

SCALDsystem™
Graphics Editor
Command Summary

Part Number 900-00061



Copyright © 1985, Valid Logic Systems, Incorporated

CONTENTS

Introduction	1
Session Control Commands	3
Commands to Add Objects	5
Commands to Modify Objects	7
Commands to Modify the Display	12
Group Manipulation Commands	13
Diagnostic Commands	14
Documentation Commands	16
Environment Commands	17
Index	21

GRAPHICS EDITOR SYNTAX SUMMARY

Commands can be entered either from the screen menu with the puck or from the keyboard. While in the Graphics Editor (GED), typing **help help** or selecting **HELP** twice from the menu displays a list of GED commands for which on-line help is available. Entering **help** once followed by any command name displays help for that feature only.

The portion of the command line in boldface type must be entered literally on the command line. Boldface type indicates the smallest unique portion of a command that the program recognizes; GED also recognizes complete command names. Variables that require the substitution of a name or a value are in italics. GED recognizes commands and variables in either lower- or upper-case. All default values are specified as part of the command definition.

This guide uses the following abbreviations:

<i>dwg</i>	drawing name
<i>dir</i>	directory name
<i>pt</i>	point
<i>lib</i>	library
<CR>	carriage return
<i>ver</i>	version

A point is usually specified with the yellow puck button.

The metacharacters * and ? serve their respective purposes of matching all characters and any one character. The name and numbers after a drawing name, as in logic.1.1, reference the drawing type, version, and page.

Square brackets [] indicate optional arguments. Syntactical lines in which the command abbreviation is repeated at the beginning of each line indicate

either/or choices among the forms of the command, as in, for example:

grid on;
grid off;
grid dots;
grid lines;

Angle brackets < > are entered literally, as are quotation marks " " that appear on the command line.

Ellipses at the end of a command line indicate that all command arguments can be repeated before the command line is terminated. Whenever a portion of the command line can be repeated, that portion is indicated by parentheses followed by ellipses.

Commands are terminated either by selecting the semicolon from the menu or by entering a carriage return. Unless specified otherwise, commands selected from the on-screen menu can be terminated by entering another command from the menu or keyboard.

SESSION CONTROL COMMANDS

diagram [*<dir>*][*dwg*][.*type*][.*ver*][.*page*]]]

Changes the name of the current drawing. The default type, version, and page are shown on the status line.

directory [*<dir>*][*dwg*][.*type*][.*ver*][.*page*]]]

Lists all drawings in the current directory; **dir** <*> lists all active directories in a user's account; **dir**<*>* lists the drawings contained within each active directory. See **help directory**.

edit [*<dir>*][*dwg*][.*type*][.*ver*][.*page*]]]

edit *pt*

Calls a drawing to the screen; the default type, version, and page is logic.1.1. The drawing can be a new one or an existing one. The **edit** command searches all of a user's active directories unless one is specified. In reference to a point, **edit** calls up the logic definition of a user-defined body.

exit

Terminates an editing session. Issuing the **exit** command twice causes unwritten modifications to be abandoned.

get [*<dir>*][*dwg*][.*type*][.*ver*][.*page*]]]

get *pt*

Gets a copy of the drawing from the disk and overwrites the current copy in main memory. All directories are searched unless one is specified. A **get** command followed by a carriage return re-edits the current drawing. If only the drawing name is specified, the default type, version, and page is logic.1.1.

ignore

ignore *dir*

ignore *libname*

Excludes the specified directory or library from the active search list. Wild cards are permitted. Bodies from the ignored libraries are made into place-

holders that are over-written if another body exists with the same name and version. Using **ignore** by itself causes the current directory to be ignored. The program asks for confirmation.

led

Allows users to compare signal names between logic design and layout. This command is for use with the SCALDstar system. To enable this operation, type **led** in the GED window and type **ged** in the Layout Editor window. See **show net**.

library [*libname*]

Adds a library to a user's active search list. Typing **library** and a carriage return lists the libraries that can be referenced.

