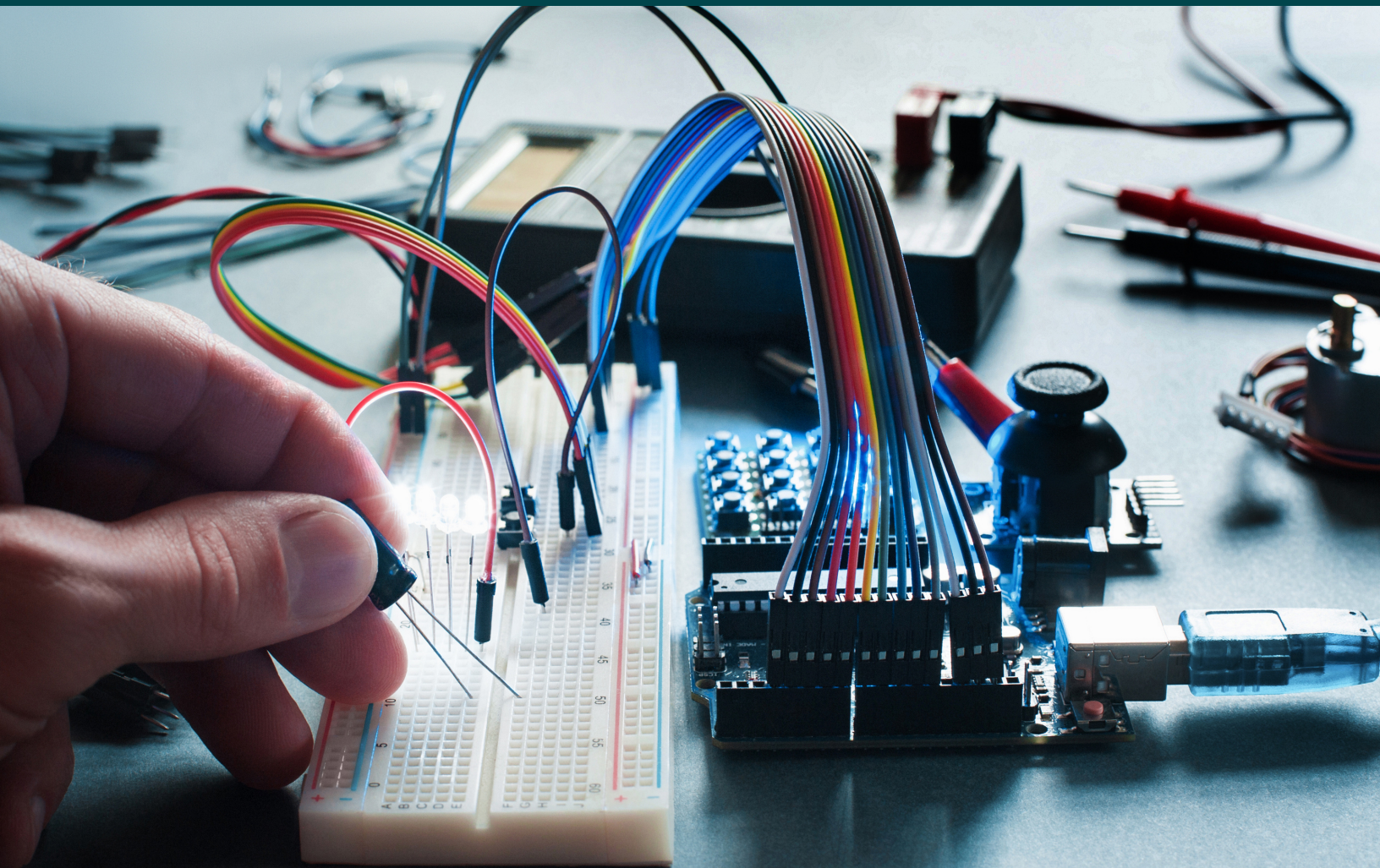


CAD*Enhance*

6 ALLEGRO HDL HACKS TO DOUBLE YOUR PRODUCTIVITY



CONTENTS

Introduction	03
Hack #1: Stretch Your Limits with the Command Console	04
Hack #2: Refreshing Logic Symbols and/or Block Symbols Without Reopening Design	07
Hack #3: Update part_table Entries Without Reopening the Design	12
Hack #4: Use your Middle Mouse Button to Change Part Connectivity during a move	14
Hack #5: Select a Group with your Middle mouse-button to improve control	15
Hack #6: Use Your Text Editor to Change Wire Names or Property Values	16

INTRODUCTION

Let's be honest, time is money. And your job as a hardware designer means you have to spend extra time to focus on menial, repetitive tasks to ensure the design is completed error-free.

As a seasoned Hardware Designer who has drawn hundreds of schematics using Cadence Allegro Tools, I'm always looking for shortcuts to help me deliver a solid design that I can stand behind.

The Cadence® Allegro Design Authoring™ Schematic Capture Tool was formerly known as Design Entry HDL, formerly known as Allegro HDL, formerly known as Concept HDL or if you are as old as me, just plain old 'Concept'. Whatever name it goes by, it is quite a very powerful Tool, but it can be intimidating to first time users. It's deceptive in its simplicity, and there are many hidden features even a grizzly old veteran can learn if they take the time to look for new tricks.

Rather than sift through hundreds of pages of documentation, I am providing you the Cliffs' Notes™ version of six useful shortcuts (aka "Hacks") enabling you to save countless hours in your design cycles. My goal is to show you tricks to produce quality designs in a fraction of the time it would take without them. Some of these Hacks you may already be familiar with, while others you may not know about.

Add these time-saving tips to your toolbox and you'll become a more proficient schematic designer while adding precious hours back into your day.

By the way, if you have a hack worth sharing, *send it to boleary@cadenhance.com and I'll send you a small reward for your efforts.*



