How to achieve consistency with Altium Designer

A technical article for Altium UK Subscription customers

The vast majority of customers we meet are interested in Design Reuse. The key is to make it easy to do the easy bits so that the customer can focus on the difficult bits. There are lots of features within Altium Designer that can help here but before you can use them efficiently, you might need to do some preparation first.

As with most engineering tasks, preparation is the key. Design your circuits with a view to re-use them and your chance of success is greatly increased. This article is the first in a series and will focus on giving you practical advice and good working practices that will provide you with efficient and consistent design processes. Success with design re-use will follow as a result.

Areas for consideration in this article include:-

1. Adopting robust net naming conventions
2. Creating custom toolbars to help with consistency
    a. Wire colours and width conventions for named nets
    b. "Standard" power objects
3. Cut, Copy & Paste techniques
4. Using Snippets to streamline rule creation
5. Ensuring robust inter-sheet connectivity

Technical article for Altium UK Subscription customers
by Tony Folan, Senior Applications Support Engineer
Premier EDA Solutions

Adopting robust net naming conventions

At schematic-level it's all about wiring and connectivity. Mistakes here propagate into the PCB-domain and this could be costly. Removing doubt and making it easy to get repeatable results are key aspects here. In support, we often get asked "Is Altium Schematic case sensitive in respect to net names?" This is a fair question but the answer we give is somewhat obvious "Who cares? Get it right!" Would you expect net "RESET" on one sheet to be connected to net "Reset" on another? How about net CLOCK on one sheet connecting to net CLOCK on another? The point here is don't trust to luck and assume the system will wipe your backside for you, just get it right!

It is important to have well defined net naming conventions to ensure consistency and to avoid unseen problems. This is especially important if you have multi-sheet designs and possibly other engineers working on those sheets (we know that one man's "0V" is another man's "GND").

You may already have your own internal standards in place but to give you a few ideas here are some possible net naming conventions.

In the latest Altium Designer reference designs you will find power nets are defined by the integer value of the voltage, followed by 'V' followed by one or two digits representing the decimal places in the nominal voltage of the net. Analogue supplies are prefixed with 'A' as are Grounds and explicitly digital supplies and grounds are prefixed with 'D' likewise. Negative voltage rails are prefixed with a minus sign.

{-}[integer]V[decimal]
e.g. 5V0, 15V0, -15V0
GND, AGND, DGND

Active low signals are suffixed with underscore '_N' and active high signals which are part of a differential pair are suffixed with '_P'. This is so that Differential Pair Directives can automate the pairing of nets on the board.

[diffpairname]_P
[diffpairname]_N

Having decided on our naming conventions, let's look at how we can use this to good effect.

Technical article for Altium UK Subscription customers
by Tony Folan, Senior Applications Support Engineer
Premier EDA Solutions

Creating custom toolbars to help with consistency

Wire colours and width conventions for named nets

We advise using specific styles for wires associated with power nets. A suggestion is to use the medium sized wire setting for DC and AC power and ground nets. Then choose a colour scheme. Perhaps something like this:

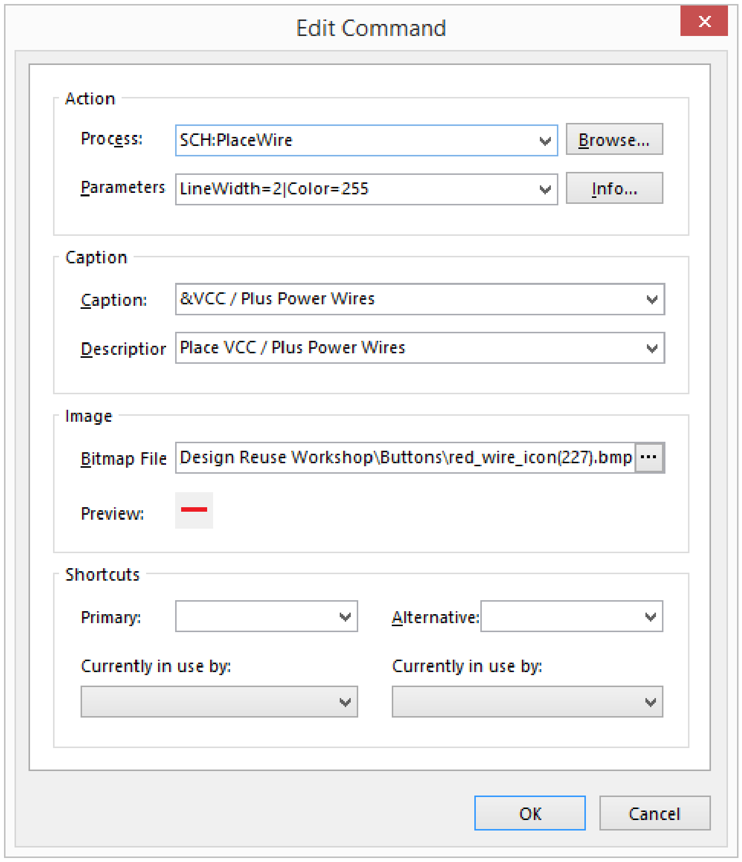
VCC / Plus Power Wires	= Red	(255)
GND / 0V Wires	= Green	(48896)
VEE Minus Power Wires	= Black	(0)
Live AC Wires	= Brown	(3039647)
Neutral AC Wires	= Blue	(16711680)

Other wires should use the small size setting and be dark blue in colour. This is the default setting in Altium Designer.

