PSpice® Quick Reference

PSpice shortcut keys

PSpice toolbars

PSpice Model Editor shortcut keys and toolbar

PSpice Stimulus Editor shortcut keys and toolbar

PSpice shortcut keys

Key	Mode or user interface item	Function or menu command
CTRL+A	Zoom menu (on the View menu)	Area
CTRL+C	Edit menu	Copy
CTRL+SHIFT+C	Cursor menu (on the Trace menu)	Display
CTRL+F	Edit menu	Find
CTRL+SHIFT+F	Cursor menu (on the Trace menu)	Freeze
CTRL+G	Edit menu	Goto Line
CTRL+H	Edit menu	Replace
CTRL+I	Zoom menu (on the View menu)	In
CTRL+SHIFT+I	Cursor menu (on the Trace menu)	Point
CTRL+L	Zoom menu (on the View menu)	Redraw
CTRL+SHIFT+L	Cursor menu (on the Trace menu)	Slope
CTRL+SHIFT+M	Cursor menu (on the Trace menu)	Min
CTRL+N	With a PSpice window active	Creates a new text file
CTRL+SHIFT+N	Cursor menu (on the Trace menu)	Next Transition
CTRL+O	File menu	Open
CTRL+P	File menu	Print
CTRL+SHIFT+P	Cursor menu (on the Trace menu)	Peak
CTRL+SHIFT+R	Cursor menu (on the Trace menu)	Previous Transition
CTRL+SHIFT+S	Cursor menu (on the Trace menu)	Search Commands
CTRL+SHIFT+T	Cursor menu (on the Trace menu)	Trough
CTRL+U	With a waveform window active	Restores the last deleted traces
CTRL+V	Edit menu	Paste
CTRL+X	Edit menu	Cut
CTRL+SHIFT+X	Cursor menu (on the Trace menu)	Max
CTRL+Y	With a waveform window active	Add a Y axis
CTRL+SHIFT+Y	With a waveform window active	Delete a Y axis
F1	Help menu	Help Topics
F3	Edit menu	Find Next
ALT+F4	File menu	Exit
F12	With a waveform window active	Restores the last waveform window
INSERT	With a waveform window active	Opens the Add Traces dialog box
DELETE	Edit menu	Delete
CTRL+DELETE	With a waveform window active	Deletes all traces in the waveform window

PSpice toolbars

PSpice toolbar in Capture

Tool	Name	Description
	New simulation profile	Opens the New Simulation dialog box. Equivalent to the New Simulation Profile command on the PSpice menu.
	Edit simulation settings	Opens the Simulation Settings dialog box. Equivalent to the Edit Simulation Settings command on the PSpice menu.
	Run	Runs the simulation. Equivalent to the Run command on the PSpice menu.
	View simulation results	Opens the simulation window. Equivalent to the View Simulation Results command on the PSpice menu.
	Voltage level marker	Places a voltage level marker on the schematic page. Equivalent to the Voltage Level command on the Markers menu (on the PSpice menu).
	Current into pin marker	Places a current into pin marker on the schematic page. Equivalent to the Current Into Pin command on the Markers menu (on the PSpice menu).
	Voltage differential marker	Places a voltage differential marker on the schematic page. Equivalent to the Voltage Differential command on the Markers menu (on the PSpice menu).

File toolbar in PSpice

Tool	Name	Description
	New	Opens a new simulation file or a new text file. Equivalent to the Simulation Profile command or Text File command on the New menu (on the File menu).
	Open	Opens a data file. Equivalent to the Open command on the File menu.
	Append file	Opens the Append dialog box, which you use to append a data file to the current waveform data. Equivalent to the Append Waveform (.DAT) command on the File menu.
	Save	Saves the active file. Equivalent to the Save command on the File menu.
	Print	Prints the active file. Similar to the Print command on the File menu.

PSpice toolbars (continued)

Edit toolbar in PSpice

Tool	Name	Description
✂	Cut	Removes the selected object and places it on the Clipboard. Equivalent to the Cut command on the Edit menu.
📋	Copy	Copies the selected object to the Clipboard. Equivalent to the Copy command on the Edit menu.
📋	Paste	Pastes the contents of the Clipboard at the cursor. Equivalent to the Paste command on the Edit menu.
↺	Undo	Undoes the last command performed. Equivalent to the Undo command on the Edit menu.
↻	Redo	Redoes the last command performed. Equivalent to the Redo command on the Edit menu.
	Toggle bookmark	Toggles a bookmark at the current line. Equivalent to the Toggle Bookmark command on the Edit menu.
	Next bookmark	Moves to the next bookmark. Equivalent to the Next Bookmark command on the Edit menu.
	Previous bookmark	Moves to the previous bookmark. Equivalent to the Previous Bookmark command on the Edit menu.
	Clear all bookmarks	Clears all bookmarks in the active window. Equivalent to the Clear Bookmarks command on the Edit menu.

