Webinar Video - Table of Contents

Altium Video Vignettes ........................................................................................................................................ 5

Altium Designer Related ...................................................................................................................................... 5

To Route or Auto Route, that is the Question ......................................................................................................... 5

3D in Altium Designer ........................................................................................................................................ 6

Improving the Altium ECAD / MCAD Connectivity Experience ........................................................................... 6

Navigating Altium Designer .................................................................................................................................. 6

Altium Designer 20 Installation ............................................................................................................................ 6

Electronic Design Process in Altium Design ........................................................................................................ 6

Wire/Cable Harness Design .................................................................................................................................. 7

What's this Vault, AVS, NEXUS, Concord Pro all about? ....................................................................................... 7

Leaving a Lasting Imprint (or in this case, footprint) ............................................................................................. 7

Symbol Creation - Setup for Success .................................................................................................................... 7

DbLink Demystified ................................................................................................................................................ 8

Linking Altium DbLink to Excel ............................................................................................................................. 8

Design Reuse in Altium Designer .......................................................................................................................... 8

DRC Revisited Part 1 .............................................................................................................................................. 8

DRC Revisited Part 2 .............................................................................................................................................. 9

Design Rule Checker .............................................................................................................................................. 9

Making the Most of Draftsman (and Blueprint) ....................................................................................................... 9

What's this Co-Design Mechatronic mishmash all about? ....................................................................................... 10

Exporting 3D of a Folded Rigid/Flex Design ........................................................................................................ 10

High Speed Design Rules .................................................................................................................................... 10

A Tour of Altium Libraries ................................................................................................................................... 11

Library – Database Management .......................................................................................................................... 11

A Robust Library Example .................................................................................................................................... 11

Mechanical Layers - Revisited ............................................................................................................................... 12

How to Tame your Mechanical Layers ................................................................................................................ 12

Multi-board Assembly in AD18 ............................................................................................................................. 12

Altium Outjob ......................................................................................................................................................... 13

PCB Translation - "Invisible Insanity" .................................................................................................................... 13

Query Language .................................................................................................................................................... 13

When Designs and Designers Need to Be ‘Flexible’ .............................................................................................. 14

Spelunking in the Altium Schematic Preferences - Part 1 .................................................................................. 14

Routing Tools in Altium Designer ........................................................................................................................ 14

General Inquiries: info@ninedotconnects.com
(214) 699-7719
Cabling & Wire Harness

Design for Manufacturing

Fab and Assembly

High Speed Design

General Inquires:
(214) 699-7719
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Library Management</td>
<td>30</td>
<td>Leaving a Lasting Imprint (or in this case, footprint)</td>
<td>30</td>
<td>Symbol Creation - Setup for Success</td>
</tr>
<tr>
<td>Symbol Creation - Setup for Success</td>
<td>30</td>
<td>Database Library Flows</td>
<td>30</td>
<td>Database Library Flows</td>
</tr>
<tr>
<td>Database Library Flows</td>
<td>30</td>
<td>Dblink Demystified</td>
<td>30</td>
<td>Fundamentals of Library Structure</td>
</tr>
<tr>
<td>Dblink Demystified</td>
<td>30</td>
<td>Fundamentals of Library Structure</td>
<td>31</td>
<td>Libraries Built to Last</td>
</tr>
<tr>
<td>Fundamentals of Library Structure</td>
<td>31</td>
<td>SOS about Library Scrubbing</td>
<td>31</td>
<td>SOS about Library Scrubbing</td>
</tr>
<tr>
<td>Libraries Built to Last</td>
<td>31</td>
<td>Linking Altium Dblink to Excel</td>
<td>32</td>
<td>Linking Altium Dblink to Excel</td>
</tr>
<tr>
<td>SOS about Library Scrubbing</td>
<td>31</td>
<td>A Robust Library Example</td>
<td>32</td>
<td>A Robust Library Example</td>
</tr>
<tr>
<td>Linking Altium Dblink to Excel</td>
<td>32</td>
<td>Tour of Altium Provided Libraries</td>
<td>33</td>
<td>Tour of Altium Provided Libraries</td>
</tr>
<tr>
<td>A Robust Library Example</td>
<td>32</td>
<td>Tour of Altium Provided Libraries</td>
<td>33</td>
<td>MCAD &amp; ECAD</td>
</tr>
<tr>
<td>Tour of Altium Provided Libraries</td>
<td>33</td>
<td>MCAD &amp; ECAD Co-Design with SOLIDWORKS PCB Connector</td>
<td>33</td>
<td>MCAD &amp; ECAD Co-Design with SOLIDWORKS PCB Connector</td>
</tr>
<tr>
<td>MCAD &amp; ECAD Co-Design with SOLIDWORKS PCB Connector</td>
<td>33</td>
<td>Improving the Altium ECAD / MCAD Connectivity Experience</td>
<td>33</td>
<td>Improving the Altium ECAD / MCAD Connectivity Experience</td>
</tr>
<tr>
<td>Improving the Altium ECAD / MCAD Connectivity Experience</td>
<td>33</td>
<td>Bridging the ECAD and MCAD Worlds</td>
<td>34</td>
<td>Bridging the ECAD and MCAD Worlds</td>
</tr>
<tr>
<td>Bridging the ECAD and MCAD Worlds</td>
<td>34</td>
<td>Desktop EDA's Altium Modeler for SolidWorks Demo</td>
<td>34</td>
<td>Desktop EDA's Altium Modeler for SolidWorks Demo</td>
</tr>
<tr>
<td>Desktop EDA's Altium Modeler for SolidWorks Demo</td>
<td>34</td>
<td>3D in Altium Designer</td>
<td>35</td>
<td>3D in Altium Designer</td>
</tr>
<tr>
<td>3D in Altium Designer</td>
<td>35</td>
<td>PCB Broadview</td>
<td>35</td>
<td>PCB Broadview</td>
</tr>
<tr>
<td>PCB Broadview</td>
<td>35</td>
<td>6 Key Considerations When Starting a New PCB Design</td>
<td>35</td>
<td>6 Key Considerations When Starting a New PCB Design</td>
</tr>
<tr>
<td>6 Key Considerations When Starting a New PCB Design</td>
<td>35</td>
<td>10 things that may surprise you about engineers in the PCB process</td>
<td>35</td>
<td>10 things that may surprise you about engineers in the PCB process</td>
</tr>
<tr>
<td>10 things that may surprise you about engineers in the PCB process</td>
<td>35</td>
<td>A Short History of the Short History of Printed Circuit Boards</td>
<td>36</td>
<td>A Short History of the Short History of Printed Circuit Boards</td>
</tr>
<tr>
<td>A Short History of the Short History of Printed Circuit Boards</td>
<td>36</td>
<td>To Route or Auto Route, that is the Question</td>
<td>36</td>
<td>ToRoute or Auto Route, that is the Question</td>
</tr>
<tr>
<td>To Route or Auto Route, that is the Question</td>
<td>36</td>
<td>A History of Printed Circuit Boards</td>
<td>37</td>
<td>AHistory of Printed Circuit Boards</td>
</tr>
<tr>
<td>A History of Printed Circuit Boards</td>
<td>37</td>
<td>PCB Elements</td>
<td>37</td>
<td>PCB Elements</td>
</tr>
<tr>
<td>PCB Elements</td>
<td>37</td>
<td>Vias: Blind, Buried, and Beyond</td>
<td>37</td>
<td>Vias: Blind, Buried, and Beyond</td>
</tr>
<tr>
<td>Vias: Blind, Buried, and Beyond</td>
<td>37</td>
<td>High Speed Topic: What is meant by a 50 ohm impedance?</td>
<td>38</td>
<td>High Speed Topic: What is meant by a 50 ohm impedance?</td>
</tr>
<tr>
<td>High Speed Topic: What is meant by a 50 ohm impedance?</td>
<td>38</td>
<td>Deep Dive - Capacitors</td>
<td>38</td>
<td>Deep Dive - Capacitors</td>
</tr>
<tr>
<td>Deep Dive - Capacitors</td>
<td>38</td>
<td>Deep Dive - Inductors</td>
<td>38</td>
<td>Deep Dive - Inductors</td>
</tr>
<tr>
<td>Deep Dive - Inductors</td>
<td>38</td>
<td>Deep Dive - Vias</td>
<td>39</td>
<td>Deep Dive - Vias</td>
</tr>
<tr>
<td>Deep Dive - Vias</td>
<td>39</td>
<td>Intuitive Stack-up Planning &amp; Impedance Control</td>
<td>39</td>
<td>Intuitive Stack-up Planning &amp; Impedance Control</td>
</tr>
<tr>
<td>Intuitive Stack-up Planning &amp; Impedance Control</td>
<td>39</td>
<td>PCB Translation - &quot;Invisible Insanity&quot;</td>
<td>40</td>
<td>PCB Translation - &quot;Invisible Insanity&quot;</td>
</tr>
<tr>
<td>PCB Translation - &quot;Invisible Insanity&quot;</td>
<td>40</td>
<td>Power Distribution Network &amp; Power Integrity</td>
<td>40</td>
<td>Power Distribution Network &amp; Power Integrity</td>
</tr>
<tr>
<td>Power Distribution Network &amp; Power Integrity</td>
<td>40</td>
<td>Test Points - Before Layout and Way Beyond Manufacturing</td>
<td>40</td>
<td>Test Points - Before Layout and Way Beyond Manufacturing</td>
</tr>
</tbody>
</table>
Nine Dot Connects Webinar Video Listing

ICD's PowerStack Suite .........................................................................................................................41
ICD's Stackup Planner .............................................................................................................................41
ICD's PDN Planner .................................................................................................................................41

**Power Supplies** .................................................................................................................................42
Powering Up on Power Supplies, Part 1 .................................................................................................42
Powering Up on Power Supplies, Part 2 .................................................................................................42
Power Distribution Network & Power Integrity .....................................................................................41

**Simulation** ........................................................................................................................................43
The accidental antenna and how to make one .......................................................................................43
Practical Aspects of Signal Integrity - Part 1 .........................................................................................44
Practical Aspects of Signal Integrity - Part 2 .........................................................................................44
Practical Aspects of Signal Integrity - Part 3 .........................................................................................45
Practical Aspects of Serial Communications - Part 4 ..........................................................................45
Beyond a Heatsink - Thermal Considerations in PCB Design ..............................................................45
Power Integrity & Power Distribution Network ....................................................................................46
ICD's Power Distribution Network Planner ............................................................................................46
Altium SPICE ........................................................................................................................................46
High Speed Design (Signal Integrity) in Altium Designer ......................................................................47
Altium Video Vignettes

A collection of short videos on useful functions and features within Altium Designer

You will learn how to install Altium Designer 20.

You will quickly learn how to change the UI (User Interface) color in Altium Designer 18 and newer versions.

Panels are widely used in Altium Designer. Starting in Altium Designer 18, many dialog boxes were converted to panels. This video provides a brief overview of the ways the panels can be configured.

If typing or copying and pasting an existing label on a primitive seems laborious and time-consuming, check out the ‘insert’ hotkey key. This feature drastically reduces the amount of time in duplicating a label for different primitives in the schematic editor.

This video explains the reference prompt when using a copy or cut function in the PCB layout editor. This video will also explain how to enable the reference feature in the schematic editor as well.

Altium Designer Related

To Route or Auto Route, that is the Question

We put this webinar together to address the complex issue of autorouting. It is a ‘high level’ presentation with the purpose of looking at the history and algorithms that are commonly used. As you will see, the effort is not simple. It's one thing to route a particular line; however, planning and prioritizing the routes are difficult. It is much like the old 'chicken and egg' conundrum. The router must know the congestion points to prioritize routing; however, it does not know these congestion points until it starts to route.

We must also understand that placement of components and the real estate also play a role in the ease or difficulty autorouting. These are factors that are controlled outside of the autorouter.

The biggest effort in autorouting is constraining it. There need to be rules in place to ensure that the routes are viable. There is a delicate balance. The more we constrain it, the more difficult it gets for the autorouter to complete the route.

More importantly, we must consider the time it takes to autoroute. It is a rare to get the routing correct on the first attempt. This is the process of reviewing the result, modifying the rules or constrains, tweaking the order of the strategy, and running it again. When the autorouter is done, the work must be reviewed. There will be odd routing formations that are ‘legal’ within the constraints, but would be considered poor routing practices if routed manually. The question is whether one wants to visually pour over the layout and correct issues or simply hand route each line and review the work as they are progressing.

In the end, we must decide whether or not to use an autorouter. This question is completely dependent on the board real estate, the number of components and their pins, the clearances allowed, the strategies available and the number of layers within the board to operate. Though autorouters are used, the vast majority of boards are still routed by hand. The human mind is still the better ‘autorouter.’