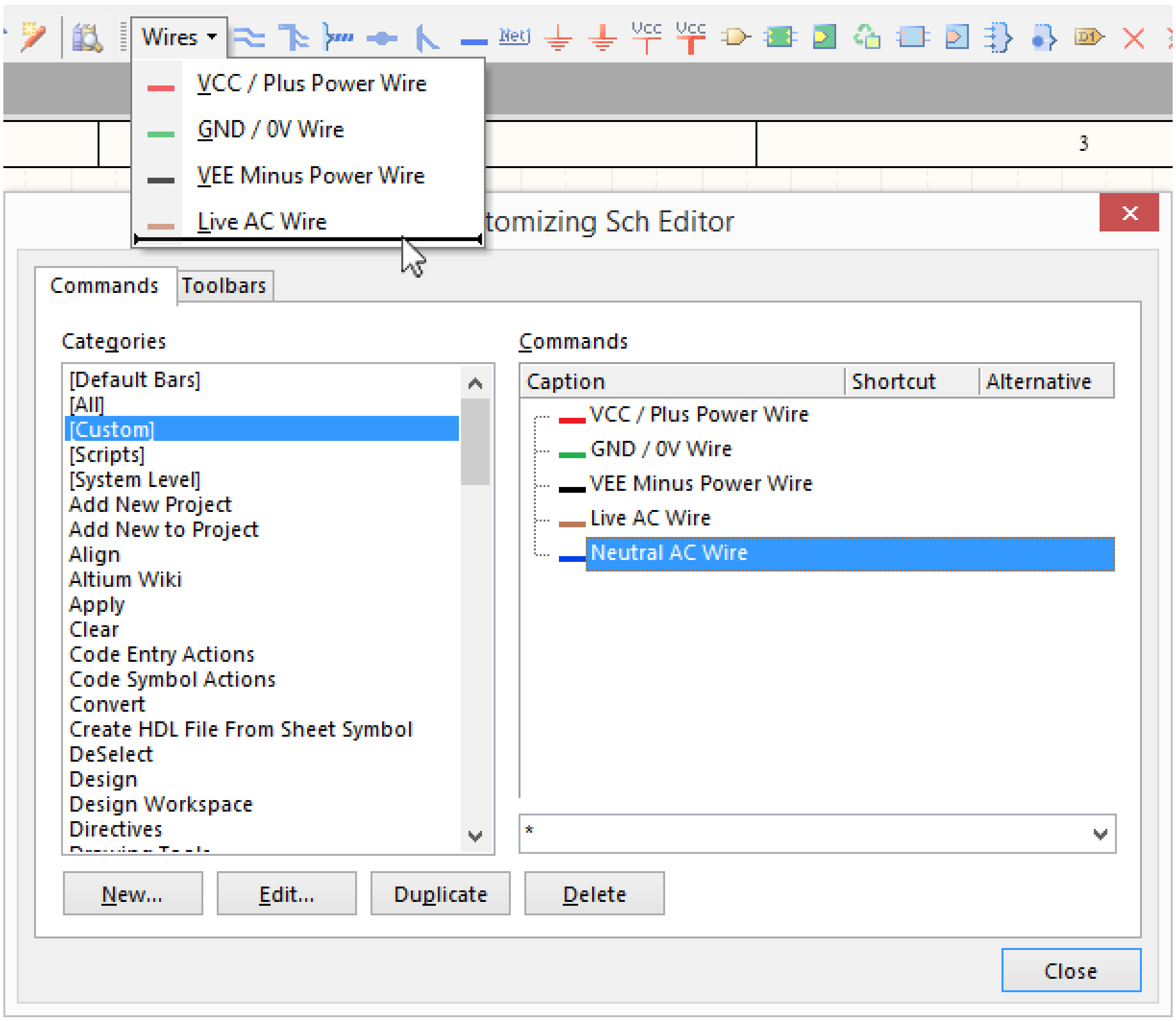
With these extra five specific wire styles, it would be pain remembering to select the right thickness and colour each time we wanted to wire a power net. We can define toolbar buttons for each and add these to the Wiring toolbar. Go to DXP » Customize » Commands » Wiring and select New. This will open the Edit Command dialog. In the Action section, click Browse to choose the process for placing a wire - SCH:PlaceWire. Then add the following Parameter. LineWidth=2|Color=255. LineWidth=2 will increase the wire size from the default setting of small, to medium. The colour number corresponds the RGB value converted from 6 digit hexidecimal number.

For example the colour blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)).

Add a Caption, Description and a Bitmap File if you wish. The bitmap file can easily be created in MS Paint. When you click ok the new command will be listed in the Custom category.



