



---

# ModelSim® Command Reference Manual

Software Version 10.5c

---

© 1991-2016 Mentor Graphics Corporation  
All rights reserved.

This document contains information that is proprietary to Mentor Graphics Corporation. The original recipient of this document may duplicate this document in whole or in part for internal business purposes only, provided that this entire notice appears in all copies. In duplicating any part of this document, the recipient agrees to make every reasonable effort to prevent the unauthorized use and distribution of the proprietary information.

This document is for information and instruction purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in written agreements between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

**MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.**

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

**U.S. GOVERNMENT LICENSE RIGHTS:** The software and documentation were developed entirely at private expense and are commercial computer software and commercial computer software documentation within the meaning of the applicable acquisition regulations. Accordingly, pursuant to FAR 48 CFR 12.212 and DFARS 48 CFR 227.7202, use, duplication and disclosure by or for the U.S. Government or a U.S. Government subcontractor is subject solely to the terms and conditions set forth in the license agreement provided with the software, except for provisions which are contrary to applicable mandatory federal laws.

**TRADEMARKS:** The trademarks, logos and service marks ("Marks") used herein are the property of Mentor Graphics Corporation or other parties. No one is permitted to use these Marks without the prior written consent of Mentor Graphics or the owner of the Mark, as applicable. The use herein of a third-party Mark is not an attempt to indicate Mentor Graphics as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A current list of Mentor Graphics' trademarks may be viewed at: [www.mentor.com/trademarks](http://www.mentor.com/trademarks).

The registered trademark Linux® is used pursuant to a sublicense from LMI, the exclusive licensee of Linus Torvalds, owner of the mark on a world-wide basis.

Mentor Graphics Corporation  
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777  
Telephone: 503.685.7000  
Toll-Free Telephone: 800.592.2210  
Website: [www.mentor.com](http://www.mentor.com)  
SupportNet: [supportnet.mentor.com/](http://supportnet.mentor.com/)

Send Feedback on Documentation: [supportnet.mentor.com/doc\\_feedback\\_form](http://supportnet.mentor.com/doc_feedback_form)

# Table of Contents

---

<b>Chapter 1</b>	
<b>Syntax and Conventions.....</b>	<b>13</b>
Documentation Conventions .....	13
File and Directory Pathnames .....	14
Design Object Names .....	15
Object Name Syntax .....	15
Tcl Syntax and Specification of Array Bits and Slices .....	16
SystemVerilog Scope Resolution Operator .....	17
Specifying Names.....	18
Environment Variables and Pathnames .....	20
Name Case Sensitivity .....	20
Extended Identifiers .....	20
Wildcard Characters .....	21
Supported Commands.....	21
Using the WildcardFilter Preference Variable .....	22
Simulator Variables .....	25
Simulation Time Units.....	26
Optionsets .....	26
Argument Files .....	27
Command Shortcuts.....	28
Command History Shortcuts .....	29
Numbering Conventions .....	30
VHDL Numbering Conventions.....	30
Verilog Numbering Conventions .....	31
GUI_expression_format.....	32
Expression Typing .....	32
Expression Syntax.....	33
Signal and Subelement Naming Conventions .....	39
Grouping and Precedence .....	39
Concatenation of Signals or Subelements .....	39
Record Field Members .....	41
Searching for Binary Signal Values in the GUI .....	41
<b>Chapter 2</b>	
<b>Commands.....</b>	<b>43</b>
abort .....	63
add dataflow.....	64
add list .....	66
add memory.....	71
add message.....	73
add watch.....	75
add wave .....	77

add_cmdhelp . . . . .	84
alias . . . . .	86
archive load . . . . .	87
archive write . . . . .	88
batch_mode . . . . .	89
bd . . . . .	90
bookmark add wave . . . . .	92
bookmark delete wave . . . . .	94
bookmark goto wave . . . . .	95
bookmark list wave . . . . .	96
bp . . . . .	97
call . . . . .	103
cd . . . . .	108
change . . . . .	109
classinfo ancestry . . . . .	111
classinfo descriptive . . . . .	112
classinfo find . . . . .	114
classinfo implements . . . . .	116
classinfo instances . . . . .	118
classinfo interfaces . . . . .	121
classinfo isa . . . . .	123
classinfo report . . . . .	124
classinfo stats . . . . .	126
classinfo trace . . . . .	128
classinfo types . . . . .	130
configure . . . . .	132
dataset alias . . . . .	138
dataset clear . . . . .	139
dataset close . . . . .	141
dataset config . . . . .	142
dataset current . . . . .	144
dataset info . . . . .	145
dataset list . . . . .	146
dataset open . . . . .	147
dataset rename . . . . .	149
dataset restart . . . . .	150
dataset save . . . . .	151
dataset snapshot . . . . .	152
delete . . . . .	155
describe . . . . .	156
disablebp . . . . .	157
do . . . . .	158
drivers . . . . .	160
dumplog64 . . . . .	162
echo . . . . .	163
edit . . . . .	164
enablebp . . . . .	165
encoding . . . . .	166
environment . . . . .	167

## Table of Contents

---

examine . . . . .	168
exit . . . . .	175
find . . . . .	176
find connections . . . . .	181
find infiles . . . . .	182
find insource . . . . .	183
force . . . . .	185
formatTime . . . . .	191
gc configure . . . . .	192
gc run . . . . .	194
help . . . . .	195
history . . . . .	196
layout . . . . .	197
log . . . . .	199
lshift . . . . .	202
lsublist . . . . .	203
mem compare . . . . .	204
mem display . . . . .	205
mem list . . . . .	208
mem load . . . . .	209
mem save . . . . .	213
mem search . . . . .	216
modelsim . . . . .	219
noforce . . . . .	220
nolog . . . . .	221
notepad . . . . .	223
noview . . . . .	224
nowhen . . . . .	225
onbreak . . . . .	226
onElabError . . . . .	229
onerror . . . . .	230
onfinish . . . . .	232
pause . . . . .	233
precision . . . . .	234
printenv . . . . .	235
process report . . . . .	236
project . . . . .	237
pwd . . . . .	240
quietly . . . . .	241
quit . . . . .	242
radix . . . . .	243
radix define . . . . .	245
radix delete . . . . .	249
radix list . . . . .	250
radix names . . . . .	251
radix signal . . . . .	252
readers . . . . .	253
report . . . . .	254
restart . . . . .	256

resume . . . . .	258
run . . . . .	259
runStatus . . . . .	262
searchlog . . . . .	264
see . . . . .	267
setenv . . . . .	268
shift . . . . .	269
show . . . . .	270
simstats . . . . .	271
simstatslist . . . . .	273
stack down . . . . .	275
stack frame . . . . .	276
stack level . . . . .	277
stack tb . . . . .	278
stack up . . . . .	279
status . . . . .	280
step . . . . .	281
stop . . . . .	283
suppress . . . . .	284
tb . . . . .	286
Time . . . . .	287
transcript . . . . .	291
transcript file . . . . .	292
transcript path . . . . .	294
transcript sizelimit . . . . .	295
transcript wrapcolumn . . . . .	296
transcript wrapmode . . . . .	297
transcript wrapwscolumn . . . . .	298
tssi2mti . . . . .	299
ui_VVMode . . . . .	300
unsetenv . . . . .	302
vcd add . . . . .	303
vcd checkpoint . . . . .	305
vcd comment . . . . .	306
vcd dumports . . . . .	307
vcd dumportsall . . . . .	310
vcd dumportsflush . . . . .	311
vcd dumportslimit . . . . .	312
vcd dumpportsoff . . . . .	314
vcd dumpportson . . . . .	315
vcd file . . . . .	316
vcd files . . . . .	318
vcd flush . . . . .	321
vcd limit . . . . .	322
vcd off . . . . .	324
vcd on . . . . .	325
vcd2wlf . . . . .	326
vcom . . . . .	328
vdel . . . . .	345

## Table of Contents

---

vdir . . . . .	347
vencrypt . . . . .	350
verror . . . . .	354
vgencomp . . . . .	356
vhencrypt . . . . .	358
view . . . . .	361
virtual count . . . . .	364
virtual define . . . . .	365
virtual delete . . . . .	366
virtual describe . . . . .	367
virtual expand . . . . .	368
virtual function . . . . .	369
virtual hide . . . . .	372
virtual log . . . . .	373
virtual nohide . . . . .	375
virtual nolog . . . . .	376
virtual region . . . . .	378
virtual save . . . . .	379
virtual show . . . . .	380
virtual signal . . . . .	381
virtual type . . . . .	385
vlib . . . . .	387
vlog . . . . .	390
vmake . . . . .	413
vmap . . . . .	415
vsim . . . . .	417
vsim<info> . . . . .	451
vsim_break . . . . .	452
vsource . . . . .	453
wave . . . . .	454
wave create . . . . .	459
wave edit . . . . .	465
wave export . . . . .	469
wave import . . . . .	471
wave modify . . . . .	472
wave sort . . . . .	477
when . . . . .	478
where . . . . .	486
wlf2log . . . . .	487
wlf2vcd . . . . .	489
wlfman . . . . .	490
wlfrecover . . . . .	497
write format . . . . .	498
write list . . . . .	501
write preferences . . . . .	502
write report . . . . .	503
write timing . . . . .	506
write transcript . . . . .	508
write tssi . . . . .	509

write wave ..... 511

## Index

## End-User License Agreement

## List of Figures

---

Figure 2-1. drivers Command Results in Transcript .....	160
Figure 2-2. find infiles Example .....	182
Figure 2-3. find insource Example .....	184
Figure 2-4. readers Command Results in Transcript.....	253



# List of Tables

---

Table 1-1. Conventions for Command Syntax .....	13
Table 1-2. Examples of Object Names .....	19
Table 1-3. Wildcard Characters in HDL Object Names .....	22
Table 1-4. WildcardFilter Arguments .....	24
Table 1-5. WildcardFilter Argument Groups .....	25
Table 1-6. Keyboard Shortcuts for Command History .....	29
Table 1-7. VHDL Number Conventions: Style 1 .....	30
Table 1-8. VHDL Number Conventions: Style 2 .....	30
Table 1-9. Verilog Number Conventions .....	31
Table 1-10. Constants Supported for GUI Expressions .....	33
Table 1-11. Array Constants Supported for GUI Expressions .....	33
Table 1-12. Variables Supported for GUI Expressions .....	34
Table 1-13. Array Variables Supported for GUI Expressions .....	34
Table 1-14. Operators Supported for GUI Expressions .....	35
Table 1-15. Precedence of GUI Expression Operators .....	36
Table 1-16. Casting Conversions Supported for GUI Expressions .....	37
Table 1-17. VHDL Logic Values Used in GUI Search .....	42
Table 1-18. Verilog Logic Values Used in GUI Search .....	42
Table 2-1. Supported Commands .....	43
Table 2-2. Message Viewer Categories .....	73
Table 2-3. Radix flag Arguments to the Examine Command .....	172
Table 2-4. runStatus Command States .....	262
Table 2-5. runStatus -full Command Information .....	262
Table 2-6. Warning Message Categories for vcom -nowarn .....	338
Table 2-7. Design Unit Properties .....	348
Table 2-8. Warning Message Categories for vlog -nowarn .....	402
Table 2-9. Wave Window Commands for Cursor .....	455
Table 2-10. Wave Window Commands for Expanded Time Display .....	455
Table 2-11. Wave Window Commands for Controlling Display .....	456
Table 2-12. Wave Window Commands for Zooming .....	456



# Chapter 1

## Syntax and Conventions

---

This chapter describes the typographical conventions used in this manual to define ModelSim command syntax.

## Documentation Conventions

The following conventions are used to define ModelSim command syntax

**Table 1-1. Conventions for Command Syntax**

Syntax notation	Description
< >	angled brackets surrounding a syntax item indicate a user-defined argument; do not enter the brackets in commands
[ ]	square brackets generally indicate an optional item; if the brackets surround several words, all must be entered as a group; the brackets are not entered <sup>1</sup>
{ }	braces indicate that the enclosed expression contains one or more spaces yet should be treated as a single argument, or that the expression contains square brackets for an index; for either situation, the braces are entered
...	an ellipsis indicates items that may appear more than once; the ellipsis itself does not appear in commands
	the vertical bar indicates a choice between items on either side of it; do not include the bar in the command
monospaced type	monospaced type is used in command examples
#	comments included with commands are preceded by the number sign (#), which you can use to add comments to DO files (macros)

1. One exception to this rule is when you are using Verilog syntax to designate an array slice. For example:

```
add wave {vector1[4:0]}
```

The square brackets in this case denote an index. The braces prevent the Tcl interpreter from treating the text within the square brackets as a Tcl command.

---

**Note**

 Command examples do not show either the prompt at the beginning of a line nor the <Enter> keystroke at the end of a line.

---

## File and Directory Pathnames

Several ModelSim commands have arguments that specify file or directory locations (pathnames). For example, the **-y** argument to the **vlog** command specifies the Verilog source library directory to search for undefined modules.

Spaces in file pathnames must be escaped or the entire path must be enclosed in quotation marks. For example:

```
vlog top.v -y C:/Documents\ and\ Settings/projects/dut
```

or

```
vlog top.v -y "C:/Documents and Settings/projects/dut"
```

## Design Object Names

---

Design objects are organized hierarchically, where various objects creates a new level in the hierarchy.

- **VHDL** — component instantiation statement, block statement, and package
- **Verilog** — module instantiation, named fork, named begin, task and function
- **SystemVerilog** — class, package, program, and interface

## Object Name Syntax

To specify object names in ModelSim, you use the following syntax:

```
[<dataset_name><datasetSeparator>] [<pathSeparator>] [<hierarchicalPath>]  
<objectName> [<elementSelection>]
```

where

- **dataset\_name** — The name mapped to the WLF file in which the object exists. The currently active simulation is the “sim” dataset. Any loaded WLF file is referred to by the logical name specified when the WLF file was loaded. Refer to the chapter [“Recording Simulation Results With Datasets”](#) in the User’s Manual for more information.
- **datasetSeparator** — The character used to terminate the dataset name. The default is colon ( : ), although you can specify a different character (except for a backslash ( \ )) as the dataset separator by using the **DatasetSeparator** variable in the *modelsim.ini* file. This character must be different than the pathSeparator character.
- **pathSeparator** — The character used to separate hierarchical object names. Normally, a backslash ( \ ) is used for VHDL and a period ( . ) is used for Verilog, although you can specify a different character (except for a backslash ( \ )) by using the **PathSeparator** variable in the *modelsim.ini* file. This character must be different than the datasetSeparator.

Neither ( . ) nor ( / ) can be used when referring to the contents of a SystemVerilog package or class.

- **hierarchicalPath** — A set of hierarchical instance names separated by a path separator and ending in a path separator prior to the objectName. For example, */top/proc/clk*.
- **objectName** — The name of an object in a design.
- **elementSelection** — Some combination of the following:
  - **Array indexing** — Single array elements are specified using either parentheses ( () ) or square brackets ( [ ] ) around a single number. You must also surround the object

and specified array element with curly braces ( {} ). Refer to [Tcl Syntax and Specification of Array Bits and Slices](#) for important information about using square brackets and parentheses in ModelSim commands.

- **Array slicing** — Slices (or part-selects) of arrays are specified using either parentheses ( () ) or square brackets ( [] ) around a range specification. A range is two numbers separated by one of the following: " to ", " downto ", or a colon ( : ). You must also surround the object and specified array slice with curly braces ( {} ). Refer to [Tcl Syntax and Specification of Array Bits and Slices](#) for important information about using square brackets and parentheses in ModelSim commands.
- **Record field selection** — A record field is specified using a period ( . ) followed by the name of the field.

## Tcl Syntax and Specification of Array Bits and Slices

Because ModelSim is based on the Tcl scripting language, you must surround objects and signals with braces ( {} ) when specifying array bits or slices with parentheses ( () ), spaces, or square brackets ( [] ).

For example:

```
toggle add {data[3:0]}
toggle add {data(3 to 0)}
force {bus1[1]} 1
```

### Further Details

Because ModelSim is based on Tcl, its commands follow Tcl syntax. One problem you may encounter with ModelSim commands is the use of square brackets ( [] ), parentheses ( () ), or spaces when specifying array bits and slices. As noted, square brackets specify slices of arrays (for example, *data[3:0]*). However, in Tcl, square brackets signify command substitution.

Consider the following example:

```
set aluinputs [find -in alu/*]
```

ModelSim evaluates the **find** command first and then sets variable *aluinputs* to the result of the **find** command. Obviously, you do not want this type of behavior when specifying an array slice, so you would use brace escape characters, as follows:

```
add wave {/s/abc/data_in[10:1]}
```

You must also use the escape characters if using VHDL syntax with spaces:

```
add wave {/s/abc/data_in(10 downto 1)}
```

For more information on Tcl syntax, refer to [Tcl Command Syntax](#).

## SystemVerilog Scope Resolution Operator

SystemVerilog offers the scope resolution operator, double colon ( :: ), for accessing classes within a package and static data within a class. The example below shows various methods of using this operator as well as alternatives using standard hierarchical references.