quit

Same as **exit**.

remove [*<dir>*]*dwg*[[*type*][*.ver*][*.page*]]]

Removes the specified drawings from the disk. The user must enter a semicolon to delete the drawings. Typing in **abort**, or a command other than a semicolon, cancels the command.

return

Returns to the previously-edited drawing from the current drawing.

script *filename*

Executes the GED commands listed within the specified text file.

simulate

Creates a Simulator window in the lower portion of the screen and establishes communication between the two windows. The simulator's open command is used interactively with signal names selected from the GED window. Compile the drawing for SIM before issuing **simulate**. To exit from the simulator, type **exit** in the simulator window.

use *dir*

Specifies the current working directory on a user's active search list. The UNIX pathname must be

given to use a SCALD directory outside of the current UNIX directory.

write

write [*<dir>*]*dwg*[[*type*][*.ver*][*.page*]]]

Writes the current drawing to the disk. Unless specified otherwise, a drawing is written to the directory it was retrieved from, or in the case of a new drawing, to the current directory.

COMMANDS TO ADD OBJECTS

add [*<dir>*]*body*[[*type*][*.ver*]]] *pt...*

Adds a body to a drawing. The default type is BODY, version 1. Unless a directory is specified, all active directories are searched.

auto *path*

Assigns path properties to bodies that do not have a path property. All bodies are automatically given path properties when the drawing is written; path properties are not transferred when a body is copied.

auto *dot*

Inserts a dot at each wire junction. See **set dots_filled** and **set dots_open**.

backannotate

Adds location designators and pin numbers assigned by the Packager to a drawing. After running the Packager with the **output backannotation**; directive, change the name of the annotation file from *pstback.dat* to *backann.cmd*, re-enter GED, and issue the **backannotate** command. Properties added with **backannotate** are not written to the connectivity file.

circle *center-pt edge-pt [arc-pt]...*

Adds circles and arcs to a drawing. The first two points define the center and edge points of a circle. An arc is entered with the edge and arc points indicating the beginning and ending of the arc (moving

counterclockwise on the circle).

copy (*source-pt destination-pt*)...

copy (*groupname destination-pt*)...

copy (*property-pt destination-pt attach-pt*)...

Copies any object or group of objects in the current drawing. The yellow button copies single objects. Either the group name or the white button copies groups. When used with properties, points are required to indicate the property to be moved, its new placement, and the object to which it is to be attached. Default properties and properties created by **section**, **pinswap**, and **backannotate** cannot be copied.

cut *pt*

cut *groupname*

Copies an object (yellow button) or a group (white button or *groupname*) from the drawing to a buffer, keeping default and user-added properties. Only one **cut** buffer can be saved at a time. See **paste**.

dot *pt...*

Adds dots to a drawing to indicate connections at wire junctions and to indicate pin connection points within bodies. See **set dots__filled**, **set dots__open**, and **auto dots**.

filenote *filename pt*

Adds contents of a text file as explanatory notes at a specified point on a drawing. These notes can be repositioned with the **move** command.

note (*textline... pt...*)...;

Inserts text strings on a drawing for documentation purposes. A semicolon exits the command.

paste <CR> *pt ...*

Transfers objects from the **cut** buffer to the specified location(s) in the current drawing. Once an object has been positioned, a subsequent press of the yellow button attaches another copy of the data to

the cursor for further placement.

property (*attach-pt (name value text-pt)*)...)...

Attaches a property name and value to a specific vertex on a pin, wire, or body. The *text-pt* designates where the text for that value will appear. The names must start with an alphabetic character. See also **display**, **reattach**, and **show properties**.

signame (*signalname... pt...*)... ;

Attaches a user-designated signal name to the wire or pin nearest the specified point. When defining bodies, **signame** is used to name pins.

wire (*pt pt...*)...