Several EDA companies like Altium have moved away from the all-encompassing autorouter. An all-purpose router becomes a no purpose router. There are too many scenarios and factors to make this viable. In recent times, they have introduced local routers which let the autorouter focus on a very specific location. Once the router has completed the task in the space provided, the user can immediately review and clean up before moving on.
3D in Altium Designer

Most companies using Altium Designer have adopted the use of 3D images into their PCB layout process. This process allows the mechanical and electrical design aspects of the design to better integrate. Not everyone designing PCBs has access to a dedicated mechanical engineer or high end mechanical tools such as AutoCAD, Solidworks, or ProE; however, this doesn't mean that the PCB designer cannot take advantage of the 3D capability with Altium Designer. We will focus on the ways a PCB designer can create or obtain 3D components in Altium Designer. Topics include:

- Why using 3D in PCB design is beneficial if not crucial?
- A brief review of the how to draw 3D bodies in Altium.
- A brief review of importing STEP files into Altium and how to orientate them.
- Demonstrate the 3D batching capability of Altium.
- Demonstrate a novel way of creating 3D objects in Altium that the 3D bodies cannot do (i.e. gull wings).
- Recommendations and sites for finding components on the web.
- A brief demonstration of the Onshape mechanical tool. This will be presented by Tom Cassidy, a member of our team who uses this tool frequently to quickly draw 3D bodies for many of our service engagements.

Improving the Altium ECAD / MCAD Connectivity Experience

In 2008 Altium revolutionized the way we handle information between the ECAD and MCAD tools. This first generation of tools can be summarized as the passing STEP files to and from the ECAD and MCAD tools. Since then, we have been stuck in this 1st generation of what we call mechatronic collaboration.

Through their partnership with SOLIDWORKS, Altium has brought forth the 2nd generation of mechatronic collaboration. This collaboration not only improves the ability to move data between the MCAD and ECAD worlds, it also removes much of the manually administrative tasks that the 1st generation of collaboration demands. It goes beyond moving STEP files and in fact, it is not even moving STEP files!

Navigating Altium Designer

What about the rest of us who need to look at the handy work of our colleagues in Altium Designer? That's what we are going to focus on in this webinar - How to navigate Altium. Topics include:

- The importance of file extensions and their meanings
- The key panels for navigation and were to find them
- Ways to zoom, pan and navigate the tool in both the schematic and PCB layouts
- How to open and view files that are under version control

A must see for anyone who tired of running to an Altium literate colleague to explore the design work of the project.

Altium Designer 20 Installation

This is an Altium Designer 20 installation guideline from the beginning to the end.

Electronic Design Process in Altium Design

Since its introduction in 2002, Altium Designer has been adopted by many designers throughout the world for their PCB design needs, from independent contractors and small businesses, to major corporations such as Siemens, Texas Instruments and SpaceX. Regardless of the size of the company or the number of designers working on the project or the complexity of the design, Altium Designer provides the tools and capabilities necessary move the design from conception to manufacturing.
Wire/Cable Harness Design

Have you ever wondered if there is a better and easier way to design wire and cable harnesses? Look no further! Cable harnesses are a challenge in the world of product design, as they require both mechanical and electrical information, yet the creation of a harness design has generally been relegated to whatever drawing tool was available within the design team. In many cases, either mechanical CAD or schematic capture tools (which were developed for the creation of PCBs) have been used to create harness and cable designs. As a result, these drawings are typically “dumb” drawings, without intelligent data nor are they easily manipulated to accommodate changes.

It may surprise you to know that Altium Designer can be used to create intelligent cable drawings. This can be useful in situations in which a handful of cables are needed for test beds or to interface one PCB to another PCB or computer. However, this requires a deep knowledge of the library, the Altium schematic tool and its limitations.

What's this Vault, AVS, NEXUS, Concord Pro all about?

In 2009, Altium embarked upon an ambitious plan to transition the design work to the manufacturing process seamlessly through a consistent workflow with real-time component information. They recognized that this process had to be repeatable, accountable, and traceable. Released in 2011, it became known as “The Vault.”

With changes to the name, the functionality, and the various offerings over the years, it’s been difficult to understand what exactly is NEXUS or Concord Pro. Is it a library database? Is it a repository? Is it a version control system? In short – it’s all of them.

This webinar will focus on:

- What it is
- How it is structured
- What it does

No marketing hype, no glossies. Just the facts.

Disclaimer: Nine Dot Connects does not have any affiliation with Altium Inc. This webinar is an independent assessment of the tool.

Leaving a Lasting Imprint (or in this case, footprint)

Where symbols are all about information in a dimensionless schematic, footprints are about the real world component in a PCB layout. All too often, it is a footprint that causes the respin of a board. This month, we will work our way through several footprint examples, both surface mount and thru hole. In addition, we will cover:

- Tips and tricks when creating unusual pads
- Demo the new Via Pad library introduced in AD15
- Provide some insight about tolerances
- Demonstrate some of the lesser known, but very practical features

Symbol Creation - Setup for Success

As you may have found with Altium Designer, the schematic symbol editor provides a number of options that allow the user to highly customize their symbols; however, maintaining this look and feel can be tedious, especially when using creating components manually.
True, Altium does have some methods that provide “power steering” to the effort; however, in many cases, it’s a matter of knowing what and where they are. More so, taking advantage of “storage” features that have been in the tool for years, yet very underutilized.

Join us as we walk you through the creation of a typical symbol in an untypical way.

DbLink Demystified

Tucked away in the File » New » Libraries sub menu of Altium Designer is a mysterious Database Link file (.DbLink). It is a much underutilized feature which can be a great benefit those with a company database and have little or no permission for direct use of it in the PCB design process. With the proper set up, the DbLink is capable of updating the parametric data of existing components in the schematic with an external database.

In this video, we will discuss:

- The differences between the DbLib and the DbLink.
- How Altium Designer matches components with database information
- How to make use of the DbLink file

This is a “must see” for anyone needing to access the company database for its information, design a library that is flexible to SPICE simulation work, or is using SOLIDWORKS PCB.

Linking Altium DbLink to Excel

In this video, we go through the linking of a database (in Excel) to a DbLib file in Altium Designer 20. This example addresses multiple footprints and library paths.

Design Reuse in Altium Designer

The term ‘design reuse’ is much touted in the field of electrical engineering. Obviously, the ability to achieve design reuse will result in both cost and time savings. Implementing it is a bit trickier. Altium Designer has two features to assist in design reuse – Snippets and Device Sheets. In this upcoming webinar, we will cover both of these features in detail.

In addition, we will briefly discuss some the challenges that comes with organizing your designs for possible reuse.

DRC Revisited Part 1

In 2013, we launched our monthly webinar series with a focus on the Design Rule Checker in Altium Designer. This was well received and is one of our most watched videos. But, we also know since then, Altium has added a number of features to enhance the DRC capabilities, both in terms of rule creation and the ability to quickly determine a flagged issue.

In Part 1 we will cover the following to demonstrate the capabilities within Altium Designer 19:

- Purpose of design rules
- Design rule dialogue box
- Preferences related to the design rules
- Pattern and detail overlays
- Design rule waiving
- PCB rules and violations panel
- Violation Details dialogue box
- Batch/Online DRC options
In 2013, we launched our monthly webinar series with a focus on the Design Rule Checker in Altium Designer. This was well received and is one of our most watched videos. But, we also know since then, Altium has added a number of features to enhance the DRC capabilities, both in terms of rule creation and the ability to quickly determine a flagged issue. Therefore we felt it was time to create Part 2 to bring it up to date.

In Part 2 we will cover:

- Classes
- Priorities
- Query creation using various methods
- Regional rules
- Special Cases

**Design Rule Checker**

The Design Rule Checker (DRC) in the PCB layout editor of Altium Design is a powerful tool to help designers constrain their designs to reduce, if not eliminate, errors that may result in manufacturing issues or other undesirable electrical effects (e.g., SI and EMI issues). Many underutilize the DRC checker capabilities because they are not intimately familiar with how to get the most out of it. In this webinar, we will dig deep into the capabilities of the DRC checker and show you the tools Altium has provided to allow you to quickly create powerful rules with a minimum of effort.

**Topics Include:**

- Regional rules (for areas that have rules that are different from the rest of the board)
- Using the PCB Filter to create rules
- Storing favorite queries for reuse
- Storing and loading a rule set
- Using of Classes to help make rule creation easy
- Using the Find Similar Objects to help build rules

**Making the Most of Draftsman (and Blueprint)**

If you have ever had to create documentation for PCB fabrication and assembly you know there is information which needs to be conveyed but not captured in the drawings or in the formatted files such as Gerber, ODB++, or IPC-2581.

We depend on mechanical layers, e-mails, Microsoft Office (Word and Excel) and phone conversations to capture this information. This is manually intensive and tedious, and done under project deadline pressures. Inevitably, something is forgotten or ill defined, and it’s another day of delay with an urgent flurry of phone calls and e-mails.

Draftsman was added to Altium Designer to reduce this effort and allow the ‘templatizing’ of this activity, therefore providing consistent data.

We will show you how to make good use of this tool and provide you some insights into the best features for your documentation.

These tool features work best when one understands the different types of documentation. We encourage you to take a look at our webinar on the topic “They want WHAT document?”

---

**General Inquires:**
info@ninedotconnects.com
(214) 699-7719
What's this Co-Design Mechatronic mishmash all about?

The ability to collaborate between ECAD and MCAD tools has been around for many years. In 2008, Altium took this collaboration to another level with their enhancements of their 3D capabilities.

With the release of the SOLIDWORKS 2019 products, collaborating between the Altium Designer and SOLIDWORKS CAD is more powerful than ever. Practically every primitive – components, copper, silk, hardware and even the board itself - can be pushed between these two tools in near real time.

In this webinar, we will discuss:

- The software tools necessary for this collaboration
- The configuration of the software tools for real time collaboration (yes, version levels play a significant role)
- A demonstration of bidirectional hand off between the Altium Designer and SOLIDWORKS CAD

Regardless of whether you have Altium Designer and/or SOLIDWORKS CAD, we will gain a better understanding of the data flow.

Exporting 3D of a Folded Rigid/Flex Design

This video provides a quick overview on how to fold the flex rigid board and then export it into a STEP.

Note that flex-rigid was introduced in version 14 of Altium Designer. The ability to export the folded view came in 14.3.

If you are new to flex-rigid, but would like to explore it, open the Bluetooth Sentinel project that Altium provides with the installation. It is usually located: C:UsersPublicDocumentsAltiumAD16Examples.

High Speed Design Rules

Buried in the PCB design rule set of Altium is a list of high speed rules. The rules themselves are rather straightforward; however, their use as it relates to high speed design is not necessarily obvious. This month's webinar will explore these rules and discuss the application of the rules as they relate to high speed design issues. Rules that are not listed under the category of "high speed design" but are very applicable to high speed design will also be discussed. In addition, we will also review the design rule features that Altium added to their PCB layout tool during the past year.

For your viewing consideration, we have enclosed a link to our 2014 webinar on the Altium's design rule checker. Though the dialogues have changed recently, the functionality and capabilities introduced in the video are still available within the tool. This is a great video for those trying to get acclimated to Altium's design rule methodology and a good refresher for those who want to take advantage of the many rule features that Altium has to offer. This video was one of the most requested webinar recordings of 2014, and we are thrilled to release this publicly.
A Tour of Altium Libraries

The Libraries can play a major factor in design process efficiency. When a good library is in place:

- Knowledge of a component is captured and shared by all members of the design team.
- The generation of the Bill of Material is automated.
- The look and feel of the schematics and PCBs of the company are uniform, professional and organized.

This webinar will briefly walk you through all of the libraries that Altium Designer offers:

- Schematic Libraries
- PCB Libraries
- Integrated Libraries
- Database Libraries
- SVN Database Libraries
- Component Libraries and their association to Vault

Library – Database Management

Vault, CIS, Arena, Agile, PLM, SQL, Access, Excel...

Regardless of what you call it or from whom you bought it, the heart of all component library management systems is a database. In this webinar, we will explore the concept and flow of a database.

We will focus on the most important factor in the successful implementation of a database – the point of component entry into the database.

We will be using the L9 database management solution to provide insight as we take this journey through a component database library.

If you are looking for ways to improve your existing library configuration or need a better understanding of component library databases, you need to attend this webinar. It will be time well spent!

A Robust Library Example

Would you like to obtain a free 'robust' library?

During the creation process for our recent webinar on DFM library considerations, we decided to pull together a library example to help demonstrate the concepts. They say a picture is worth a thousand words, so we decided that a good library example should be worth a handful of specifications...

The library contains 34 components of various types, from discrete components to mechanical hardware. All components have been placed on the schematic along with their corresponding footprints on a PCB layout sheet to allow you to see the results of the library when the information is consistent. In addition, we wish to highlight the following:

Symbol Library:

- Method of handling the power on discrete logic gates
- A method for naming symbols
- A method for description information
- Key Static parameters
Nine Dot Connects Webinar Video Listing

- Visibility and order of certain parameters for schematic display

**Footprint Library:**

- Etched components such as an inductor
- Courtyard for all footprints
- Assembly layer information for all footprints
- Thermal pads designed to avoid Via-In-Pad

**Mechanical Layers - Revisited**

In 2017, we conducted a webinar on mechanical layers in Altium Designer. During that webinar we introduced a mechanical layer set which many of you have used.