It is now easier to define more toolbar buttons for the other wire styles as we can duplicate the one we have just defined. Once you have all your wire styles setup you can simply drag these from the Customising Sch Editor dialog directly on to the Wiring toolbar. We won't do that as it is possible to go a step further. We will setup a customised drop down menu instead. As Customising Sch Editor dialog is still open we can right click on the Wiring toolbar and click Insert Drop Down. You can now name and/or assign an image for the representation on the toolbar. Call it 'Wires' and click OK. Now drag your custom wire commands from the Customising Sch Editor dialog directly on to Wires drop down.

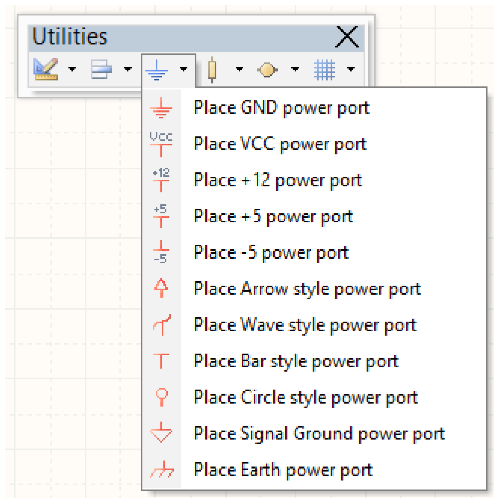


The setup of the drop down menu is complete. We can now select and place wires according to our chosen design style guidelines quickly and easily.

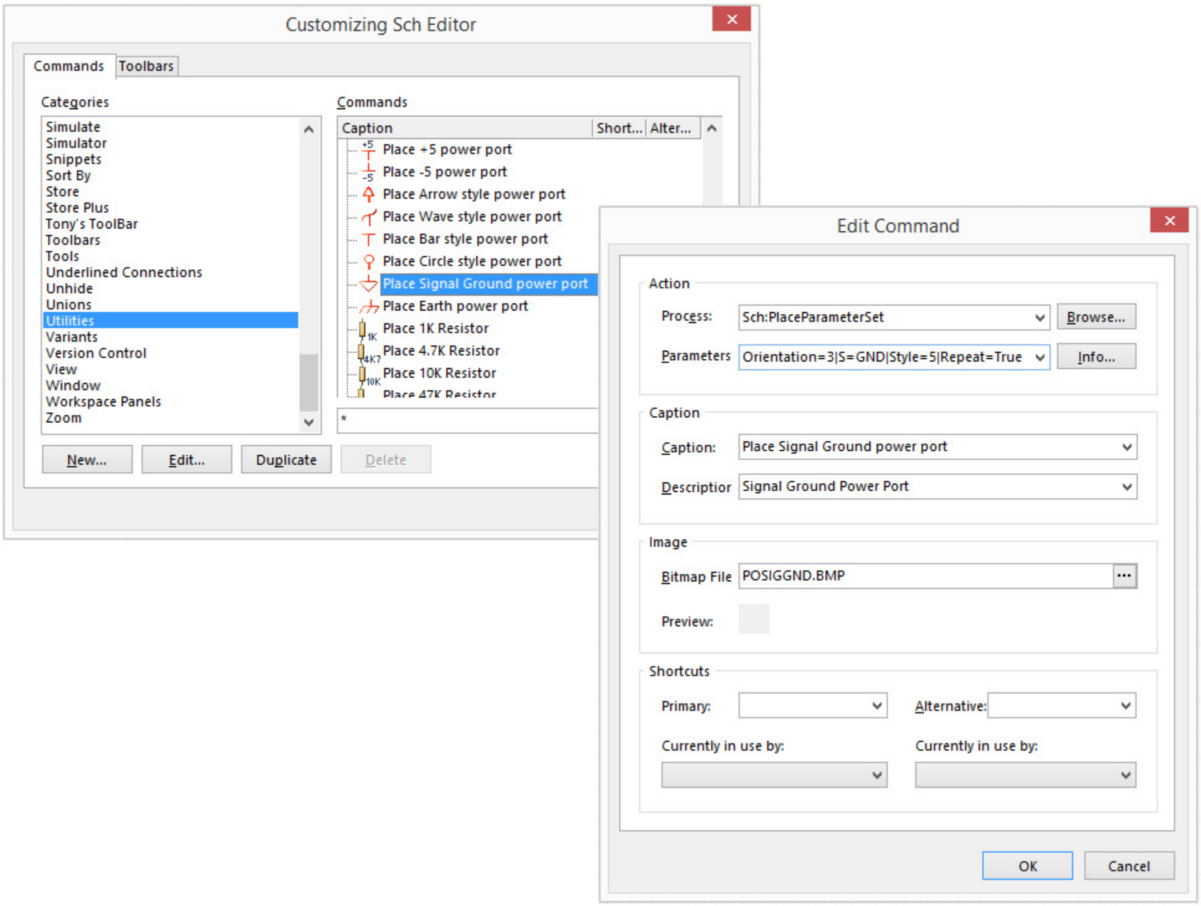
It would make sense to colour code ratsnests in PCB to match the schematic wire colour. Currently, there is no automatic facility in Altium Designer to do this, however it should not be too much of a burden to set these colours manually.