Adds wires to a drawing. A press of the yellow button places an endpoint of a wire at the nearest grid intersection. Two presses of the yellow button ends a wire at the nearest grid intersection. One press of the blue button starts or ends a wire at a pin. A press of a white or green button changes the wiring mode.

COMMANDS TO MODIFY OBJECTS

backannotate See Commands to Add Objects

bubble *pt...*

Toggles bubbled pins between input and output. The pins that can be bubbled are defined when the body is created.

change *pt...*

Edits text strings in a drawing with a line editor. Control-I displays a help menu for the line editor. Alternatively, the screen-oriented editor vi is invoked with ^V and exited with :wq or ZZ. Note that user-added properties are labelled !PROP, and default or body properties are labelled !DEF.

delete *pt...*

delete *groupname...*

Deletes the object or group nearest to the cursor. The yellow button deletes single objects. Either the group name or the white button deletes groups.

display

A command that changes the way objects are displayed.

display name *pt*

display name "*groupname*"

display value *pt*

display value "*groupname*"

display both *pt*

display both "*groupname*"

display invisible *pt*

display invisible "*groupname*"

Controls the display of property names and values.

Either or both the name and value can be displayed, or both can be suppressed. The puck buttons or *groupname* define the properties affected. Note that double quotes are required around the *groupname*.

display default *pt*

display default "*groupname*"

display realnumber *pt*

display realnumber "*groupname*"

Controls the size of notes and property names in a drawing. The default option returns the text to its default size, which is 12 characters per horizontal inch. The *realnumber* option alters the size. For example, text strings can be reduced to 80% with a **display** number of **0.8**, or doubled with **2.0**.

display heavy *pt*

display heavy "*groupname*"

display thin *pt*

display thin "*groupname*"

Alters the width of wires on the screen. Heavy wires are often used for displaying bus signals, and thin wires are used for scalars.

display pattern *patternnumber* *pt*

display pattern *patternnumber* "*groupname*"

Alters the style of wires displayed on the screen.

A solid line is the default pattern for wires (*patternnumber* 1). An additional five patterns, comprised of dots and dashes, are available by entering a digit between 2 and 6 for the *patternnumber* argument. The entire net is affected in all drawings except BODY and DOC drawings.

See **help display**.

display filled *pt*

display filled "*groupname*"

display open *pt*

display open "*groupname*"

Changes the dots in a drawing from open to filled and vice versa.

display left *pt*

display left "*groupname*"

display right *pt*

display right "*groupname*"

Changes the location of the origin point for text notes and properties from the lower left side of the string to the lower right. Left justification is the default.

find See entry in Diagnostic Commands

group [*groupname*] *pt* *pt...*

Defines a group with an arbitrary polygon drawn around the objects to be contained in the group. The blue button may be used to automatically close the group boundary. Objects within the group are then highlighted. By default, a letter of the alphabet is assigned as the *groupname* unless otherwise specified.

mirror *pt...*

Flips a body about the Y-axis. Mirroring should be done with caution since bodies with unmarked pins, such as merge bodies, can have problematic results if bits are reversed. See **rotate** and **spin**.

move (*source-pt destination-pt*)...

move (*groupname destination-pt*)...

Moves any object or group of objects in the drawing. The yellow button moves single objects. Either the group name or the white button moves groups.

paint *colorlabel pt ...*

paint *colorlabel groupname ...*

Colors objects interactively. The object or group to be painted is selected with a point or *groupname*. The following is a list of supported colors, the names of which are either typed literally or selected from the on-screen menu:

aqua	green	pink	skyblue
blue	grey	purple	violet
brown	mono	red	white
peach	orange	salmon	yellow

The default for all objects is a **monochromatic** depiction. See **set color** in Environment Commands.

pinswap *pinnumber pt...*

pinswap *pt pt...*

Swaps pin numbers on a body that are defined to be in the same pin group. Pinswaps may only be performed after a section assignment. See **section**.