A number of innovations have been added to Altium Designer that make it easier to propagate established mechanical sets to new libraries and layouts. We will demonstrate these features in this webinar. In addition, we will introduce a revised version of the mechanical layers. For those of you who are using our existing layer set, have no fear. The new features in Altium make it easy to stick with what you have already have established.

**How to Tame your Mechanical Layers**

Let’s face it - mechanical layers are an important ‘nuisance’ in the design process. Getting everyone to use the layers consistently (or at all) is certainly half the battle; however, organizing the mechanical layers for documentation purposes can be another hassle all together. All too often, we are constantly turning on and off layers to see what we need to see during the layout process.

It may surprise you that there are ways in most EDA tools to organize all layers so that viewing the different aspects of the design are practically instantaneous. We will demonstrate this concept in Altium Designer. We will also provide valuable collateral at the end of the webinar for your use in your design tools.

We provide the actual Altium Designer project files that we use in the webinar along with the 6-page Mechanical Layer Configuration Guide.

**Multi-board Assembly in AD18**

Altium Designer 18 introduced a feature that had been requested by the user community for years – The ability to quickly and intelligently combine multiple boards. We will explore this new project type known as the PrjMbd, as shown below:

- Demonstrate the minimal prep work to stage a PCB project for a multi-board project
- Show the multi-board schematic editor and its primitives
- Examine the connector manager and its ability to modify names for consistent naming between labels
- Use the assembly to show the physical connection between the PCBs

Below is the 3D image of the RangeFinder project created in the multi-board project assembly editor that we will showcased in this webinar.
Altium Outjob

How important is it to create correct and consistent documents for your PCB designs? Do you have a process and tools to perform the task of correct, consistent PCB documentation creation in an efficient and automated way?

The completion of a board layout is simply a milestone; it does not mark the end of a project, but rather, the beginning of an effort to get everything documented. Documentation provides the information necessary to communicate the designer’s intent to those who will fabricate, assemble, test, and in some cases, manufacture in high volume. In addition, it is also necessary to allow you and your colleagues the ability to revisit, revise, and improve existing designs.

Altium Designer provides a batching capability called the Outjob file. This file allows a team or designer to set up the configuration of their documentation so that it generates all the PCB documentation in an exact and consistent fashion for each project. Bill of Material, Gerbers, ODB++, Pick and Place files, Drill files, 3D drawings, PCB layout drawings, and schematics can all be set up in with the Outjob files to ensure consistent and professional documentation.

PCB Translation - “Invisible Insanity”

The famous science fiction writer Isaac Asimov was quite fond of telling jokes. He once joked about a computer developed to translate between English and Russian. As a test case, they decided to input the phrase “out of sight, out of mind.” They immediately took the result in Russian and fed it back into the computer. The result: “Invisible insanity.” This, at times, describes the process of translating a PCB design from one tool to another.

EDA tools are like spoken languages. As a result, the effort to migrate a design between tools is rarely simple. There are several “gotchas” that need to be realized and avoided, and nuances to be understood and managed. This assumes that one was able to get the bulk of the design translated, to begin with.

The purpose of this webinar is to walk through a translation performed on a Beaglebone board. While this will be imported into Altium, this is must-see for anyone who is either:

- Contemplating a move to another software tool, or
- Wanting to take an existing board and translate it to the tool of their interest, or
- Wanting to hire a contractor who uses another tool.

Query Language

In our last two webinars pertaining to DRC rules, we have focused on the features within Altium Designer to create queries for the creation of rules and for selecting primitives within the layout. As seen in our last webinar, one can make use of several tools in order to compose a query.

However, the knowing handful of queries can make the task of creating rules or searching for features easier. In this webinar, we will explore some of the more practical queries in addition to understanding the syntax.
When Designs and Designers Need to Be 'Flexible'

There have been many webinars focused on rigid-flex design. Many of these webinars come from manufacturers of rigid-flex boards. They explain what is necessary for us to achieve success in manufacturing these boards. We have also seen a few webinars from ECAD software companies showing us the various capabilities of their tools.

What is missing? They are rigid-flex considerations from a designer’s point of view. Our tools can do it, and our manufacturing can create it, but the real question is - how do we design it? That will be the focus of our upcoming webinar.

We will explore the following rigid-flex design topics:

- Board outline and bending
- Design rules pertaining to the flex circuit
- Component placement on a flex circuit
- Copper types on a flex circuit (pours, fill and vias)
- Routing on a flex circuit

Spelunking in the Altium Schematic Preferences - Part 1

What's interesting about the preferences in Altium Designer is they are to provide options and flexibility, making the tool more custom to the user.

However, many of us do not go spelunking in the preferences because we do not feel we have full command of the tool, or we simply choose to accept the default settings. This defeats the purpose of the preferences!!

It dawned on us at Nine Dots that the preferences are a powerful feature of Altium Designer, but are generally overlooked. We also realized that to show all the preferences in 1 hour would not allow us to cover them adequately. In this webinar, our focus will be specific to the schematic preferences. We intend to demonstrate and explain all of these schematic preferences.

We covered the following topics and feel free to download the preference sheets that we showcased during the webinar:

- General
- Grids
- Break Wire
- Default Units

Routing Tools in Altium Designer

Since its introduction in 2002, Altium continues to add to and improve its flagship product, Altium Designer. Its routing tool is no exception. Altium rarely removes a routing feature, knowing that entrenched habits (a.k.a. muscle memory) play a key role in a user's ability to navigate the tool. For example, the hotkey sequence "P T" is still available even though the routing features were moved from the Place menu (hence the use of the letter 'P' for the start of the hotkey sequence) to the Routing menu.

There are so many little but useful features and shortcuts, that someone with years of experience in the tool may have forgotten them or may not have ever known about them.

This webinar will go through the features of the interactive routing tool in the PCB editor of Altium Designer. This includes the tilde key (~) and those tools that are available in the properties panel while interactively routing.

Whether you are new to Altium Designer or very well-seasoned, there is always a useful feature or shortcut to be discovered.

General Inquires: info@ninedotconnects.com
(214) 699-7719
You may download a document that contains tilde (~) commands for routing in Altium Designer.

We provide the Altium project that was used in the webinar. Feel free to download.

Spelunking in the Altium Schematic Preferences - Part 2

In Part 1 we began our exploration of the Altium Designer schematics preferences. This included a thorough look at the General, Grids, Break Wire and Default Units pages. This month we will look at the following schematic preference pages in detail:

- Graphical Editing
- Compiler
- AutoFocus
- Library Autozoom
- Default Primitive Pages

Exploring Scripting in Altium Designer

BEFORE YOU WATCH – the purpose of this video is to provide a very high-level overview of scripting in Altium Designer. This is not a “how to”. It provides you an overview of what you will need to know going into this effort. Unfortunately, this is not a trivial effort.

Along with this video, please consider the following commentary:

Many high-end tools offer the ability to script. It is a feature that interests most engineers who are always looking for a way to automate repetitive or boring tasks. Embarking on a script, especially one that has to do with evaluating the position of primitives, is much like writing a novel. It may be exciting at first, but it can take a good deal of time and perseverance to get it up and running flawlessly. More often than not, even though we have the interest, work obligations get in the way.

For example, if one can repeat key clicks and mouse motions 100 times and do it in 5 minutes, is it worth investing 20 minutes researching if a certain feature exists and if it does, how to use it? Then, is it worth investing 1 or 2 hours if it needs to be coded? If mashing a bunch of keys and mouse clicks dozens of times is a one-time proposition, then most of us just grin and bear it. If we have to do it again at some time in the future or if we know that our colleagues could also benefit from automating this task, then we may consider the other options.

This question was posed to one of our experts in Altium scripting. As if scripting was not difficult enough, Altium makes it more difficult by not providing all the processes and their options. When someone approaches Nine Dot Connects about a script, it is carefully reviewed to ensure that scripting is viable option.

As for whether one should pursue a script, consider one or more of the following criteria:

- Does it meet (a) specific need(s) which typical users believe is needed, in which they have difficulty remembering or implementing the process without making mistakes?
- Does it significantly speed up routine processes therefore saving valuable time?
- Does it automate a manual process that is so complex, obscure or esoteric that it is very difficult to remember correctly?
- Does it automate or incorporate a feature which is too complex (computationally or otherwise) for the user to do manually?
- Would it be simple and fast enough so it saves significantly more time that it takes to use it?

A good script is more than pressing a run button. It needs to provide feedback. It should have control and reporting dialog boxes to confirm it is being run, allow the setting of options, and summarize immediate results.
And, if the script is going to be sold, it needs to provide much more value than it costs to invest in the tool.

In all cases it must not conflict with procedures a company has already established (such as file naming and storage protocols, configuration managements, etc.).

It also must be something which is done fairly regularly so the user does not forget how to use it. For example a script to draw schematic templates (with borders, zones, title block, parameters, fields, section indicators, different sheet sizes, top vs. continuation sheets, logo insertion, etc.) could be really nice for a design service, but not of a lot of value to a customer at one company who would only do it once.

It would be of little value if its sole purpose replicated an intrinsic function of the tool, such as checking for connectivity. It is much safer to check and report on conditions than to modify or create layout because of the esoteric relations to other things.

In order to be created and used correctly and effectively, scripts should do relatively simple and well-bounded tasks (although repetition is straightforward). For example, a script to move vias onto the nearest grid would be simple but fixing the aftermath would be very complex and perhaps even impossible.

And finally, make certain that all other workarounds have been sought or considered before embarking on this effort.

**High Speed Design (Signal Integrity) in Altium Designer**

The term "High Speed" design a popular term in the EDA industry, used by many EDA companies to express the idea that their tool has functionality that makes it better at producing electrically stable designs. However like analog design, high speed design is part art form and part science. Though Altium Designer provides functionality to assist in high speed design, it still requires an understanding in order to properly apply those capabilities.

Altium Designer provides core capabilities to assist in analyzing, constraining, and routing high speed interconnects.

**In this webinar, we will explore:**

- The Layer Stack Manager, as it relates to proper stack up for design
- Noise and how to mitigate it through proper layer stack up
- The Signal Integrity tools that Altium provides, including cross talk and reflections
- The Signal Integrity rules and how they relate to Signal Integrity capabilities

**SOS about Library Scrubbing**

Altium's marketing push for Concord Pro as a component management system has shed light onto the gorilla in the room known as library management. They have motivated design groups to have a much needed and long delayed conversation about organizing their libraries. As a result, we at Nine Dot Connects have been fielding an increasing number of questions regarding library management. We are elated that this critical topic is finally being seriously addressed by a number of companies.

However, there is a misconception that a component management tool will create an organized library by importing existent data. Like any software process, garbage-in equals garbage-out. Regardless of the management system, the components must be prepared and formatted for the new system. More importantly, design team members must agree about how the components will be maintained moving forward. In short, a good component management system relies on a definitive setup procedure and continued consistent maintenance that follows that procedure.
In this video, we will explore:

- The general structure of a library management system
- What it takes to scrub a library
- Look at the time commitment required to scrub a library
- The effort to maintain it thereafter

SOLIDWORKS PCB, Altium Designer, and the Big Picture

Any software package that has longevity goes through an amazing evolution. For example, Microsoft Word has progressed from a word processing tool to a suite completely capable of formatting a book for publication. Intuit's Quicken went from passive computerized accounting sheets to a financial system that can actively participate in the financial transactions. When one thinks about it, any tool that you have used for years has added features and abilities to make life a little easier or more productive.

But jumping into a fully grown and mature product can be daunting and a bit intimidating, if you're new to the tool. There are so many buttons, so many options, and manuals that run 300+ pages. Sometimes one wishes that the bells and whistles can be set aside to focus on the core capabilities that makes the tool so great.

Fortunately, when it comes to Altium Designer, there is such a tool. Developed jointly by SOLIDWORKS and Altium Inc, SOLIDWORKS PCB was designed for those designers and engineers who want the power of Altium Designer's core capabilities and the added benefits of seamless MCAD-ECAD collaboration.

We will introduce SOLIDWORKS PCB (which we affectionately call 'SWPCB') and discuss the use models each represents.

Altium SPICE

This video is a brief overview of the SPICE capability in Altium Designer. It provides a general understanding of the way a user would add models to the components in addition to setting up and running the analysis.

Altium provides several SPICE specific libraries that a user will need in order to provide a stimulus to the circuit (ie, VDC, sine waves, pulses, etc). They are generally located in the example directory created during the installation of Altium Designer. The path to this directory will be something similar to C:\Users\Public\Documents\AltiumADxx.x\Library\Simulation, where ADxx.x is a version number.

The 5 libraries provided (all of which are Integrated Libraries):

- Simulation Math Function
- Simulation PSPICE Functions
- Simulation Sources
- Simulation Special Functions
- Simulation Transmission Line

Install these libraries as you would with any library in Altium.