"Standard" power objects

Users often find themselves having to create items such as a Power Ports to find that they need to place many more. What users tend to do in this situation is to copy the last one they placed. Then paste this down multiple times, changing it along the way. We can further exploit the customisation of toolbars to help in this situation. The ideal scenario is that the user can choose from a variety of predefined items. The Utilities Toolbar is good for this. By default, an array of pre-defined power ports, discrete components and simulation sources are available.



All of these can be duplicated and modified to better suit the organisation needs. Perhaps your organisation follows the IEC 60617 standard for graphical symbols. If so, the current toolbar setup does not match the standard. For example, the symbol used to represent the Earth power port should use the same symbol used to represent the GND power port. The current representation actually represents a Frame or Chassis according to the standard. To make changes, enter the customisation mode by right clicking in the toolbar area. Then choose the Customize option. You can now access the Edit Command dialog for anything on the toolbar by double clicking on it. Alternatively, you can use the menu system and select DXP » Customize » Commands » Utilities.

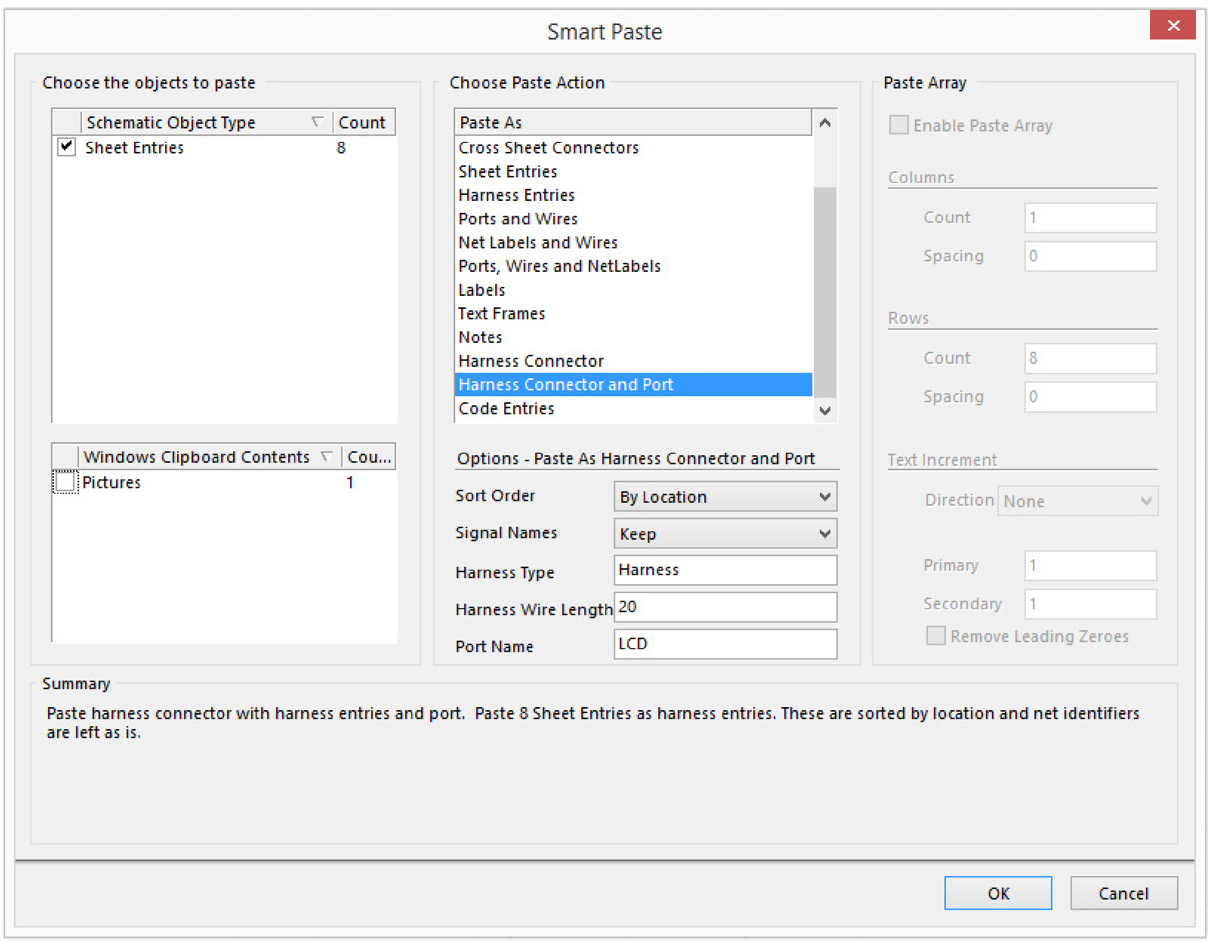


Toolbar information is stored along with the Altium Designer preferences. To make toolbars and other customisations available to multiple users, simply Save your preferences to a file (button at the bottom of the DXP » Preferences dialog box) and re-load on the other user's PC.