reattach *text-pt reattach-pt...*

Reattaches properties from one object to another with the exception of properties generated by **backannotate**, **section**, and **pinswap**. After **reattach** is executed, the user may **move** the reattached property.

replace [*<dir>*]*body[.ver] pt...*

Replaces a body (default version 1) for another body. Default properties are retained, as are pins with the same name or position. If pin names do

not match, a pin property becomes a body property.

rotate *pt...*

Rotates a body or text string 90 degrees for each press of the puck button. A mirroring effect occurs for 180 and 270 degree rotations of bodies, not text. See **mirror** and **spin**.

section [*pinnumber*] *pt...*

Displays different sections of a body with their pin numbers and assigns path properties. Note that pin numbers can only be deleted or moved. See **pinswap** and **help section**.

smash *pt...*

Breaks a body into separate lines, arcs, and notes; the body properties are eliminated. This command is useful for creating library parts and producing documentation.

spin *pt...*

Provides true rotations, not mirrors, of a body. See **mirror** and **rotate**.

split *pt pt...*

Either adds a segment to an existing wire or separates objects with common vertices.

swap *text-pt text-pt...*

Swaps the position of two lines of text (properties or notes). A note cannot be swapped for a property, or vice versa.

version *pt...*

Selects an alternate version of a body, if available. Repeated pressing of the puck cycles through all versions. For example, a NAND gate can be viewed as an INVERT OR gate with the **version** command.

COMMANDS TO MODIFY THE DISPLAY

grid

A command that alters the grid display. Use the **set** command to change units to the metric system.

grid on;

grid off;

Turns the grid display on or off.

grid dots;

grid lines;

Displays the grid as dots or lines.

grid gridsize;

grid gridsize gridmultiple;

Alters the size and display of the grid. The *gridsize* defines the minimum spacing between grid points; the defaults are 0.05" for BODY drawings, 0.166" for DOC drawings, and 0.1" for all others. The *gridmultiple* defines the number of grid points displayed to allow user control of grid density. The grid size should be changed with caution, and should not be changed within an existing drawing or in drawings that will be later combined.

redo

Reverses the effects of **undo** according to the log maintained since the last read or write of the drawing.

show See entry under Diagnostic Commands

undo

Undoes the operation of the previous command. Repeated applications of **undo** will undo commands according to a log recorded since the last read or write of the drawing. Window and file operations cannot be undone.

window

A command that alters the view of the current drawing.

window;

Refreshes the screen and clears error and status messages.

window pt;

Defines a point on the drawing as the new screen center.

window pt pt;

Defines two points as the opposite corners of a rectangle, the contents of which are then expanded to fill the screen.

window pt1 pt2 pt3

Changes the center and scale of a drawing. The new center is *pt1*. The scale of the drawing is determined by the ratio of the distance between *pt1* and *pt3* to that between *pt1* and *pt2*. The drawing will be enlarged if the distance from *pt1* to *pt3* is greater than the distance from *pt1* to *pt2*.

window fit;

Fits the entire drawing to the screen window.

window +integer;

window -integer;

window realnumber;

Specifies relative scaling of a drawing. For example, +2 enlarges the drawing by a factor of 2, -2 reduces the scale by 1/2, and 1.5 enlarges the scale one and a half times.

GROUP MANIPULATION COMMANDS

The following commands operate on groups as well as single objects. See **group** command.

copy See entry in Commands to Add Objects

cut See entry in Commands to Add Objects

delete See entry in Commands to Modify Objects

display See entry in Commands to Modify Objects

find See entry in Diagnostic Commands

paste See entry in Commands to Add Objects

DIAGNOSTIC COMMANDS

check

Checks for connectivity problems and general errors on the current drawing. The location of errors can be viewed with the **error** command.

error

Takes data from **check** and redraws the screen with the detected error located at screen center. An asterisk and short error message appear beside the error.

find pattern

Searches the current drawing and finds all text that matches a given pattern and places it in a group. Note that *pattern* is case-sensitive and that wild cards are permitted. The location of each object found can be individually viewed with the **next** command.

next

Centers each object found by **find** on the screen and displays its origin.

show

This command temporarily displays drawing information until a **window**; command refreshes the screen. Type **show** followed by a carriage return for a list of options.

show attach

Displays attachments between properties and objects.

show body pt

Lists the name of the selected body and the SCALD directory in which it is stored.

show color pt...