### Example 1-1. SystemVerilog Scope Resolution Operator Example

```
package myPackage;
    class packet;
        static int a[0:1] = {1, 2};
        int b[0:1];
        int c;

        function new;
            b[0] = 3;
            b[1] = 4;
            c = a[0];
        endfunction
    endclass
endpackage : myPackage

module top;
    myPackage::packet my = new;
    int myint = my.a[1];
endmodule
```

The following examples of the [examine](#) command access data from the class *packet*.

```
examine myPackage::packet::a
examine /top/my.a
```

Both of the above commands return the contents of the static array *a* within class *packet*.

```
examine myPackage::packet::a(0)
examine /top/my.a(0)
```

Both of the above commands return the contents of the first element of the static array *a* within class *packet*.

```
examine /top/my.b
```

Return the contents of the instance-specific array *b*.

```
examine /top/my.b(0)
```

Return the contents of the first element of the instance-specific array *b*.

When referring to the contents of a package or class, you cannot use the standard path separators, a period ( . ) or a forward slash ( / ).

## Specifying Names

ModelSim distinguishes between four "types" of object names: simple, relative, fully-rooted, and absolute.

- **Simple name** — does not contain any hierarchy. It is simply the name of an object (such as *clk* or *data[3:0]*) in the current context.
- **Relative name** — does not start with a path separator and may or may not include a dataset name or a hierarchical path (such as *u1/data* or *view:clk*). A relative name is relative to the current context in the current or specified dataset.
- **Fully-rooted name** — starts with a path separator and includes a hierarchical path to an object (e.g., */top/u1/clk*). There is a special case of a fully-rooted name where the top-level design unit name can be unspecified (such as */u1/clk*). In this case, the first top-level instance in the design is assumed.
- **Absolute name** — is an exactly specified hierarchical name containing a dataset name and a fully rooted name (such as *sim:/top/u1/clk*).

The current dataset is used when accessing objects where a dataset name is not specified as part of the name. The current dataset is determined by the dataset currently selected in the Structure window or by the last dataset specified in an [environment](#).

The current context in the current or specified dataset is used when accessing objects with relative or simple names. The current context is either the current process, if any, or the current instance if there is no current process, or the current process is not in the current instance. The situation of the current process not being in the current instance can occur, for example, by selecting a different instance in the Structure tab or by using the [environment](#) to set the current context to a different instance.

The current context is also the activation level of an automatic task, function, or block. Different levels of activation may be selected by using the Call Stack window, or by using the 'stack up' or 'stack down' commands.

For example, when you set a breakpoint on line 5 of the following code:

```
package p;
  int I;
  function automatic int factorial(int n);
    if(n==0)
      return 1;
    else
      return n * factorial(n - 1);
  endfunction : factorial
endpackage : p

module top;
  initial begin
    p::I=p::factorial(3);

    $display(p::I);
    $display(p::factorial(4));
  end
endmodule: top
```

When you issue the command:

**examine n**

the transcript returns:

0

However, when you issue the command:

**stack up;examine n**

the transcript returns:

1

[Table 1-2](#) contains examples of various ways of specifying object names.

**Table 1-2. Examples of Object Names**

Object Name	Description
<i>clk</i>	specifies the object <i>clk</i> in the current context
<i>/top/clk</i>	specifies the object <i>clk</i> in the top-level design unit.
<i>/top/block1/u2/clk</i>	specifies the object <i>clk</i> , two levels down from the top-level design unit
<i>block1/u2/clk</i>	specifies the object <i>clk</i> , two levels down from the current context
<i>array_sig[4]</i>	specifies an index of an array object

**Table 1-2. Examples of Object Names (cont.)**

Object Name	Description
<code>{array_sig(1 to 10)}</code>	specifies a slice of an array object in VHDL; see <a href="#">Tcl Syntax and Specification of Array Bits and Slices</a> for more information
<code>{mysignal[31:0]}</code>	specifies a slice of an array object in Verilog; see <a href="#">Tcl Syntax and Specification of Array Bits and Slices</a> for more information
<code>record_sig.field</code>	specifies a field of a record

## Environment Variables and Pathnames

You can substitute environment variables for pathnames in any argument that requires a pathname.

For example:

```
vlog -v $lib_path/und1
```

Assuming you have defined \$lib\_path on your system, vlog will locate the source library file *und1* and search it for undefined modules. Refer to [Environment Variables](#) for more information.

## Name Case Sensitivity

Name case sensitivity is different for VHDL and Verilog. VHDL names are not case-sensitive except for extended identifiers in VHDL 1076-1993 or later. In contrast, all Verilog names are case-sensitive.

Names in ModelSim commands are case-sensitive when matched against case-sensitive identifiers; otherwise, they are not case-sensitive.

## Extended Identifiers

The following are supported formats for extended identifiers for any command that takes an identifier.

```
{\ext ident!\} }  
# Note that trailing space before closing brace is required  
  
\ext\ ident\!\\  
# All non-alpha characters escaped
```

# Wildcard Characters

---

You can use wildcard characters in HDL object names in many simulator commands.

## Supported Commands

There are a number of commands that support wildcard characters.

The following is a partial list of the commands:

- add dataflow
- add list
- add memory
- add watch
- add wave
- describe
- dumports
- examine
- find (see the Examples section in the [find](#) command for wildcard searches in foreach loops to be applied with commands that do not accept wildcards.)
- log
- vcd add

When you execute any of these commands with a wildcard, the default behavior is to exclude the following object types:

- VHDL shared variables in packages and design units, constants, generics, and immediate assertions
- Verilog parameters, specparams, memories
- SystemVerilog multi-dimensional arrays and class objects
- Signals in cells
- Non-dynamic objects of a size equal to or greater than the level specified in the WildcardSizeThreshold *modelsim.ini* variable if the variable has been enabled. Refer to the [WildcardSizeThreshold](#) and [WildcardSizeThresholdVerbose](#) *modelsim.ini* variables for more information.

You can alter these exclusions with the WildcardFilter preference variable. Refer to the section “[Using the WildcardFilter Preference Variable](#)” for more information.

Table 1-3 identifies these supported wildcard characters.

**Table 1-3. Wildcard Characters in HDL Object Names**

Character Syntax	Description
*	matches any sequence of characters
?	matches any single character
[]	matches any one of the enclosed characters; a hyphen can be used to specify a range (for example, a-z, A-Z, 0-9); can be used <i>only</i> with the <a href="#">find</a> command

---

**Note**

 A wildcard character does not match a path separator. For example, `/dut/*` will match `/dut/siga` and `/dut/clk`. However, `/dut*` will not match either of those.

---

## Using the WildcardFilter Preference Variable

The WildcardFilter preference variable controls which object types are excluded when performing wildcard matches with simulator commands. The WildcardFilter preference variable is a Tcl List and can be modified using Tcl commands.

The default object types are defined with the [WildcardFilter](#) *modelsim.ini* variable and load at each invocation of the simulator. You can add both individual ([Table 1-4](#)) and group objects ([Table 1-5](#)) to the current variable list, and you can remove individual objects from the current list.

### Determining the Current WildcardFilter Variable Settings

Enter one of the following commands:

**set WildcardFilter**

or

**echo \$WildcardFilter**

which returns the list of currently set variables.

### Changing the WildcardFilter Settings from the Command Line

Refer to the list of WildcardFilter arguments in [Table 1-4](#) and [Table 1-5](#) to determine what you want to include in the wildcard matches.

- To define a new list of values enter the following command:

```
set WildcardFilter "<arg1 arg2 ...>"
```

Note that you must enclose the space-separated list of arguments in quotation marks.

- To add one or more values to the current list enter the following command:

```
lappend WildcardFilter <arg1 arg2 ...>
```

Note that you must not enclose the space-separated list of arguments in quotation marks.

- To remove a value from the filter use the set command with the Tcl lsearch command to create the new list from the existing list. For example:

```
set WildcardFilter [lsearch -not -all -inline $WildcardFilter Endpoint]
```

#### Changing the WildcardFilter Settings back to the Default

Enter the following command:

```
set WildcardFilter default
```

#### Changing the WildcardFilter settings from the GUI

1. Choose **Tools > Wildcard Filter** from the main menu.
2. Select the individual Filters you want to exclude from wildcard searches ([Table 1-4](#) describes each option), or select Composite Filters to activate related filters ([Table 1-5](#) describes each composite option).
3. Click OK.

Refer to the Tcl man pages ([Help>Tcl Man Pages](#)) for more information about the lsearch and set commands.

#### Changing the default WildcardFilter settings

1. Open the modelsim.ini file for editing (refer to [Making Changes to the modelsim.ini File](#))
2. Select the individual Filters you want to exclude from wildcard searches ([Table 1-4](#) describes each option), or select Composite Filters to activate related filters ([Table 1-5](#) describes each composite option).
3. Edit the WildcardFilter variable
4. Save the modelsim.ini file to your working directory.

#### WildcardFilter Argument Descriptions

[Table 1-4](#) provides a list of the WildcardFilter arguments.

**Table 1-4. WildcardFilter Arguments**

Argument	Description
Alias	VHDL Alias
CellInternal	Signals in cells, where a cell is defined as 1) a module within a 'celldefine 2) a Verilog module found with a library search (using either vlog -v or vlog -y) and compiled with vlog +libcell or 3) a module containing a specify block
Class	Verilog class declaration
ClassReference	SystemVerilog class reference
Compare	Waveform comparison signal
Constant	VHDL constant
Generic	VHDL generic
ImmediateAssert	VHDL immediate assertions
Integer	VHDL integer
Memory	Verilog memories
NamedEvent	Verilog named event
Net	Verilog net
Parameter	Verilog parameter
Real	Verilog real registers
Reg	Verilog register
Signal	VHDL signal
SpecParam	Verilog specparam
Time	Verilog time registers
Transaction	Transaction stream and stream arrays
Variable	VHDL shared variables in packages and design units.
VHDLFile	VHDL files
VirtualExpr	Virtual expression
VirtualSignal	Virtual signal

[Table 1-5](#) provides a list of the group aliases of WildcardFilter arguments. You can set a group value with the set command. The expanded list of values is returned.

**Table 1-5. WildcardFilter Argument Groups**

Group Argument	Specific arguments included
AllVHDL	Architecture, Block, Generate, Package, Foreign, Process, Signal, Variable, Constant, Generic, Alias, Subprogram, VHDLFile
AllVerilogVars	Parameter, Reg, Integer, Time, Real, SpecParam, Memory, NamedEvent, ClassReference
AllVerilog	Net, Parameter, Reg, Integer, Time, Real, SpecParam, Memory, NamedEvent, Class, Cross, Covergroup, Coverpoint, ClassReference
VirtualSignals	VirtualSignal, VirtualExpr
AllHDLSignals	Signal, Variable, Constant, Generic, Alias, Net, Parameter, Reg, Integer, Time, Real, SpecParam, Memory, NamedEvent, VirtualSignal, VirtualExpr, ClassReference
AllVariables	Variable, Constant, Generic, Alias, Parameter, Reg, Integer, Time, Real, SpecParam, Memory, NamedEvent, ClassReference
AllHDLSignalsVars	Signal, Variable, Constant, Generic, Alias, Net, Parameter, Reg, Integer, Time, Real, SpecParam, Memory, NamedEvent, VirtualSignal, VirtualExpr, ClassReference
AllSignals	Signal, Net, Parameter, Reg, Integer, Time, Real, SpecParam, Memory, NamedEvent, VirtualSignal, VirtualExpr, Endpoint, ClassReference
AllSignalsVars	Signal, Variable, Constant, Generic, Alias, Net, Parameter, Reg, Integer, Time, Real, SpecParam, Memory, NamedEvent, VirtualSignal, VirtualExpr, Endpoint, ScVariable, ClassReference
AllConstants	Constant, Generic, Parameter, SpecParam
Default	Variable, Constant, Generic, Parameter, SpecParam, Memory, Assertion, Cover, Endpoint, ScVariable, CellInternal, ImmediateAssert VHDLFile

## Simulator Variables

You can reference ModelSim variables in a simulator command by preceding the name of the variable with the dollar sign (\$) character.

ModelSim uses global variables for simulator state variables, simulator control variables, simulator preference variables, and user-defined variables. Refer to [modelsim.ini Variables](#) in the User's Manual for more information on variables.

The [report](#) command returns a list of current settings for either the simulator state or simulator control variables.

## Simulation Time Units

You can specify the time unit for delays in all simulator commands that have time arguments.

For example:

```
force clk 1 50 ns, 1 100 ns -repeat 1 us
run 2 ms
```

Note that all the time units in a ModelSim command need not be the same.

Unless you specify otherwise as in the examples above, simulation time is always expressed using the resolution units that are specified by the [UserTimeUnit](#) variable.

By default, the specified time units are assumed to be relative to the current time unless the value is preceded by the character @, which signifies an absolute time specification.

## Optionsets

By defining and calling optionsets, you can easily use and combine common command line options.

The executable expands these optionsets and passes them to the tool as if they appeared directly on the command line. The behavior is similar to the **-f <file>** option.

### Defining an Optionset

Define your optionsets in the [DefineOptionset] section of the *modelsim.ini* file, where the syntax is:

```
<optionset_name> = <command_arguments>
```

- <optionset\_name> — a string that begins with a letter, and contains only letters, numbers, or underscores. The name is case-insensitive.
- <command\_arguments> — a list of arguments as you would specify them on the command line. This list of arguments can:
  - Refer to another <optionset\_name>, enclosed in percent-signs (%).

- Include shell environment variables, preceded by a dollar-sign (\$). If you embed the variable in a string, you must surround it with parentheses.

You can instruct the executable to return all the values of any optionsets as they are read with the following entry in the [optionsets] section.

```
PRINT_OPTIONSET_VALUE = 1
```

### Calling an Optionset

Call your defined optionsets with the -optionset argument to the commands: vlog, vcom and vsim.

The syntax of -optionset is:

```
<command> -optionset <optionset_name>
```

## Argument Files

You can load additional arguments into some commands by using argument files, which are specified with the -f argument.

The following commands support the -f argument:

- vlog
- vcom
- vencrypt
- vmake
- vsim

The -f <filename> argument specifies a file that contains additional command line arguments. The following conventions describe some syntax rules for argument files.

- Single Quotes (‘ ’) — Allows you to group arbitrary characters so that no character substitution occurs within the quotes, such as environment variable expansion or escaped characters.

```
+acc=rn+'\mymodule'  
//does not treat the '\' as an escape character
```

- Quotation marks (“ ”) — Allows you to group arbitrary characters so that Tcl-style backslash substitution and environment variable expansion is performed.

```
+acc=rn+"\\mymodule\\$VAR"  
// escapes the path separators (\\) and substitutes
```

```
// your value of '$VAR'
```

- Unquoted — The following are notes on what occurs when some information is not quoted:

- Backslash substitution — Any unquoted backslash ( \ ) will be treated as an escape character.

```
+acc=rn\\mymodule
```

```
// the leading '\\' is considered an escape character
```

- Environment variable expansion — Any unquoted environment variable, such as \$envname, will be expanded. You can also use curly braces ( { } ) in your environment variable, such as \${envname}.

```
+acc=rn\\$MODULE
```

```
// the leading '\\' is considered an escape character and the
```

```
// variable $MODULE is expanded
```

- Newline Character — You can specify arguments on separate lines in the argument file with a backslash ( \ ), which is the line continuation character. You must use a space before the backslash.
- Comments — Comments within the argument files follow these rules:
  - All text in a line beginning with // to its end is treated as a comment.
  - All text bracketed by /\* ... \*/ is treated as a comment.
  - All text in a line beginning with # to its end is treated as a comment.

## Command Shortcuts

The following shortcut techniques are available on the command line.

- You can abbreviate command syntax, but the minimum number of characters required to run a command are those that make it unique. This means the addition of new commands may prevent an older shortcut from working. For this reason, ModelSim does not allow command name abbreviations in macro files. This minimizes your need to update macro files as new commands are added.
- You can enter multiple commands on one line if they are separated by semi-colons (;). For example:

```
ModelSim> vlog -nodebug=ports level3.v level2.v ; vlog -nodebug top.v
```

The return value of the last function executed is the only one printed to the transcript. This may cause some unexpected behavior in certain circumstances. Consider this example:

```
vsim -c -do "run 20 ; simstats ; quit -f" top
```

Although it seems as if the **simstats** results should display in the Transcript window, they do not because the last command is **quit -f**. To see the return values of intermediate commands, you must explicitly print the results. For example:

```
vsim -do "run 20 ; echo [simstats] ; quit -f" -c top
```

## Command History Shortcuts

You can review simulator command history or rerun previous commands by using keyboard shortcuts at the ModelSim/VSIM prompt.

Table 1-6 contains a list of these shortcuts.