Subversion (aka SVN)

The term Subversion (also known as SVN) is thrown around quite a bit. It is a power tool that is free of charge. As of Altium Designer Version 10, the SVN engine is included. In fact, every time you start Altium Designer, the SVN engine is idling, awaiting for you to invoke its commands.
Unfortunately, Subversion's methodology requires a bit of knowledge; otherwise it will feel a bit scattered. How do I set it up in Altium? What's a repo and where do I put it? Why do I need this tool called Tortoise when I already have SVN in my Altium Designer? Do I have to use the SVN database library in order to use SVN for my documentation? Do I need to use the Altium vault?

During this monthly webinar, we are going to present a primer on SVN. We will walk through the set up and usage of SVN as it relates to Altium Designer. In addition, we will demonstrate the use (and need) for Tortoise. A number of common scenarios will be performed.

NOTE: Though we may touch upon SVN database libraries for clarification purposes, we will be primarily focused on SVN as it relates to project documentation

Test Points - Before Layout and Way Beyond Manufacturing

Design for Test (DFT) does not occur naturally in design, especially with the advent of BGAs and components so small that one must use tweezers to grasp them let alone probe them. Yet, one of the easiest items to add to a design for the purpose of the test in fabrication, assembly, and in the labs is the test points.

In this webinar we will cover:

- The benefits of test points
- Recommended copper structures
- The various approaches to using test points within Altium Designer

Unique Id in Altium Designer

Buried in the properties of each component that is placed in any of the Altium Designer editors is a property called the Unique Id. This is a randomly generated 8 alpha character code provided to each and every component to assist Altium Designer in its net list management.

The Unique Id plays a major role in Altium Designer’s ability to maintain continuity between the schematic and PCB net lists, yet, tends to do its job so well, that most users don’t even know it’s there!

However, a little knowledge about these Unique Ids goes a long way in assisting users in establishing connectivity in imported designs, connecting schematic and PCB snippets, and even reconciling potential issues when it comes to duplicating schematic files in the same project.

xSignals and Length Tuning

Nine Dot Connects is no stranger to complex high-speed designs with very tight timing constraints. While length matched signals may be aesthetically pleasing, the effort required to obtain the end result can be daunting. The concepts involved include matched single and differential signals, T-branches, daisy-chain (fly-by) routing, and serial terminations. In order to efficiently deal with these issues the proper use of tools such as Length Tuning and xSignals is critical.

This webinar will show the functionality of both Length Tuning and xSignals in Altium Designer. Given that we have not covered the creation of differential signals in Altium Designer in prior webinars, a brief primer will be given.

In addition to some tool instruction, Tom Cassidy, our resident expert in DDR3/4 and other high speed copper configurations, will demonstrate ‘real world’ uses of both features. There are many tips and tricks that can make the process more efficient.
Cabling & Wire Harness

Cable Design 101

Creating a cable can be a frustrating endeavor. Being an electro-mechanical device by nature, its mechanical considerations are just as important as their electrical aspects. Let's face it. It wasn't something discussed in the engineering classroom.

Just to muddy the waters further, most companies don't have tools for the purposes of cable design. Many of these designs are done in all-purpose drawing tools such as Visio, Paint, or drawn in an EDA or MCAD tool in which one pushes the tool beyond its intended capabilities.

We will take a good look at cabling and the basic aspects to consider:

- The different types of cable and wire available
- The different connector types available
- The information that one needs to consider putting on a cable drawing to ensure that it will be built as envisioned

Zuken E3 will be shown as part of this process, though the information being presented can be applied to any drawing tool.

Ethernet Cable Design with Zuken E3

This video demonstrates the relative ease of pulling together a custom cable in the Zuken E3 tool. In this video, we draw an Ethernet cable consisting of two RJ-45 connectors and a CAT-6 cable.

Though Zuken E3 has much of the look and feel of a schematic tool that one would use for PCB schematic design, this tool was constructed to handle both the mechanical and electrical aspects of system level design. One must keep in mind that EDA tools are geared for electrical schematic designs, which are dimensionless by their nature. This is why it is so difficult to make a cable design with any intelligence (i.e., properties, parameters, attributes, etc.) in an EDA tool.

The E3 tool is truly a system engineering tool; cabling is simply one of several core capabilities.

We have a webinar recording that goes with this demo video, and you can find more information at http://ninedotconnects.com/webinar-request-cable.

Wire/Cable Harness Design in Zuken E3

Have you ever wondered if there is a better and easier way to design wire and cable harnesses? Look no further!

Cable harnesses are a challenge in the world of product design, as they require both mechanical and electrical information, yet the creation of a harness design has generally been relegated to whatever drawing tool was available within the design team. In many cases, either mechanical CAD or schematic capture tools (which were developed for the creation of PCBs) have been used to create harness and cable designs. As a result, these drawings are typically “dumb” drawings, without intelligent data nor are they easily manipulated to accommodate changes.

What if you are designing machinery in which there are literally dozens of cables? In addition, what if the physical layout of those cables must be understood in order for the mechanical and electrical teams to constrain their designs? To add to this complexity, consider that there could be numerous subsystems that need to be interconnected. The E3.series solution by Zuken was developed to handle these large cable harness design efforts. Mil/Aero, automotive, robotics, and heavy machinery companies have turned to E3.series in an effort to take cabling and system engineering design beyond static drawings. To add to this, E3.series can also effectively handle design of hydraulic and pneumatic systems along with electrical, so that an entire system design can be achieved in one design environment.
Nine Dot Connects Webinar Video Listing

The E3.series presentation will cover the various capabilities of the E3.series solution including:

- Creation of Schematics and Interconnection Diagrams
- Rapid creation of cable connections
- Use of "objects" to define different views of the interconnections, e.g. interconnect diagrams, wiring diagrams, manufacturing documentation...
- Seamless integration between schematics, cable and formboard designs
- Integration to 3D Mechanical Design tools for cable routing
- Auto-generation of reports, including BOM's, Cable Lists and Connection Lists

**Design for Manufacturing**

**6 Key Considerations When Starting a New PCB Design**

How many times have you said to yourself, "I wish I would have known that earlier!" It's a fact that there are hundreds of decisions that need to be made when starting a PCB design. The real struggle is which decisions need to be made first? A missed (or wrong) choice is similar to casting concrete. Once it's poured and cured, there isn't much that can be changed without a jackhammer. All too often many PCB designs have been delayed or scrapped because of an overlooked decision.

This webinar will cover 6 key considerations that need to be made before starting any new PCB design:

1. Fabrication  
2. Electrical  
3. Mechanical  
4. Regulatory  
5. Procurement  
6. DFM / DFT (Design for Manufacturing; Design for Test)

A cautionary note was brought to our attention regarding the bypass capacitors. "Usually generosity with bypass caps is a great thing, but beware LDO's as most go unstable if you go beyond the specified capacitance in the datasheet. There are a few out there that are stable at any capacitance." One example comes from Analog Devices:


**Mechanical Layers - Revisited**

In 2017, we conducted a webinar on mechanical layers in Altium Designer. During that webinar we introduced a mechanical layer set which many of you have used.

A number of innovations have been added to Altium Designer that make it easier to propagate established mechanical sets to new libraries and layouts. We will demonstrate these features in this webinar. In addition, we will introduce a revised version of the mechanical layers. For those of you who are using our existing layer set, have no fear. The new features in Altium make it easy to stick with what you have already established.

**How to Tame your Mechanical Layers**

Let's face it - mechanical layers are an important 'nuisance' in the design process. Getting everyone to use the layers consistently (or at all) is certainly half the battle; however, organizing the mechanical layers for documentation purposes can be another hassle all together. All too often, we are constantly turning on and off layers to see what we need to see during the layout process.
Nine Dot Connects Webinar Video Listing

It may surprise you that there are ways in most EDA tools to organize all layers so that viewing the different aspects of the design are practically instantaneous. We will demonstrate this concept in Altium Designer. We will also provide valuable collateral at the end of the webinar for your use in your design tools.

We provide the actual Altium Designer project files that we use in the webinar along with the 6-page Mechanical Layer Configuration Guide.

**Fab & Assembly Documentation**

In spite of the ability in modern design tools to quickly generate documentation for assembly and fabrication, in many cases, the mentality of document delivery is to still “throw it over the wall.” Sometimes we give far more information than needed. Other times, the fabricator or assembler comes to us asking for some odd document that we may have never heard of, let alone created.

This webinar is going to discuss what documents are needed for fabrication, assembly, and test; where it needs to be generated within the PCB process; and what aspects of the schematics, libraries and PCB layout lend themselves to their creation.

If you are doing any of this documentation by hand, please join us to see what you can do to make quick generation of accurate documentation much easier.

This presentation is small slice of the PCB Fundamentals training class that we have developed to assist designers to understand all of the aspects of PCBs for successful manufacturing.

**DFM Part 1 (Post-Layout Considerations)**

If you are like most electronic engineers, you may find yourself trying to further refine your skills and knowledge of PCB layout, in addition to wrestling with the manufacturing and assembly aspects of it. In the past 20 years, there have been 4 monumental changes to the electronics industry, much of which we bear the brunt of, yet seem to be ignored by the industry:

- The electronics engineer's function is now “cradle to grave.” In the past, the electronics engineer could hand off to another department; now the EE is responsible for the whole project.
- The electronics engineer was not taught layout concepts, let alone high speed design concepts.
- The electronics engineer was not taught about manufacturing or assembly.
- Electronics used to start with the electronics itself. Marketing and Mechanical are now demanding the form, fit and function.

Fortunately, DFM (Design For Manufacturing) is not that complex. It simply takes a bit of understanding to know what goes on when your designs are moving through manufacturing and assembly. With this understanding, you can design so that your product will not be hampered by the "gotchas" and in doing so provide you with better schedule and budget predictability and enable you to compress both.

**DFM Part 2 (Pre-Layout Considerations)**

Previously we aired the DFM (Design for Manufacturing) introduction as it relates to PCB layout considerations, and we received a great number of requests for not only the webinar slides and recording, but also the additional coverage of the topic.

We present DFM (Pre-layout Considerations) as part 2 of this series. Like the construction of any building, there must be a blueprint to implement; however, all too often in the electronics industry, in our rush to get to PCB layout and beyond, we dive into the schematic with little or no game plan. Though a great deal of DFM concepts are implemented in the PCB layout, the foundation for DFM lies in the pre-layout considerations.

General Inquiries:  
(214) 699-7719  
info@ninedotconnects.com
The webinar will include discussions on:

- The need for specifications and how to write a requirement
- Schematic “etiquette” as it relates to the readability and formatting of the schematics
- The necessary elements of a footprint to generate good assembly and fabrication document
- 

DFM Part 3 (Library Considerations)

In our prior webinar, we touched briefly on DFM (Design for Manufacturing) considerations as they relate to libraries. We believe that this topic needs to be covered in greater detail, given its importance in DFM. Therefore, our next webinar will be focused on the component library.

Please note that though the topic is generally applicable to any EDA tool, examples will be based on the Altium Designer symbol and footprint editors.

Footprints:

- Mechanical layers - what should be included on each layer and the purpose of the layer
- Special cases - inductors, jumpers, and other commonly used footprints which could cause difficulties with the netlist
- Naming conventions - ensuring that names allow for understanding
- 3D - its importance in the libraries and how it impacts the component clearance rules

Symbols:

- Parameters - understanding the importance of the name given and the format of the value to ensure a BOM that will be complete and correct upon completion of the schematic. We will also cover the key parameters that are critical for part identification
- Multi module components - treatment of the pins and some tips to allow modification to the symbol once in the schematic
- General tips to ensure that the symbols help your schematic tell a story

This webinar “takes the mystery out of creating a library component.”

DFM Part 4 (Stackup Considerations)

Printed Circuit Board (PCB) stack ups are a lot like a car. We use them, but most of us generally bring a car to someone to repair it. The less we know about what’s happening under the hood, the more reliant we are upon others to provide recommendations or to perform repairs and services. In a way, we are at their mercy.

We do a similar thing with PCBs - we generally hand it off to someone to deal with it. However, like a car, the more we know about the PCB fabrication process, the better informed we are about making decisions and calling the shots rather than throwing it over the wall and simply hoping for the best.

As we continue with our Design For Manufacturing (DFM) series, we are going to dig deeper on the topic of the PCB layer stack up. We’ll start by reviewing of the manufacturing process. This provides us the foundation for the many decisions we have to make.

Topics include:

- How fabrication materials impact impedance
- The types of fabrication materials and why one must be cautious about their selection
Nine Dot Connects Webinar Video Listing

- Via types and why there must be careful planning when using them

As a side note, we will be using the ICD's PowerStack Suite when demonstrating the concepts presented.

**Beyond the Design & Layout of Your PCB**

For many of us in the design and layout of a PCB, the manufacturing and assembly of our design is outsourced. We provide a documentation set to our contract manufacturer and expect them to work their magic.

However, throwing documentation over the wall does not guarantee success. Design Rule Checkers in all PCB tools are geared for electrical connectivity, not necessarily manufacturing. Quick turn PCB companies build “as is”. Without a proper understanding of the design through manufacturing flow, more time and money is spent in higher class level work and respins.

