feature sizes, particularly via hole sizes and trace widths
- Clearances between different objects in the board
- Solder mask expansion on pads

The above list contains some minimum aspects of DFM that apply to just about any circuit board. However, more specialized boards or components may carry specific requirements in any of the above areas to ensure high quality during fabrication and assembly. If you plan to use this feature to collaborate with your manufacturing partner, talk to them and ask for their DFM requirements, they'll tell you what information they look at to assess manufacturability.

Where's My Board Information Report?

The report you generate won't be automatically added to your project, but it will be created in the project folder on your local disk. The report exists as a text file and an HTML file; you will need to include both files in your project. Make sure to add it to your project; once it's in your project you can **add it to version control in Altium 365**. This means each time the project is changed and a new report is generated, Altium 365 will track these changes throughout the history of the project.



Your Board Information Report will be generated in the same folder as your .PcbPrj file for your project.

Process for Using Board Information Reports

Here's one process you can follow when using the Board Report feature as a DFM report:

1. Generate the required Board Information report for your project. You'll generally want to focus on the areas in the above list so that your fabricator can quickly scan through your board information and spot any DFM violations.
2. Navigate to the project folder on your local disk. Use the **Add Existing to Project** feature to add both files into your project. They will appear in the Projects Panel.
3. Once the text and HTML files are added into the project, save the project to your Altium 365 Workspace.
4. Share the project data with your manufacturer.

Just like any of the steps you need to follow for design and project management, there is no set method to use any of the project data you generate in Altium Designer. However, when planning for manufacturing, it's important to get some kind of DFM report in front of your manufacturer before you finalize the design. The above process is a great way to do this as it gives your manufacturer everything needed to quickly compare your board with their capabilities. This also applies to complex designs with thousands of nets and components.

How you complete Step 4 above is your choice, and you'll have several options available through your Altium 365 Workspace. These options for sharing your board information as a DFM report are convenient as they can be used with any manufacturer, even manufacturers who don't use Altium Designer.

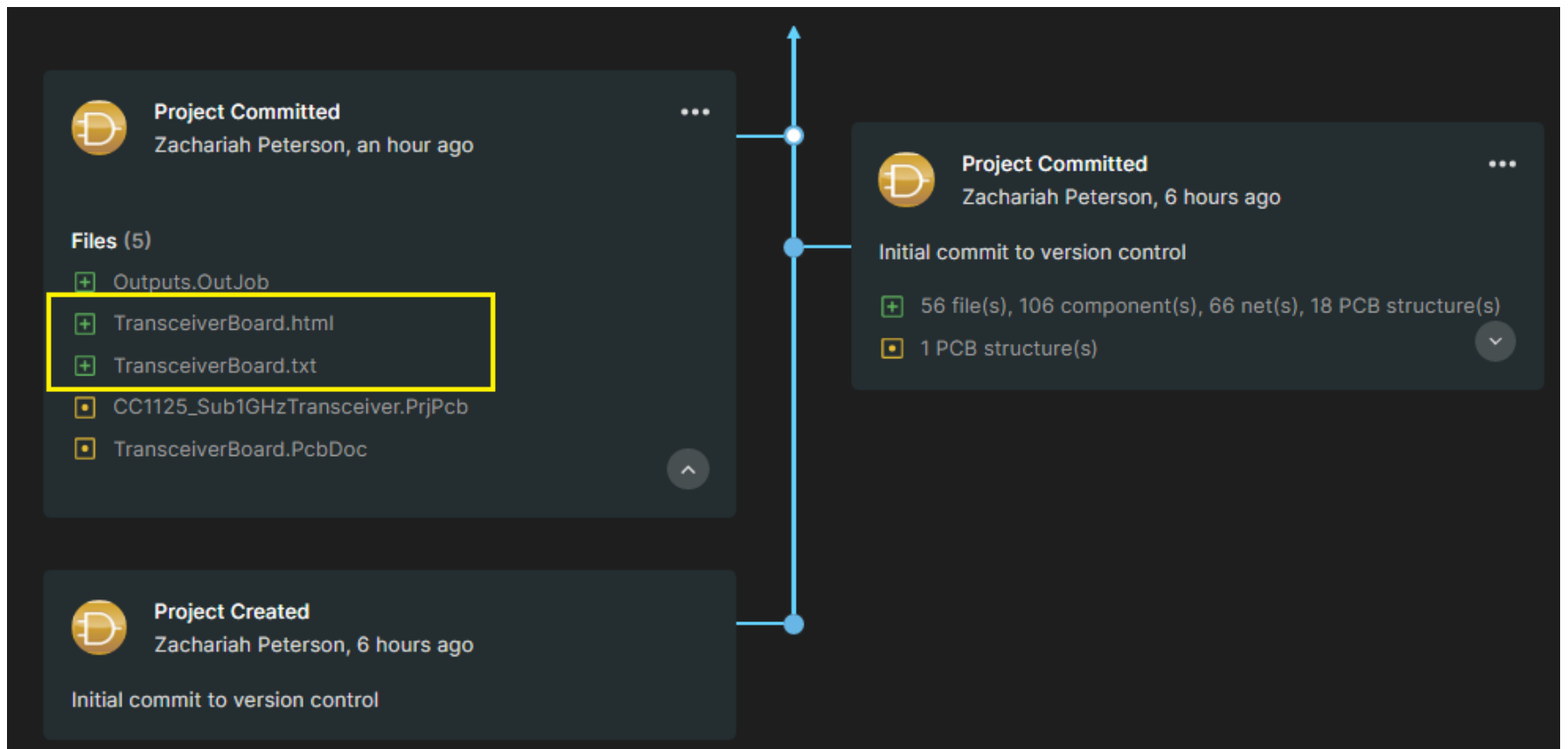
How to Share the Board Report With Your Manufacturer