**Table 1-6. Keyboard Shortcuts for Command History**

Shortcut	Description
!!	repeats the last command
!n	repeats command number n; n is the VSIM prompt number (for example, for this prompt: VSIM 12>, n =12)
!<string>	shows a list of executed commands that start with <string>; Use the up and down arrows to choose from the list
!abc	repeats the most recent command starting with "abc"
^xyz^ab^	replaces "xyz" in the last command with "ab"
up arrow and down arrow keys	scrolls through the command history
Ctrl-N (UNIX only)	scroll to the next command
Ctrl-P (UNIX only)	scroll to the previous command
click on prompt	left-click once on a previous ModelSim or VSIM prompt in the transcript to copy the command typed at that prompt to the active cursor
his or history	shows the last few commands (up to 50 are kept)

## Numbering Conventions

Numbers in ModelSim can be expressed in either VHDL or Verilog style. You can use two styles for VHDL numbers and one for Verilog.

### VHDL Numbering Conventions

There are two types of VHDL number styles:

#### VHDL Style 1

```
[ - ] [ radix # ] value [ # ]
```

**Table 1-7. VHDL Number Conventions: Style 1**

Element	Description
-	indicates a negative number; optional
radix	can be any base in the range 2 through 16 (2, 8, 10, or 16); by default, numbers are assumed to be decimal; optional
value	specifies the numeric value, expressed in the specified radix; required
#	is a delimiter between the radix and the value; the first # sign is required if a radix is used, the second is always optional

A hyphen (-) can also designate a "don't care" element when you search for a signal value or expression in the List or Wave window. If you want the '-' to be read as a "don't care" element, rather than as a negative sign, be sure to enclose the number in quotation marks. For instance, you would type "-0110--" as opposed to -0110--. If you do not include the quotation marks, ModelSim will read the '-' as a negative sign. For example:

```
16#FFca23#
2#11111110
-23749
```

#### VHDL Style 2

```
base "value"
```

**Table 1-8. VHDL Number Conventions: Style 2**

Element	Description
base	specifies the base; binary: B, octal: O, hex: X; required
"value"	specifies digits in the appropriate base with optional underscore separators; default is decimal; required

For example:

```
B"11111110"
X"FFca23"
```

#### Searching for VHDL Arrays in the Wave and List Windows

Searching for signal values in the Wave or List window may not work correctly for VHDL arrays if the target value is in decimal notation. You may get an error that the value is of incompatible type. Since VHDL does not have a radix indicator for decimal, the target value may get misinterpreted as a scalar value. Prefixing the value with the Verilog notation 'd should eliminate the problem, even if the signal is VHDL.

## Verilog Numbering Conventions

Verilog numbers are expressed in the style:

```
[ - ] [ size ] [ base ] value
```

**Table 1-9. Verilog Number Conventions**

Element	Description
-	indicates a negative number; optional
size	the number of bits in the number; optional
base	specifies the base; binary: 'b or 'B, octal: 'o or 'O, decimal: 'd or 'D, hex: 'h or 'H; optional
value	specifies digits in the appropriate base with optional underscore separators; default is decimal; required

A hyphen (-) can also designate a "don't care" element when you search for a signal value or expression in the List or Wave windows. If you want the '-' to be read as a "don't care" element, rather than a negative sign, be sure to enclose the number in double quotes. For instance, you would type "-0110--" as opposed to 7'b-0110--. If you don't include the double quotes, ModelSim will read the '-' as a negative sign. For example:

```
'b11111110          8'b11111110
'Hffca23          21'H1fca23
-23749
```

## GUI\_expression\_format

---

The GUI\_expression\_format is an option of several simulator commands that operate within the ModelSim GUI environment. The expressions help you locate and examine objects within the List and Wave windows (expressions may also be used through the **Edit > Search** menu in both windows). The commands that use the expression format are:

[configure](#)

[examine](#)

[searchlog](#)

[virtual function](#)

[virtual signal](#)

.

## Expression Typing

GUI expressions are typed. The supported types consist of the following scalar and array types.

### Scalar Types

The scalar types are as follows: boolean, integer, real, time (64-bit integer), enumeration, and signal state. Signal states are represented by the nine VHDL std\_logic states: 'U' 'X' '0' '1' 'Z' 'W' 'L' 'H' and '-'.

Verilog states 0, 1, x, and z are mapped into these states and the Verilog strengths are ignored. Conversion is done automatically when referencing Verilog nets or registers.

### Array Types

The supported array types are signed and unsigned arrays of signal states. This would correspond to the VHDL std\_logic\_array type. Verilog registers are automatically converted to these array types. The array type can be treated as either UNSIGNED or SIGNED, as in the IEEE std\_logic\_arith package. Normally, referencing a signal array causes it to be treated as UNSIGNED by the expression evaluator; to cause it to be treated as SIGNED, use casting as described below. Numeric operations supported on arrays are performed by the expression evaluator via ModelSim's built-in numeric\_standard (and similar) package routines. The expression evaluator selects the appropriate numeric routine based on SIGNED or UNSIGNED properties of the array arguments and the result.

The enumeration types supported are any VHDL enumerated type. Enumeration literals may be used in the expression as long as some variable of that enumeration type is referenced in the expression. This is useful for sub-expressions of the form:

```
(/memory/state == reading)
```

# Expression Syntax

GUI expressions generally follow C-language syntax, with both VHDL-specific and Verilog-specific conventions supported. These expressions are not parsed by the Tcl parser, and so do not support general Tcl; parentheses should be used rather than braces. Procedure calls are not supported.

A GUI expression can include the following elements: Tcl macros, constants, array constants, variables, array variables, signal attributes, operators, and casting.

## Tcl Macros

Macros are useful for pre-defined constants or for entire expressions that have been previously saved. The substitution is done only once, when the expression is first parsed.

Macro syntax is:

`$<name>`

Substitutes the string value of the Tcl global variable `<name>`.

## Constants

**Table 1-10. Constants Supported for GUI Expressions**

Type	Values
boolean value	true false TRUE FALSE
integer	[0-9]+
real number	<int> ([<int>].<int>[exp]) where the optional [exp] is: (e E)[+ -][0-9]+
time	integer or real optionally followed by time unit
enumeration	VHDL user-defined enumeration literal
single bit constants	expressed as any of the following: 0 1 x Z U H L W 'U' 'X' '0' '1' 'Z' 'W' 'L' 'H' '-' 1'b0 1'b1

## Array Constants, Expressed in Any of the Following Formats

**Table 1-11. Array Constants Supported for GUI Expressions**

Type	Values
VHDL # notation	<int>#<alphanum>[#] Example: 16#abc123#

**Table 1-11. Array Constants Supported for GUI Expressions (cont.)**

Type	Values
VHDL bitstring	"(U X 0 1 Z W L H )*" Example: "11010X11"
Verilog notation	[-][<int>](b B o O d D h H) <alphanum> (where <alphanum> includes 0-9, a-f, A-F, and '-') Example: 12'hc91 (This is the preferred notation because it removes the ambiguity about the number of bits.)
Based notation	0x..., 0X..., 0o..., 0O..., 0b..., 0B... ModelSim automatically zero fills unspecified upper bits.

## Variables

**Table 1-12. Variables Supported for GUI Expressions**

Variable	Type
Name of a signal	The name may be a simple name, a VHDL or Verilog style extended identifier, or a VHDL or Verilog style path. The signal must be one of the following types: -- VHDL signal of type INTEGER, REAL, or TIME -- VHDL signal of type std_logic or bit -- VHDL signal of type user-defined enumeration -- Verilog net, Verilog register, Verilog integer, or Verilog real
NOW	Returns the value of time at the current location in the WLF file as the WLF file is being scanned (not the most recent simulation time).

## Array variables

**Table 1-13. Array Variables Supported for GUI Expressions**

Variable	Type
Name of a signal	-- VHDL signals of type bit_vector or std_logic_vector -- Verilog register -- Verilog net array  A subrange or index may be specified in either VHDL or Verilog syntax. Examples: mysignal(1 to 5), mysignal[1:5], mysignal (4), mysignal [4]

## Signal attributes

```
<name>'event
<name>'rising
<name>'falling
<name>'delayed()
<name>'hasX
```

The 'delayed attribute lets you assign a delay to a VHDL signal. To assign a delay to a signal in Verilog, use “#” notation in a sub-expression (e.g., #-10 /top/*signalA*).

The hasX attribute lets you search for signals, nets, or registers that contains an X (unknown) value.

See [Examples of Expression Syntax](#) below for further details on 'delayed and 'hasX.

## Operators

**Table 1-14. Operators Supported for GUI Expressions**

Operator	Description	Kind
+	arithmetic add	arithmetic
/	arithmetic divide	arithmetic
mod/MOD	arithmetic modulus	arithmetic
*	arithmetic multiply	arithmetic
rem/REM	arithmetic remainder	arithmetic
-	arithmetic subtract	arithmetic
&	concat	arithmetic
<name>'delayed(<time>)	delayed signal (<time>)	attributes
<name>'falling	Falling edge	attributes
<name>'rising	Rising edge	attributes
<name>'event	Value change	attributes
<name>'hasX	Value has an X	attributes
and, AND	bitwise and	bitwise logical
nand, NAND	bitwise nand	bitwise logical
nor, NOR	bitwise nor	bitwise logical
or, OR	bitwise or	bitwise logical
xnor, XNOR	bitwise xnor	bitwise logical

**Table 1-14. Operators Supported for GUI Expressions (cont.)**

Operator	Description	Kind
xor, XOR	bitwise xor	bitwise logical
rol, ROL	rotate left	bitwise logical
ror, ROR	rotate right	bitwise logical
sla, SLA	shift left arithmetic	bitwise logical
sll, SLL	shift left logical	bitwise logical
sra, SRA	shift right arithmetic	bitwise logical
srl, SRL	shift right logical	bitwise logical
not, NOT, ~	unary bitwise inversion	bitwise logical
&&	boolean and	boolean
!	boolean not	boolean
	boolean or	boolean
==	equal	boolean
====	exact equal <sup>1</sup>	boolean
!=	exact not equal	boolean
>	greater than	boolean
>=	greater than or equal	boolean
<	less than	boolean
<=	less than or equal	boolean
!=, /=	not equal	boolean
&<vector_expr>	AND reduction	reduction
<vector_expr>	OR reduction	reduction
^<vector_expr>	XOR reduction	reduction

1. This operator is allowed to be compatible with other simulators.

**Table 1-15. Precedence of GUI Expression Operators**

Operator	Kind
delayed(), 'falling, 'rising, 'event, 'hasX	attributes
&,  , ^	unary
!, not, NOT, ~	boolean

**Table 1-15. Precedence of GUI Expression Operators (cont.)**

Operator	Kind
/, mod, MOD, *, rem, REM	arithmetic
nand, NAND, nor, NOR	bitwise logical
and, AND	bitwise logical
xor, XOR, xnor, XNOR	bitwise logical
or, OR	bitwise logical
+, -	arithmetic
&	concat
rol, ROL, ror, ROR, sla, SLA, sll, SLL, sra, SRA, srl, SRL	bitwise logical
>, >=, <, <=	boolean
==, ===, !=, !=, /=	boolean
&&	boolean
	boolean

**Note**

 Arithmetic operators use the std\_logic\_arith package.

## Casting

**Table 1-16. Casting Conversions Supported for GUI Expressions**

Casting	Description
(bool)	convert to boolean
(boolean)	convert to boolean
(int)	convert to integer
(integer)	convert to integer
(real)	convert to real
(time)	convert to 64-bit integer
(std_logic)	convert to 9-state signal value
(signed)	convert to signed vector
(unsigned)	convert to unsigned vector
(std_logic_vector)	convert to unsigned vector

## Examples of Expression Syntax

```
/top/bus & $bit_mask
```

This expression takes the bitwise AND function of signal */top/bus* and the array constant contained in the global Tcl variable *bit\_mask*.

```
clk'event && (/top/xyz == 16'hffae)
```

This expression evaluates to a boolean true when signal *clk* changes and signal */top/xyz* is equal to hex ffae; otherwise is false.

```
clk'rising && (mystate == reading) && (/top/u3/addr == 32'habcd1234)
```

Evaluates to a boolean true when signal *clk* just changed from low to high and signal *mystate* is the enumeration *reading* and signal */top/u3/addr* is equal to the specified 32-bit hex constant; otherwise is false.

```
(/top/u3/addr and 32'hff000000) == 32'hac000000
```

Evaluates to a boolean true when the upper 8 bits of the 32-bit signal */top/u3/addr* equals hex ac.

```
/top/signalA'delayed(10ns)
```

This expression returns */top/signalA* delayed by 10 ns.

```
/top/signalA'delayed(10 ns) && /top/signalB
```

This expression takes the logical AND of a delayed */top/signalA* with */top/signalB*.

```
virtual function { (#-10 /top/signalA) && /top/signalB}
mySignalB_AND_DelayedSignalA
```

This evaluates */top/signalA* at 10 simulation time steps before the current time, and takes the logical AND of the result with the current value of */top/signalB*. The '#' notation uses positive numbers for looking into the future, and negative numbers for delay. This notation does not support the use of time units.

```
((NOW > 23 us) && (NOW < 54 us)) && clk'rising && (mode == writing)
```

Evaluates to a boolean true when WLF file time is between 23 and 54 microseconds, *clk* just changed from low to high, and signal mode is enumeration writing.

```
searchlog -expr {dbus'hasX} {0 ns} dbus
```

Searches for an 'X' in *dbus*. This is equivalent to the expression: *dbus(0) == 'x' // dbus(1) == 'x'* . . . . This makes it possible to search for X values without having to write a type specific literal.

## Signal and Subelement Naming Conventions

ModelSim supports naming conventions for VHDL and Verilog signal pathnames, VHDL array indexing, Verilog bit selection, VHDL subrange specification, and Verilog part selection.

Examples in Verilog and VHDL syntax:

```
top.chip.vlogsig
/top/chip/vhdsig
vlogsig[3]
vhdsig(9)
vlogsig[5:2]
vhdsig(5 downto 2)
```

## Grouping and Precedence

Operator precedence generally follows that of the C language, but liberal use of parentheses is recommended.

## Concatenation of Signals or Subelements

Elements in the concatenation that are arrays are expanded so that each element in the array becomes a top-level element of the concatenation. But for elements in the concatenation that are records, the entire record becomes one top-level element in the result.

To specify that the records be broken down so that their subelements become top-level elements in the concatenation, use the **concat\_flatten** directive. Currently, leaving full arrays as elements in the result is not supported. (Please contact Mentor Graphics if you need that option.)

If the elements being concatenated are of incompatible base types, a VHDL-style record will be created. The record object can be expanded in the Objects and Wave windows just like an array of compatible type elements.

### Concatenation Syntax for VHDL

```
<signalOrSliceName1> & <signalOrSliceName2> & . . .
```

### Concatenation Syntax for Verilog

```
&{<signalOrSliceName1>, <signalOrSliceName2>, . . . }
&{<count>{<signalOrSliceName1>}, <signalOrSliceName2>, . . . }
```

Note that the concatenation syntax begins with "&{" rather than just "{". Repetition multipliers are supported, as illustrated in the second line. The repetition element itself may be an arbitrary concatenation subexpression.

### Concatenation Directives

A concatenation directive (as illustrated below) can be used to constrain the resulting array range of a concatenation or influence how compound objects are treated. By default, ModelSim will create the concatenation with a descending index range from  $(n-1)$  down to 0, where  $n$  is the number of elements in the array.

```
(concat_range 31:0)<concatenationExpr> # Verilog syntax  
(concat_range (31:0))<concatenationExpr> # Also Verilog syntax  
(concat_range (31 downto 0))<concatenationExpr> # VHDL syntax
```

The **concat\_range** directive completely specifies the index range.

```
(concatAscending) <concatenationExpr>
```

The **concatAscending** directive specifies that the index start at zero and increment upwards.

```
(concatFlatten) <concatenationExpr>
```

The **concatFlatten** directive flattens the signal structure hierarchy.

```
(concatNoFlatten) <concatenationExpr>
```

The **concatNoFlatten** directive groups signals together without merging them into one big array. The signals become elements of a record and retain their original names. When expanded, the new signal looks just like a group of signals. The directive can be used hierarchically with no limits on depth.

```
(concatSortWildAscending) <concatenationExpr>
```

The **concatSortWildAscending** directive gathers signals by name in ascending order (the default is descending).

```
(concatReverse) <concatenationExpr>
```

The **concatReverse** directive reverses the bits of the concatenated signals.

### Examples of Concatenation

```
&{ "mybusbasename*" }
```

Gathers all signals in the current context whose names begin with "mybusbasename", sorts those names in descending order, and creates a bus with index range  $(n-1)$  downto 0, where  $n$  is the number of matching signals found. (Note that it currently does not derive the index name from the tail of the one-bit signal name.)

```
(concat_range 13:4) &{ "mybusbasename*" }
```

Specifies the index range to be 13 downto 4, with the signals gathered by name in descending order.

```
(concat_descending) &{ "mybusbasename*" }
```

Specifies an ascending range of 0 to  $n-1$ , with the signals gathered by name in descending order.

```
(concat_descending) ((concat_sort_wild_descending) &{ "mybusbasename*" })
```

Specifies an ascending range of 0 to  $n-1$ , with the signals gathered by name in ascending order.

```
(concat_reverse) (bus1 & bus2)
```

Specifies that the bits of bus1 and bus2 be reversed in the output virtual signal.