HACK #1

STRETCH YOUR LIMITS WITH THE COMMAND CONSOLE

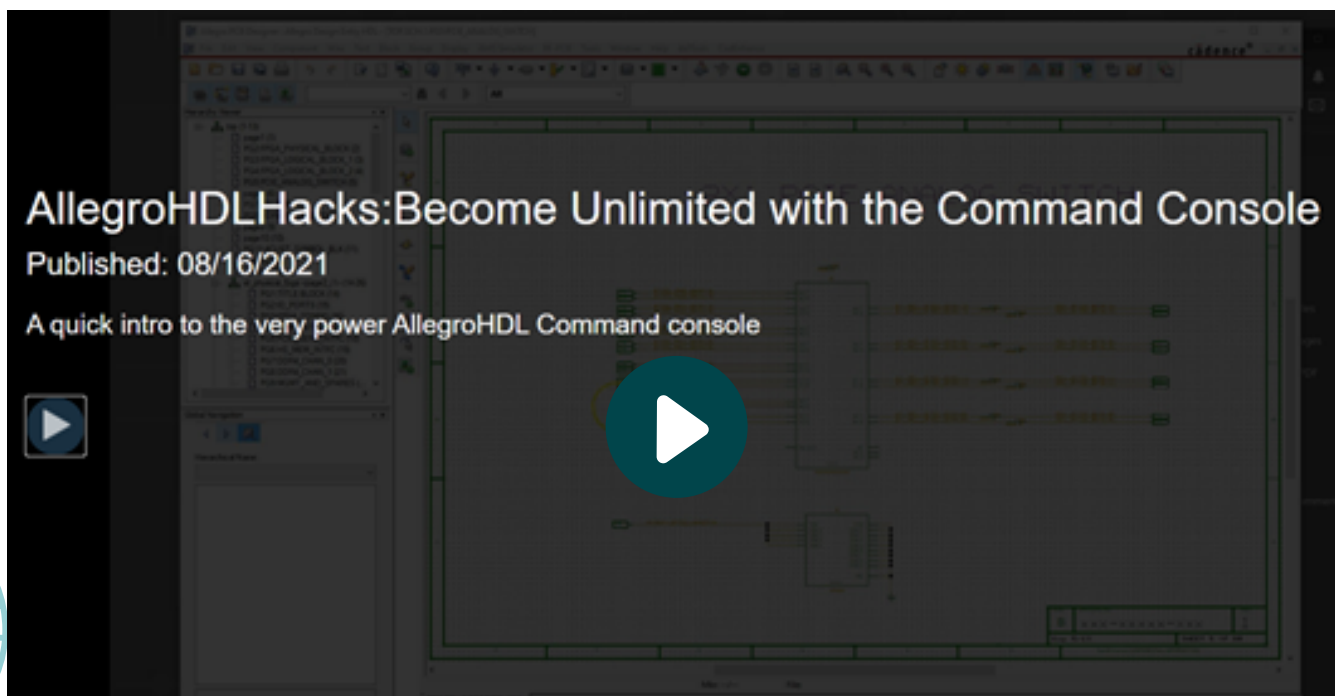
The Allegro HDL provides a console (which must be enabled) to let you quickly interact with your schematic pages. The console commands are rich, and powerful, and once you get used to them, they become much easier to type than to find the corresponding buttons in the GUI.

FREE DOWNLOAD

Download a copy of the command reference from our website at

[cadence_conref-Cmds_17.2.pdf](#)

GET STARTED WITH THE ALLEGRO HDL CONSOLE





It's worth mentioning that most commands have three or four letter shortcuts. Many of them have modifiers that make the command way more powerful than what is available in the button you normally use in the GUI.

If you haven't already, be sure to learn to use commands like:

- **EDit**, and **GET** - *open or reload design pages and blocks*
- **SELEct**, **INCLude**, **EXCLude**, **FINd** - *select and group bodies, wires and properties*
- **MOVE**, **DELEte**, **CHANGe**, **MODIfy**, **REPLace**, **VERsion** - *to work with the selections*
- **ADD** - *add part or component*
- **BADd** - *add a hierarchical block*
- **WIRe** - *add a wire with an optional name*
- **DISPlay** - *changes the appearance of an object or property on the page*
- **COPy** - *an extremely powerful command, especially with the extra count option which lets you copy items with a fixed offset between them*
- **PROPerty** - *quickly add properties*
- **ATTribute** - *shows all of the parts' properties*
- **SET** - *gives you quick access to the Tools/Options Menu*
- **SHOW** - *(deserves more explanation, keep reading...)*

The “**SHOw**” command lets you query many different pieces of information. Here are a few of my favorites:

- **Attach** - *shows where properties like sigName are attached to wires or pins*
- **Body** - *pops up a window showing the library name of the selected component and you can copy the text from the window*
- **Coord** - *shows the actual coordinate of where you click in the console window*
- **Modified** - *shows a list of all drawings that have been modified and not saved*
- **Origin** - *shows the origin of all objects on the current drawing*
- **Size** - *shows the size of the text which you can use to apply to another piece of text with the **TEXTSize** command*





- **VPAdd, VPDel** - add and delete new windows to the view so you can cut and paste between two schematic pages
- **WRite HIEr_write** - Save and Save Hierarchy from the block you are in and below


While looking up these commands, I came across these two **AUTO** sub-commands that I never knew about:

- **AUTO** netprop <group_name> property_name=property_value
- **AUTO** prop <group_name> property_name=property_value will add the same property to every symbol in the specified group. So if you wanted to add BOM_IGNORE to a selected set of symbols in group A use:

```
AUTO prop A BOM_IGNORE=TRUE
```

Earlier, we referenced [this pdf which provides the details of ALL the console commands](#). It details some very powerful modifiers that can be added to the commands. This allows you to do much more with one command than you can do with any of the menu buttons.

*I think most designers who are familiar with using the console would have this one dream:
...If only Allegro HDL's console had a way to recall command history!
As it stands, you can get around pretty quickly by cut and paste commands within the console.*



*Have you looked at the [CadEnhance CE-HDL Plug-in for Allegro Hdl](#)?
Our CE-HDL Tool makes extensive use of these console commands to automate the painful tasks of connecting FPGAs and other large parts in a fraction of the time it takes to do it manually. CE-HDL also provides utilities to clean up your schematic to provide a professional looking product.*