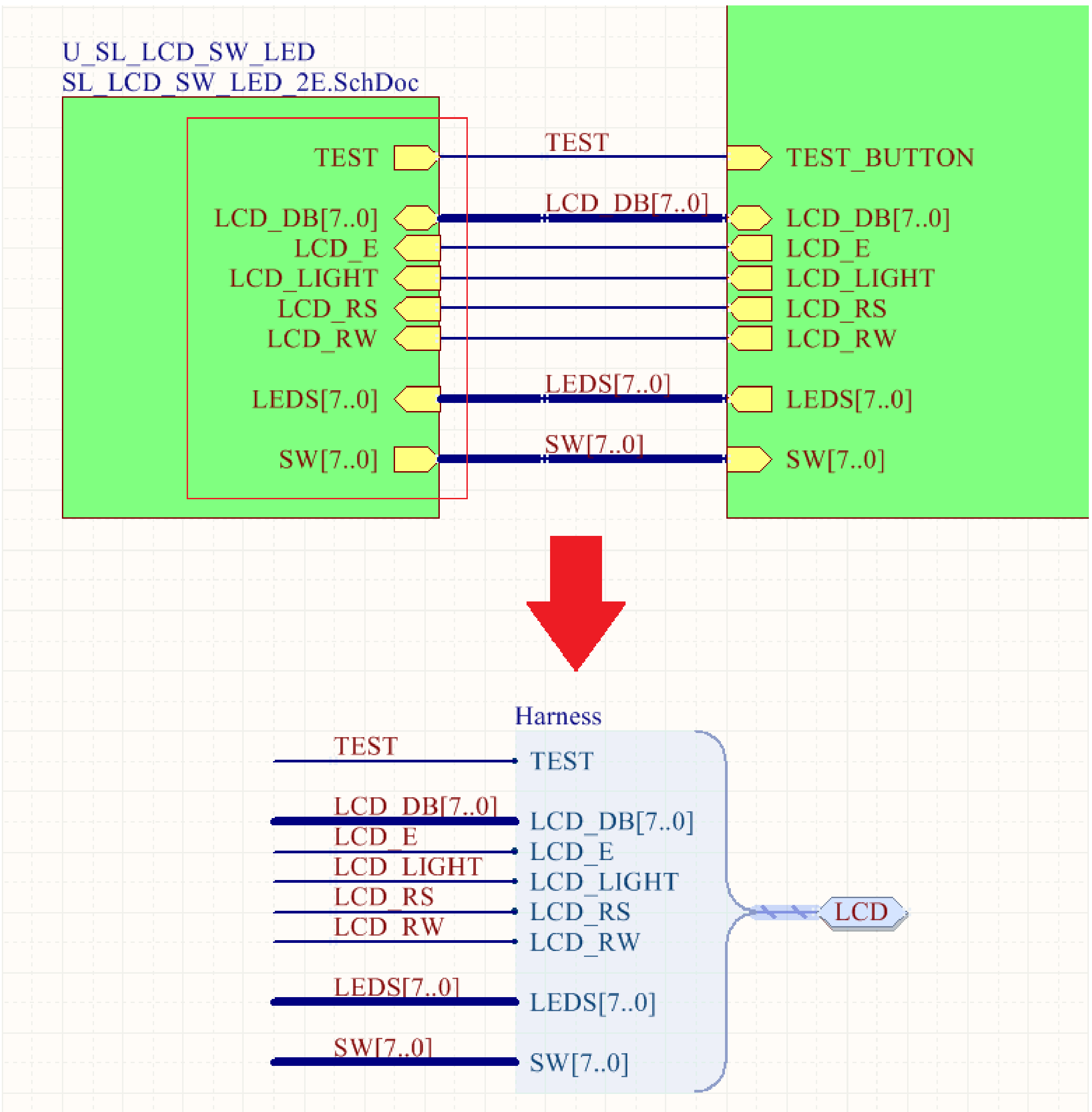
Cut, Copy & Paste techniques

Smart Paste is command that takes the familiar copy-and-paste concept to an entirely new level. It has the ability to transform objects from one type to another. This is very useful in goal of turning an existing schematic sheet into a reusable sheet. For example, we can copy existing Buses or Sheet Entries and convert them into Signal Harnesses.