## Record Field Members

Arbitrarily-nested arrays and records are supported, but operators will only operate on one field at a time. That is, the expression  $\{a == b\}$  where  $a$  and  $b$  are records with multiple fields, is not supported.

This would have to be expressed as:

```
{(a.f1 == b.f1) && (a.f2 == b.f2) ...}
```

Examples:

```
vhdsig.field1
vhdsig.field1.subfield1
vhdsig.(5).field3
vhdsig.field4(3 downto 0)
```

## Searching for Binary Signal Values in the GUI

When you use the GUI to search for signal values displayed in 4-state binary radix, you should be aware of how ModelSim maps between binary radix and std\_logic. The issue arises because there is no “un-initialized” value in binary, while there is in std\_logic. So, ModelSim relies on

mapping tables to determine whether a match occurs between the displayed binary signal value and the underlying std\_logic value.

This matching algorithm applies only to searching using the GUI. It does not apply to VHDL or Verilog test benches.

For comparing VHDL std\_logic/std\_ulogic objects, ModelSim uses the table shown below. An entry of “0” in the table is “no match”; an entry of “1” is a “match”; an entry of “2” is a match only if you set the Tcl variable **STDLOGIC\_X\_MatchesAnything** to 1. Note that X will match a *U*, and - will match anything.

**Table 1-17. VHDL Logic Values Used in GUI Search**

Search Entry	Matches as follows:								
	U	X	0	1	Z	W	L	H	-
U	1	1	0	0	0	0	0	0	1
X	1	1	2	2	2	2	2	2	1
0	0	2	1	0	0	0	1	0	1
1	0	2	0	1	0	0	0	1	1
Z	0	2	0	0	1	0	0	0	1
W	0	2	0	0	0	1	0	0	1
L	0	2	1	0	0	0	1	0	1
H	0	2	0	1	0	0	0	1	1
-	1	1	1	1	1	1	1	1	1

For comparing Verilog net values, ModelSim uses the table shown below. An entry of “2” is a match only if you set the Tcl variable “VLOG\_X\_MatchesAnything” to 1.

**Table 1-18. Verilog Logic Values Used in GUI Search**

Search Entry	Matches as follows:			
	0	1	Z	X
0	1	0	0	2
1	0	1	0	2
Z	0	0	1	2
X	2	2	2	1

# Chapter 2

## Commands

---

This chapter describes ModelSim commands that you can enter either on the command line of the Main window or in a DO file. Some commands are automatically entered on the command line when you use the graphical user interface.

Note that, in addition to the simulation commands listed in this chapter, you can also use the Tcl commands described in the Tcl man pages (use the Main window menu selection: **Help > Tcl > Man Pages**).

**Table 2-1** provides a brief description of each ModelSim command and whether the command is supported for use in batch simulation mode (**vsim -batch**), and/or command-line mode (**vsim -c**). Refer to [General Modes of Operation](#) for more information about batch and command-line simulation. For more information on command details, arguments, and examples, click the link in the Command name column.

**Table 2-1. Supported Commands**

Command name	Action	-batch	-c
<a href="#">abort</a>	This command halts the execution of a DO file interrupted by a breakpoint or error.	Y	Y
<a href="#">add dataflow</a>	This command adds the specified process, signal, net, or register to the Dataflow window. Wildcards are allowed.	N	N
<a href="#">add list</a>	This command adds the following objects and their values to the List window:	Y	Y
<a href="#">add log</a>	also known as the <b>log</b> command; see “ <a href="#">log</a> ” on page 199	Y	Y
<a href="#">add memory</a>	This command displays the contents and sets the address and data radix of the specified memory in the MDI frame of the Main window.	N	N
<a href="#">add message</a>	This command is used within a DO file or script and specifies a user defined runtime message that is sent to the transcript and .wlffiles. Messages are displayed in the Message Viewer window in the GUI. Refer to “ <a href="#">Message Viewer Window</a> for information.	Y	Y
<a href="#">add watch</a>	This command adds signals and variables to the Watch window in the Main window.	N	N

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<code>add wave</code>	This command adds the following objects to the Wave window:	Y	Y
<code>add_cmdhelp</code>	This command adds the specified command name, description, and command arguments to the command-line help. You can then access the information using the help command.	N	Y
<code>alias</code>	This command displays or creates user-defined aliases. Any arguments passed on invocation of the alias will be passed through to the specified commands.	Y	N
<code>archive load</code>	The archive load command allows you to load an archived debug database (.dbar) file that was previously created with the archive write command. The archived file may include a number of WLF files, design source files, and a DBG file.	N	N
<code>archive write</code>	The archive write command allows you to create a debug archive file, with the file extension .dbar, that contains one or more WLF files, debug information captured from the design library, an optional connectivity debug database file, and optional HDL source files. With this archived file, you can perform post-simulation debugging in different location from that which the original simulation was run.	N	N
<code>batch_mode</code>	This command returns “1” if Questa SIM is operating in batch mode, otherwise it returns “0.” It is typically used as a condition in an if statement.	Y	Y
<code>bd</code>	This command deletes a breakpoint. You can delete multiple breakpoints by specifying separate information groupings on the same command line.	Y	Y
<code>bookmark add wave</code>	This command creates a named reference to a specific zoom range and scroll position in the specified Wave window. Bookmarks are saved in the wave format file and are restored when the format file is read.	N	N
<code>bookmark delete wave</code>	This command deletes bookmarks from the specified Wave window.	N	N

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">bookmark goto wave</a>	This command zooms and scrolls a Wave window using the specified bookmark.	N	N
<a href="#">bookmark list wave</a>	This command displays a list of available bookmarks in the Transcript window.	N	N
<a href="#">bp</a>	This command sets either a file-line breakpoint or returns a list of currently set breakpoints. It allows enum names, as well as literal values, to be used in condition expressions.	Y	Y
<a href="#">call</a>	This command calls the following types of functions/tasks.	Y	Y
<a href="#">change</a>	This command modifies the value of a: VHDL constant, generic, or variable; Verilog register or variable; or C variable if running C Debug.	Y	Y
<a href="#">classinfo ancestry</a>	This command returns class inheritance hierarchy for a named class type.	Y	Y
<a href="#">classinfo descriptive</a>	This command returns the descriptive class name for the specified authoritative class name.	Y	Y
<a href="#">classinfo find</a>	This command reports on the current state of a specified class instance, whether it exists, has not yet been created, or has been destroyed.	Y	Y
<a href="#">classinfo implements</a>	This command displays a list of which classes implement SystemVerilog interface classes. The type of the class argument affect the contents of this list.	Y	Y
<a href="#">classinfo instances</a>	This command reports the list of existing class instances of a specific class type. You can use this to determine what class instances to log or examine. It can also help in debugging problems where class instances are not being cleaned up as they should be resulting in excessive memory usage.	Y	Y
<a href="#">classinfo interfaces</a>	This command lists the interface class types that match or do not match a specified pattern. Finds all interface classes that match a regular expression and determines the full path of interface class types.	Y	Y
<a href="#">classinfo isa</a>	This command returns to the transcript a list of all classes extended from the specified class type.	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">classinfo report</a>	This command prints detailed reports on class instance usage. The command displays columns for class type names and their current, peak and total class instance counts. The columns may be arranged, sorted, or eliminated using the command arguments.	Y	Y
<a href="#">classinfo stats</a>	This command prints statistics about the total number of class types and total, peak, and current class instance counts during the simulation.	Y	Y
<a href="#">classinfo trace</a>	This command displays the active references to the specified class instance. This is very useful in debugging situations where class instances are not being destroyed as expected because something in the design is still referencing them. Finding those references may lead to uncovering bugs in managing these class references which often lead to large memory savings.	Y	Y
<a href="#">classinfo types</a>	This command lists the class types that match or do not match a specified pattern. Finds all classes that match a regular expression and determines the full path of class types.	Y	Y
<a href="#">configure</a>	The configure command invokes the List or Wave widget configure command for the current default List or Wave window.	N	Y
<a href="#">dataset alias</a>	This command maps an alternate name (alias) to an open dataset. A dataset can have any number of aliases, but all dataset names and aliases must be unique even when more than one dataset is open. Aliases are not saved to the .wlf file and must be remapped if the dataset is closed and then re-opened.	N	Y
<a href="#">dataset clear</a>	All event data is removed from the current simulation WLF file, while retaining all currently logged signals. Subsequent run commands will continue to accumulate data in the WLF file.	N	Y
<a href="#">dataset close</a>	This command closes an active dataset. To open a dataset, use the dataset open command.	N	Y
<a href="#">dataset config</a>	This command configures WLF parameters for an open dataset and all aliases mapped to that dataset.	N	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">dataset current</a>	This command activates the specified dataset and sets the GUI context to the last selected context of the specified dataset. All context dependent GUI data is updated and all context dependent CLI commands start working with respect to the new context.	N	Y
<a href="#">dataset info</a>	This command reports a variety of information about a dataset. Arguments to this command are order dependent. Please read through the argument descriptions for more information.	N	Y
<a href="#">dataset list</a>	This command lists all active datasets.	N	Y
<a href="#">dataset open</a>	This command opens a WLF file (either the currently running vsim.wlf or a saved WLF file) and/or UCDB file (representing coverage data) and assigns it the logical name that you specify.	N	Y
<a href="#">dataset rename</a>	This command changes the name of a dataset to the new name you specify. Arguments to this command are order dependent. Follow the order specified in the Syntax section.	N	Y
<a href="#">dataset restart</a>	This command unloads the specified dataset or currently active dataset and reloads the dataset using the same dataset name. The contents of Wave and other coverage windows are restored for UCDB datasets after a reload.	N	Y
<a href="#">dataset save</a>	This command writes data from the current simulation to the specified file. This lets you save simulation data while the simulation is still in progress.	N	Y
<a href="#">dataset snapshot</a>	This command saves data from the current WLF file (vsim.wlf by default) at a specified interval. It provides you with sequential or cumulative "snapshots" of your simulation data.	N	Y
<a href="#">delete</a>	This command removes objects from either the List or Wave window. Arguments to this command are order dependent.	N	Y
<a href="#">describe</a>	This command displays information about simulation objects and design regions in the Transcript window.	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">disablebp</a>	This command turns off breakpoints and when commands. To turn on breakpoints or when commands again, use the enablebp command.	Y	Y
<a href="#">do</a>	This command executes the commands contained in a DO file.	Y	Y
<a href="#">drivers</a>	This command displays the names and strength of all drivers of the specified object.	Y	Y
<a href="#">dumplog64</a>	This command dumps the contents of the specified WLF file in a readable format to stdout. The WLF file cannot be opened for writing in a simulation when you use this command. This command cannot be used in a DO file.		
<a href="#">echo</a>	This command displays a specified message in the Transcript window.	Y	Y
<a href="#">edit</a>	This command invokes the editor specified by the EDITOR environment variable. By default, the specified filename will open in the Source window.	N	N
<a href="#">enablebp</a>	This command turns on breakpoints and when commands that were previously disabled.	Y	Y
<a href="#">encoding</a>	This command translates between the 16-bit Unicode characters used in Tcl strings and a named encoding, such as Shift-JIS.	Y	Y
<a href="#">environment</a>	This command has two forms, environment and env. It allows you to display or change the current dataset and region/signal environment.	Y	Y
<a href="#">examine</a>	This command has two forms, examine and exa. It examines one or more objects and displays current values (or the values at a specified previous time) in the Transcript window.	Y	Y
<a href="#">exit</a>	This command exits the simulator and the ModelSim application.	Y	Y
<a href="#">find</a>	This command locates objects by type and name. Arguments to the command are grouped by object type.	N	Y
<a href="#">find connections</a>	This command returns the set of nets that are electrically equivalent to a specified net. It is only available during a live simulation.	N	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">find infiles</a>	This command searches for a string in the specified file(s) and prints the results to the Transcript window. The results are individually hotlinked and will open the file and display the location of the string.	Y	Y
<a href="#">find insource</a>	This command searches for a string in the source files for the current design and prints the results to the Transcript window. The results are hotlinked individually and will open the file and display the location of the string. When you execute this command in command-line mode from outside of the GUI, the results are sent to stdout with no hotlinks.	Y	Y
<a href="#">force</a>	This command allows you to apply stimulus interactively to VHDL signals, Verilog nets and registers.	Y	Y
<a href="#">formatTime</a>	This command provides global format control for all time values displayed in the GUI. When specified without arguments, this command returns the current state of the three arguments.	Y	Y
<a href="#">gc configure</a>	This command specifies when the System Verilog garbage collector will run. The garbage collector may be configured to run after a memory threshold has been reached, after each simulation run command completes, and/or after each simulation step command. The default settings are optimized to balance performance and memory usage for either regular simulation or class debugging (vsim -classdebug). Returns the current settings when specified without arguments.	N	Y
<a href="#">gc run</a>	This command runs the SystemVerilog garbage collector.	N	Y
<a href="#">help</a>	This command displays in the Transcript window a brief description and syntax for the specified command.	N	Y
<a href="#">history</a>	This command lists the commands you have executed during the current session. History is a Tcl command. For more information, consult the Tcl Man Pages (Help > Tcl Man Pages).	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">layout</a>	This command allows you to perform a number of editing operations on custom GUI layouts, such as loading, saving, maximizing, and deleting.	N	N
<a href="#">log</a>	This command creates a wave log format (WLF) file containing simulation data for all HDL objects whose names match the provided specifications. Objects that are displayed using the add list and add wave commands are automatically recorded in the WLF file. By default the file is named vsim.wlf and stored in the current working directory. You can change the default name using the vsim -wlf option of the vsim command or by setting the WLFFilename variable in the modelsim.ini file.	Y	Y
<a href="#">lshift</a>	This command takes a Tcl list as an argument and shifts it in-place, one place to the left, eliminating the left-most element.	Y	Y
<a href="#">lsublist</a>	This command returns a sublist of the specified Tcl list that matches the specified Tcl glob pattern. Arguments to this command are order dependent. Follow the order specified in the Syntax section.	Y	Y
<a href="#">mem compare</a>	This command compares a selected memory to a reference memory or file. Must have the "diff" utility installed and visible in your search path in order to run this command. Arguments to this command are order dependent. Please read through the argument descriptions for more information.	Y	Y
<a href="#">mem display</a>	This command prints to the Transcript window the memory contents of the specified instance. If the given instance path contains only a single array signal or variable, the signal or variable name need not be specified.	Y	Y
<a href="#">mem list</a>	This command displays a flattened list of all memory instances in the current or specified context after a design has been elaborated.	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">mem load</a>	This command updates the simulation memory contents of a specified instance. You can upload contents either from a memory data file, a memory pattern, or both. If both are specified, the pattern is applied only to memory locations not contained in the file.	Y	Y
<a href="#">mem save</a>	This command saves the contents of a memory instance to a file in any of the supported formats: Verilog binary, Verilog hex, and MTI memory pattern data.	Y	Y
<a href="#">mem search</a>	This command finds and prints to the screen the first occurring match of a specified memory pattern in the specified memory instance. Shorthand instance names are accepted. Optionally, you can instruct the command to print all occurrences. The search pattern can be one word or a sequence of words.	Y	Y
<a href="#">modelsim</a>	The modelsim command starts the ModelSim GUI without prompting you to load a design.	?	?
<a href="#">noforce</a>	This command removes the effect of any active force commands on the selected HDL objects. and also causes the object's value to be re-evaluated.	Y	Y
<a href="#">nolog</a>	This command suspends writing of data to the wave log format (WLF) file for the specified signals.	Y	Y
<a href="#">notepad</a>	This command opens a simple text editor. It may be used to view and edit ASCII files or create new files.	N	N
<a href="#">noview</a>	This command closes a window in the ModelSim GUI. To open a window, use the view command.	N	N
<a href="#">nowhen</a>	This command deactivates selected when commands.	Y	Y
<a href="#">onbreak</a>	This command is used within a DO file and specifies one or more scripts to be executed when running a script that encounters a breakpoint in the source code.	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">onElabError</a>	This command specifies one or more commands to be executed when an error is encountered during the elaboration portion of a vsim command. The command is used by placing it within a DO file script. Use the onElabError command without arguments to return to a prompt.	Y	Y
<a href="#">onerror</a>	This command is used within a DO file script before a run command; it specifies one or more commands to be executed when a running script encounters an error.	Y	Y
<a href="#">onfinish</a>	This command controls simulator behavior when encountering \$finish or sc_stop() in the design code. When you specify this command without an argument, it returns the current setting.	Y	Y
<a href="#">pause</a>	This command interrupts the execution of a macro and allows you to perform interactive debugging of a macro file. The command is placed within the macro to be debugged.	Y	Y
<a href="#">precision</a>	This command determines how real numbers display in the graphic interface (e.g., Objects, Wave, Locals, and List windows). It does not affect the internal representation of a real number and therefore precision values over 17 are not allowed. Executing the precision command without any arguments returns the current precision setting.	Y	Y
<a href="#">printenv</a>	This command prints to the Transcript window the current names and values of all environment variables. If variable names are given as arguments, returns only the names and values of the specified variables.	Y	Y
<a href="#">process report</a>	This command creates a textual report of all processes displayed in the Process Window.	Y	Y
<a href="#">project</a>	This command is used to perform common operations on projects.	N	Y
<a href="#">pwd</a>	This Tcl command displays the current directory path in the Transcript window.	Y	Y
<a href="#">quietly</a>	This command turns off transcript echoing for the specified command.	Y	Y
<a href="#">quit</a>	This command exits the simulator.	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<code>radix</code>	This command specifies the default radix to be used for the current simulation. Specifying the command with no argument returns the current radix setting.	Y	Y
<code>radix define</code>	This command is used to create or modify a user-defined radix. A user definable radix is used to map bit patterns to a set of enumeration labels or setup a fixed or floating point radix. User-defined radices are available for use in the most windows and with the examine command.	Y	Y
<code>radix delete</code>	This command will remove the radix definition from the named radix.	Y	Y
<code>radix list</code>	This command will return the complete definition of a radix, if a name is given. If no name is given, it will list all the defined radices.	Y	Y
<code>radix names</code>	This command returns a list of currently defined radix names.	Y	Y
<code>radix signal</code>	This command sets or inspects radix values for the specified signal in the Objects, Locals, Schematic, and Wave windows. When no argument is used, the radix signal command returns a list of all signals with a radix.	Y	Y
<code>report</code>	This command displays information relevant to the current simulation.	Y	Y
<code>restart</code>	This command reloads the design elements and resets the simulation time to zero. Only design elements that have changed are reloaded. (Note that SDF files are always reread during a restart.)	Y	Y
<code>resume</code>	This command is used to resume execution of a macro (DO) file after a pause command or a breakpoint.	Y	Y
<code>run</code>	This command advances the simulation by the specified number of timesteps.	Y	Y
<code>runStatus</code>	This command returns the current state of your simulation to stdout after issuing a run or step command.	Y	Y
<code>searchlog</code>	This command searches one or more of the currently open logfiles for a specified condition.	N	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">see</a>	This command displays the specified number of source file lines around the current execution line and places a marker to indicate the current execution line. If specified without arguments, five lines will be displayed before and four lines after.	Y	Y
<a href="#">setenv</a>	This command changes or reports the current value of an environment variable. The setting is valid only for the current ModelSim session. Arguments to this command are order dependent. Please read the argument descriptions for more information.	Y	Y
<a href="#">shift</a>	This command shifts macro parameter values left one place, so that the value of parameter \\$2 is assigned to parameter \\$1, the value of parameter \\$3 is assigned to \\$2, and so on. The previous value of \\$1 is discarded.	Y	Y
<a href="#">show</a>	This command lists HDL objects and subregions visible from the current environment.	Y	Y
<a href="#">simstats</a>	This command returns performance-related statistics about elaboration and simulation. The statistics measure the simulation kernel process (vsimk) for a single invocation of vsim. If you invoke vsim a second time, or restart the simulation, the current statistics are discarded and new values are collected.	Y	Y
<a href="#">simstatslist</a>	This command returns performance-related statistics about elaboration and simulation. Use this command in place of the simstats command to produce the original statistics output format as a list instead of on separate lines.	Y	Y
<a href="#">stack down</a>	This command moves down the call stack.	Y	Y
<a href="#">stack frame</a>	This command selects the specified call frame.	Y	Y
<a href="#">stack level</a>	This command reports the current call frame number.	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<code>stack tb</code>	This command displays a stack trace for the current process in the Transcript window. This lists the sequence of HDL function calls that have been entered to arrive at the current state for the active process. The <code>stack tb</code> command is an alias for the <code>tb</code> command.	Y	Y
<code>stack up</code>	This command moves up the call stack.	Y	Y
<code>status</code>	This command lists summary information about currently interrupted macros.	Y	Y
<code>step</code>	The <code>step</code> command is an alias for the <code>run</code> command with the <code>-step</code> switch. Steps the simulator to the next HDL.	Y	Y
<code>stop</code>	This command is used with the <code>when</code> command to stop simulation in batch files. The <code>stop</code> command has the same effect as hitting a breakpoint. The <code>stop</code> command may be placed anywhere within the body of the <code>when</code> command.	Y	Y
<code>suppress</code>	This command prevents one or more specified messages from displaying. You cannot suppress Fatal or Internal messages. The <code>suppress</code> command used without arguments returns the message numbers of all suppressed messages.	Y	Y
<code>tb</code>	This (traceback) command (traceback) displays a stack trace for the current process in the Transcript window. This lists the sequence of HDL function calls that have been entered to arrive at the current state for the active process.	Y	Y
<code>Time</code>	These commands allow you to perform comparisons between, operations on, and conversions of, time values.	Y	Y
<code>transcript</code>	This command controls echoing of commands executed in a macro file. If no option is specified, the current setting is reported.	Y <sup>1</sup>	Y
<code>transcript file</code>	This command sets or queries the current <code>PrefMain(file)</code> Tcl preference variable. You can use this command to clear a transcript in batch mode or to limit the size of a transcript file. It offers an alternative to setting the <code>PrefMain(file)</code> Tcl preference variable through the GUI.	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">transcript path</a>	This command returns the full pathname to the current transcript file.	Y	Y
<a href="#">transcript sizelimit</a>	This command sets or queries the current preference value for the transcript fileSizeLimit value. If the size limit is reached, the transcript file is saved and simulation continues.	Y	Y
<a href="#">tssi2mti</a>	This command is used to convert a vector file in TSSI Format into a sequence of force and run commands.	N	Y
<a href="#">unsetenv</a>	This command deletes an environment variable. The deletion is not permanent – it is valid only for the current ModelSim session.	Y	Y
<a href="#">ui_VVMode</a>	This command specifies behavior when encountering UI registration calls used by verification packages, such as AVM or OVM. Returns the current setting when specifies without an argument.	Y	Y
<a href="#">vcd add</a>	This command adds the specified objects to a VCD file.	Y	Y
<a href="#">vcd checkpoint</a>	This command dumps the current values of all VCD variables to the specified VCD file. While simulating, only value changes are dumped. Related Verilog tasks: \$dumpall, \$fdumpall	Y	Y
<a href="#">vcd comment</a>	This command inserts the specified comment in the specified VCD file. Arguments to this command are order dependent. Please read the argument descriptions for more information.	Y	Y
<a href="#">vcd dumpports</a>	This command creates a VCD file that includes port driver data.	Y	Y
<a href="#">vcd dumpportsall</a>	This command creates a checkpoint in the VCD file which shows the value of all selected ports at that time in the simulation, regardless of whether the port values have changed since the last timestep. Related Verilog task: \$dumpportsall	Y	Y
<a href="#">vcd dumpportsflush</a>	This command flushes the contents of the VCD file buffer to the specified VCD file. Related Verilog task: \$dumpportsflush	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">vcd dumpportslimit</a>	This command specifies the maximum size of the VCD file (by default, limited to available disk space). When the size of the file exceeds the limit, a comment is appended to the file and VCD dumping is disabled.	Y	Y
<a href="#">vcd dumpportoff</a>	This command turns off VCD dumping and records all dumped port values as x.	Y	Y
<a href="#">vcd dumpporton</a>	This command turns on VCD dumping and records the current values of all selected ports. This command is typically used to resume dumping after invoking vcd dumpportoff. Related Verilog task: \$dumpporton	Y	Y
<a href="#">vcd file</a>	This command specifies the filename and state mapping for the VCD file created by a vcd add command. The vcd file command is optional. If used, it must be issued before any vcd add commands.	Y	Y
<a href="#">vcd files</a>	This command specifies filenames and state mapping for VCD files created by the vcd add command. The vcd files command is optional. If used, it must be issued before any vcd add commands. Related Verilog task: \$fdumpfile	Y	Y
<a href="#">vcd flush</a>	This command flushes the contents of the VCD file buffer to the specified VCD file. This command is useful if you want to create a complete VCD file without ending your current simulation. Related Verilog tasks: \$dumpflush, \$fdumpflush	Y	Y
<a href="#">vcd limit</a>	This command specifies the maximum size of a VCD file (by default, limited to available disk space).	Y	Y
<a href="#">vcd off</a>	This command turns off VCD dumping to the specified file and records all VCD variable values as x. Related Verilog tasks: \$dumpoff, \$fdumpoff	Y	Y
<a href="#">vcd on</a>	This command turns on VCD dumping to the specified file and records the current values of all VCD variables.	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">vcd2wlf</a>	This command is a utility that translates a VCD (Value Change Dump) file into a WLF file that you can display in ModelSim using the vsim -view argument. This command only works on VCD files containing positive time values.	Y	Y
<a href="#">vcom</a>	The vcom command compiles VHDL source code into a specified working library (or to the work library by default).	Y	Y
<a href="#">vdel</a>	This command deletes a design unit from a specified library. This command provides additional information with the -help switch.	Y	Y
<a href="#">vdir</a>	This command lists the contents of a design library and checks the compatibility of a vendor library. If vdir cannot read a vendor-supplied library, the library may not be compatible with ModelSim.	Y	Y
<a href="#">vencrypt</a>	This command encrypts Verilog and SystemVerilog code contained within encryption envelopes. The code is not pre-processed before encryption, so macros and other `directives are unchanged. This allows IP vendors to deliver encrypted IP with undefined macros and `directives.	Y	Y
<a href="#">verror</a>	This command prints a detailed description about a message number. It may also point to additional documentation related to the error. This command provides additional information with the -help or -h switch.	Y	Y
<a href="#">vgencomp</a>	Once a Verilog module is compiled into a library, you can use this command to write its equivalent VHDL component declaration to standard output.	Y	Y
<a href="#">vhencrypt</a>	This command encrypts VHDL code contained within encryption envelopes. The code is not compiled before encryption, so dependent packages and design units do not have to exist before encryption.	Y	Y
<a href="#">view</a>	This command opens the specified window. If you specify this command without arguments it returns a list of all open windows in the current layout.	N	N