Probably the simplest way to do this is to invite your manufacturer to your Workspace. Even if you add your manufacturer as a team member, you can still limit what they can see within your workspace to only their allowed projects. This way, if your manufacturer is an Altium Designer user, they can open the project in Altium Designer and view the board report directly. There are some other options your manufacturer can use if they don't have an Altium license.



Download a Snapshot From the Project History

One quick way a manufacturer can get access to the board report is through the Project History feature in your Altium 365 Workspace. From here, the manufacturer can download the board report and run their own DFM checks without opening up the project files. To get access to the board report, just navigate to the [Project History](#) area and download a snapshot of the project.



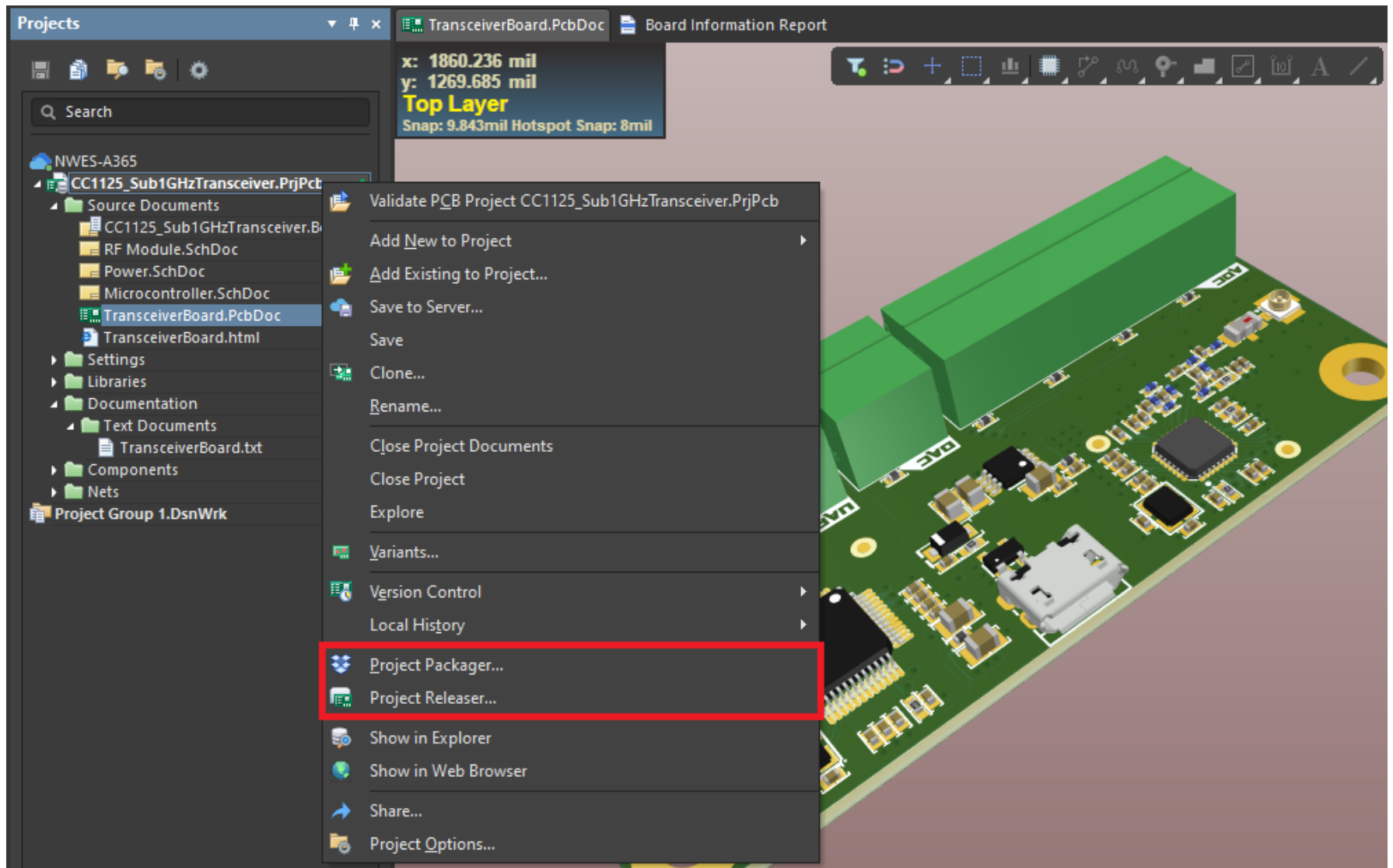
Click the 3 dots at the top of a Project History entry to download a snapshot of the project.

