

<b><i>Navigating</i></b>			
Shortcut	Does	Domain	Value
CTRL-x	Descends into selected instance	sch	****
CTRL-ALT-d	Descends into hierarchical instance given by the selected text (which is expected to be of the form xinst1/xinst2/xinst3 (will take you into schematic for xinst3). If no text selected, then place text corresponding to the most recent Return-to-top	sch	****
CTRL-i	Locates selected instance in table view of the hierarchy editor	sch	****
CTRL-ALT-s	If editing cellName_sim, then it will place an instance of cellName (if it exists) Also, locates cellName_sim in the Library Manager	sch	****
ALT-e	Return to top	sch	****
ALT-x	Descend back to depth from which last ALT-e was executed	sch	****
CTRL-ALT-t	Find the test-bench (if current library name is PROJ_something, it will look in PROJ_TB) for the selected instance (if it is "aCell" then it will look for aCell_tb	sch	**
ALT-SHIFT-s	locate the master of the selected instance in the IC Manage library manger	sch	***

<b><i>Configuring</i></b>			
Shortcut	Does	Domain	Value
CTRL-t	Loads the display.drf_thickLine display resources - to give cadetBlue wires and thick highlighting (which color scheme carries over to ViVa if plotting using CTRL-4	sch	****
ALT-Up/Dn Arrow	Double/Halve grid snap spacing	sch sym	****
CTRL-F10	Toggle grid	sch	****
CTRL-m	Toggle the split mode - whether segments are used to split imported devices in Layout XL	lay	****
CTRL-SHIFT-t	Toggle bindkey-setup - that is, go back to Cadence defaults if so desired	sch	**

<b><i>Launching/Quitting</i></b>			
Shortcut	Does	Domain	Value
ALT-SHIFT-h	Launches hierarchy editor (if you're viewing a schematic that's owned by a configuration view)	sch	***
ALT-c	If ADE-L is linked to the schematic window you're viewing, then launch the calculator	sch	**
CTRL-SHIFT-n	The selected instance's master schematic will be opened in a new window	sch	****
ALT-SHIFT-n	If the schematic is bound to an ADE-L session, the netlist will be generated and the CIW will have a note on how to use the netlist	sch	***
CTRL-ALT-q	Quit the ADE-L that this schematic is attached to	sch	**

<b><i>Editing</i></b>			
Shortcut	Does	Domain	Value
CTRL-ALT-c	Fix @Partname --> @cellName	sym	**
ALT-scroll-up/dn	Wire : increase/decrease thickness Label or instance : increase iteration number (Eg : I20 --> I20<1:0>)	sch	***
ALT-SHIFT-scroll up/dn	transpose bus : <1> --> <2> , <1:0> --> <2:1>	sch	**
CTRL-2	Made Editable	sch sym lay	****
CTRL-Up/Dn Arrow	Change (all) pin name font sizes	sch	**
ALT-SHIFT-q	Launch properties form with "All Selected"	sch	
CTRL-ALT-i	With two or more instances selected, will name the instances selected 2nd, 3rd, etc after the first. So, if the first was M1, the new names will be M1A, M1B, etc.. If it was M1<0>, the new names will be M1<0>, M1<1>, etc	sch	****
CTRL-ALT-r	With two resistors selected, the first being a PDK resistor and the 2nd being an analogLib/res, the resistor value from the first will be copied to the ideal one. Very useful for quickly building a sch_model that is correct by construction, to eliminate unnecessary node count	sch	***
CTRL-PageUp	With text instances selected and the first one being either a single letter (like 'A') or a single number (like '1'), then there rest will be made successors if the first (i.e., B,C,D.. Or 2,3,4..)	sch	**
CTRL-PageDown	Like CTRL-PageUp, but decrementing rather than incrementing	sch	***
CTRL-ALT-n	If a cell has been selected in the Library Manager, then place the selected view (if no view selected, place symbol view) in the current schematic window. Much nicer than using the Component browser in my O :)	sch	****

<b><i>Selecting</i></b>			
Shortcut	Does	Domain	Value
CTRL-m	Toggle between full enclosure and partial enclosure to select	sch	***
CTRL-p	Toggle between being able to select on pin and all visible objects associated with an instance (such as instance name label, etc)	sch	***

CTRL-ALT-h	select all of the same master	sch lay	**
------------	-------------------------------	---------	----

#### Displaying and Plotting

Shortcut	Does	Domain	Value
9	If wire already selected, highlight that, else, prompt user for a wire to highlight	sch lay	****
0	Discard all highlights	sch lay	***
8	Prompt user for a wire whose highlight to discard	sch lay	**
CTRL-t	Load the display.drf_thickLine and use those styles - thick highlighting	sch lay	****
CTRL-l ("el")	Load the display.drf_layout and use those styles (filled)	lay	
CTRL-SHIFT-b	Load the display.drf_blink and use blinking highlight style	sch lay	
CTRL-4	Enter the select outputs to plot (voltages and currents). After selecting, press Esc to plot. The sch window must be linked to an ADE-L session window	sch	****
ALT-e	Launch Print Preview	ViVA	
CTRL-B	Launch Print Preview (needs a cdsplotinit file)	sch	****
6	Display OP info	sch	
ALT-6	Display OP Node Voltages	sch	
CTRL-6	Display defaults for devices (W, L, etc..) (get rid of term net names!)	sch	***
1	Prompt user for a device whose OP info will be displayed in new viewer	sch	
3	Dismiss the viewer that is showing OP infor - IF IT EXISTS - else, one of your existing schematic windows will be nuked	sch	
F7	Restore wire color display (only) to default (doesn't edit the schematic)	sch	***
CTRL-F7	Restore wire color display (only) to stored-database values (doesn't edit the schematic)	sch	
7	Prompt user to select net which will be sent to Calculator as VS("<netname>") so user can issue plot command to see result from DC sweep	sch	*
ALT-7	Same as "7", but prompts for a terminal (for current)	sch	*
5	Plot (in ViVa) the expression currently in the Calculator buffer	sch	**
ALT-SHIFT-d	Launch plot-transient-difference command (user prompted to click on two nets)	sch	***
CTRL-7	Launch plot DC signal comand	sch	***
CTRL-d	Close current plot window (sometimes, when you go into select-to-plot command mode (CTRL-4), ViVA opens up a new window and you can end up with unwanted empty windows piling up	ViVA	**
CTRL-SHIFT-o	Save the operating point results to param.txt in HSPICE format	sch	**