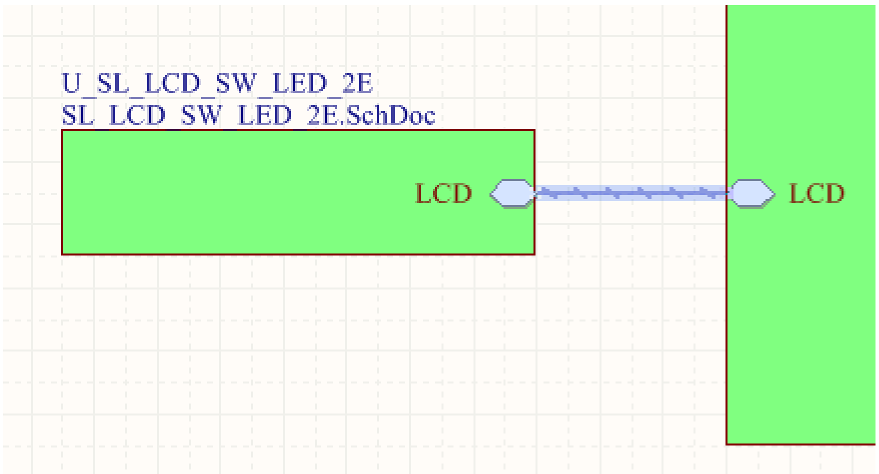
Signal Harnesses extend on Bus and Wire connectivity by allowing you to assemble logical groupings of any signals. This greatly simplifies the wiring traffic, enhancing readability, and streamlines the structure of your Schematic design. The use of Signal Harnesses should be used where possible, since the standard Harness itself is a reusable design element.



In the example below the Sheet Entries (highlighted with the red square) were copied. Then the Edit >> Smart Paste tool was opened. The result was completed in two pastes. The first paste took the Sheet Entry information and pasted it down as a Harness Connector with Harness Entries and a Port. The second paste generated the Net Labels, Buses and Wires.



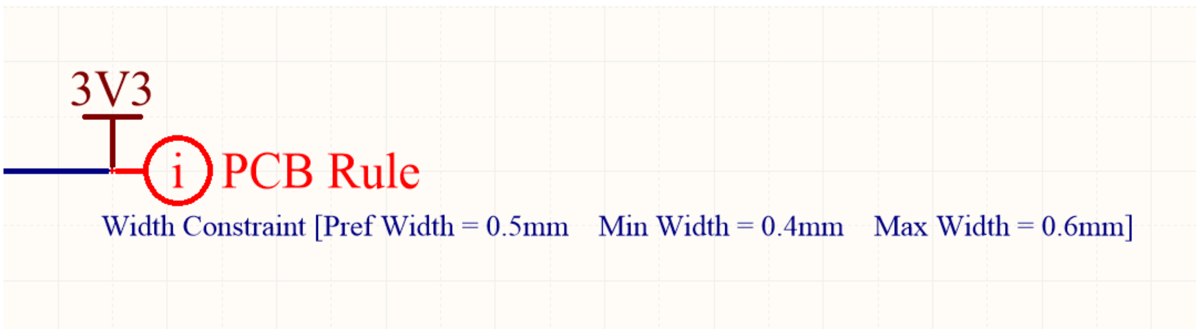
This can be cut and pasted on both child sheets. The existing Ports are to be deleted in the process. We have now greatly simplified the readability of the design and taken our first steps on the road to making that sheet reusable.



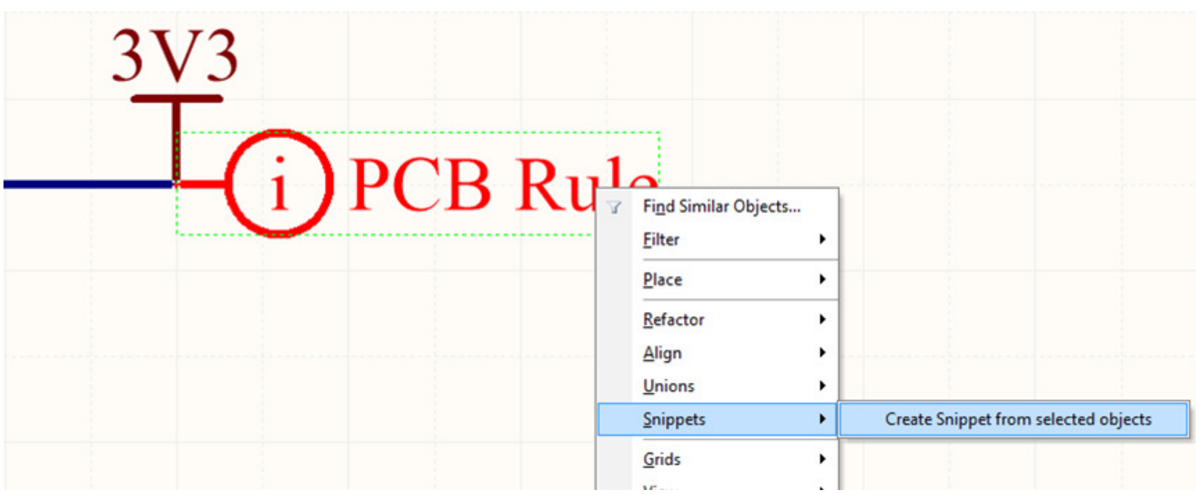
Using Snippets to streamline rule creation

Other items users will commonly copy and paste are Directives.