If all your manufacturer needs to do is open up the board report, then this is a quick and easy option. Once the files are downloaded, the board report can be viewed by opening up the HTML file in the file package. This not only helps your manufacturer identify design choices that violate their capabilities, it also [helps them quote your board](#) as they can see via sizes and via counts. This is also a quick way to start the quotation process before the design is finalized and fabrication files are generated.

If there are any problems found in the board report, your manufacturer could send an email outlining the required changes. A better option is to go to the Design area in the 365 Workspace via the web browser and leave a comment directly in the project files. This is preferable because the comment history will be saved and trackable along with the rest of the design data.

Project Packager and Project Releaser

Two other sharing options we've discussed in the past are the Project Packager feature and the Project Releaser feature. The Project Packager would be used by the designer within Altium Designer to create a ZIP file with all the project files. This could then be emailed directly to your manufacturer, or it could be shared through a proprietary file sharing system, FTP portal, or a typical cloud solution like Google Drive.



The Project Packager and Project Releaser features are two ways you can send your project files out to your manufacturer for DFM checks. This includes your board information report.

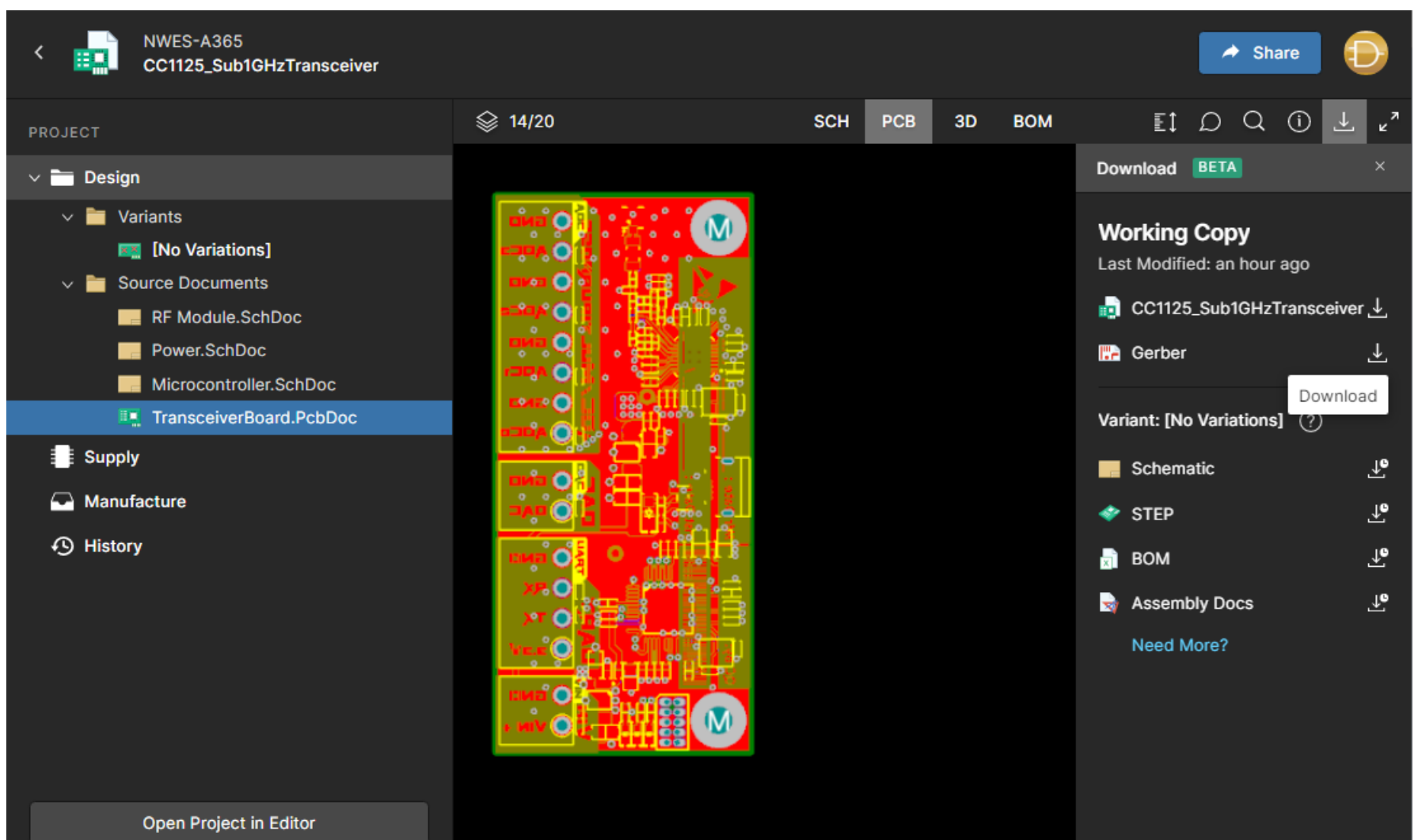
The Project Release feature is preferable if you're like me and you want to cut down on all the back-and-forth emailing that tends to happen on complex PCB projects. Once a Project Release is created, you can go into your 365 Workspace and use the **Send to Manufacturer** feature to give them access to the entire set of project files. This is a great option as the manufacturer doesn't need to be added to your workspace, and you don't create any risks that any sensitive project data gets exposed to an unauthorized party.

Fixing Common Clearance Problems

One of the most common DFM error I've encountered in projects is clearance errors. These are usually hole-to-copper (antipad diameter), pad-to-copper, or trace-to-copper clearances, and often for specific nets like power and ground. Thankfully, it only takes a couple minutes to fix these problems in Altium Designer by modifying the design rules and repouring your polygons. Using the Board Report feature as the start for your DFM report process lets you and your manufacturer instantly spot these clearance problems just by reading through a list.