Knowledge of what is required after the Gerbers or ODB++ files are generated is essential.

At this webinar:

**Paul Taubman Nine Dot Connects**

Paul will discuss Gerber files, ODB++, IPC-2581 and their history, construct and effective use

**Ray Fugitt DownStream Technologies**

Ray will discuss DFMstream to allow you the ability to capture manufacturing issues prior to release to manufacturing

**David Hoover TTM Technologies**

David will discuss issues that are commonly detected in the manufacturing process

If you are interested in increasing your knowledge of the PCB manufacturing process and learning how to control the results better while minimizing the related risks then you will enjoy this technical video.
Fab and Assembly

Beyond the Design & Layout of Your PCB

For many of us in the design and layout of a PCB, the manufacturing and assembly of our design is outsourced. We provide a documentation set to our contract manufacturer and expect them to work their magic.

However, throwing documentation over the wall does not guarantee success. Design Rule Checkers in all PCB tools are geared for electrical connectivity, not necessarily manufacturing. Quick turn PCB companies build “as is”. Without a proper understanding of the design through manufacturing flow, more time and money is spent in higher class level work and respins.

Knowledge of what is required after the Gerbers or ODB++ files are generated is essential.

At this webinar:

**Paul Taubman Nine Dot Connects**
Paul will discuss Gerber files, ODB++, IPC-2581 and their history, construct and effective use

**Ray Fugitt DownStream Technologies**
Ray will discuss DFMstream to allow you the ability to capture manufacturing issues prior to release to manufacturing

**David Hoover TTM Technologies**
David will discuss issues that are commonly detected in the manufacturing process

If you are interested in increasing your knowledge of the PCB manufacturing process and learning how to control the results better while minimizing the related risks then you will enjoy this technical video.

Blueprint-PCB

Does your EDA tool dictate your entire PCB design process?

EDA companies do not play well with each other. We all know this and have accepted this reality. But, what if the old saying about "having your cake and eating, too" was applicable to EDA? Now, it is! Blueprint-PCB by Downstream Technologies was specifically developed to take the manufacturing and assembly data from your EDA tools to generate a common documentation package for your fabricators and assemblers.

What does this mean to you?

Use the EDA tool you want! No more tool specific mandates that reduce productivity during EDA transitions and ramp up times.

No more laborious documentation efforts. With your ability to create robust templates, documentation is simply a matter of feeding your Gerbers, ODB++ or IPC-2581 data into Blueprint-PCB and generating a documentation package that is uniform and complete in minutes!
High Speed Design

The accidental antenna and how to make one

There are antennas... and then there are accidental antennas. Accidental antennas are the noisemakers of your PCB design. They are created when the copper of the board takes on a role beyond its job of connecting point A to point B.

If an accidental antenna did not wreak havoc on the test bench or with the initial prototype, their presence all too often will be felt during FCC testing, when the project is already a dollar short and a day late.

This webinar will examine accidental antennas and provide some guidance as to how to avoid creating them. The three objectives of this webinar:

Does a properly created microstrip behave as an antenna?
What changes can be made to increase/decrease such behavior?
What makes a good shield?

As much as we would like to boil this down to a simple “rules of thumb”, some concepts should first be reviewed. We encourage you to review our recent publicly released videos that will provide additional information that you may find beneficial:

- Transmission Line AC Impedance
- What is 50 Ohm Impedance?
- Q&A from the webinar

Practical Aspects of Signal Integrity - Part 1

"There are two kinds of engineer: those who have signal integrity problems, and those that will." - Eric Bogatin

We at Nine Dot Connects will be the first to admit that Signal Integrity (SI) analysis has not been one of our main concerns. For years we believed that following "best practices" would be sufficient for all of our high-speed design needs. And it also seemed more cost effective to do a respin or two rather than spend the money to purchase and learn advanced SI tools.

However, times change and reality has reared its ugly head. Faster signal transitions, tighter layouts, lower voltage levels, costly high-density parts, and shorter production cycles are all conspiring to make SI analysis an absolute necessity for a successful product. Lifecycle Insights (September 2018) found that the average number of respin was 2.9 and the average cost of a respin was an incredible $28,482.

In response, we have come up with a webinar to present how SI analysis can be applied to real-world design examples. Tom Cassidy, who has years of experience doing high-speed design and layout, will present a series of common layout paradigms and how SI can be used to identify potential problems and provide solutions. The webinar will culminate with a finished DDR3 design, showing how SI analysis can help ensure a working layout the first time.

Although this webinar will be using a board designed in Altium and analyzed in Cadence Sigrity, the concepts are applicable to any tier-one analysis tool.

Practical Aspects of Signal Integrity - Part 2

In part one of our webinar on “Practical Aspects of Signal Integrity”, Tom Cassidy provided the basic design concepts to consider when laying out a board for high speed signals to ensure the signal's integrity. If you missed it, we encourage you to watch the recording of this webinar. The concepts presented build upon each other.

In part two, Tom will be presenting on several design use cases, including:
Nine Dot Connects Webinar Video Listing

- Reference Planes
- Stripline vs. Microstrip
- Differential Pairs
- Termination
- Crosstalk
- Fly-by vs. T-Branch
- Complete DDR3 layout

Although this webinar will be using a board designed in Altium Designer and analyzed in Cadence Sigrity, the concepts are applicable to any tier-one analysis tool.

**Practical Aspects of Signal Integrity - Part 3**

In part 1 and 2 of our webinar on "Practical Aspects of Signal Integrity", Tom Cassidy provided the basic design concepts to consider when laying out a board for high speed signals to ensure the signal’s integrity. If you missed one or both, we encourage you to watch the recording of these webinars. The concepts presented build upon each other.

In Part 3 of this webinar series Tom will present the simulation and analysis of a functional FPGA-based DDR3 memory design. In addition, he will examine some of the common layout paradigms inherent to DDR3 and explore their signal integrity implications.

- Xilinx Zynq FPGA and Micron DDR3 Memory
- "Power-aware" Simulation and Analysis
- JEDEC Compliance Reports
- Signal Termination
- Differential Pair Routing Concerns
- Alternate Routing Paradigms

Although this webinar will be using a board designed in Altium Designer and analyzed in Cadence Sigrity, the concepts are applicable to any tier-one analysis tool.

**Practical Aspects of Serial Communications - Part 4**

Serial communication is a lot like the display of cascading dominoes. The success of the cascade (energy propagating from one end to the other) relies upon each domino falling in the right direction with enough momentum to cause the next domino to fall. Serial communication is not much different when it comes to propagating an energy wave, which we call a “signal”, along a trace. But unlike cascading dominoes which make for interesting Rube Goldberg machines, we should not have to hold our breath and hope for the best when it comes to the signal integrity of our serial communication lines on a PCB.

Part 4 of the Nine Dot Connects "Practical Aspects" webinar series will focus on serial communications. We will cover several communication paradigms including SPI, PCIe, and USB3.0, using Cadence's Sigrity toolset, to demonstrate these concepts. Additionally, this webinar will explore how aspects of high-speed serial bus design are crossing over into the advanced parallel design world

**High Speed Topic: The stuff that makes up signal integrity - AC Impedance**

We introduced the concept of impedance as it relates to both cables and printed circuit boards. However, that conversation was kept with respect to Direct Current (DC) to allow us to establish the fundamental concept of impedance. If you missed this webinar or would like to refresh yourself on the topic, please check out the link and request the video recording:
In this video, we will continue to discuss impedance from the AC perspective. This is as imperative since high-speed signal integrity is completely reliant on knowing the impedance issues and requirements.

During this video, we will compare AC and DC impedances, take a closer look at the composition of a square wave, delve into the S-parameters (scatter parameters) $S12$ for isolation, and $S21$ for insertion loss, and then see how these aspects apply to the PCB.

Sean Kelly, one of senior RF/high-speed engineers here at Nine Dot Connects, will be presenting.

**High Speed Topic: What is meant by a 50 ohm impedance?**

In our past videos, we have discussed characteristic impedance as it relates to the layer stack-up of the PCB. However, we have yet to address the notion of impedance itself. Impedance has always been part of the signal traces on a PCB but not necessarily a concern. With high-speed designs becoming more prevalent and significantly faster-rising edge times now being common on all digital components (even the classic 7400 logic gate series), the understanding of impedance is now becoming a critical concern. Many of the intermittent failures that one may experience can be traced back to impedance issues.

In this deep dive, the focus will be on fundamentals of impedance:

Review the classic approach of the voltage/current relationship and explain why the inductance to capacitance relationship creates the same kind of relationship, hence the term ‘characteristic impedance.’

Discuss the three functions of impedance: power transfer, reflections, and radiation. How understanding and managing these three critical functions can help avoid attenuation, signal distortion, and cross talk.

Review of T-line theory as an infinite transmission line

This presentation will be given by Sean Kelly. Sean has been in the electronics industry for 30 years with 20 years of experience in the field of PCB design. In addition, Sean is an instructor and consultant on topics such as high-speed design and RF.

**Q&A’s from Webinar**

**Practical DDR3- Part 1**

There’s a lot of talk about the 3rd generation of Double Data Rate memory known as DDR3. We at Nine Dot Connects have laid out several DDR3 boards in the past three months. There is quite a bit of detail to know about DDR3 design and layout and unfortunately, there is also a lot of misinformation out there. We have waded through and analyzed the literature. We wish to share our findings and understanding with you.

In our two-part series on this topic, we will first cover key concepts necessary for proper signal integrity and general DDR3 design.

**Topics to be covered this month are:**

- Brief history of the DDR concept
- Comparison between the different generations of DDR
- The signaling and timing requirements for DDR3
- Understanding match length versus match delay
- Compensating for typical routing delay
- Using the iCD Stackup Planner to assist in delay matching calculation

In part 2, we will build upon this foundation by demonstrating the practical aspects of DDR3 layout techniques.
Practical DDR3 - Part 2

In part 1 of our DDR 3 webinar, we established some of the key points to consider when handling the timing issues of DDR3. In part 2, we will introduce the following copper related aspects DDR 3:

- Serpentine
  - Why they are needed
  - When to use them
  - Proper methods
  - Issue that occur when not done correctly
- Signal Group and swapping
- Pin delays and swapping
- On-Die-Termination (ODT)
  - What is it
  - Why it is important
- Vias

We will demonstrate these concepts in addition to addressing rules, schematic issues, length matching, and xSignals in Altium Designer. Please note that many of these concepts are applicable to other EDA tools.

What Drives a High Speed Signal: Voltage or Current?

In the January 2014 webinar High Speed Design Rules, we showed an experiment whose results seemed to defy the rule ofthumb that states electricity takes the path of least resistance.

Connect a power supply to a metal table and run the wire output of the power supply parallel to the table. After a 2m run, bend the wire 90 degrees and run it for 1.4m before bending it again onto the table. Now power up the circuit and measure the current density. One would expect the current return path to run the hypotenuse of the circuit triangle. Yet, this does not happen!

So what's the deal? Is this a current or voltage phenomenon? Actually, it is a fields phenomenon. The key to controlling all forms of interference, including EMI, is to control and contain the electric and magnetic fields in our printed circuit boards. Many of the common PCB design practices used by designers allow fields to spread, creating the issues we often encounter.

Our webinar this month, PC Board Design to Contain and Control Fields, will be presented by Rick Hartley. Rick has decades of high-speed design experience and has led the way in many of the bespoke practices used today.

This 1-hour webinar will help you understand the best methods to minimize interference. Topics covered:

- Location of the Energy in a circuit
- What causes Energy to spread
- Transmission Lines and Return Current Paths
- Proper and Improper use of Planes in a PC board
- Trace Routing to Contain Fields
- PCB Layer Stack-ups that cause problems
- PCB Layer Stack-ups that eliminate problems
High Speed Design Series - RF Part 1

Though Nine Dot Connects is known as a Value Added Reseller (VAR) of electrical engineering design tools, we have been rapidly expanding our capabilities to provide design services in all aspects of the PCB design flow. We have a number of talented individuals who have mastered the field of PCB design, and their stories and experiences embody the engineering spirit that brought each of us into this industry. But more so, they wish to impart their knowledge to the engineering community on topics that many of us wish to tackle and understand.

This is the first of a 3-part series on High Speed Design concepts.

In this part 1 of our 3 part series, we introduce to you Gary Gaddie, a +30 year veteran of the PCB industry. Gary has worked in both the military and commercial spaces. He is involved with the IPC and is CID++. Gary keeps abreast of the changes and innovations in PCB manufacturing and assembly technologies, allowing him to literally “Design for Manufacturing” (DFM). Gary will be discussing the basics of RF concepts in PCB design.

Topics will cover practical issues including (but not limited to):

- Antenna placement
- Waveguides
- Via stitching and via spacing
- Antipads

High Speed Design Series - EM/SI Part 2

This is part 2 of our 3-part series on High Speed Design concepts. In this episode, Jeff Condit will focus on noise mitigation in high speed design. Jeff has over 40 years of experience in many facets of electrical engineering such as hardware, software, systems, RF, digital and analog, and work experiences that range from military to commercial and everything in between. His talents extend to the ability to present engineering concepts in a concise, logical way.