Lists color of indicated object.

show coordinate pt...

Lists GED coordinates of the indicated point.

show connections

Displays wire connections in the drawing.

show group groupname

show group pt

Highlights a specified group and lists the number of bodies, notes, arcs, and wires within the group.

show history

Lists the drawings that have been edited during the current GED session. Drawings that have been modified but not written are noted. See also **return**.

show keys

Lists function key assignments.

show net pt

show net netname

Lists the name of a net and highlights its segments. The net can be designated either with the puck or by its signal name. See **led**.

show origins

Displays the origins of bodies.

show pins

Displays the pin connection points on bodies.

show properties

Displays the name and value of all properties.

show pwd

Lists the UNIX directory from which the current GED session originated.

show release

Displays the software release number of GED.

show size pt

Lists the scale of a text string.

show vectors pt

Displays the pin names from the body definition for a selected body.

DOCUMENTATION COMMANDS

filenote See Commands to Add Objects

format *filename* <CR> *dwg*

Creates a DOC drawing by combining a text file with referenced drawings. Drawing locations within the file are noted by an ampersand followed by the drawing name. The drawing is scaled to fit into the number of blank lines following the drawing name.

hardcopy

hardcopy *pagesize* ;

hardcopy *realnumber* ;

hardcopy *pagesize* [*<dir>*]*dwg*[[*.type*][*.ver*][*.pg*]]

hardcopy *realnumber* [*<dir>*]*dwg*[[*.type*][*.ver*][*.pg*]]

Hardcopies drawings to an electrostatic or pen plotter. The drawing can either be adjusted to fit a specified page size (A - E) or scaled to a multiple of a real number (up to 2.5). If a drawing is not specified, the current drawing is plotted. Wild cards are permitted in the specification of the drawing.

note See Commands to Add Objects

scale *pt pt dwg*

Smashes a drawing and fits it within a rectangular space defined by two points in the current drawing. See **filenote** and **format**.

vectorize

Takes the current drawing and creates a file in vector plot format with the name *vector.dat*.

ENVIRONMENT COMMANDS

assign *key "command"*

Assigns the left, right, and top function keys on the keyboard. After typing in **assign**, press the desired function key and enter the command string followed by a carriage return. Note the required double quotes around the command string. Default values,

listed in */u0/editor/softkeyassign*, should be changed with care. The user should never change LF1, LF10, LF11 and LF12. Definitions can also be stored in *startup.ged*. See **help assign** and **show keys**.

set

A command that sets default options used within GED. Any of these commands, for instance, **set grid_on**, may be placed within the *startup.ged* file to set the user's editing environment. Typing **set** followed by a carriage return lists all current settings and options.

set size *scalefactor*

Sets the size of text in a drawing as *n* times the default of 0.082 inches high. *Scalefactor* cannot be a value greater than 24 (2 inches).

set ascii

set noascii

set binary

set nobinary

set conn

set noconn

set dependency

set nodependency

Sets the types of files written when a drawing is saved. Default values for **write** are ASCII, binary, connectivity, and dependency. Any unwanted file types can be turned off temporarily, then reset.

set orthog_wire

set direct_wire

Sets the wiring mode. The default value is orthogonal wiring. See **wire** command.

set stop_at_pin

set go_at_pin

Governs wire-to-pin mode. The default, **stop_at_pin**, ends a wire when it reaches a pin; **go_at_pin** continues the wire. The wire is then stopped by pressing the yellow cursor button twice.

set move_orthog

set move_direct

Controls wire modes during moves. The default, **move_orthog**, preserves the bends in the wires when an object or group is moved; **move_direct** uses diagonal wires.