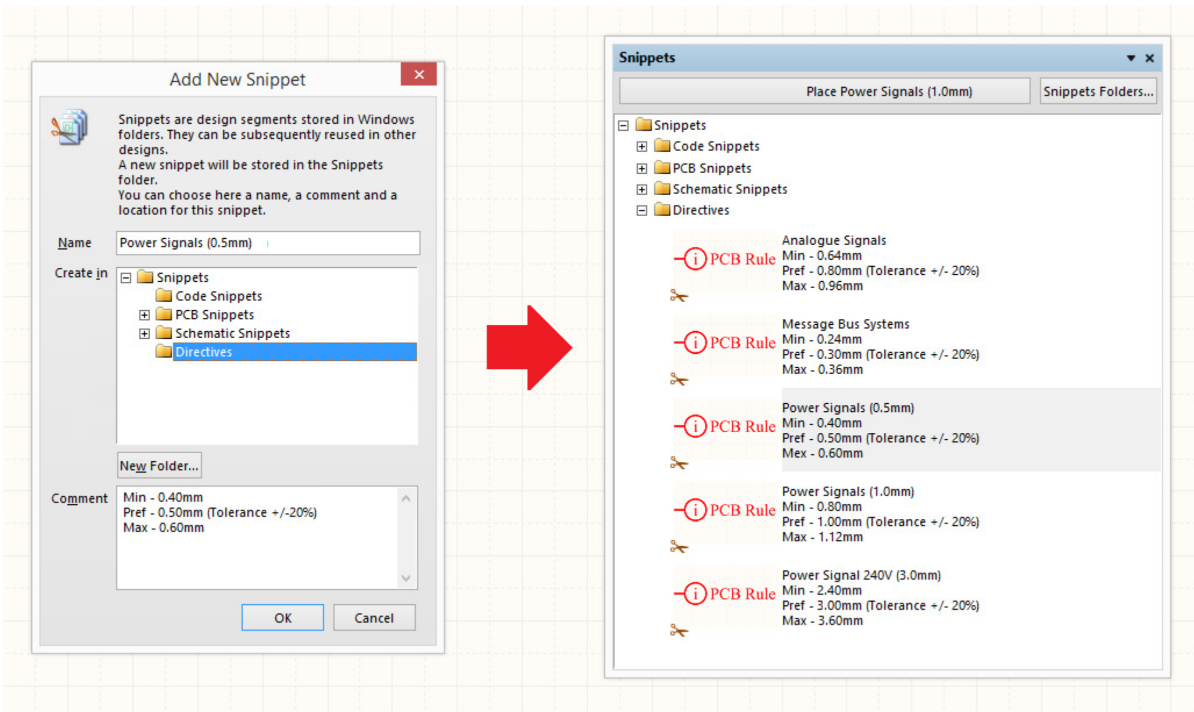
A Directive is an object placed solely on the Schematic that instructs other parts of the software. As the image below demonstrates, it is possible to instruct the PCB to create a design rule for a specific net. This is done by placing a Parameter Set Directive on a wire belonging to that net. As well as wires, Directives can also be placed on buses, harness and blankets to create design rules that will target a class of nets.



To avoid having to copy and paste Directives, they can be stored as Design Snippets. Think of them like an advanced clipboard feature which allows us save and catalogue common design data and easily reuse it in other designs. To create a Snippet of a Directive, simply select it, right click and choose Snippets » Create Snippet from selected objects.



The software will prompt the user to name, choose a location for and write a comment for the Snippet. The Snippet will then become available for placement from the Snippets panel. For your reference, you can find the Snippets panel under View » Workspace Panels » System » Snippets.



Snippets are stored on the system as standard design files. This makes it easy to share Snippets amongst a pool of users. For example, you can store them in a centralised network location. All user machines should then be setup to point to this location. This way, when a new Snippet is added, it will instantly be available to all users.

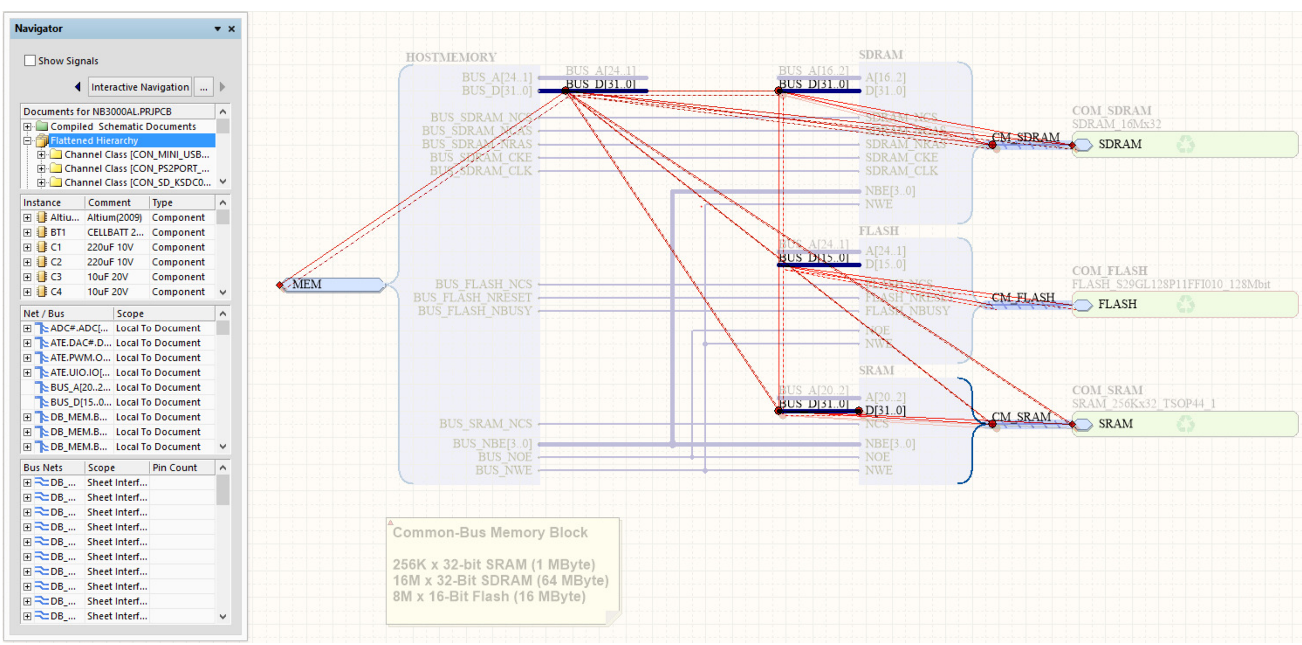
Ensuring robust inter-sheet connectivity