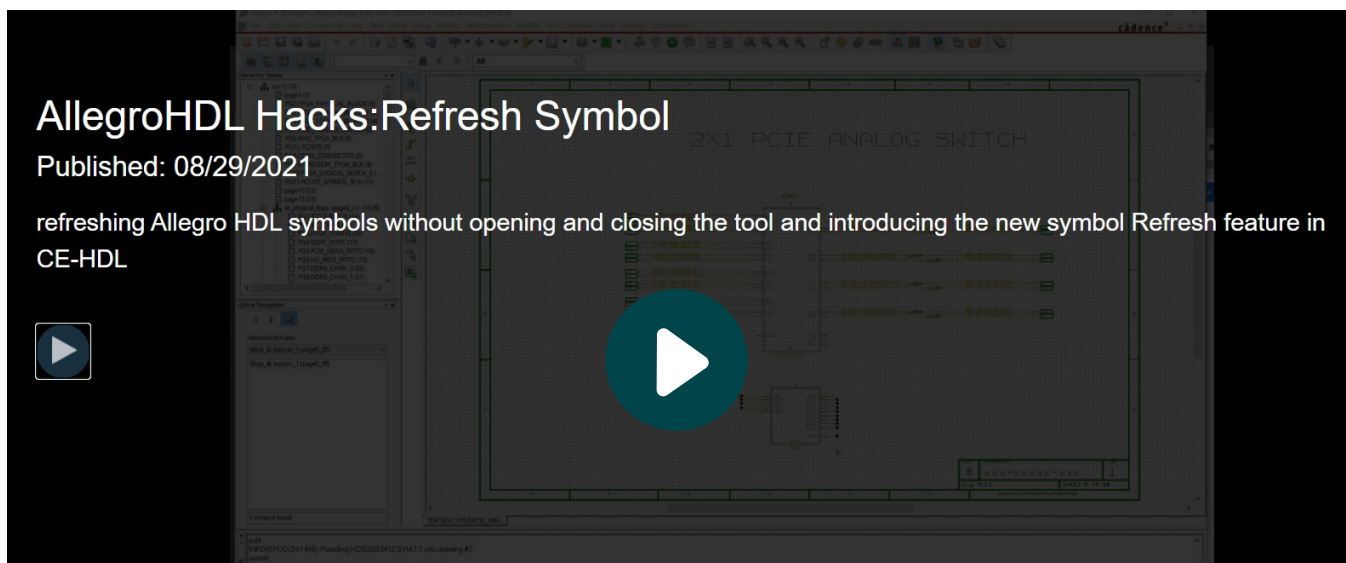


HACK #2

REFRESHING LOGIC SYMBOLS AND/OR HBLOCK SYMBOLS WITHOUT REOPENING DESIGN

When logic symbols for a component get changed inside or outside of Allegro HDL, Users usually end up closing and re-opening the tool to get the new symbol to refresh in the schematic. Reopening the design does work to refresh the most common symbol changes, but it does not fully update the symbols in cases where pin properties changed for the symbols or the number of symbols for the component actually changes. Reopening is also time consuming and forces the user to save their work at a point where they may not want to.

Here we present a faster and more reliable way to refresh a symbol after the library part has been updated.



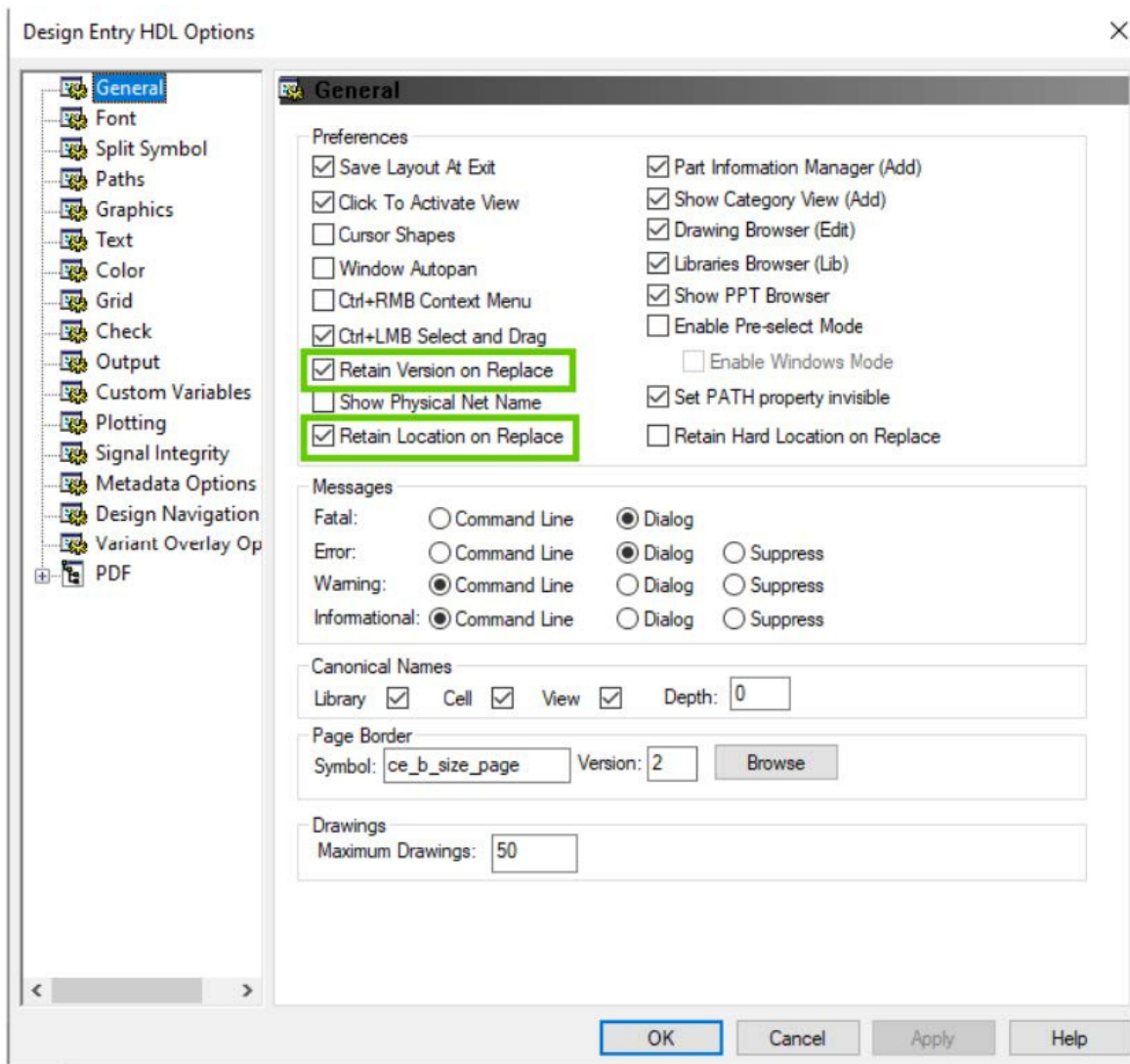
Let's say you updated the LTC2975C symbol in your ce_ic lib
\$ALLEGRO_LIB_ROOT/ce_lib/concept/ic



In the command window type:

- **lib ce_ic** - tells AHDL to reload the ce_ic library from disk
- **symread LTC2975C** - this is the magic that reloads the symbol into AllegroHDL Memory
- **repl LTC2975C** - and then click on the symbol you want to update

If it was a multi-symbol part, you would need to run the '**REPL**ace' command for each version. It's always a good idea to enable the 'Retain Location and Retain Version On Replace in the General Tab of the Options popup which you can get to by typing '**SET**' in the console.





If **Retain Version on Replace** is set, you can select multiple symbol versions on the same page using a **SELE**ct or **GRO**up or **FIND** command and then issue one replace command without specifying a version.

For instance, if there are 3 versions of the HDS3SS3412 part on the current page, you can type:

- FIND HDS3SS3412*
 - WARNING(SPCOCN-1200): Using group B
 - INFO(SPCOCN-1195): Group "B" contains: 3 bodies 0 properties 0 notes 0 wires 0 dots 0 images
 - Version: 1

In this case Group B now contains the 3 symbols. If you want to replace the part in a group, you need to enter the replace command first, and then enter the group name separately

- REPLace HDS3SS3412
- B
 - INFO(SPCOCN-1680): Replaced 3 bodies.

If **Retain Version on Replace** is not set, you would have to issue the

- **REPLace HDS3SS3412..1** and click on version 1
- **REPLace HDS3SS3412..2** and click on version 2
- **REPLace HDS3SS3412..3** and click on version 3

When a large part is fractured into many symbols and placed across multiple pages, the Refresh Symbol Process can become a tedious, error-prone task. You need to manually edit each page and issue the **FIND/REPL**ace commands, and its very easy to miss a symbol along the way.