Simulate toolbar in PSpice

Tool	Name	Description
▶	Run	Runs the current simulation. Equivalent to the Run command on the Simulation menu.
❚❚	Pause	Pauses the simulation run. Equivalent to the Pause command on the Simulation menu.

PSpice toolbars (continued)

Probe toolbar in PSpice

Tool	Name	Description
	Zoom in	Zooms in on a specified point. Equivalent to the In command on the Zoom menu (on the View menu).
	Zoom out	Zooms out from a specified point. Equivalent to the Out command on the Zoom menu (on the View menu).
	Zoom area	Zooms in on a selected area of a graph. Equivalent to the Area command on the Zoom menu (on the View menu).
	Zoom fit	Zooms to show all traces and labels. Equivalent to the Fit command on the Zoom menu (on the View menu).
	Log X axis	Toggles the X axis between log and linear scaling. Equivalent to selecting the Log option or Linear option in the X Axis tab in the Axis Settings dialog box (accessed by choosing the Axis Settings command on the Plot menu).
	Fourier	Toggles between displaying the Fourier transform of all analog traces in the selected plot. Equivalent to the Fourier (or End Fourier) command on the Trace menu.
	Performance analysis	Toggles the performance analysis on and off. Equivalent to the Performance Analysis command on the Trace menu. NOTE: This toolbar button and corresponding menu command are not available in PSpice A/D Basics.
	Log Y axis	Toggles the Y axis between log and linear scaling. Equivalent to selecting the Log option or Linear option in the Y Axis tab in the Axis Settings dialog box (accessed by choosing the Axis Settings command on the Plot menu).
	Add trace	Opens the Add Traces dialog box, which you use to add a trace (or traces) to a selected plot. Equivalent to the Add Trace command on the Trace menu.
	Eval goal function	Opens the Evaluate Goal Function(s) dialog box, which you use to evaluate a goal function value. Equivalent to the Eval Goal Function command on the Trace menu. NOTE: This toolbar button and corresponding menu command are not available in PSpice A/D Basics.
	Text label	Opens the Text Label dialog box, which you use to add a text label to a graph. Equivalent to the Text command on the Label menu (on the Plot menu).
	Mark data points	Marks data points on analog traces. Equivalent to selecting the Mark Data Points option in the Probe Options dialog box (accessed by choosing the Options command on the Tools menu).
	Toggle cursor	Toggles the display of the Probe cursor on and off. Equivalent to the Display command on the Cursor menu (on the Trace menu).

PSpice toolbars (continued)

Cursor toolbar in PSpice

Tool	Name	Description
	Cursor peak	Positions the cursor at the next peak value. Equivalent to the Peak command on the Cursor menu (on the Trace menu).
	Cursor trough	Positions the cursor at the next trough. Equivalent to the Trough command on the Cursor menu (on the Trace menu).
	Cursor slope	Positions the cursor at the next slope. Equivalent to the Slope command on the Cursor menu (on the Trace menu).
	Cursor min	Positions the cursor at the next minimum value. Equivalent to the Min command on the Cursor menu (on the Trace menu).
	Cursor max	Positions the cursor at the next maximum value. Equivalent to the Max command on the Cursor menu (on the Trace menu).
	Cursor point	Positions the cursor at the next data point. Equivalent to the Point command on the Cursor menu (on the Trace menu).
	Cursor search	Opens the Search Command dialog box, which you use to position the cursor at a specific place along a trace. Equivalent to the Search Commands command on the Cursor menu (on the Trace menu).
	Cursor next tran	Positions the cursor at the next digital transition. Equivalent to the Next Transition command on the Cursor menu (on the Trace menu).
	Cursor prev tran	Positions the cursor at the previous digital transition. Equivalent to the Previous Transition command on the Cursor menu (on the Trace menu).
	Mark label	Marks the current cursor location with a value. Equivalent to the Mark command on the Label menu (on the Plot menu).

PSpice Model Editor shortcut keys and toolbar

NOTE: PSpice Model Editor is not available in PSpice A/D Basics.

PSpice Model Editor shortcut keys

Key	Mode or user interface item	Function or menu command
CTRL+C	Edit menu	Copy
CTRL+N	File menu	New
CTRL+O	File menu	Open
CTRL+P	File menu	Print
CTRL+S	File menu	Save
CTRL+V	Edit menu	Paste
CTRL+X	Edit menu	Cut
F1	Help menu	Help Topics
ALT+F4	File menu	Exit
DELETE	Edit menu	Delete

PSpice Model Editor toolbar

Tool	Name	Description
	New library	Creates a new library. Equivalent to the New command on the File menu.
	Open library	Opens an existing library. Equivalent to the Open command on the File menu.
	Save library	Saves the active library. Equivalent to the Save command on the File menu.
	Print model directly	Prints the active model. Similar to the Print command on the File menu.
	Print preview	Displays print preview pages of the model data. Equivalent to the Print Preview command on the File menu.
	Cut	Removes the selected object and places it on the Clipboard. Equivalent to the Cut command on the Edit menu.
	Copy	Copies the selected object to the Clipboard. Equivalent to the Copy command on the Edit menu.
	Paste	Pastes the contents of the Clipboard at the cursor. Equivalent to the Paste command on the Edit menu.