Jeff will address the need for noise mitigation in high speed PCB designs:

- Identifying unintended antennas and resonators on your board
- Brief review of the fundamentals as to why noise comes about
- Concerns one must address when laying out digital designs
- Transforming impedances without creating reflections
- Practices one can use to mitigate noise on the board

High Speed Design Series - PCB/SI Part 3

This is part 3 of our 3-part series on High Speed Design concepts. In this episode, Jeff Condit will continue his discussion on noise mitigation in high speed design. The September webinar focused on the fundamentals necessary in order to gain an intuitive feel for High Speed Design.

Jeff will address the need for noise mitigation in high speed PCB designs:

- Concerns one must address when laying out digital designs
- Transforming impedances without creating reflections
- Practices one can use to mitigate noise on the board
Jeff has over 40 years of experience in many facets of electrical engineering such as hardware, software, systems, RF, digital and analog, and work experiences that range from military to commercial and everything in between. His talents extend to the ability to present engineering concepts in a concise, logical way.

**Library Management**

*Leaving a Lasting Imprint (or in this case, footprint)*

Where symbols are all about information in a dimensionless schematic, footprints are about the real world component in a PCB layout. All too often, it is a footprint that causes the respin of a board. This month, we will work our way through several footprint examples, both surface mount and thru hole. In addition, we will cover:

- Tips and tricks when creating unusual pads
- Demo the new Via Pad library introduced in AD15
- Provide some insight about tolerances
- Demonstrate some of the lesser known, but very practical features

**Symbol Creation - Setup for Success**

As you may have found with Altium Designer, the schematic symbol editor provides a number of options that allow the user to highly customize their symbols; however, maintaining this look and feel can be tedious, especially when using creating components manually.

True, Altium does have some methods that provide “power steering” to the effort; however, in many cases, it’s a matter of knowing what and where they are. More so, taking advantage of “storage” features that have been in the tool for years, yet very underutilized.

Join us as we walk you through the creation of a typical symbol in an untypical way.

**Database Library Flows**

Vault, CIS, Arena, Agile, PLM, SQL, Access, Excel....

Regardless of what you call it or from whom you bought it, the heart of all component library management systems is a database. In this webinar, we will explore the concept and flow of a database.

We will focus on the most important factor in the successful implementation of a database – the point of component entry into the database.

We will be using the L9 database management solution to provide insight as we take this journey through a component database library.

If you are looking for ways to improve your existing library configuration or need a better understanding of component library databases, you need to attend this webinar. It will be time well spent!

**DbLink Demystified**

Tucked away in the File » New » Libraries sub menu of Altium Designer is a mysterious Database Link file (.DbLink). It is a much underutilized feature which can be a great benefit those with a company database and have little or no permission for
Nine Dot Connects Webinar Video Listing

direct use of it in the PCB design process. With the proper set up, the DbLink is capable of updating the parametric data of existing components in the schematic with an external database.

In this webinar, we will discuss:

The differences between the DbLib and the DbLink.
How Altium Designer matches components with database information
How to make use of the DbLink file

This is a "must see" for anyone needing to access the company database for its information, design a library that is flexible to SPICE simulation work, or is using SOLIDWORKS PCB.

**Fundamentals of Library Structure**

The Libraries can play a major factor in design process efficiency.

When a good library is in place:

Knowledge of a component is captured and shared by all members of the design team.
The generation of the Bill of Material is automated.
The look and feel of the schematics and PCBs of the company are uniform, professional and organized.

There is no doubt that every designer wants a good library; however the big question is, "What's necessary to make a good library?" In this webinar series, we will introduce the necessary building blocks to create a library structure that can help your organization improve their work flow.

**This tool-neutral video will focus on the structure of the library:**

- The data necessary to a library and its purpose
- The necessity of good description fields and what data should be provided
- The parameters necessity and their formatting

**Libraries Built to Last**

Component libraries represent a huge accumulated investment in the design content that underpins every electronic assembly that you create. But with many different library management strategies, how do you decide which strategy best suites your organization so that the components and library structure you create today have lasting value?

In this video, we'll examine the common errors made when a library is first being developed and more importantly, we'll outline the various strategies you can take to ensure that your library infrastructure can grow as your organizational needs evolve. By keeping this clear picture of the end in mind, this webinar will help ensure your libraries are built to last.

**SOS about Library Scrubbing**

Altium's marketing push for Concord Pro as a component management system has shed light onto the gorilla in the room known as library management. They have motivated design groups to have a much needed and long delayed conversation about organizing their libraries. As a result, we at Nine Dot Connects have been fielding an increasing number of questions regarding library management. We are elated that this critical topic is finally being seriously addressed by a number of companies.

However, there is a misconception that a component management tool will create an organized library by importing existent data. Like any software process, garbage-in equals garbage-out. Regardless of the management system, the components
must be prepared and formatted for the new system. More importantly, design team members must agree about how the components will be maintained moving forward. In short, a good component management system relies on a definitive setup procedure and continued consistent maintenance that follows that procedure.

**Linking Altium DbLink to Excel**

In this video, we go through the linking of a database (in Excel) to a DbLib file in Altium Designer 20. This example addresses multiple footprints and library paths.

**A Robust Library Example**

Would you like to obtain a free 'robust' library?

During the creation process for our recent webinar on DFM library considerations, we decided to pull together a library example to help demonstrate the concepts. They say a picture is worth a thousand words, so we decided that a good library example should be worth a handful of specifications...

The library contains 34 components of various types, from discrete components to mechanical hardware. All components have been placed on the schematic along with their corresponding footprints on a PCB layout sheet to allow you to see the results of the library when the information is consistent. In addition, we wish to highlight the following:

**Symbol Library:**
- Method of handling the power on discrete logic gates
- A method for naming symbols
- A method for description information
- Key Static parameters
- Visibility and order of certain parameters for schematic display

**Footprint Library:**
- Etched components such as an inductor
- Courtyard for all footprints
- Assembly layer information for all footprints
- Thermal pads designed to avoid Via-In-Pad

**What you will learn:**
- Different library strategies and their implications
- How to choose the right library strategy for your organization
- Best-practice component management techniques for each strategy
- How ADLib and CADmium by Solution Quadrant can bridge the library process gaps

**Who should attend?**
- Engineering managers
- Designers
- Component Librarians
- Component Procurers
Tour of Altium Provided Libraries

The Libraries can play a major factor in design process efficiency.

**When a good library is in place:**

- Knowledge of a component is captured and shared by all members of the design team.
- The generation of the Bill of Material is automated.
- The look and feel of the schematics and PCBs of the company are uniform, professional and organized.

There is no doubt that every designer wants a good library; however the big question is, "What's necessary to make a good library?" In this webinar series, we will introduce the necessary building blocks to create a library structure that can help your organization improve their work flow.

**This video will briefly walk you through all of the libraries that Altium Designer offers:**

- Schematic Libraries
- PCB Libraries
- Integrated Libraries
- Database Libraries
- SVN Database Libraries
- Component Libraries and their association to Vault

**MCAD & ECAD**

**ECAD-MCAD Co-Design with SOLIDWORKS PCB Connector**

The ability to collaborate between ECAD and MCAD tools has been around for many years. In 2008, Altium took this collaboration to another level with their enhancements of their 3D capabilities.

With the release of the SOLIDWORKS 2019 products, collaborating between the Altium Designer and SOLIDWORKS CAD is more powerful than ever. Practically every primitive – components, copper, silk, hardware and even the board itself - can be pushed between these two tools in near real time.

In this webinar, we will discuss:

- The software tools necessary for this collaboration
- The configuration of the software tools for real time collaboration (yes, version levels play a significant role)
- A demonstration of bidirectional hand off between the Altium Designer and SOLIDWORKS CAD

Regardless of whether you have Altium Designer and/or SOLIDWORKS CAD, we encourage you to join us so you can gain a better understanding of the data flow.

**Improving the Altium ECAD / MCAD Connectivity Experience**

In 2008 Altium revolutionized the way we handle information between the ECAD and MCAD tools. This first generation of tools can be summarized as the passing STEP files to and from the ECAD and MCAD tools. Since then, we have been stuck in this 1st generation of what we call mechatronic collaboration.
Through their partnership with SOLIDWORKS, Altium has brought forth the 2nd generation of mechatronic collaboration. This collaboration not only improves the ability to move data between the MCAD and ECAD worlds, it also removes much of the manually administrative tasks that the 1st generation of collaboration demands. It goes beyond moving STEP files and in fact, it is not even moving STEP files!

**Bridging the ECAD and MCAD Worlds**

In 2008, with the release of Altium Designer version AD6.8, Altium revolutionized the way the ECAD world interfaces with the MCAD world. By viewing the electronics from a 3D perspective, it allowed these 2 worlds to talk. Desktop EDA has further enhanced this capability through their line of ECAD/MCAD integration products that work nicely with Altium Designer, Mentor PADS, SolidWorks, SolidEdge, and Inventor.

There is more to the interaction between ECAD and MCAD than simply passing 3D graphics. ECAD and MCAD tools serve their respective domains, and what's important to one domain may not have an equivalent or bearing on the other. Understanding these translation issues is key to making this bridge work.

Though our primary intent in this video is to show you the power of Desktop EDA, we also invite you to this webinar to learn about the nuances of bridging the communication between the ECAD and MCAD worlds.

This presentation will provide the following:

- An overview of methods of communication between ECAD and MCAD
- A brief history of the various interchange formats (IDF, IDX, and live interaction)
- A primer on library management as it relates to 3D
- A demonstration of the Desktop EDA tools between Altium and SolidWorks to re-enforce the concepts discussed in the presentation.

Our guest presenter is **Roger Castro**, a 45 year veteran in the field of engineering, of which 25 years have been focused on PCB related issues. He is well versed in both electrical and mechanical concepts and is pleased to have this opportunity to candidly speak on this subject. He is an expert in Desktop-EDA's tool set in addition to Altium Designer and SolidWorks.

**Desktop EDA's Altium Modeler for SolidWorks Demo**

DesktopEDA has been specializing in ECAD and MCAD collaboration since the 1990s. With a firm understanding of both the mechanical domain and the electrical domain, DesktopEDA provides a number of software solutions for those who wish to establish a streamlined approach to collaborating these 2 different domains without compromising the integrity that each domain demands.

This introduction video provides an overview of DesktopEDA as it relates to Altium and Solidworks. This is a solution for an electromechanical engineer or designer who has both Altium Designer and Solidworks on their machine; HOWEVER, this is only one of several solutions offered.

The following are offerings geared for specific engineers with specific tools. By using a shared network drive folder, members of the design team can pass information in real time using their domain specific tools without the need to export and e-mails files (or to learn the workings of another tool that is outside of their design domain)

**For ECAD:**
- IDF Modeler for Altium Designer
- PADS ASCII Modeler

**For MCAD:**
Nine Dot Connects Webinar Video Listing

- Solidworks IDF Modeler
- Inventor IDF Modeler
- SolidEdge IDF Modeler

**3D in Altium Designer**

Most companies using Altium Designer have adopted the use of 3D images into their PCB layout process. This process allows the mechanical and electrical design aspects of the design to better integrate.

Not everyone designing PCBs has access to a dedicated mechanical engineer or high end mechanical tools such as AutoCAD, Solidworks, or ProE; however, this doesn't mean that the PCB designer cannot take advantage of the 3D capability with Altium Designer.

We will focus on the ways a PCB designer can create or obtain 3D components in Altium Designer. **Topics include:**

- Why using 3D in PCB design is beneficial if not crucial?
- A brief review of the how to draw 3D bodies in Altium.
- A brief review of importing STEP files into Altium and how to orientate them.
- Demonstrate the 3D batching capability of Altium.
- Demonstrate a novel way of creating 3D objects in Altium that the 3D bodies cannot do (i.e. gull wings).
- Recommendations and sites for finding components on the web.
- A brief demonstration of the Onshape mechanical tool. This will be presented by Tom Cassidy, a member of our team who uses this tool frequently to quickly draw 3D bodies for many of our service engagements.

**PCB Broadview**

6 Key Considerations When Starting a New PCB Design

How many times have you said to yourself, "I wish I would have known that earlier!" It's a fact that there are hundreds of decisions that need to be made when starting a PCB design. The real struggle is which decisions need to be made first? A missed (or wrong) choice is similar to casting concrete. Once it's poured and cured, there isn't much that can be changed without a jackhammer. All too often many PCB designs have been delayed or scrapped because of an overlooked decision.

This webinar will cover 6 key considerations that need to be made before starting any new PCB design:

1. Fabrication
2. Electrical
3. Mechanical
4. Regulatory
5. Procurement
6. DFM / DFT (Design for Manufacturing; Design for Test)

A cautionary note was brought to our attention regarding the bypass capacitors. "Usually generosity with bypass caps is a great thing, but beware LDO's as most go unstable if you go beyond the specified capacitance in the datasheet. There are a few out there that are stable at any capacitance." One example comes from Analog Devices:


10 things that may surprise you about engineers in the PCB process

At Nine Dot Connects, we make an effort to address what we call the "pain points" in the PCB design flow, from functional capabilities of an EDA tool to the PCB process itself. We are capable of doing this because all of us at Nine Dot Connects have walked a mile or two in your shoes.