The next version of the CadEnhance CE-HDL tool has a button you can use to handle the whole refresh process for single and multi-symbol parts with one click (a massive time saver!).

[Check Out Our Video](#)

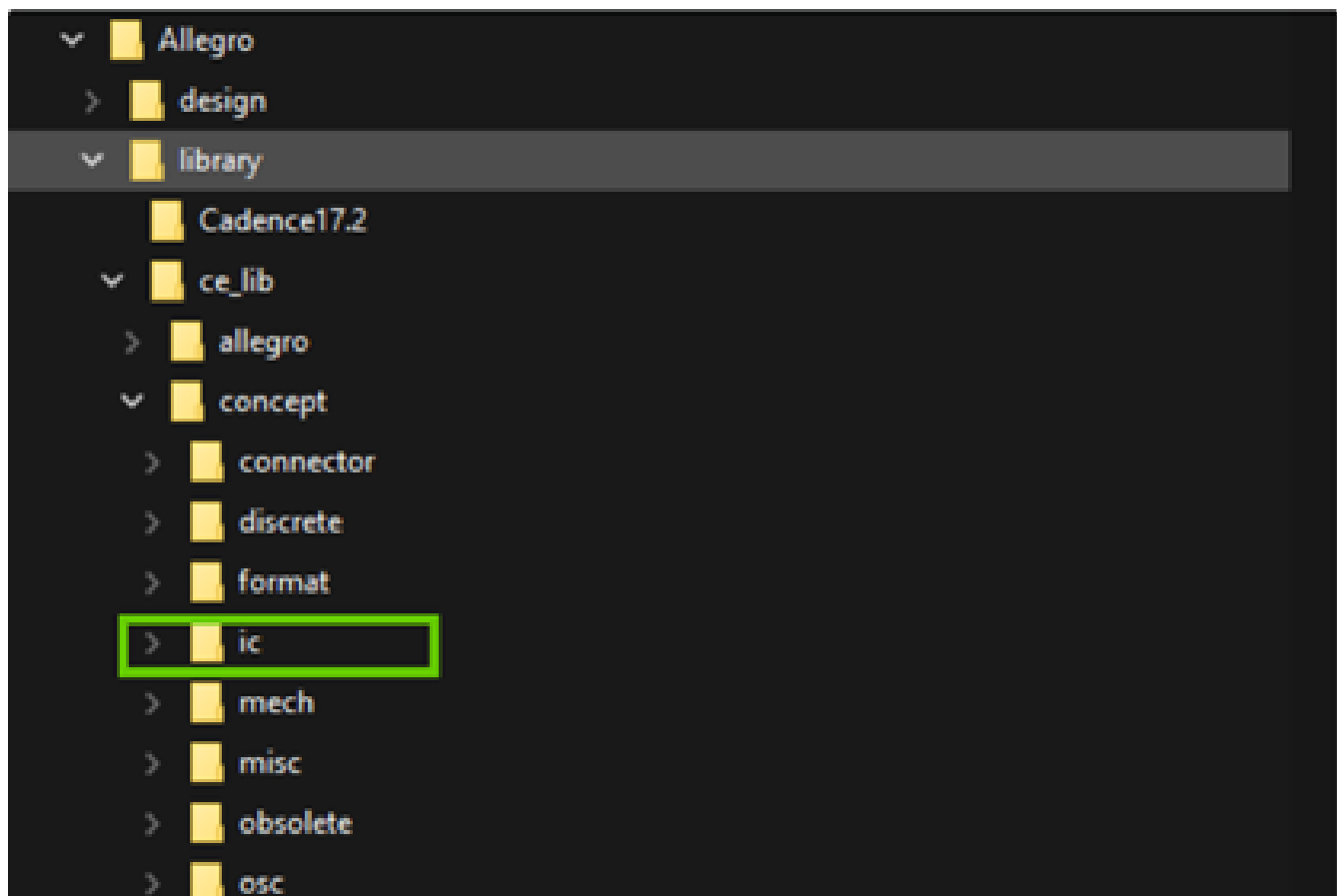
If you'd like to be the first to take advantage of this new feature, email me at boleary@cadenhance.com and I will let you know as soon as it becomes available.



BACKGROUND ON SYMBOL LIBRARIES

Symbols are stored in directories that Allegro calls “libraries”. The top_level schematic of a design is a library and a hierarchical block is also a library.

- The cds.lib file in your design directory tells Allegro HDL where to find your symbol and schematic libraries and maps them to a unique name (and a cds.lib file can include several other cds.lib files to create a bigger library)
- If you update a symbol in any library, you need to know the name of the library the symbol is stored in, for instance:
 - if you keep all of your integrated circuits in a directory (c:\allegro\library\ce_lib\concept\ic)





- You could have a cds.lib file in a setup directory that looks like this:
 - `$ cat $ALLEGRO_LIB_ROOT/ce_lib/setup/cds.lib`
 - `DEFINE ce_discrete $ALLEGRO_LIB_ROOT/ce_lib/concept/discrete`
 - `DEFINE ce_format $ALLEGRO_LIB_ROOT/ce_lib/concept/format`
 - `DEFINE ce_mech $ALLEGRO_LIB_ROOT/ce_lib/concept/mech`
 - `DEFINE ce_misc $ALLEGRO_LIB_ROOT/ce_lib/concept/misc`
 - `DEFINE ce_connector $ALLEGRO_LIB_ROOT/ce_lib/concept/connector`
 - **`DEFINE ce_ic $ALLEGRO_LIB_ROOT/ce_lib/concept/ic`**
 - `DEFINE ce_osc $ALLEGRO_LIB_ROOT/ce_lib/concept/osc`
 - `DEFINE standard $CONCEPT_INST_DIR/share/library/standard`
 - `INCLUDE $ALLEGRO_LIB_ROOT/ce_lib/setup/customer_cds.lib`

The cds.lib file in your schematic design directory can **INCLUDE** the cds.lib for this:

- `$ cat ./design/CE_SCHEMATICS/17_2/FOX_SGLD/REV-02X/027_REV_1/cds.lib`
- `DEFINE top_lib worklib`
- `DEFINE userlib $ALLEGRO_LIB_ROOT/userlib/concept`
- **`INCLUDE $ALLEGRO_LIB_ROOT/ce_lib/setup/cds.lib`**

When the design is opened, Allegro HDL reads the local cds.lib, it sees the INCLUDE so it goes and loads that cds.lib file (note that ALLEGRO_LIB_ROOT is an environment variable that points to the place where the Allegro/library directory lives on your computer (or on a shared drive)).

- **`DEFINE ce_ic $ALLEGRO_LIB_ROOT/ce_lib/concept/ic`** - tells Allegro HDL to refer to `c:\allegro\library\ce_lib\concept\ic` as `ce_ic`





HACK #3

UPDATE PART_TABLE ENTRIES WITHOUT REOPENING THE DESIGN

Entries for a select part in the part_table files may also be updated while the design is open.

These entries can be changed in the local .ptf file in the part_name/part_table directory) or in a global part_table file that encompasses all parts of a class like resistors.ptf.