Instantly Generate Gerbers After Inspecting the Board Report

If everything in the DFM report looks good, you or your manufacturer can immediately generate Gerbers directly in the 365 Workspace through the web browser. Simply open up the Design section of the project, and on the right side of the screen, you can open up the Download area. From here, your manufacturer can generate Gerber files for the project.



Click the 3 dots at the top of a Project History entry to download a snapshot of the project.

The person who generates the files from the web browser will receive a download link once the files have finished generating. This is extremely convenient for a collaborator who doesn't have an Altium Designer license as they can download all the important documentation they'll need without [creating an Outjob file](#).

The list of tiles that can be generated through a web browser isn't limited to Gerber files. Notice along the right-hand side of the screen, there are several files you can create without using Altium Designer. You'll have more customizability for any deliverables if they are opened and created in Altium Designer, while Altium 365 only generates a standard set of files.

However, this does give a manufacturer a quick and easy way to get the board to the next stage of a DFM check and run some more DFM analysis. This is often done with specialized CAM software or proprietary software, and the manufacturer will eventually need the Gerber files to complete these final checks. If everything checks out, they can instantly put the board into fabrication.

The [Altium 365™](#) platform gives you, your team, and your manufacturer the tools you need to quickly get your board through DFM report checks and help you move immediately into fabrication. You'll have all the tools you need to share, store, and manage all of your PCB design data in a secure cloud platform. Altium 365 is the only cloud collaboration platform specifically for PCB design and manufacturing, and all the features in Altium 365 integrate with the world-class design tools in [Altium Designer®](#).

We have only scratched the surface of what is possible to do with Altium Designer on Altium 365. You can check the [product page](#) for a more in-depth feature description or one of the [On-Demand Webinars](#).