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">virtual count</a>	This command reports the number of currently defined virtuals that were not read in using a macro file.	N	Y
<a href="#">virtual define</a>	This command prints to the transcript the definition of the virtual signals, functions, or regions in the form of a command that can be used to re-create the object.	N	Y
<a href="#">virtual delete</a>	This command removes the matching virtuals.	N	Y
<a href="#">virtual describe</a>	This command prints to the transcript a complete description of the data type of one or more virtual signals. Similar to the existing describe command.	N	Y
<a href="#">virtual expand</a>	This command prints to the transcript a list of all the non-virtual objects contained in the specified virtual signal(s). You can use this to create a list of arguments for a command that does not accept or understand virtual signals.	N	Y
<a href="#">virtual function</a>	This command creates a new signal, known only by the GUI (not the kernel), that consists of logical operations on existing signals and simulation time, as described in <expressionString>.	N	Y
<a href="#">virtual hide</a>	This command causes the specified real or virtual signals to not be displayed in the Objects window. This is used when you want to replace an expanded bus with a user-defined bus. You make the signals reappear using the virtual nohide command.	N	Y
<a href="#">virtual log</a>	This command causes the simulation-mode dependent signals of the specified virtual signals to be logged by the kernel. If wildcard patterns are used, it will also log any normal signals found, unless the -only option is used. You unlog the signals using the virtual nolog command.	N	Y
<a href="#">virtual nohide</a>	This command reverses the effect of a virtual hide command, causing the specified real or virtual signals to reappear the Objects window.	N	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">virtual nolog</a>	This command reverses the effect of a virtual log command. It causes the simulation-dependent signals of the specified virtual signals to be excluded ("unlogged") by the kernel. If wildcard patterns are used, it will also unlog any normal signals found, unless the -only option is used.	N	Y
<a href="#">virtual region</a>	This command creates a new user-defined design hierarchy region.	N	Y
<a href="#">virtual save</a>	This command saves the definitions of virtuals to a file named virtual.do in the current directory.	N	Y
<a href="#">virtual show</a>	This command lists the full path names of all explicitly defined virtuals.	N	Y
<a href="#">virtual signal</a>	This command creates a new signal, known only by the GUI (not the kernel), that consists of concatenations of signals and subelements as specified in <expressionString>.	N	Y
<a href="#">virtual type</a>	This command creates a new enumerated type known only by the GUI, not the kernel. Virtual types are used to convert signal values to character strings. The command works with signed integer values up to 64 bits.	N	Y
<a href="#">vlib</a>	This command creates a design library. You must use vlib rather than operating system commands to create a library directory or index file.	Y	Y
<a href="#">vlog</a>	The vlog command compiles Verilog source code and SystemVerilog extensions into a specified working library (or to the work library by default). Compressed SystemVerilog source files (those compressed with zlib) are accepted.	Y	Y
<a href="#">vmake</a>	The vmake utility allows you to use a MAKE program to maintain individual libraries. You run vmake on a compiled design library. This utility operates on multiple source files per design unit; it supports Verilog include files as well as Verilog and VHDL PSL vunit files.	Y	Y
<a href="#">vmap</a>	The vmap command defines a mapping between a logical library name and a directory by modifying the modelsim.ini file.	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
<a href="#">vsim</a>	The vsim command invokes the VSIM simulator, which you can use to view the results of a previous simulation run (when invoked with the -view switch)	Y	Y
<a href="#">vsim&lt;info&gt;</a>	The vsim<info> commands return information about the current vsim executable.	Y	Y
<a href="#">vsim_break</a>	Stop (interrupt) the current simulation before it runs to completion. To stop a simulation and then resume it, use this command in conjunction with run -continue.	Y	Y
<a href="#">vsource</a>	This command specifies an alternative file to use for the current source file. This command is used when the current source file has been moved. The alternative source mapping exists for the current simulation only..	Y	Y
<a href="#">wave</a>	A number of commands are available to manipulate and report on the Wave window.	N	N
<a href="#">wave sort</a>	This command sorts signals in the Wave window by name or full path name.	N	N
<a href="#">when</a>	This command instructs ModelSim to perform actions when the specified conditions are met.	Y	Y
<a href="#">where</a>	This command displays information about the system environment. It is useful for debugging problems where ModelSim cannot find the required libraries or support files.	Y	Y
<a href="#">wlf2log</a>	This command translates a ModelSim WLF file (vsim.wlf) to a QuickSim II logfile. It reads the vsim.wlf WLF file generated by the add list, add wave, or log commands in the simulator and converts it to the QuickSim II logfile format.	Y	Y
<a href="#">wlf2vcd</a>	This command translates a ModelSim WLF file to a standard VCD file. Complex data types that are unsupported in the VCD standard (records, memories, etc.) are not converted.	Y	Y
<a href="#">wlfman</a>	This command allows you to get information about and manipulate saved WLF files.	Y	Y

**Table 2-1. Supported Commands (cont.)**

Command name	Action	-batch	-c
wlfrecover	This command attempts to "repair" WLF files that are incomplete due to a crash or if the file was copied prior to completion of the simulation. Use this command if you receive a "bad magic number" error message when opening a WLF file. You can run the command from the VSIM> or ModelSim> prompt or from a shell.	Y	Y
write format	This command records the names and display options of the HDL objects currently being displayed in the Analysis, List, Memory, Message Viewer, Test Browser, and Wave windows.	N	Y
write list	This command records the contents of the List window in a list output file.	N	Y
write preferences	This command saves the current GUI preference settings to a Tcl preference file. Settings saved include Wave, Objects, and Locals window column widths; Wave, Objects, and Locals window value justification; and Wave window signal name width.	N	Y
write report	This command prints a summary of the design being simulated including a list of all design units (VHDL configurations, entities, and packages, and Verilog modules) with the names of their source files. The summary includes a list of all source files used to compile the given design.	Y	Y
write timing	This command displays path delays and timing check limits, unadjusted for delay net delays, for the specified instance.	Y	Y
write transcript	This command writes the contents of the Transcript window to the specified file. The resulting file can then be modified to replay the transcribed commands as a DO file (macro).	N	Y
write tssi	This command records the contents of the List window in a "TSSI format" file.	Y	Y
write wave	This command records the contents of the Wave window in PostScript format. The output file can then be printed on a PostScript printer.	N	N

1. transcript on | off only are supported.

# abort

This command halts the execution of a DO file interrupted by a breakpoint or error.

When DO files are nested, you may choose to abort the last DO file script only, abort a specified number of nesting levels, or abort all DO files. You can specify this command within a DO file to return early.

## Syntax

abort [<n> | all]

## Arguments

- <n>  
(optional) The number of nested DO file script levels to abort. Specified as an integer greater than 0, where the default value is 1.
- all  
(optional) Instructs the tool to abort all levels of nested DO files.

## add dataflow

This command adds the specified process, signal, net, or register to the Dataflow window. Wildcards are allowed.

Arguments to this command are order dependent. Please read the argument descriptions for more information.