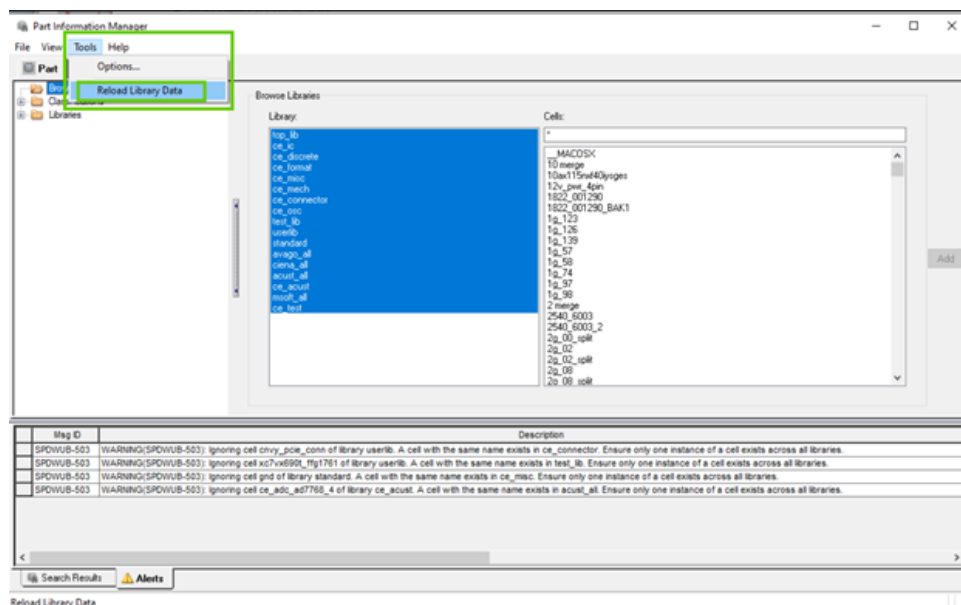
This sequence allows the user to update the new entries into Allegro HDL without needing to reopen the design.

There's no single command I know of to do this. If anyone does know of one please share.

You'll need to type:

- **ADD**

Which is the Add command with no arguments. That will pop up the interactive Part Information Manager where you can select Tools/Reload Library Data.



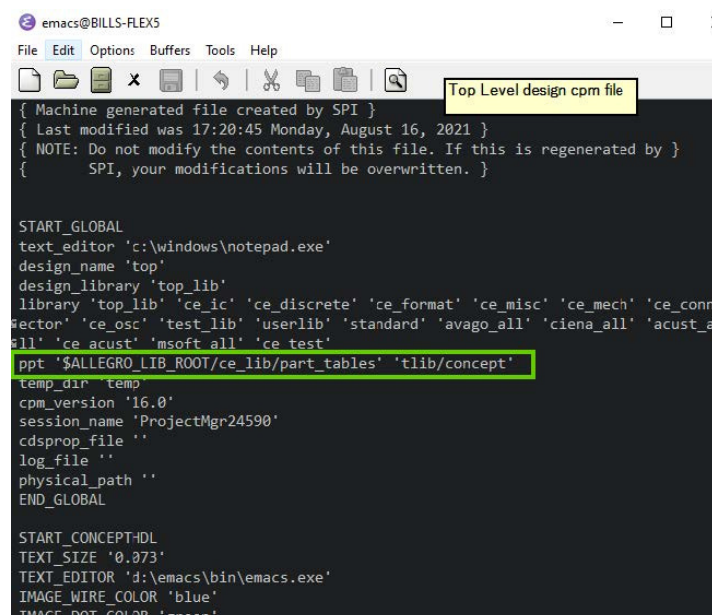
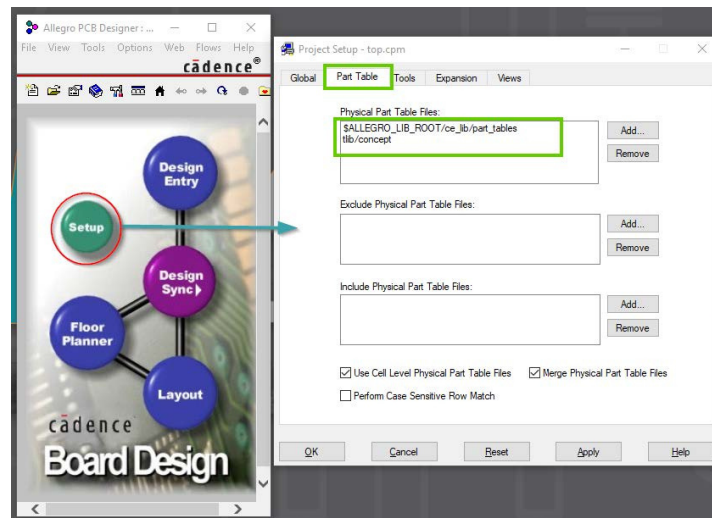
And that will refresh all the part_table files that Allegro HDL is tracking.



Note that Allegro HDL uses the 'ppt' setting in the top_level design.cpm file in your working directory to specify where to look for the ppt files.

There are 2 ways to change this setting, And if you do change it to add or remove a new Part Table File, **you will need to restart Allegro HDL.**

- Use the Setup Button in the Proj manager, open the Part Table tab and add paths to the Physical Part Table Files Section.
- Edit the ppt setting directly in the .cpm file with a text Editor.