When breaking up Schematics into smaller functional blocks there are some overriding principles that should be adhered to. The key ones are that each reusable block should feature a single or small group of key components focusing on a specific function. Also, all possible supporting circuitry should be included in the sheet. In order to do this we need to be able to traverse though the logical representation of our Schematic project easily. A vital tool for this is the Navigator panel. It's an extremely useful and powerful way of locating objects and nets across multi-sheet schematics in the active project.

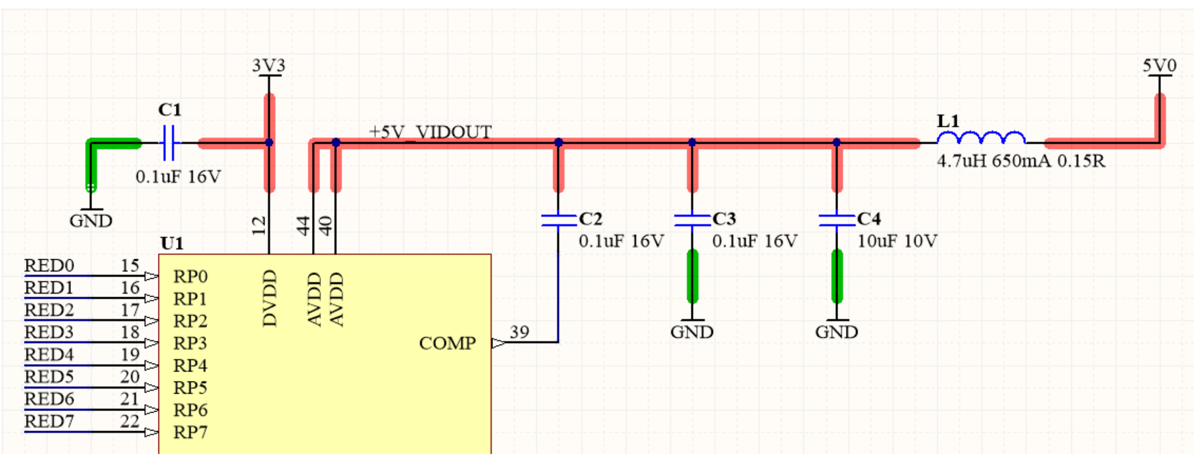
The Navigator panel is docked with the Projects panel by default. If it is not available you can find it under View » Workspace Panels » Design Compiler » Navigator.

In the Documents box you will see listed the Complied Schematic Sheets, the Flattened Hierarchy (when selected lists all objects within the project) and finally the logical view of your project. As well as the Mask, Zoom, Select options in the System - Navigator preferences, we also have an option called Connective Graph. The perfect tool for highlighting connectivity in your design. When the Connective Graph option is enabled and you are browsing net objects on a schematic sheet, a solid red line in the design editor window indicates a physical connection between net objects. A dotted red line indicates a logical connection between objects in the net.

Hold the Alt key as you click on an entry in the panel to also highlight that object in the PCB.



While navigating you can use the Highlighter tool to mark the important nets that you are looking to separate. This is a separate tool to the Navigator panel. In the lower right hand corner of the screen, there's an icon of a marker tip with two coloured boxes, click this, and then select a wire on your Schematic. It's a visual tool, it does not alter the net in any way.
Other options are Shift click to highlight more than one net with the same colour. You can press the Spacebar to change the highlight colour (a handful of pre-set choices). Finally, press Ctrl+Shift+C to clear the highlighting.



Putting it all together

So hopefully we've stimulated some thought and given you some ideas on how to set up Altium Designer to better suit your need for consistency. The next article builds upon these concepts and will show how to implement some of the many Design Re-use capabilities in Altium Designer.