### Syntax

```
add dataflow <object> ... [-connect <source_net> <destination_net>]  
  {[-in] [-out] [-inout] | [-ports]} [-internal] [-nofilter] [-recursive]
```

### Arguments

- <object> ...  
(required unless specifying -connect) Specifies a process, signal, net, or register to add to the Dataflow window. Wildcards are allowed. Multiple objects are specified as a space separated list, Refer to the section “[Wildcard Characters](#)” on page 21 for wildcard usage as it pertains to the add commands. Must be specified as the first argument to the add dataflow command.
- -connect <source\_net> <destination\_net>  
(optional) Computes and displays in the Dataflow window all paths between two nets.  
    <source\_net> — The net that originates the path search.  
    <destination\_net> — The net that terminates the path search.
- -in  
(optional) Specifies to add ports of mode IN.
- -inout  
(optional) Specifies to add ports of mode INOUT.
- -out  
(optional) Specifies to add ports of mode OUT.
- -ports  
(optional) Specifies to add all ports. This switch has the same effect as specifying -in, -out, and -inout together.
- -internal  
(optional) Specifies to add internal (non-port) objects.
- -nofilter  
(optional) Specifies that the *WildcardFilter* Tcl preference variable be ignored when finding signals or nets.

The *WildcardFilter* Tcl preference variable identifies types to ignore when matching objects with wildcard patterns.

- **-recursive**

(optional) Specifies that the scope of the search is to descend recursively into subregions. If omitted, the search is limited to the selected region.

You can specify **-r** as an alias to this switch.

## Examples

- Add all objects in the design to the dataflow window.

**add dataflow -r /\***

- Add all objects in the region to the dataflow window.

**add dataflow \***

## Related Topics

[Automatically Tracing All Paths Between Two Nets](#)

[Dataflow Window](#)

## add list

This command adds objects and their values to the List window. Arguments to this command are order-dependent. Please read the argument descriptions for more information.

### Syntax

```
add list {<object> ... | <object_name> {sig ...} [-allowconstants] [-depth <level>]
          [-filter <f> | -nofilter <f>] {[-in] [-inout] [-out] | [-ports]} [-internal] [-label <name>]
          [-nodelta] [-<radix_type> | -radix <type>]
          [-radixenumnumeric | -radixenumsymbolic] [-recursive] [-trigger | -nottrigger]
          [-width <integer>]
```

### Description

Use add list to display the following types of objects and their values in the List window:

- VHDL signals and variables
- Verilog nets and registers
- User-defined buses

If you do not specify a port mode, such as -in or -out, this command displays all objects in the selected region with names matching the object name specification.

Refer to [Wildcard Characters](#) for wildcard usage as it pertains to the add commands.

### Arguments

- <object> ...  
(required when <object\_name>{sig ...} is not specified.) Specifies the name of the object to be listed. Multiple objects are entered as a space separated list. Wildcards are allowed. Refer to the section “[Wildcard Characters](#)” for wildcard usage as it pertains to the add commands. Must be specified as the first argument to the add list command.

Note that the WildcardFilter Tcl preference variable identifies types to ignore when matching objects with wildcard patterns.

You can add variables as long as they are preceded by the process name. For example:

```
add list myproc/int1
```

You must specify the <object> argument as the first argument to the add list command.

- <object\_name> {sig ...}  
(required when <object> is not specified) Creates a user-defined bus with the specified object name containing the specified signals (sig) concatenated within the user-defined bus. Arguments, must be enclosed in braces ({}). Must be specified as the second argument to the add list command.

**sig** — A space-separated list of signals, enclosed in braces ( { } ), that are included in the user-defined bus. The signals may be either scalars or various sized arrays as long as they have the same element enumeration type.

For example:

```
add list {mybus {a b y}}
```

- **-allowconstants**

(optional) *For use with wildcard searches.* Specifies that constants matching the wildcard search should be added to the List window.

This command does not add constants by default because they do not change.

- **-depth <level>**

(optional) Restricts a recursive search, as specified with **-recursive**, to a certain level of hierarchy.

<level> — an integer greater than or equal to zero.

For example, if you specify **-depth 1**, the command descends only one level in the hierarchy.

- **-filter <f> | -nofilter <f>**

(optional) Allows a one-time modification of the WildcardFilter in the command invocation. The **add list** command can take as many **[-filter <f>]** and **[-nofilter <f>]** arguments as you would like to specify. Valid filters, <f>, are exactly the same set of words that can be applied to the WildcardFilter. The filter(s) used during a command starts with the WildcardFilter and then applies the user specified filters, if any. The **-filter** values are added to the filter, the **-nofilter** values are removed from the filter. They are applied in the order specified so conflicts are resolved with the last specified wins.

- **-in**

(optional) *For use with wildcard searches.* Specifies that the scope of the search is to include ports of mode IN if they match the *object* specification.

- **-inout**

(optional) *For use with wildcard searches.* Specifies that the scope of the search is to include ports of mode INOUT if they match the *object* specification.

- **-out**

(optional) *For use with wildcard searches.* Specifies that the scope of the search is to include ports of mode OUT if they match the *object* specification.

- **-ports**

(optional) For use with wildcard searches. Specifies that the scope of the search is to include all ports. This switch has the same effect as specifying **-in**, **-out**, and **-inout** together.

- **-internal**  
(optional) *For use with wildcard searches.* Specifies that the scope of the search is to include internal objects (non-port objects) if they match the *object* specification. VHDL variables are not selected.
- **-label <name>**  
(optional) Specifies an alternative signal name to be displayed as a column heading in the listing.  
  
    <name> — Specifies the label to be used at the top of the column. You must enclose <name> in braces ( { } ) if it includes any spaces.  
  
    This alternative name is not valid in a [force](#) or [examine](#) command.
- **-nodefault**  
(optional) Specifies that the delta column not be displayed when adding signals to the List window. Identical to [configure](#) list -delta none.
- **-<radix\_type>**  
(optional) Specifies the radix type for the objects that follow in the command. Valid entries (or any unique abbreviations) are: binary, ascii, unsigned, decimal, octal, hex, sfixed, symbolic, ufixed, time, and default.  
  
    If no radix is specified for an enumerated type, the default radix is used. You can change the default radix for the current simulation using the [radix](#) command. You can change the default radix permanently by editing the [DefaultRadix](#) variable in the *modelsim.ini* file.  
  
    If you specify a radix for an array of a VHDL enumerated type, ModelSim converts each signal value to 1, 0, Z, or X.
- **-radix <type>**  
(optional) Specifies a user-defined radix. The -radix <type> switch can be used in place of the -<radix\_type> switch. For example, -radix hexadecimal is the same as -hex.  
  
    <type> — binary, ascii, unsigned, decimal, octal, hex, sfixed, symbolic, time, ufixed, and default.  
  
    Refer to the [radix](#) command for information about sfixed and ufixed radix types.  
  
    This option overrides the global setting of the default radix (the [DefaultRadix](#) variable in the *modelsim.ini* file) for the current simulation only.
- **-radixenumnumeric**  
This option overrides the global setting of the default radix (the [DefaultRadix](#) variable in the *modelsim.ini* file).
- **-radixenumsymbolic**  
(optional) Reverses the action of -radixenumnumeric and sets the global setting of the default radix (the [DefaultRadix](#) variable in the *modelsim.ini* file) to symbolic.

- **-recursive**  
(optional) *For use with wildcard searches.* Specifies that the scope of the search is to descend recursively into subregions. If omitted, the search is limited to the selected region. You can use the **-depth** argument to specify how far down the hierarchy to descend. You can use "**-r**" as an alias to this switch.
- **-trigger | -notrigger**  
(optional) Specifies whether objects should be updated in the List window when the objects change value.
  - trigger** — (default) Update objects in the List Window when their values change.
  - notrigger** — Do not update objects in the List Window when their values change.
- **-width <integer>**  
(optional) Formats the column width. The maximum width, when not specifying this argument is 30,000 characters, which you can override with this switch.  
**integer** — A positive integer specifying the column width in characters.

## Examples

- List all objects in the design.  
**add list -r /\***
- List all objects in the region.  
**add list \***
- List all input ports in the region.  
**add list -in \***
- Display a List window containing three columns headed *a*, *sig*, and *array\_sig(9 to 23)*.  
**add list a -label sig /top/lower/sig {array\_sig(9 to 23)}**
- List *clk*, *a*, *b*, *c*, and *d* only when *clk* changes.  
**add list clk -notrigger a b c d**
- Lists *clk*, *a*, *b*, *c*, and *d* every 100 ns.  
**config list -strobeperiod {100 ns} -strobestart {0 ns} -usestrobe 1**  
**add list -notrigger clk a b c d**
- Creates a user-defined bus named "mybus" consisting of three signals; the bus is displayed in hex.  
**add list -hex {mybus {msb {opcode(8 downto 1)} data}}**
- Lists the object *vec1* using symbolic values, lists *vec2* in hexadecimal, and lists *vec3* and *vec4* in decimal.  
**add list vec1 -hex vec2 -dec vec3 vec4**

## Related Topics

[add wave](#)

# add memory

This command displays the contents and sets the address and data radix of the specified memory in the MDI frame of the Main window.

Refer to “Wildcard Characters” for wildcard usage as it pertains to the add commands.

Arguments to this command are order dependent. Please read the argument descriptions for more information.

## Syntax

```
add memory [-addressradix {decimal | hex}] [-dataradix <type>]  
[-radixenumnumeric | -radixenumsymbolic] [-wordsperline <num>] <object_name> ...
```

## Arguments

- **-addressradix {decimal | hex}**  
(optional) Specifies the address radix for the memory display.
  - decimal — (default) Sets the radix to decimal. You can abbreviate this argument to "d".
  - hex — Sets the radix to hexadecimal. You can abbreviate this argument to "h".
- **-dataradix <type>**  
(optional) Specifies the data radix for the memory display. If you do not specify this switch, the command uses the global default radix.  
<type> — binary, ascii, unsigned, decimal, octal, hex, sfixed, symbolic, time, ufixed, and default.

Refer to the [radix](#) command for information about sfixed and ufixed radix types.

If you do not specify a radix for an enumerated type, the command uses the symbolic representation.

You can change the default radix for the current simulation using the [radix](#) command. You can change the default radix permanently by editing the [DefaultRadix](#) variable in the *modelsim.ini* file. Changing the default radix does not change the radix of the currently displayed memory. Use the add memory command to re-add the memory with the desired radix, or change the display radix from the Memory window Properties dialog.

- **-radixenumnumeric**  
(optional) Causes Verilog enums to be displayed as numbers (formatted by the current radix). This overrides the default behavior of always showing enums symbolically.
- **-radixenumsymbolic**  
(optional) Restores the default behavior of displaying Verilog enums as symbols by reversing the action of the -radixenumnumeric option.

- `-wordsperline <num>`

(optional) Specifies how many words are displayed on each line in the memory window. By default, the information displayed will wrap depending on the width of the window.

    num — Any positive integer

- `<object_name> ...`

(required) Specifies the hierarchical path of the memory to be displayed. Multiple memories are specified as a space separated list. Must be specified as the final argument to the add memory command.

Wildcard characters are allowed.

---

**Note**

 The *WildcardFilter* Tcl preference variable identifies types to ignore when matching objects with wildcard patterns.)

---

## Related Topics

[Memory List Window](#)

# add message

This command is used within a DO file or script and specifies a user-defined runtime message that is sent to the transcript and .wlf files. Messages are displayed in the Message Viewer window in the GUI. Refer to the *GUI Reference Manual* for more information on this window.

## Syntax

```
add message <message_body> [-category <category>] [-efftime <time>] [-file <filename>]  
  [-id <id_number>] [-inline] [-line <linenumber>] [-noident] [-nolevel] [-objects <list>]  
  [-region region] [-severity {error | note | warning}]
```

## Arguments

- <message\_body>  
(required) User specified message.
- -category <category>  
(optional) Sets the category for the message in the Message Viewer window where the default is USER. The Message Viewer window Category column recognizes the following keywords:

**Table 2-2. Message Viewer Categories**

DISPLAY	FLI	PA
PLI	SDF	TCHK
VCD	VITAL	WLF
MISC	USER	<user-defined>

- -efftime <time>  
(optional) Specifies the simulation time when the message is saved to the log file. The time specified is listed in the Message Viewer window Time column when the message is called. Useful for placing messages at specific times in the simulation.  
    <time> — Specified as an integer or decimal number.
- -file <filename>  
(optional) Displays a user specified string in the File Info column of the Message Viewer window.
- -id <id\_number>  
(optional) Assigns an identification number to the message.  
    <id\_number> — Any non-negative integer from 0 - 9999 where the default is 0. The number specified is added to the base identification number of 80000.

- **-inline**  
(optional) Causes the message to also be returned to the caller as the return value of the add message command.
- **-line <linenumber>**  
(optional) Displays the user specified number in File Info column of the Message Viewer window.
- **-noident**  
(optional) Prevents return of the ID number of the message.
- **-nolevel**  
(optional) Prevents return of the severity level of the message.
- **-objects <list>**  
(optional) List of related design items shown in the Objects column of the Message Viewer window.  
  
*<list>* — A space separated list enclosed in curly braces ({} ) or quotation marks ("").
- **-region region**  
(optional) Message is displayed in the Region column of the Message Viewer window.
- **-severity {error | note | warning}**  
(optional) Sets the message severity level.  
  
error — ModelSim cannot complete the operation.  
note — (default) The message is informational.  
warning — There may be a problem that will affect the accuracy of the results.

## Examples

- Create a message numbered 80304.

**add message -id 304 <message>**

## Related Topics

[displaymsgmode](#)

[msgmode](#)

[Message Viewer Window](#)

# add watch

This command adds signals and variables to the Watch window in the Main window.

Refer to “Wildcard Characters” for wildcard usage as it pertains to the add commands.

Arguments to this command are order-dependent. Please read the argument descriptions for more information.

## Syntax

```
add watch <object_name> ... [-radix <type>] [-radixenumnumeric | -radixenumsymbolic]
```

## Arguments

- <object\_name> ...

(required) Specifies the name of the object to be added. Multiple objects are entered as a space-separated list. Must be specified as the first argument to the add watch command.

Wildcard characters are allowed. (Note that the *WildcardFilter* Tcl preference variable identifies types to ignore when matching objects with wildcard patterns.)Wildcard expansion is limited to 150 items. If you exceed this limit, a dialog box will ask you to accept the limit or cancel the operation.

Variables must be preceded by the process name. For example,

```
add watch myproc/int1
```

- -radix <type>

(optional) Specifies a user-defined radix. If you do not specify this switch, the command uses the global default radix.

<type> — binary, ascii, unsigned, decimal, octal, hex, sfixed, symbolic, time, ufixed, and default.

Refer to the [radix](#) command for information about sfixed and ufixed radix types.

You can change the default radix for the current simulation using the [radix](#) command. You can change the default radix permanently by editing the [DefaultRadix](#) variable in the *modelsim.ini* file.

- -radixenumnumeric

(optional) Causes Verilog enums to be displayed as numbers (formatted by the current radix). This overrides the default behavior of always showing enums symbolically.

- -radixenumsymbolic

(optional) Restores the default behavior of displaying Verilog enums as symbols by reversing the action of the -radixenumnumeric option.

## Related Topics

[Watch window](#)

[DefaultRadix](#)

## add wave

This command adds objects to the Wave window. Arguments to this command are order-dependent. Please read the argument descriptions for more information.

### Syntax

```
add wave [-allowconstants] [-clampanalog {0 | 1}] [-color <standard_color_name>]
[-depth <level>] [[-divider <divider_name> ...] [-expand <signal_name>]
[-filter <f> | -nofilter <f>] [-format <type> | -<format>]
[-group <group_name> [<sig_name1> ...]] [-height <pixels>]
{[-in] [-inout] [-out] | [-ports]} [-internal] [-label <name>][-max <real_num>]
[-min <real_num>][-noupdate]
[-numdynitem <int>] [-position <location>] [-queueends]
[-<radix_type> | -radix <type>] [-radixenumnumeric | -radixenumsymbolic] [-recursive]
[-startdynitem <int>] [-time] [<object_name> ...]
[{\<object_name> {sig1 sig2 ...}}]
```

### Description

Use add wave to display the following types of objects in the Wave window:

- VHDL signals and variables
- Verilog nets and registers
- SystemVerilog class objects
- Dividers and user-defined buses.

If no port mode is specified, this command will display all objects in the selected region with names matching the object name specification.

Refer to “[Wildcard Characters](#)” on page 21 for wildcard usage as it pertains to the add commands.

Arguments to this command are order dependent. Please read through the argument descriptions for more information.

### Arguments

- **-allowconstants**

(optional) *For use with wildcard searches.* Specifies that constants matching the wildcard search should be added to the Wave window.

By default, constants are ignored because they do not change.

- `-clampanalog {0 | 1}`

(optional) Clamps the display of an analog waveform to the values specified by `-max` and `-min`. Specifying a value of 1 prevents the waveform from extending above the value specified for `-max` or below the value specified for `-min`.