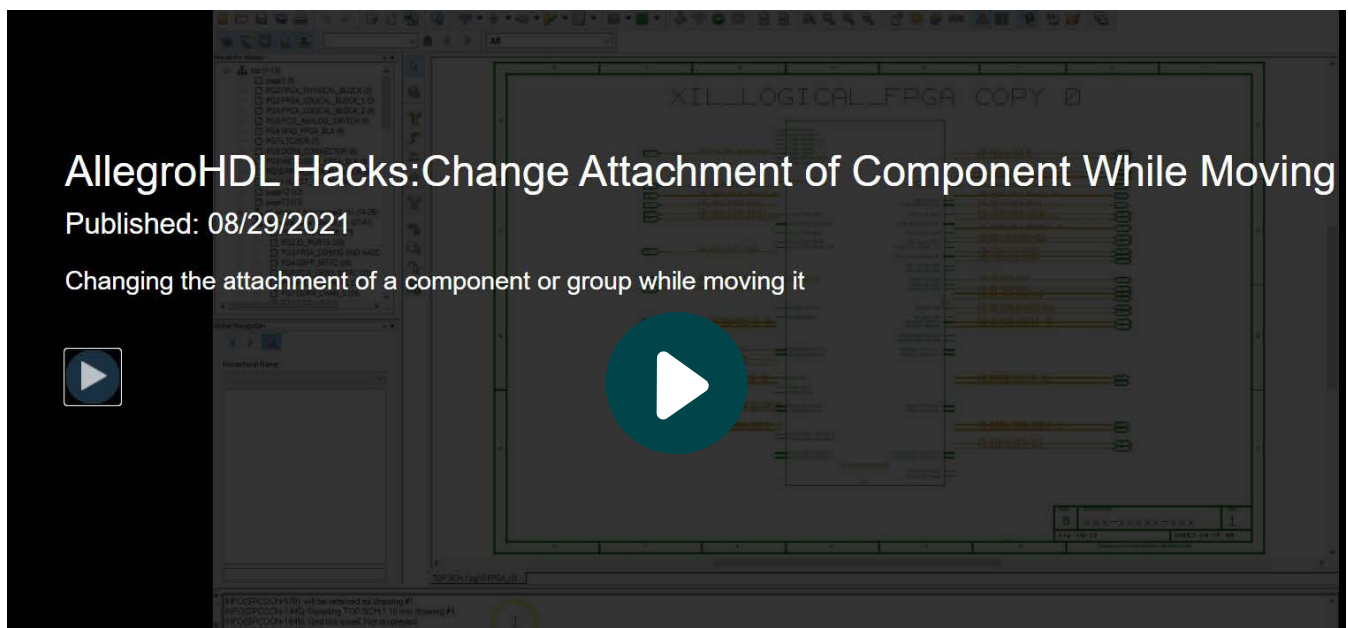




HACK #4

USE YOUR MIDDLE MOUSE BUTTON TO CHANGE PART CONNECTIVITY DURING A MOVE

- Did you ever create a messy collection of shorted signals when you tried to move a component around on your schematic page? The middle mouse button comes to the rescue.



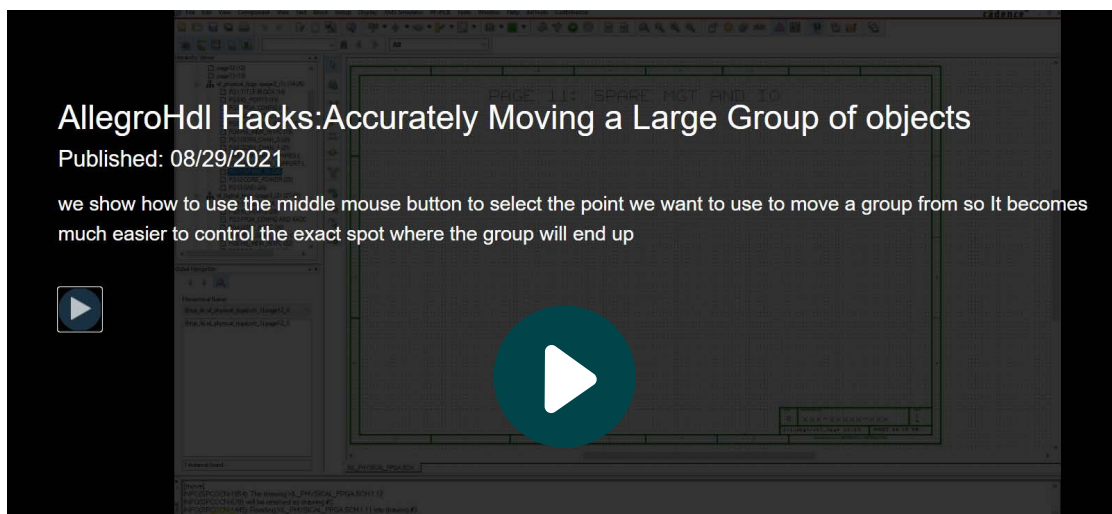
- When you select a part to move, it automatically starts moving the part and dragging all the wires that are attached to it.
- More often than not, you'll want to move the part out of the way and leave the wires where they are. This is easy to do if you click the middle mouse button to cycle through the attach types. You can get to the point where the symbol is disconnected from all the wires and then move it.



HACK #5

SELECT A GROUP WITH YOUR MIDDLE MOUSE-BUTTON TO IMPROVE CONTROL

You may find it cumbersome to try and move Large Groups of selected objects to a precise point on your schematic page.



If you just type

`MOVE {GROUP Letter}`

In the console, The group gets picked up in the center of the extents of the group.

When the group is large, it becomes difficult to move it to the exact place you want.

Instead, If you type:

MOVE

Without the Group Letter.

Then middle-click your mouse close to a pin or end of a wire in the selected group, Your group will get picked up and move relative to that point.

This makes it much easier to accurately locate a point in your group exactly where you want in the design window.

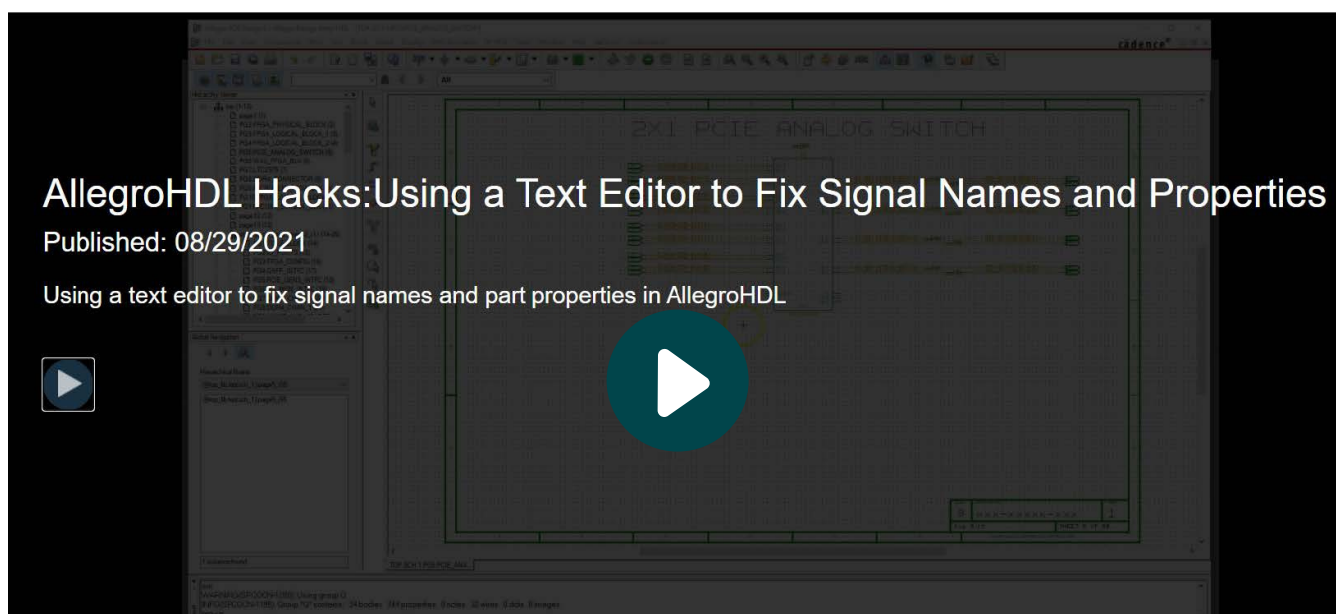
Real-world example: you can get the top wire in the group to end up right at the top pin of a part.