General Inquires: info@ninedotconnects.com
(214) 699-7719
General Inquiries:  
info@ninedotconnects.com  
(214) 699-7719

We have learned (the hard way through experience) that engineering and business mix like oil and water. Much like many things in life, the PCB process that we logically believe should happen is not the PCB process in place. By understanding what makes us engineers tick, we can address the PCB process from a very different, positive, and productive angle.

This webinar will focus on several observations that we have made here at Nine Dot Connects as we have interfaced with numerous engineers and managers. These observations are so obvious, that quite frankly, they are not obvious. Come join us for a webinar meant to cordially explain those engineering idiosyncrasies that you and your colleagues were never comfortable discussing with management.

**A Short History of the Short History of Printed Circuit Boards**

Departing from our traditional in-depth technical topics, we want to cover a short of history of Printed Circuit Boards (PCB). We want to you to relax and enjoy the information.

In an odd sense, PCBs and their manufacturing advancements has been a response to history itself - The product innovations of the roaring 1920s, the foundation for communication equipment during the 2nd World War and beyond, the backbone for the innovations brought on by the inventions of the transistor and the IC, and the "right stuff" for the electronics needed for landing a man on the moon. And of course, the "glue" that propelled us into the internet age.

Please join us for a bit of history on the PCB. And remember, those who fail to learn from history are destined to repeat it (or in our industry, respin it!)

**To Route or Auto Route, that is the Question**

We put this webinar together to address the complex issue of autorouting. It is a 'high level' presentation with the purpose of looking at the history and algorithms that are commonly used. As you will see, the effort is not simple. It's one thing to route a particular line; however, planning and prioritizing the routes are difficult. It is much like the old 'chicken and egg' conundrum. The router must know the congestion points to prioritize routing; however, it does not know these congestion points until it starts to route.

We must also understand that placement of components and the real estate also play a role in the ease or difficulty autorouting. These are factors that are controlled outside of the autorouter.

The biggest effort in autorouting is constraining it. There need to be rules in place to ensure that the routes are viable. There is a delicate balance. The more we constrain it, the more difficult it gets for the autorouter to complete the route.

More importantly, we must consider the time it takes to autoroute. It is a rare to get the routing correct on the first attempt. This is the process of reviewing the result, modifying the rules or constrains, tweaking the order of the strategy, and running it again. When the autorouter is done, the work must be reviewed. There will be odd routing formations that are ‘legal’ within the constraints, but would be considered poor routing practices if routed manually. The question is whether one wants to visually pour over the layout and correct issues or simply hand route each line and review the work as they are progressing.

In the end, we must decide whether or not to use an autorouter. This question is completely dependent on the board real estate, the number of components and their pins, the clearances allowed, the strategies available and the number of layers within the board to operate. Though autorouters are used, the vast majority of boards are still routed by hand. The human mind is still the better ‘autorouter.’

Several EDA companies like Altium have moved away from the all-encompassing autorouter. An all-purpose router becomes a no purpose router. There are too many scenarios and factors to make this viable. In recent times, they have introduced local routers which let the autorouter focus on a very specific location. Once the router has completed the task in the space provided, the user can immediately review and clean up before moving on.
A History of Printed Circuit Boards

Departing from our traditional in-depth technical topics, we want to cover a short of history of Printed Circuit Boards (PCB). We want you to relax and enjoy the information.

In an odd sense, PCBs and their manufacturing advancements has been a response to history itself - The product innovations of the roaring 1920s, the foundation for communication equipment during the 2nd World War and beyond, the backbone for the innovations brought on by the inventions of the transistor and the IC, and the “right stuff” for the electronics needed for landing a man on the moon. And of course, the “glue” that propelled us into the internet age.

Please join us for a bit of history on the PCB. And remember, those who fail to learn from history are destined to repeat it (or in our industry, respin it!)

PCB Elements

Vias: Blind, Buried, and Beyond

The Z direction on a PCB multi-layer is remarkably restrictive. We simply do not draw copper from one layer to the next. We must make use of a copper structure called the via. Over time, to accommodate small components and densely populated boards, more via options have been made available.

In this webinar, we will discuss:

- Sanity check - Make sure that the desired via type works with the stack up
- Annular rings - the equation to ensure the right fit / hit
- Explore non-traditional vias types - microvias, back drilled vias, vias-in-pad
- Documentation - what automatically gets generated and what needs to be provided as a fabrication note

High Speed Topic: The stuff that makes up signal integrity - AC Impedance

We introduced the concept of impedance as it relates to both cables and printed circuit boards. However, that conversation was kept with respect to Direct Current (DC) to allow us to establish the fundamental concept of impedance. If you missed this webinar or would like to refresh yourself on the topic, please check out the link and request the video recording:

DC Impedance Video

In this video, we will continue to discuss impedance from the AC perspective. This is as imperative since high-speed signal integrity is completely reliant on knowing the impedance issues and requirements.

During this video, we will compare AC and DC impedances, take a closer look at the composition of a square wave, delve into the S-parameters (scatter parameters) S12 for isolation, and S21 for insertion loss, and then see how these aspects apply to the PCB.

Sean Kelly, one of senior RF/high-speed engineers here at Nine Dot Connects, will be presenting.
High Speed Topic: What is meant by a 50 ohm impedance?

In our past videos, we have discussed characteristic impedance as it relates to the layer stack-up of the PCB. However, we have yet to address the notion of impedance itself. Impedance has always been part of the signal traces on a PCB but not necessarily a concern. With high-speed designs becoming more prevalent and significantly faster-rising edge times now being common on all digital components (even the classic 7400 logic gate series), the understanding of impedance is now becoming a critical concern. Many of the intermittent failures that one may experience can be traced back to impedance issues.

In this deep dive, the focus will be on fundamentals of impedance:

Review the classic approach of the voltage/current relationship and explain why the inductance to capacitance relationship creates the same kind of relationship, hence the term ‘characteristic impedance.’
Discuss the three functions of impedance: power transfer, reflections, and radiation. How understanding and managing these three critical functions can help avoid attenuation, signal distortion, and cross talk.
Review of T-line theory as an infinite transmission line

This presentation will be given by Sean Kelly. Sean has been in the electronics industry for 30 years with 20 years of experience in the field of PCB design. In addition, Sean is an instructor and consultant on topics such as high-speed design and RF.

Q&A’s from Webinar

Deep Dive - Capacitors

For many of us, the introduction to capacitors was in the form of equations as they related to the charge plates. When it comes to selecting sizes and types of capacitors for real designs, much of it tends to be based on experiences, tribal knowledge, and copying and pasting from existing similar designs.

We will present a deep dive webinar on CAPACITORS.

Our discussion includes:

- The uses for capacitors
- The various configurations
- The various materials and their advantages / disadvantages

This deep dive will be presented by our PCB expert, Jeff Condit, who has 40+ years' experience in the PCB design domain.

Deep Dive - Inductors

Inductors were widely used decades ago, for they were far easier to make than capacitors. In fact, a good deal of inductors were wound by the engineer or technician. Over time, capacitors took on a popularity that even encouraged engineers to avoid the use of inductors whenever possible.

In this webinar, we will dig deep into the concepts and uses for inductors, not only revisiting known circuits that use inductors, but to explore other ways of using them that may not be so obvious.

A review of the fundamentals will also be given, in addition to touching on eddy currents and hysteresis.

This Webinar will be presented by Jeff Condit, whose 40+ years of industry knowledge includes a great deal of experience in winding and using inductors in many of his power and analog projects.
Deep Dive - Vias

If you have attended our webinars in the past, we are always pressed to keep the presentations under an hour. With any given topic, there is far more information than what we can present. How many times have we said, “There’s more to it, but…” or “We don’t have the time to go into detail”?

Therefore, we will hold our first deep dive session. From the feedback we have received from our webinar on PCB stack ups, there is an interest on the topic of vias, in particular, their purpose; types of vias available; stack up, DFM and layout considerations; and so on.

The format of this webinar is simple enough – we will speak on the topic for 20 minutes. Though we may show pictures to help illustrate a point, this not a PowerPoint or formal presentation. In fact, we want this webinar to be driven by participant questions.

This deep dive will be presented by our PCB expert, Jeff Condit, who has 40+ years’ experience in the PCB design domain.

Intuitive Stack-up Planning & Impedance Control

How do you intuitively engineer good impedance controlled PCB stack-up and routing plans for each new design which meets all of their specific impedance control design requirements?

- Search on the Internet?
- Do hand calculations?
- Ask a friend?
- Always force the design into the same one or two because they worked in the past?
- Buy a very expensive and complex tool like Polar?
- “Feel” the best solution
- Hire some “expert” to do it?

Can you intuitively visualize and understand what it's doing and how to reliably use it?

How do you efficiently approach gathering the raw data to make the final choices?

As signal speeds increase the shortcomings of materials (like FR-4) become critical. Dielectric constants aren't constant resulting in dispersion. Loss increases and is frequency dependent resulting in signal degradation over longer runs. Gathering information on lots of materials, tabulating all the information at various frequencies along with the specific thicknesses available, selecting optimal materials, and adjusting stack-ups to compensate can be very time consuming and error-prone.

In this video, we will show you intuitively how various elements of a PCB’s structure and routing affect the electromagnetic fields and in turn the couplings, energies, and characteristic impedance of traces on it. Practical suggestions, methods, alternatives, and solutions will be discussed.
PCB Translation - “Invisible Insanity”

The famous science fiction writer Isaac Asimov was quite fond of telling jokes. He once joked about a computer developed to translate between English and Russian. As a test case, they decided to input the phrase "out of sight, out of mind." They immediately took the result in Russian and fed it back into the computer. The result: “Invisible insanity.” This, at times, describes the process of translating a PCB design from one tool to another.

EDA tools are like spoken languages. As a result, the effort to migrate a design between tools is rarely simple. There are several “gotchas” that need to be realized and avoided, and nuances to be understood and managed. This assumes that one was able to get the bulk of the design translated, to begin with.

The purpose of this webinar is to walk through a translation performed on a Beaglebone board. While this will be imported into Altium, this is must-see for anyone who is either:

- Contemplating a move to another software tool, or
- Wanting to take an existing board and translate it to the tool of their interest, or
- Wanting to hire a contractor who uses another tool.

Download Translation Checklist

Power Distribution Network & Power Integrity

Though signals tend to be the focus of impedance issues, the power distribution network (PDN) on the board is just as susceptible to impedance. The PCB layer stack up is a capacitor all its own and with the addition of bypass capacitors and inductors on the board, the ability to distribute power to avoid power starvation or to reduce ripple becomes an integral part of the design.

Topics include:

- The power distribution network and how impedances interrupt and starve components of the power they require.
- Dive deeper into the impedances that are caused by capacitors and inductors, and more importantly, how they interact with each other.
- How to manage the impedance by a careful selection of capacitors.

We will use the ICD PDN Planner tool to visually see the results as we present.

Test Points - Before Layout and Way Beyond Manufacturing

Design for Test (DFT) does not occur naturally in design, especially with the advent of BGAs and components so small that one must use tweezers to grasp them let alone probe them. Yet, one of the easiest items to add to a design for the purpose of the test in fabrication, assembly, and in the labs is the test points.

In this webinar we will cover:

- The benefits of test points
- Recommended copper structures
- The various approaches to using test points within Altium Designer
**ICD's PowerStack Suite**

When it comes to high speed design, the name of the game is understanding and controlling impedance...

At first glance, you may wonder why ICD put together software that seems to address two nearly unrelated areas. The stack up planner addresses the layers. The PDN planner addresses the use of capacitors. What gives?

When we really look at high speed design, we are dealing with the issue of the effect of impedance, both on the traces and on the power provided to the board. The trace impedances are heavily influenced by the layer stack up.

On the power planes, we also deal with impedances. Any noise that makes its way onto the Power Distribution Network (PDN) (a.k.a the power planes) needs to be given a low impedance path to the ground plane, regardless of the frequencies of the noises. Thus the need to determine which values of capacitors are needed, what kind, and how many. The PDN Planner provides over 6500 capacitors in the library. More will be added as they become available. The user can add capacitors to the library as well.

The ICD stackup planner provides accurate impedance information through its built-in field solver. By using the information in the stack up planner, the PDN planner can, in turn, provide accurate information for all noise frequencies of concern. With the ICD stack up planner, the user has access to a library of over 23,230 materials (core, prepreg, and solder mask) up to 100 GHz.

**ICD's Stackup Planner**

It may not be obvious, however, the layer stackup completely influences the trace impedances. The question is, how does one derive rules for constraints such as trace space and trace width? Some EDA tools provide equations, while others do not. In the end, this is not something that can be guessed. That's where the ICD stack up planner comes in.