PSpice Model Editor shortcut keys and toolbar (continued)

PSpice Model Editor toolbar (continued)

Tool	Name	Description
	Zoom in	Zooms in about the center of the plot. Equivalent to the In command on the View menu.
	Zoom out	Zooms out about the center of the plot. Equivalent to the Out command on the View menu.
	Zoom area	Zooms to fit a selected area. Equivalent to the Area command on the View menu.
	Zoom to fit data	Zooms to show all data in the plot. Equivalent to the Fit command on the View menu.
	New model	Inserts a new model into the library. Equivalent to the New command on the Model menu.
	Toggle X	Toggles the X axis between log and linear scaling. Equivalent to selecting the Log option or Linear option in the Axis Settings dialog box (accessed by choosing the Axis Settings command on the Plot menu).
	Toggle Y	Toggles the Y axis between log and linear scaling. Equivalent to selecting the Log option or Linear option in the Axis Settings dialog box (accessed by choosing the Axis Settings command on the Plot menu).
	Extract	Extracts new parameters from your specifications. Equivalent to the Extract Parameters command on the Tool menu.
	Sync splitters	Synchronizes the splitter bars in the model data windows within the Spec Entry document window. Equivalent to selecting the Synchronize Graph Splitter Window option in the Options dialog box (accessed by choosing the Options command on the Tools menu).
	Update graph	Redraws the graph after you add or edit values. Equivalent to the Redraw command on the View menu.
	Auto refresh	Refreshes a graph automatically after a change is made. Equivalent to selecting the Automatically Update Graph option in the Options dialog box (accessed by choosing the Options command on the Tools menu).

PSpice Stimulus Editor shortcut keys and toolbar

NOTE: PSpice Stimulus Editor is not available in PSpice A/D Basics.

PSpice Stimulus Editor shortcut keys

Key	Mode or user interface item	Function or menu command
ALT+A	Edit menu	Add
CTRL+A	View menu	Area
CTRL+D	With a plot window active	Delete
CTRL+I	View menu	In
CTRL+L	View menu	Redraw
ALT+N	Stimulus menu	New
CTRL+N	View menu	Fit
CTRL+O	View menu	Out
CTRL+P	View menu	Previous
CTRL+T	Edit menu	Attributes
F1	Help menu	Help Topics
ALT+F4	File menu	Exit
F12	File menu	Save As
CTRL+F12	File menu	Open
SHIFT+F12	File menu	Save
CTRL+SHIFT+F12	File menu	Print
DELETE	Edit menu	Delete
ALT+DELETE	Stimulus menu	Remove
INSERT	Stimulus menu	Get

PSpice Stimulus Editor toolbar

Tool	Name	Description
	New	Opens a new plot window. Equivalent to the New command on the File menu.
	Open	Opens an existing stimulus library. Equivalent to the Open command on the File menu.
	Save	Saves the current stimulus library. Equivalent to the Save command on the File menu.
	Print	Prints the active plot window. Similar to the Print command on the File menu.

PSpice Stimulus Editor shortcut keys and toolbar (continued)

PSpice Stimulus Editor toolbar (continued)

Tool	Name	Description
	Zoom in	Zooms in and centers plot at cursor location. Equivalent to the In command on the View menu.
	Zoom out	Zooms out and centers plot at cursor location. Equivalent to the Out command on the View menu.
	View area	Zooms in on an area you define. Equivalent to the Area command on the View menu.
	View fit	Zooms to the default view level. Equivalent to the Fit command on the View menu.
	Axis settings	Opens the Axis Settings dialog box, which you use to specify the axis settings. Equivalent to the Axis Settings command on the Plot menu.
	New stimulus	Opens the New Stimulus dialog box, which you use to create a new stimulus. Equivalent to the New command on the Stimulus menu.
	Get stimulus	Opens the Get Stimulus dialog box, which you use to add stimuli to the plot. Equivalent to the Get command on the Stimulus menu.
	Edit attributes	Opens the Edit Digital Transition dialog box, which you use to modify the values and settings of the selected stimuli. Equivalent to the Attributes command on the Edit menu.
	Add	Adds a new point or transition to the stimuli at the cursor location. Equivalent to the Add command on the Edit menu.