HACK #6

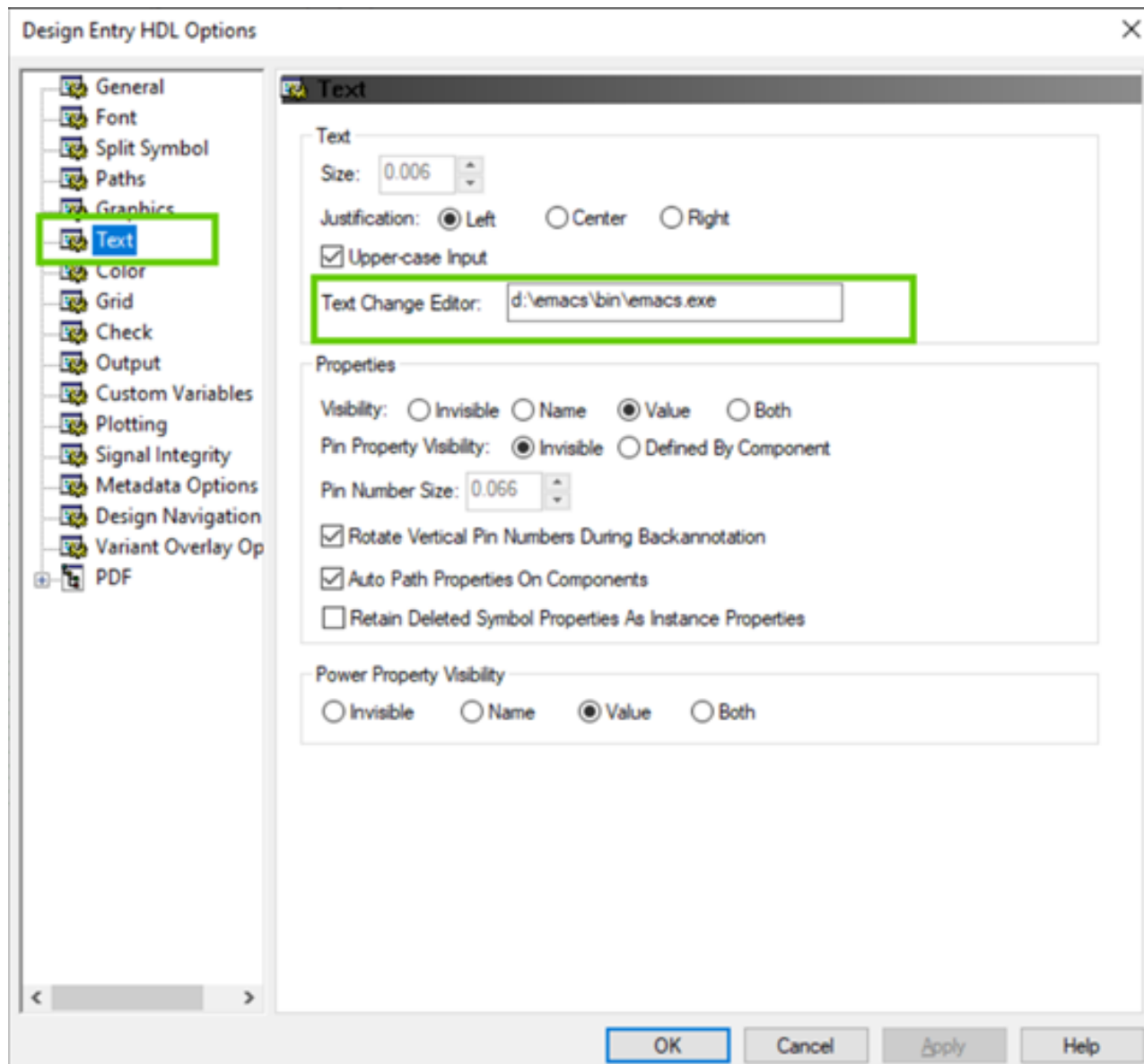
USE YOUR TEXT EDITOR TO CHANGE WIRE NAMES OR PROPERTY VALUES

Using your favorite Text Editor to edit your signal names and property values can save countless hours as you copy and paste elements in your design.



If you don't have your favorite text editor setup, type "set" in the command window and a configuration will pop up.





On the text tab look for the text editor selection.

There is no button to let you browse to select your Editor, so it's best to browse to find your editor in another explorer window, select the .exe file and then use Shift+Right-Click + copy path to copy the path of the file to the clipBoard, so you can copy it into the Text Change Editor block entry

I think it's unfortunate that Allegro HDL cannot use some environment Variable to choose your text editor. Instead, it stores your textEditor in the design.cpm file. If you change it, it changes for other users and they have to change it back to choose their editor.



Once your editor is setup, use **select** to interactively select some sigNames,

- Select A Properties
- Select Command Info from Cadence Docs
 - SElect [group_name | DEFault][type...]{selection...| group_name |ALL}
 - where type is:
 - BODies | PROPERTIES | NOTes | Wires | DOTs
 - and selection is:
 - {Lpoint Lpoint} | Ctrl+Rpoint | Mpoint

Or use the powerful **FIND** command,

- FIND *sig_name=*
- will find all sig_names
- it will put them into a group and tell you what group they selected
- or you can use set tell find what group to put them in
 - SET nextgroup A
 - FIND *sig_name=*

Once you have a group, use the **CHAnge** command,

- **CHAnge** A
- Type Ctrl-e to popup your Editor
 - use search/replace to make any changes you want in the editor
 - save the file and close the editor
 - the wire names will be updated in the design