One should always consult their fabricator on the stack up from the very start of the PCB layout. Sure, they can provide the impedance numbers for you in addition to a stack up itself. However, there are a lot of variables to play with, and sometimes, it makes sense to investigate other materials, dielectric thicknesses, different heights, different trace widths and different trace spacing.

With the ICD stack up planner, the user has access to a library of over 23,230 materials (core, prepreg, and solder mask) up to 100 GHz. Frequency capability of the materials is also important. When placing a core material it should have the same weight of copper on both sides. Although one can specifically order any combo (at a price) it is economical to use 1oz or ½ oz on each side.

In addition, with its built-in field solver, the ICD can provide accurate impedance measurements for each layer in addition to providing rules such as trace space and trace width. In the end, you not only control what you believe to be the optimal stack up, you have the ability to have an intelligent conversation with your fabricator.

**ICD's PDN Planner**

In the high speed design, the focus tends to be on signal integrity aspects of the traces; however, the power planes are also subject to integrity issues at high frequencies. The fact of the matter is that the Power Distribution Network (PDN) of a board is more than just large areas of copper for the power and ground. Even the bypass capacitors chosen may not only fail to mitigate noise at certain frequencies, the improper selection of the capacitor values can contribute to the noise levels. As a result, a board will not only have issues of its own, but will be at the mercy of the external environment in which it must operate.
The ICD PDN planner was developed to give the layout designer the ability to select bulk and bypass capacitor values that will optimally provide low impedance paths for noise at all frequencies of concern on the PDN. Since the PDN planner can access the data of the ICD stack-up planner, the effects of the board can be added to the field solver to provide accurate results.

The ICD PDN Planner provides over 5,600 capacitors in the library. More will be added as they become available. The user can add capacitors to the library as well.

Power Supplies

Powering Up on Power Supplies, Part 1

Nine Dot Connects invites you to attend our new design series on power supply design. This video series was developed with the intent of providing an intuitive understanding of power design so that one can get a feel for the concept. Our approach will be from the ground up, starting with the key elements needed in power supply and working our way to the structure and types of power supplies.

Our first video will focus on the common components that make up a power supply.

Topics include:

- An in-depth review of LRCs (primarily, the inductor and capacitor aspects)
- Parasitic effects of the LRCs
- Voltage and current sources
- Demonstration of the concepts using SPICE modeling in Altium Designer

This design series will be presented by Jeff Condit, a senior engineer at Nine Dot Connects with over 40 years of experience in the field of electrical engineering. Jeff recently presented a 2-part series on high-speed PCB design. If you are interested in viewing these presentations, click here to view. Be sure to look for the recordings of the webinar for "High-Speed Design Series - EM/SI Part 2 and PCB/SI Part 3.

Future webinars in this design series will cover topics such as:

- Limitations and nonlinearities of inductors and transformers, such as B-H loops, winding resistance, and eddy and hysteresis losses
- Design criteria for transformers, inductors, and chokes
- Ideal basic power converter topologies and how they work (this includes where the current flows and when and what happens when things switch)
- SPICE simulations of MOSFET, a Schottky rectifier, and several sources for both buck and boost power design

Powering Up on Power Supplies, Part 2

Nine Dot Connects invites you to attend our Part 2 of the "Power Supply Design Series." This video series was developed with the intent of providing an intuitive understanding of power design so that one can get a feel for the concept. This session will build off of the theory covered in Part 1 (recording below and to the right) and focus on specific configurations of power supply designs. We will analyze two power circuits in the video, the buck, and boost power circuits.
Nine Dot Connects Webinar Video Listing

**Topics include:**

- An overview of how the circuit works
- A SPICE simulation (performed in Altium Designer)
- A PCB layout of the concept

Each circuit we analyze will have a Q and A portion to ensure that all aspects of the circuit being presented have been covered.

This design series will be presented by Jeff Condit, a principal engineer at Nine Dot Connects with over 40 years of experience in the field of electrical engineering. Jeff presented the Part 1 of the current series last month, and a 2 part series on high speed PCB design a few months back. If you are interested in viewing these 2 part series, [click here to view](#).

**Power Distribution Network & Power Integrity**

Though signals tend to be the focus of impedance issues, the power distribution network (PDN) on the board is just as susceptible to impedance. The PCB layer stack up is a capacitor all its own and with the addition of bypass capacitors and inductors on the board, the ability to distribute power to avoid power starvation or to reduce ripple becomes an integral part of the design.

**Topics include:**

- The power distribution network and how impedances interrupt and starve components of the power they require.
- Dive deeper into the impedances that are caused by capacitors and inductors, and more importantly, how they interact with each other.
- How to manage the impedance by a careful selection of capacitors.

We will use the iCD PDN Planner tool to visually see the results as we present.

**Simulation**

**The accidental antenna and how to make one**

There are antennas... and then there are accidental antennas. Accidental antennas are the noisemakers of your PCB design. They are created when the copper of the board takes on a role beyond its job of connecting point A to point B.

If an accidental antenna did not wreak havoc on the test bench or with the initial prototype, their presence all too often will be felt during FCC testing, when the project is already a dollar short and a day late.

This webinar will examine accidental antennas and provide some guidance as to how to avoid creating them. The three objectives of this webinar:

- Does a properly created microstrip behave as an antenna?
- What changes can be made to increase/decrease such behavior?
- What makes a good shield?

As much as we would like to boil this down to a simple “rules of thumb”, some concepts should first be reviewed. We encourage you to review our recent publicly released videos that will provide additional information that you may find beneficial:

- Transmission Line AC Impedance
- What is 50 Ohm Impedance?
- Q&A from the webinar

General Inquires: info@ninedotconnects.com  
(214) 699-7719
Practical Aspects of Signal Integrity - Part 1

"There are two kinds of engineer: those who have signal integrity problems, and those that will." - Eric Bogatin

We at Nine Dot Connects will be the first to admit that Signal Integrity (SI) analysis has not been one of our main concerns. For years we believed that following "best practices" would be sufficient for all of our high-speed design needs. And it also seemed more cost effective to do a respin or two rather than spend the money to purchase and learn advanced SI tools.

However, times change and reality has reared its ugly head. Faster signal transitions, tighter layouts, lower voltage levels, costly high-density parts, and shorter production cycles are all conspiring to make SI analysis an absolute necessity for a successful product. Lifecycle Insights (September 2018) found that the average number of respin was 2.9 and the average cost of a respin was an incredible $28,482.

In response, we have come up with a webinar to present how SI analysis can be applied to real-world design examples. Tom Cassidy, who has years of experience doing high-speed design and layout, will present a series of common layout paradigms and how SI can be used to identify potential problems and provide solutions. The webinar will culminate with a finished DDR3 design, showing how SI analysis can help ensure a working layout the first time.

Although this webinar will be using a board designed in Altium and analyzed in Cadence Sigrity, the concepts are applicable to any tier-one analysis tool.

Practical Aspects of Signal Integrity - Part 2

In part one of our webinar on "Practical Aspects of Signal Integrity", Tom Cassidy provided the basic design concepts to consider when laying out a board for high-speed signals to ensure the signal’s integrity. If you missed it, we encourage you to watch the recording of this webinar. The concepts presented build upon each other.

In part two, Tom will be presenting on several design use cases, including:

- Reference Planes
- Stripline vs. Microstrip
- Differential Pairs
- Termination
- Crosstalk
- Fly-by vs. T-Branch
- Complete DDR3 layout

Although this webinar will be using a board designed in Altium Designer and analyzed in Cadence Sigrity, the concepts are applicable to any tier-one analysis tool.
Practical Aspects of Signal Integrity - Part 3

“There are two kinds of engineer: those who have signal integrity problems, and those that will.” - Eric Bogatin

Part 1 and 2 of our webinar on “Practical Aspects of Signal Integrity”, Tom Cassidy provided the basic design concepts to consider when laying out a board for high speed signals to ensure the signal's integrity. If you missed one or both, we encourage you to watch the recording of these webinars. The concepts presented build upon each other.

In Part 3 of this webinar series Tom will present the simulation and analysis of a functional FPGA-based DDR3 memory design. In addition, he will examine some of the common layout paradigms inherent to DDR3 and explore their signal integrity implications.

- Xilinx Zynq FPGA and Micron DDR3 Memory
- "Power-aware" Simulation and Analysis
- JEDEC Compliance Reports
- Signal Termination
- Differential Pair Routing Concerns
- Alternate Routing Paradigms

Although this webinar will be using a board designed in Altium Designer and analyzed in Cadence Sigrity, the concepts are applicable to any tier-one analysis tool.

Practical Aspects of Serial Communications - Part 4

Serial communication is a lot like the display of cascading dominoes. The success of the cascade (energy propagating from one end to the other) relies upon each domino falling in the right direction with enough momentum to cause the next domino to fall. Serial communication is not much different when it comes to propagating an energy wave, which we call a “signal”, along a trace. But unlike cascading dominoes which make for interesting Rube Goldberg machines, we should not have to hold our breath and hope for the best when it comes to the signal integrity of our serial communication lines on a PCB.

Part 4 of the Nine Dot Connects “Practical Aspects” webinar series will focus on serial communications. We will cover several communication paradigms including SPI, PCIe, and USB3.0, using Cadence's Sigrity toolset, to demonstrate these concepts. Additionally, this webinar will explore how aspects of high-speed serial bus design are crossing over into the advanced parallel design world

Beyond a Heatsink - Thermal Considerations in PCB Design

“Thermal” is a design item that we might expect only the mechanical engineers to consider and use flow simulation tools such as those found in SOLIDWORKS and other mechanical design tools. One must keep in mind that component selection and placement has a tremendous impact on the simulation results. The old saying “nip it in the bud” is quite germane here, and catching thermal issues early is a perfect example of this phrase.

Understanding thermal issues doesn't require a degree or training in thermodynamics. It is simply understanding what needs to be exported from your PCB design and how to set up the simulation in the proper tools.

This webinar will explore the work flow for testing thermal flow for a PCB. This webinar will be presented by Bill Dempsey, a PCB design engineer who has provided thermal simulation services to multiple clients.
Power Integrity & Power Distribution Network

Though signals tend to be the focus of impedance issues, the power distribution network (PDN) on the board is just as susceptible to impedance. The PCB layer stack up is a capacitor all its own and with the addition of bypass capacitors and inductors on the board, the ability to distribute power to avoid power starvation or to reduce ripple becomes an integral part of the design.

Topics include:

- The power distribution network and how impedances interrupt and starve components of the power they require.
- Dive deeper into the impedances that are caused by capacitors and inductors, and more importantly, how they interact with each other.
- How to manage the impedance by a careful selection of capacitors.

We will use the ICD PDN Planner tool to visually see the results as we present.

ICD’s Power Distribution Network Planner

In the high speed design, the focus tends to be on signal integrity aspects of the traces; however, the power planes are also subject to integrity issues at high frequencies. The fact of the matter is that the Power Distribution Network (PDN) of a board is more than just large areas of copper for the power and ground. Even the bypass capacitors chosen may not only fail to mitigate noise at certain frequencies, the improper selection of the capacitor values can contribute to the noise levels. As a result, a board will not only have issues of its own, but will be at the mercy of the external environment in which it must operate.

The ICD PDN planner was developed to give the layout designer the ability to select bulk and bypass capacitor values that will optimally provide low impedance paths for noise at all frequencies of concern on the PDN. Since the PDN planner can access the data of the ICD stack-up planner, the effects of the board can be added to the field solver to provide accurate results.

The ICD PDN Planner provides over 5,600 capacitors in the library. More will be added as they become available. The user can add capacitors to the library as well.

Altium SPICE

This video is a brief overview of the SPICE capability in Altium Designer. It provides a general understanding of the way a user would add models to the components in addition to setting up and running the analysis.

Altium provides several SPICE specific libraries that a user will need in order to provide a stimulus to the circuit (ie, VDC, sine waves, pulses, etc). They are generally located in the example directory created during the installation of Altium Designer. The path to this directory will be something similar to C:UsersPublicDocumentsAltiumADxx.xLibrarySimulation, where ADxx.x is a version number.

The 5 libraries provided (all of which are Integrated Libraries):

- Simulation Math Function
- Simulation PSPICE Functions
- Simulation Sources
- Simulation Special Functions
- Simulation Transmission Line

Install these libraries as you would with any library in Altium.
High Speed Design (Signal Integrity) in Altium Designer

The term "High Speed" design a popular term in the EDA industry, used by many EDA companies to express the idea that their tool has functionality that makes it better at producing electrically stable designs. However like analog design, high speed design is part art form and part science. Though Altium Designer provides functionality to assist in high speed design, it still requires an understanding in order to properly apply those capabilities.

Altium Designer provides core capabilities to assist in analyzing, constraining, and routing high speed interconnects.

In this webinar, we will explore:

- The Layer Stack Manager, as it relates to proper stack up for design
- Noise and how to mitigate it through proper layer stack up
- The Signal Integrity tools that Altium provides, including cross talk and reflections
- The Signal Integrity rules and how they relate to Signal Integrity capabilities