set dots_open

set dots_filled

Alters display of dots. The default, **dots_open**, displays dots as small, open circles; **dots_filled** displays dots filled.

set sticky_off

set sticky_on

Determines whether default properties on BODY drawings that have been deleted are also deleted from existing instances on logic drawings (**sticky_off**) or whether they are converted into non-default properties (**sticky_on**). The default is **sticky_off**. **Sticky_on** is to be used with caution.

set capslock_off

set capslock_on

Causes GED to see all input as upper case when the caps lock feature is turned on. The default is **capslock_off**.

set grid_off

set grid_on

Governs the display of the grid. The grid is turned off, by default, when GED is entered and on each new edit of a drawing. Enter **set grid_on** to display the grid during editing sessions.

set decimal

Defines GED as having 500 internal units per inch on the Versatec.

set metric

Causes GED to use 20 internal units per Versatec millimeter. Libraries have pins on 2.5mm centers.

set fractional

Resets the default internal division of 500 units per inch on the Versatec to 400, thus allowing a 1/8 inch grid and pin center.

set default_grid gridsize

Sets the default grid size for drawing types other than BODY and DOC. The default is 0.05 for BODY, 0.166 for DOC and 0.1 for all others.

set hpr

set vgb

Defines the method for obtaining hardcopy plots. The **hardcopy printout** option rasterizes files with the aid of the CPU, thereby freeing the terminal for other input. The **video graphics board** option is the default for monochromatic screens. Color terminals and Benson plotters require **hpr**.

set W11versatec

set W22versatec

set W36versatec

set W42versatec

set mono_hpplot

set b9424

Initializes the system for different plotters: Versatec, Hewlett-Packard, and Benson. The default is an 11-inch Versatec.

set local_plot

set spooled_plot

Determines whether spooling is local or remote. Local plotting is the default. The **set spooled** option spools the output for a Versatec (but not an HP) for later use or for remote system use. To plot the files stored in **vw.spool** on a local printer, use the UNIX command **vpl vw.spool**.

set double_width

set single_width

Governs the darkness of plotted lines. The default is double width lines.

set left_justification

set right_justification

Sets justification for text strings. The default is left justification.

set user_sim simfile

Defines an alternate Simulator file. The default is: /u0/scald/simulator/sim.

set color_arc colorlabel

set color_body colorlabel

set color_dot colorlabel

set color_note colorlabel

set color_prop colorlabel

set color_wire colorlabel

Defines the colors of objects. The name of each colorlabel is typed in literally:

aqua	green	pink	skyblue
blue	grey	purple	violet
brown	mono	red	white
peach	orange	salmon	yellow

The default for all objects is a **monochromatic** depiction. See **paint**.

INDEX

add.....	5
assign.....	16
auto	5
backannotate	5
bubble.....	7
change.....	7
check	14
circle.....	5
copy	6
cut.....	6
delete	8
diagram	3
directory	3
display.....	8
dot.....	6
edit.....	3
error	14
exit	3
filenote	6
find.....	14
format.....	16
get	3
grid.....	12
group.....	9
hardcopy	16
ignore.....	3
led.....	4
library.....	4
mirror	10
move.....	10
next.....	14
note.....	6
paste	6
paint.....	10
pinswap	10
property	7

NOTES

quit	4
reattach	10
redo	12
remove	4
replace	11
return	4
rotate	11
scale	16
script	4
section	11
set	17
show	14
signame	7
simulate	4
smash	11
spin	11
split	11
swap	11
undo	12
use	4
vectorize	16
version	11
window	13
wire	7
write	5

NOTES



Valid Logic Systems, Incorporated
2820 Orchard Parkway, San Jose, California 95134
(408) 945-9400 Telex 371 9004

Valid International
39 Windsor Road, Slough, Berkshire, SL1 2EE
44 753 820101 Telex 847 318

Printed in U.S.A.