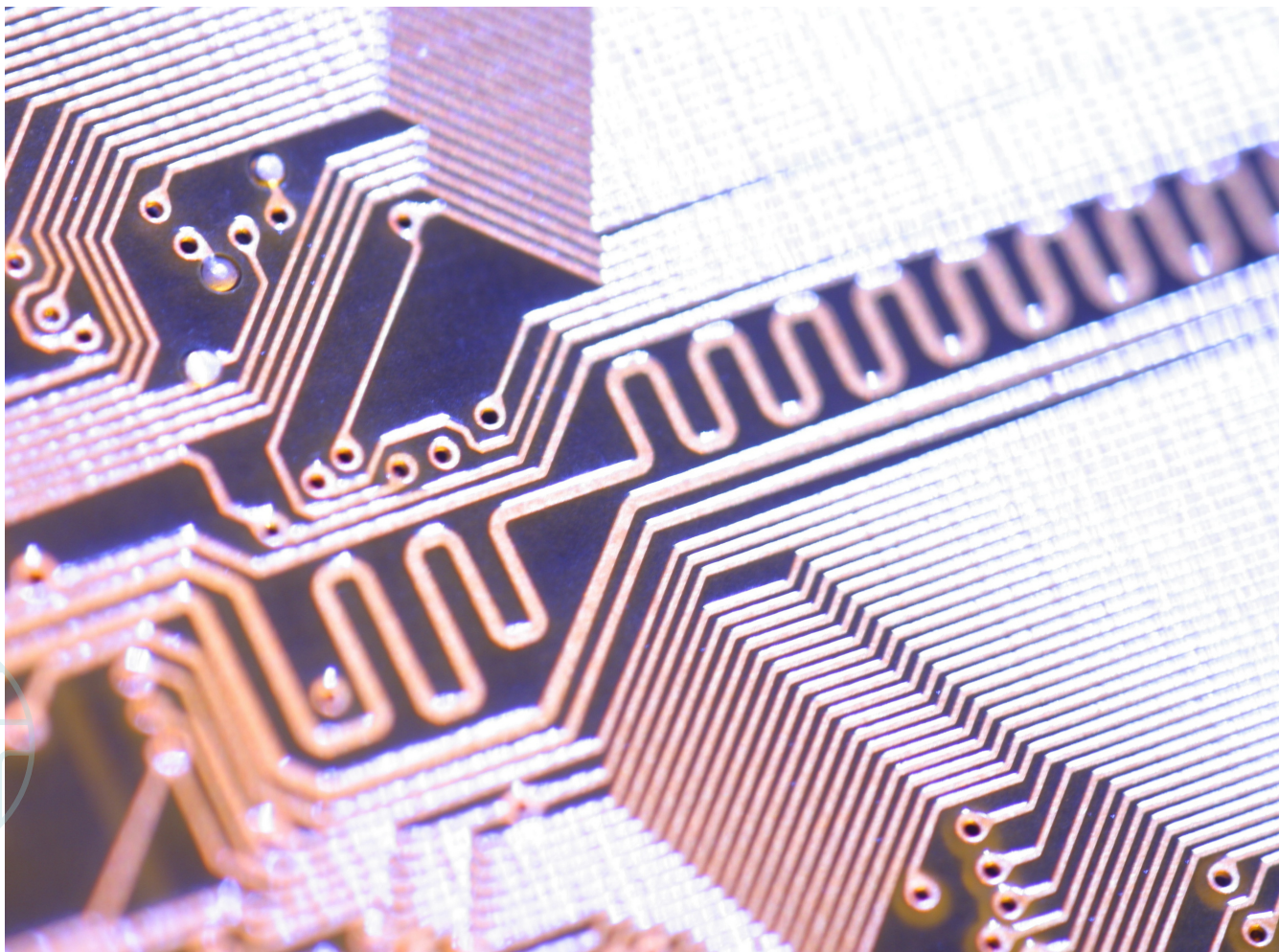


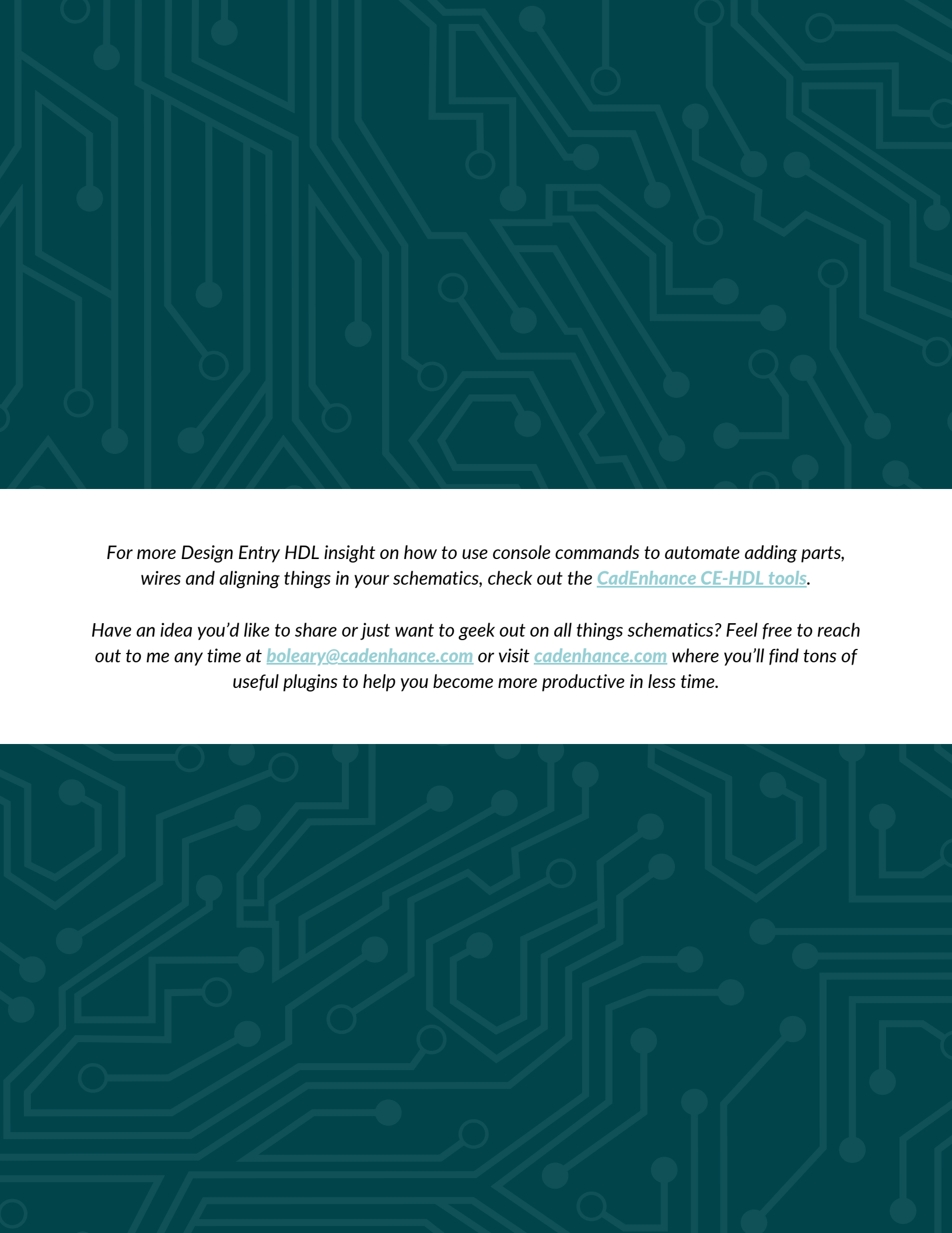
If your editor supports regular expressions, you can wield much more control in your replacements

- To reset all symbol Locations in a group use
 - FIND *LOCATION=*
 - CHA A
 - CTRL-e
 - In the editor use regular expression to
 - replace \$LOCATION=.* with \$LOCATION=?

Never Heard of regular expressions or are your regular expressions skills rusty?

Check out the [video](#) we provided above and for even more background try our quick intro guide at [CadEnhance Regular Expression Discussion](#).





For more Design Entry HDL insight on how to use console commands to automate adding parts, wires and aligning things in your schematics, check out the [CadEnhance CE-HDL tools](#).

Have an idea you'd like to share or just want to geek out on all things schematics? Feel free to reach out to me any time at boleary@cadenhance.com or visit cadenhance.com where you'll find tons of useful plugins to help you become more productive in less time.