0 — not clamped

1 — (default) clamped

- `-color <standard_color_name>`

(optional) Specifies the color used to display a waveform.

`<standard_color_name>` — You can use either of the following:

standard X Window color name — enclose 2-word names in quotes (""), for example:

`-color "light blue"`

rgb value — for example:

`-color #357f77`

- `-depth <level>`

(optional) Restricts a recursive search, as specified with `-recursive` to a specified level of hierarchy.

`<level>` — Any integer greater than or equal to zero. For example, if you specify `-depths 1`, the command descends only one level in the hierarchy.

- `-divider [<divider_name> ...]`

(optional) Adds a divider to the Wave window. If you do not specify this argument, the command inserts an unnamed divider.

`<divider_name> ...` — Specifies the name of the divider, which appears in the pathnames column. Multiple objects entered as a space separated list.

When you specify more than one `<divider_name>` the command creates a divider for each name.

You can begin a name with a space, but you must enclose the name within quotation marks ("") or braces ({}). You cannot begin a name with a hyphen (-).

- `-expand <signal_name>`

(optional) Instructs the command to expand a compound signal immediately, but only one level down.

`<signal_name>` — Specifies the name of the signal. This string can include wildcards.

- `-filter <f> | -nofilter <f>`

(optional) Allows a one-time modification of the WildcardFilter in the command invocation. The add list command can take as many `[-filter <f>]` and `[-nofilter <f>]` arguments as you would like to specify. Valid filters, `<f>`, are exactly the same set of words that can be applied to the WildcardFilter. The filter(s) used during a command starts with

the WildcardFilter and then applies the user specified filters, if any. The -filter values are added to the filter, the -nofilter values are removed from the filter. They are applied in the order specified so conflicts are resolved with the last specified wins.

- **-format <type> | -<format>**

(optional) Specifies the display format of the objects. Valid entries are:

<b>-format &lt;type&gt;</b>	<b>-&lt;format&gt;</b>	Display Format
<b>-format literal</b>	<b>-literal</b>	Literal waveforms are displayed as a box containing the object value.
<b>-format logic</b>	<b>-logic</b>	Logic signals may be U, X, 0, 1, Z, W, L, H, or '-'.
<b>-format analog-step</b>	<b>-analog-step</b>	Analog-step changes to the new time before plotting the new Y.
<b>-format analog-interpolated</b>	<b>-analog-interpolated</b>	Analog-interpolated draws a diagonal line.
<b>-format analog-backstep</b>	<b>-analog-backstep</b>	Analog-backstep plots the new Y before moving to the new time.
<b>-format event</b>	<b>-event</b>	Displays a mark at every transition.

The Y-axis range of analog signals is bounded by -max and -min switches.

- **-group <group\_name> [<sig\_name1> ...]**

(optional) Creates a wave group with the specified group\_name.

**<group\_name>** — Specifies the name of the group. You must enclose this argument in quotation marks ("") or braces ({} ) if it contains any spaces.

**<sig\_name> ...** — Specifies the signals to add to the group. Multiple signals are entered as a space separated list. This command creates an empty group if you do not specify any signal names.

- **-height <pixels>**

(optional) Specifies the height of the waveform in pixels.

**<pixels>** — Any positive integer.

- **-in**

(optional) *For use with wildcard searches.* Specifies that the scope of the search is to include ports of mode IN if they match the object\_name specification.

- **-out**

(optional) *For use with wildcard searches.* Specifies that the scope of the search is to include ports of mode OUT if they match the object\_name specification.

- **-inout**  
(optional) *For use with wildcard searches.* Specifies that the scope of the search is to include ports of mode INOUT if they match the object\_name specification.
- **-ports**  
(optional) *For use with wildcard searches.* Specifies that the scope of the listing is to include ports of modes IN, OUT, or INOUT.
- **-internal**  
(optional) *For use with wildcard searches.* Specifies that the scope of the search is to include internal objects (non-port objects) if they match the object\_name specification.
- **-label <name>**  
(optional) Specifies an alternative name for the signal being added. For example,

```
add wave -label c clock
```

adds the *clock* signal, labeled as "c".

This alternative name is not valid in a [force](#) or [examine](#) command.

- **-max <real\_num>**  
(optional) Specifies the maximum Y-axis data value to be displayed for an analog waveform. Used in conjunction with the -min switch; the value you specify for -max must be greater than the value you specify for -min.  
  
    <real\_num> — Any integer that is greater than the value specified for -min.
- **-min <real\_num>**  
(optional) Specifies the minimum Y-axis data value to be displayed for an analog waveform. Used in conjunction with the -max switch; the value you specify for -min must be less than the value you specify for -max.  
  
    <real\_num> — Any integer that is less than the value specified for -max.

For example, if you know the Y-axis data for a waveform varies between 0.0 and 5.0, you could add the waveform with the following command:

```
add wave -analog -min 0 -max 5 -height 100 my_signal
```

---

**Note**



Although -offset and -scale are still supported, the -max and -min arguments provide an easier way to define upper and lower limits of an analog waveform.

---

- **-noupdate**  
(optional) Prevents the Wave window from updating when a series of add wave commands are executed in series.

- **-numdynitem <int>**  
(optional) Specifies the number of child elements of a queue or dynamic array to display in the Wave window. For example, if the value 3 is specified, then only three elements will be displayed in the Wave window.  
    <int> — Any non-negative integer from 0 to the number of elements of the specified queue or dynamic array.
- **-position <location>**  
(optional) Specifies where the command adds the signals.  
    <location> — Can be any of the following:  
        top — Adds the signals to the beginning of the list of signals.  
        bottom | end — Adds the signals to the end of the list of signals.  
        before | above — Adds the signals to the location before the first selected signal in the wave window.  
        after | below — Adds the signals to the location after the first selected signal in the wave window.  
        <integer> — Adds the signals beginning at the specified point in the list of signals.
- **-queueends**  
(optional) Adds a SystemVerilog queue to the Wave window and displays the first and last elements of the queue.  
    <queue> — The relative or full path to a queue.
- **-<radix\_type>**  
(optional) Specifies the radix type for the objects that follow in the command. Valid entries (or any unique abbreviations) are: binary, ascii, unsigned, decimal, octal, hex, sfixed, symbolic, time, ufixed, and default.  
If no radix is specified for an enumerated type, the default radix is used. You can change the default radix for the current simulation using the [radix](#) command. You can change the default radix permanently by editing the [DefaultRadix](#) variable in the *modelsim.ini* file.  
If you specify a radix for an array of a VHDL enumerated type, ModelSim converts each signal value to 1, 0, Z, or X.
- **-radix <type>**  
(optional) Specifies a user-defined radix. The **-radix <type>** switch can be used in place of the **-<radix\_type>** switch. For example, **-radix hexadecimal** is the same as **-hex**.  
    <type> — binary, ascii, unsigned, decimal, octal, hex, sfixed, symbolic, time, ufixed, and default.  
Refer to the [radix](#) command for information about sfixed and ufixed radix types.  
This option overrides the global setting of the default radix (the [DefaultRadix](#) variable in the *modelsim.ini* file) for the current simulation only.

- **-radixenumnumeric**  
(optional) Causes Verilog enums to be displayed as numbers (formatted by the current radix). This overrides the default behavior of always showing enums symbolically.
- **-radixenumsymbolic**  
(optional) Restores the default behavior of displaying Verilog enums as symbols by reversing the action of the **-radixenumnumeric** option.
- **-recursive**  
(optional) For use with wildcard searches. Specifies that the scope of the search is to descend recursively into subregions.  
If you do not specify this switch, the search is limited to the selected region. You can use the **-depth** argument to specify how far down the hierarchy to descend.
- **-startdynitem <int>**  
(optional) Specifies the index of a queue or dynamic array from where the Wave window starts displaying the data. For example, if a queue has 10 elements and **-startdynitem 3** is specified, the display starts from `q[3]`.  
`<int>` — Any non-negative integer where 0 is the default.
- **-time**  
(optional) Use time as the radix for Verilog objects that are register-based types (register vectors, time, int, and integer types).
- **<object\_name> ...**  
(required unless specifying `{<object_name> {sig1 sig2 ...}}`) Specifies the names of objects to be included in the Wave window. Must be specified as the final argument to the `add wave` command. Wildcard characters are allowed. Multiple objects are entered as a space separated list. Note that the *WildcardFilter* Tcl preference variable identifies types to ignore when matching objects with wildcard patterns.

Variables may be added if preceded by the process name. For example,

```
add wave myproc/int1
```

- **{<object\_name> {sig1 sig2 ...}}**  
(required unless specifying `<object_name>`) Creates a user-defined bus with the specified object name containing the specified signals (sig1 and so forth) concatenated within the user-defined bus. Must be specified as the final argument to the `add wave` command.

`sig` — A space-separated list of signals, enclosed in braces (`{ }`), that are included in the user-defined bus. The signals may be either scalars or various sized arrays as long as they have the same element enumeration type.

---

**Note**

 You can also select **Wave > Combine Signals** (when the Wave window is selected) to create a user-defined bus.

---

## Examples

- Display an object named *out2*. The object is specified as being a logic object presented in gold.

```
add wave -logic -color gold out2
```

- Display a user-defined, hex formatted bus named *address*.

```
add wave -hex {address {a_7 a_6 a_5 a_4 a_3 a_2 a_1 a_0}}
```

- Add all wave objects in the region.

```
add wave *
```

- Add all wave input ports in the region.

```
add wave -in *
```

- Create a user-defined bus named "mybus" consisting of three signals. *Scalar1* and *scalar2* are of type std\_logic and *vector1* is of type std\_logic\_vector (7 downto 1). The bus is displayed in hex.

```
add wave -hex {mybus {scalar1 vector1 scalar2}}
```

Slices and arrays may be added to the bus using either VHDL or Verilog syntax. For example:

```
add wave {vector3(1)}  
add wave {vector3[1]}  
add wave {vector3(4 downto 0)}  
add wave {vector3[4:0]}
```

- Add the object *vec1* to the Wave window using symbolic values, adds *vec2* in hexadecimal, and adds *vec3* and *vec4* in decimal.

```
add wave vec1 -hex vec2 -dec vec3 vec4
```

- Add a divider with the name "-Example-". Note that for this to work, the first hyphen of the name must be preceded by a space.

```
add wave -divider " -Example- "
```

- Add an unnamed divider.

```
add wave -divider  
add wave -divider ""  
add wave -divider {}
```

## Related Topics

[add list](#)

[Wave Window](#)

## **add\_cmdhelp**

This command adds the specified command name, description, and command arguments to the command-line help. You can then access the information using the help command.

To delete an entry, invoke the command with an empty command description and arguments. See examples.

The arguments to this command are order-dependent. Please read through the argument descriptions for more information.

### **Syntax**

`add_cmdhelp {<command_name>} {<command_description>} {<command_arguments>}`

### **Arguments**

- `{<command_name>}`  
(required) Specifies the command name that will be entered as an argument to the help command. Must be enclosed in braces (`{ }` ). The command\_name must not interfere with an already existing command\_name. Must be specified as the first argument to the add\_cmdhelp command.
- `{<command_description>}`  
(required) Specifies a description of the command. Must be enclosed in braces (`{ }` ). Must be specified as the second argument to the add\_cmdhelp command.
- `{<command_arguments>}`  
(required) A space-separated list of arguments for the command. Must be enclosed in braces (`{ }` ). If the command doesn't have any arguments, enter `{ }` . Must be specified as the third argument to the add\_cmdhelp command.

### **Examples**

- Add a command named "date" with no arguments.

`add_cmdhelp date {Displays date and time.} {}`

Entering:

**VSIM> help date**

Returns:

Displays date and time.  
Usage: date

- Add the change date command.

`add_cmdhelp {change date} {Modify date or time.} {-time|-date <arg>}`

Entering:

**VSIM> help change date**

Returns:

Modify data or time  
Usage: change date -time|-date <arg>

- Deletes the change date command from the command-line help.

**add\_cmdhelp {change date} {} {}**

## alias

This command displays or creates user-defined aliases. Any arguments passed on invocation of the alias will be passed through to the specified commands.

Returns nothing. Existing commands (for example, run, env, and so forth) cannot be aliased.

### Syntax

**alias** [**<name>** [**"<cmds>"**]]

### Arguments

- **<name>**  
(optional) Specifies the new procedure name to be used when invoking the commands.
- **"<cmds>"**  
(optional) Specifies the command or commands to be evaluated when the alias is invoked. Multiple commands are specified as a semicolon (;) separated list. You must enclose the string in quotes ("").

### Examples

- List all aliases currently defined.

**alias**

- List the alias definition for the specified name if one exists.

**alias <name>**

- Create a Tcl procedure, "myquit", that when executed, writes the contents of the List window to the file *mylist.save* by invoking write list, and quits ModelSim by invoking quit.

**alias myquit "write list ./mylist.save; quit -f"**

# archive load

The archive load command allows you to load an archived debug database (.dbar) file that was previously created with the archive write command. The archived file may include a number of WLF files, design source files, and a DBG file.

## Syntax

```
archive load <archive_name> [-dbgDir <directory_name>] -wlf <wlf_file_name>
```

## Arguments

- <archive\_name>  
(required) Specifies the name of the archived file to be opened for reading. A suggested suffix is .dbar.
- -dbgDir <directory\_name>  
(optional) Specifies a location to extract files into. Files are extracted on-demand when ModelSim needs them. If you do not specify this switch, the command extracts to the current working directory.
- -wlf <wlf\_file\_name>  
(required) Specifies the WLF files to open for analysis.  
<wlf\_file\_name> — can be a single file or a list of files. A list of file names must be enclosed in curly braces {}. The name of the wlf file must be exactly the same as that specified in the archive write command, including the pathname, if provided.

## Related Topics

[archive write](#)

## archive write

The archive write command allows you to create a debug archive file, with the file extension .dbar, that contains one or more WLF files, debug information captured from the design library, an optional connectivity debug database file, and optional HDL source files. With this archived file, you can perform post-simulation debugging in different location from that which the original simulation was run.

### Syntax

```
archive write <archive_name> -wlf <wlf_file_name> [-include_src] [-dbg <dbg_file_name>]
```

### Arguments

- <archive\_name>  
(required) Specifies the name of the archive file to be created. A suggested suffix is .dbar.
- -wlf <wlf\_file\_name>  
(required) Specifies the name of the WLF file to use for post-simulation analysis.  
    <wlf\_file\_name> — can be a single file or a list of files enclosed in curly braces {} if you want to capture more than one WLF file in the archive.
- -include\_src  
(optional) Indicate if source files should be captured in the archive. This is off by default, which means no source will be in the archive.
- -dbg <dbg\_file\_name>  
(optional) Specifies the name of an existing debug database (.dbg) file to be included in the archive.

## batch\_mode

This command returns “1” if ModelSim is operating in batch mode, otherwise it returns “0.” It is typically used as a condition in an if statement.

### Syntax

batch\_mode

### Arguments

None

### Examples

Some GUI commands do not exist in batch mode. If you want to write a script that will work in or out of batch mode, you can use the batch\_mode command to determine which command to use. For example:

```
if [batch_mode] {  
    log /*  
} else {  
    add wave /*  
}
```

### Related Topics

[General Modes of Operation](#)

## bd

This command deletes a breakpoint. You can delete multiple breakpoints by specifying separate information groupings on the same command line.

Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

### Syntax

**bd** {<filename> <line\_number>}

**bd** {<id\_number> | <label>} ...

### Arguments

- <filename>  
(required when not specifying <id\_number> or <label>.) A string that specifies the name of the source file in which the breakpoint is to be deleted. The filename must match the one used previously to set the breakpoint, including whether you used a full pathname or a relative name. Must be specified as the first argument to the **bd** command.
- <line\_number>  
(required) A string that specifies the line number of the breakpoint to be deleted.
- <id\_number> | <label>  
(required when not specifying <filename>.) Specifies the identification of breakpoints using markers assigned by the **bp** command. Must be specified as the first argument to the **bd** command.
  - <id\_number> — A string that specifies the identification number of the breakpoint to be deleted. The identification number is set with the -id argument to the **bp** command.
  - <label> — A string that specifies the label of the breakpoint to be deleted. The label is set with the -label switch to the **bp** command.

### Examples

- Delete the breakpoint at line 127 in the source file named *alu.vhd*.

**bd alu.vhd 127**

- Delete the breakpoint with id# 5.

**bd 5**

- Delete the breakpoint with the label *top\_bp*

**bd top\_bp**

- Delete the breakpoint with id# 6 and the breakpoint at line 234 in the source file named *alu.vhd*.

**bd 6 alu.vhd 234**

## Related Topics

[bp](#)

## bookmark add wave

This command creates a named reference to a specific zoom range and scroll position in the specified Wave window. Bookmarks are saved in the wave format file and are restored when the format file is read.

You can also interactively add a bookmark through the GUI by selecting the **Wave > Bookmarks > Bookmarks** menu item.

Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

### Syntax

```
bookmark add wave <label> [[<range_start> [<unit>]] [<range_end> [<unit>]] [<topindex>]]
```

### Arguments

- **<label>**

(required) A string that specifies the name for the bookmark. Must be specified as the first argument to the bookmark add wave command.

- **<range\_start> [<unit>]**

(optional) Specifies the beginning point of the zoom range where the default starting point is zero (0).

<unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <limit> and <unit> within curly braces ({}).

The complete grouping of <range\_start> and <range\_end> must also be enclosed in braces ({} {}) or quotes (" "), for example:

```
{ {100 ns} {10000 ns} }  
{10000}
```

- **<range\_end> [<unit>]**

(optional) Specifies the end point of the zoom range.

<unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <limit> and <unit> within curly braces ({}).

- **<topindex>**

(optional) An integer that specifies the vertical scroll position of the window. You must specify a zoom range to specify topindex. The number identifies which object the window should be scrolled to. For example, specifying 20 means the Wave window will be scrolled down to show the 20th object.

## Examples

- Add a bookmark named "foo" to the current default Wave window. The bookmark marks a zoom range from 10ns to 1000ns and a scroll position of the 20th object in the window.

```
bookmark add wave foo {{10 ns} {1000 ns}} 20
```

## Related Topics

[bookmark delete wave](#)

[bookmark goto wave](#)

[bookmark list wave](#)

## bookmark delete wave

This command deletes bookmarks from the specified Wave window.

You can also interactively delete a bookmark through the GUI by selecting the **Wave > Bookmarks > Bookmarks** menu item.

Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

### Syntax

```
bookmark delete wave {<label> | -all}
```

### Arguments

- <label> | -all

(required) Controls which bookmarks to delete. Must be specified as the first argument to the bookmark delete wave command.

<label> — Specifies the name of the bookmark to delete.

-all — Specifies that all bookmarks in the window be deleted.

### Examples

- Delete the bookmark named "foo" from the current default Wave window.

```
bookmark delete wave foo
```

### Related Topics

[bookmark add wave](#)

[bookmark goto wave](#)

[bookmark list wave](#)

# bookmark goto wave

This command zooms and scrolls a Wave window using the specified bookmark.

You can also interactively navigate between bookmarks through the GUI by selecting the **Wave > Bookmarks > Bookmarks** menu item.

Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

## Syntax

`bookmark goto wave <label>`

## Arguments

- `<label>`

(required) Specifies the bookmark to go to. Must be specified as the first argument to the bookmark goto wave command.

## Related Topics

[bookmark add wave](#)

[bookmark delete wave](#)

[bookmark list wave](#)

## bookmark list wave

This command displays a list of available bookmarks in the Transcript window.

### Syntax

`bookmark list wave`

### Arguments

none

### Related Topics

[bookmark add wave](#)

[bookmark delete wave](#)

[bookmark goto wave](#)

# bp

This command sets either a file-line breakpoint or returns a list of currently set breakpoints. It allows enum names, as well as literal values, to be used in condition expressions.

Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

## Syntax

### Setting an HDL breakpoint

```
bp {[<filename>] <line_number> | <filename>:<line_number> | in} [-ancestor] [-appendinst] [-cond "<condition_expression>"] [-disable] [-id <id_number> | -label "<label>"] [-inst <region> [-inst <region> ...]] [-uvm] [<command>...]
```

### Querying a breakpoint

```
bp [-query <filename> [<line_number>]]
```

### Reporting all breakpoints

#### Note

 If you specify this command with no arguments, it returns a list of all breakpoints in the design containing information about each breakpoint. For example, the command “bp” returns

```
# bp top.vhd 70;# 2
```

## Arguments

- <filename>  
(optional) Specifies the name of the source file in which to set the breakpoint. If you do not specify a filename, the command will use the source file of the current context.
- <line\_number>  
(required to set an HDL breakpoint) Specifies the line number where the breakpoint is to be set.
- in  
(required for task or function breakpoints) Supports the lookup of Verilog and SystemVerilog task and function names as an alternative to file name and line numbers. Places a breakpoint on the first executable line of the specified task or function. Does not work for VHDL or SystemC.
- -ancestor  
(optional) Stops the simulation only when any ancestor parent of the process matches the given process-name.

- `-appendinst`  
(optional) When specifying multiple breakpoints with `-inst`, append each instance-path condition to the earlier condition. This overrides the default behavior, in which each condition overwrites the previous one.
- `-disable`  
(optional) Sets the breakpoint to a disabled state. You can enable the breakpoint later using the `enablebp` command. This command enables breakpoints by default.
- `<command>...`  
(optional, must be specified as the final argument) Specifies one or more commands that are to be executed at the breakpoint. You must separate multiple commands with semicolons (;) or place them on multiple lines. Braces are required only if the string contains spaces.

---

**Note**

 You can also specify this command string by choosing **Tools > Breakpoints...** from the main menu and using the **Modify Breakpoints** dialog box.

---

Any commands that follow a `run` or `step` command are ignored. A run or step command terminates the breakpoint sequence. This rule also applies if you use a DO file script within the command string.

If many commands are needed after the breakpoint, you could place them in a DO file script.

- `-cond "<condition_expression>"`  
(optional) Specifies one or more conditions that determine whether the breakpoint is hit.  
" `<condition_expression>` " — A conditional expression that results in a true/false value. You must enclose the condition expression within braces ( { } ) or quotation marks (" ") when the expression makes use of spaces. Refer to the note below when setting breakpoints in the GUI.

If the condition is true, the simulation stops at the breakpoint. If false, the simulation bypasses the breakpoint. A condition cannot refer to a VHDL variable (only a signal).

The `-cond` switch re-parses expressions each time the breakpoint is hit. This allows expressions with local references to work. Condition expressions referencing items outside the context of the breakpoint must use absolute names. This is different from the behavior in previous ModelSim versions where a relative signal name was resolved at the time the `bp` command was issued, allowing the breakpoint to work even though the relative signal name was inappropriate when the breakpoint was hit.

---

**Note**

 You can also specify this expression by choosing **Tools > Breakpoints...** from the main menu and entering the expression in the **Breakpoint Condition** field of the **Modify Breakpoints** dialog box. Do not enclose the condition expression in quotation marks (" ") or braces ( { } ).

---

The condition expression can use the following operators:

Operation	Operator Syntax
equals	<code>==, =</code>
not equal	<code>!=, /=</code>
AND	<code>&amp;&amp;, AND</code>
OR	<code>  , OR</code>

The operands may be object names, signalname'event, or constants. Subexpressions in parentheses are permitted. The command will be executed when the expression is evaluated as TRUE or 1. The formal BNF syntax for an expression is:

```

condition ::= Name | { expression }
expression ::= expression AND relation
             | expression OR relation
             | relation
relation ::= Name = Literal
            | Name /= Literal
            | Name ' EVENT
            | ( expression )
Literal ::= '<char>' | "<bitstring>" | <bitstring>

```

The "`=`" operator can occur only between a Name and a Literal. This means that you cannot compare the value of two signals (for example, `Name = Name` is not valid).

You can construct a breakpoint such that the simulation breaks when a SystemVerilog Class is associated with a specific handle, or address:

```

bp <filename> <line_number> -cond "this==<class_handle>"
bp <filename> <line_number> -cond "this!=<class_handle>"

```

where you can obtain the class handle with the [examine](#) -handle command. The string "this" is a literal that refers to the specific *line\_number*.

You can construct a breakpoint such that the simulation breaks when a line number is of a specific class type or extends the specified class type:

```
bp <filename> <line_number> -cond "this ISA <class_type_name>"
```

where *class\_type\_name* is the actual class name, not a variable.

- `-id <id_number> | -label "<label>"`

(optional) Attempts to assign an id number or label to the breakpoint. The command returns an error if the id number you specify is already assigned.

`-id <id_number>` — Any positive integer that is not already assigned.

`-label "<label>"` — Associates a name or label with the specified breakpoint. Adds a level of identification to the breakpoint. The label may contain special characters.

---

Quotation marks (" ") or braces ({ }) are required only if <label> contains spaces or special characters.

---

**Note**

 Id numbers for breakpoints are assigned from the same pool as those used for the **when** command. So even if you have not specified a given id number for a breakpoint, that number may still be used for a when command.

---

- -inst <region> [-inst <region> ...]

(optional) Sets an HDL breakpoint so it applies only to the specified instance.

To apply multiple instance-path conditions on a single breakpoint, specify -inst <region> multiple times. By default, this overrides the previous breakpoint condition (you can use the -appendinst argument to append conditions instead).

<region> — The full path to the instance specified.

---

**Note**

 You can also specify this instance by choosing **Tools > Breakpoints...** from the main menu and using the **Modify Breakpoints** dialog box.

---

- -query <filename> [<line\_number>]

(optional) Returns information about the breakpoint(s) set in the specified file. The information returned varies depending on the condition of the breakpoint(s) in the specified file. Returns a complete list of all breakpoints and whether they are enabled or not when specified without <line\_number>. Returns nothing if <line\_number> is not executable.

<filename> — The name of the file containing the breakpoint.

<line\_number> — The line number where a breakpoint has been set.

The output contains six fields of information. For example:

```
bp -query top.vhd 70
```

Returns

```
# 1 1 top.vhd 70 2 1
```

- {1 | 0} — Indicates whether a breakpoint exists at the location.
  - 0 — Breakpoint does not exist.
  - 1 — Breakpoint exists.
- 1 — always reports a 1.
- <file\_name>
- <line\_number>
- <id\_number>

- {1 | 0} — Indicates whether the breakpoint is enabled.
  - 0 — Breakpoint is not enabled.
  - 1 — Breakpoint is enabled.
- **-uvm**  
Specifies UVM-style instance name(s) for setting source breakpoints on class instances in the UVM hierarchy. Must be followed by the **-inst <instance\_name>** option.

## Examples

- List all existing breakpoints in the design, including the source file names, line numbers, breakpoint id numbers, labels, and any commands that have been assigned to the breakpoints.

**bp**

- Set a breakpoint in the source file *alu.vhd* at line 147.

**bp alu.vhd 147**

- Set a breakpoint at line 153 of the source file of the current context:

**bp 153**

- Execute the *macro.do* DO file when the breakpoint is hit.

**bp alu.vhd 147 {do macro.do}**

- Set a breakpoint on line 22 of *test.vhd*. When the breakpoint is hit, the values of variables *var1* and *var2* are examined. This breakpoint is initially disabled; it can be enabled with the [enablebp](#) command.

**bp -disable test.vhd 22 {echo [exa var1]; echo [exa var2]}**

- Set a breakpoint so that the simulation pauses whenever *clk=1* and *prdy=0*:

**bp test.vhd 14 -cond {clk=1 AND prdy=0}**

- Set a breakpoint with the label *top\_bp*

**bp top.vhd 14 -label top\_bp**

- Set a breakpoint for line 15 of *a.vhd*, but only for the instance *a2*:

**bp a.vhd 15 -inst "/top/a2"**

- Set multiple breakpoints in the source file *test.vhd* at line 14. The second instance will overwrite the conditions of the first.

**bp test.vhd 14 -inst /test/inst1 -inst /test/inst2**

- Set multiple breakpoints at line 14. The second instance will append its conditions to the first.

**bp test.vhd 14 -inst /test/inst1 -inst /test/inst2 -appendinst**

- Set a breakpoint for a specific variable of a particular class type:  
**set x [examine -handle my\_class\_var]**  
**bp top.sv 15 -cond {this == \$x}**
- Set a breakpoint on the first executable line of the function /uvm\_pkg::set\_config\_int.  
**bp in /uvm\_pkg::set\_config\_int**
- List the line number and enabled/disabled status (1 = enabled, 0 = disabled) of all breakpoints in *testadd.vhd*.  
**bp -query testadd.vhd**
- List details about the breakpoint on line 48.  
**bp -query testadd.vhd 48**
- List all executable lines in *testadd.vhd* between lines 2 and 59.  
**bp -query testadd.vhd 2 59**

## Related Topics

[bd](#)

[Editing File-Line Breakpoints](#)

# call

This command calls the following types of functions/tasks:

- SystemVerilog static functions and class functions directly from the vsim command line in live simulation mode. Tasks are not supported.
- PLI and VPI system tasks and system functions.

Function return values are returned to the vsim shell as a Tcl string. If the function returns a class reference, the class instance ID is returned.

## Syntax

---

### Note

---

 Note the grouping of the following arguments, which indicates slightly different usage depending on how you want the command to perform a search on a system task or function name.

- <pathToFunction> <classInstancePath>  
SystemVerilog static functions and class functions
- -usertf -builtin <systfName>  
PLI and VPI system tasks and system functions (restricts the search for the task/function name to either the user-added PLI/VPI routines or to the built-in routines):

---

```
call [-env <hierEnvPath>] [{<pathToFunction> [<classInstancePath>]}] [{-usertf | -builtin <systfName>}] [<arg1> [<arg2>] ... [<argN>]]
```

## Arguments

- -env <hierEnvPath>  
(optional) Hierarchical environment path, to be used as the starting scope for the object name lookups. If present, must appear before actual function name.
- <pathToFunction>  
(required when calling a System Verilog static or class function) The name of a function, which you can qualify in any of the following ways:
  - By specifying the path to the function declaration, through the structural hierarchy, or declaration hierarchy. Hierarchical paths must be specified as a full path to a function or a function that exists relative to the current context (as shown in the Structure window, or returned by the environment command).
  - By specifying a class instance hierarchical path.
  - By specifying a class instance id string.

- <classInstancePath>

(optional) Must be specified if the function path is a declaration path and the function is a non-static class function. Conversely, the class instance path name must not be specified if the given function path is a class instance variable reference or a class instance name in the format @<class\_type>@nnn. This is because the class instance information can be extracted from the pathname itself.

- -builtin

(optional) Search only built-in system functions or task names. (The \$ is understood to be a prefix.)

- -usertf

(optional) Search only user-defined system functions or task names. (The \$ is understood to be a prefix.)

- <systfName>

(required when calling a system task or function) The system task or function to be executed. You can specify the sysfName according to the following syntax rules:

- \\${<systfName>} (for example, \\$display or \\$mytask)
- With –usertf or –builtin flags (as the case may be).  
(for example, –usertf mytask or –builtin display)
- <systfName> (for example, display or mytask) In this case SystemVerilog static functions and class functions will be searched first and then the PLI/VPI system tasks/functions, for a match.

- <arg1> [<arg2>] ...[<argN>]

(optional) All arguments required by the function are specified in a space-separated list in declaration order. If a function has default arguments, the arguments may be omitted from the command line provided that the arguments occur at the end of the declaration list.

Function input arguments can be constant values including integers, enumerated values, and strings. A string containing spaces or special characters must be enclosed in quotation marks (" ") or braces ( { } ) or Tcl will try to interpret the string. For example: "my string" or {my string}. Arguments can also be design objects. Class references can be arguments, specified by either their design instance path or class instance id string. If a function has type inout, out, or ref arguments, suitable user design objects must be passed in as arguments. Any passed in argument will first be tested to determine if it is an appropriate constant value. If it is not, then the argument will be tested to determine if it is a design object. Consequently, where there is ambiguity between a constant string and the name of a design object, the constant will be given precedence. If in this case the design object is desired, the full hierarchical path to the object can be supplied to differentiate it from the constant string.

## Examples

### Calling a System Verilog Static of Class Function

- Call using a static declaration path, where the function *sf\_voidstring()* is a static class function that accepts a string:

```
call sim:/user_pkg::myfcns::sf_voidstring first_string
```

- Call using a class instance path to specify the function, where the function *f\_intint()* of the class type */utop/tmyfcns* accepts an integer:

```
call /utop/tmyfcns.f_intint 37
```

- Call using a class instance path to specify the function, and pass in a class instance (*/utop/tmyfcns* is a class handle):

```
call /utop/tmyfcns.f_voidclasscolor /utop/tmyfcns
```

- Call using a class instance path, and pass in a class instance as an argument using a class instance id string

```
call /utop/tmyfcns.f_voidclasscolor @myType@3
```

- Call using a class instance id string to specify the function:

```
call @myType@543.get_full_name
```

- Call using a declaration path, where the function is non-static so a class instance must also be supplied. The member function *f\_voidstring()* accepts a string:

```
call sim:/user_pkg::myfcns::f_voidstring /my/class/instance "some string"
```

- Call using a class instance id string to specify the function where the function returns a string:

```
call @uvm_sequencer_3@3.get_full_name
```

Returns:

```
# test.e2_a.sequencer
```

- Call using a relative class hierarchical name to specify the function where the function returns a class handle:

```
call moduleX.who_am_i
```

Returns:

```
# @myClassX@4
```

### Calling a System Task

- Call \$display with literal values:

```
call \$display {"%0s"} {"Hello from TCL!"}
```

---

Returns:

```
# Hello from TCL!
call -builtin display {"%0d"} 'd2999
```

Returns:

```
# 2999
call -usertf display {"%0d"} 'd3999
```

Returns:

```
# ** Error: Expected user-defined system task $display not found in
the context (/top2).
```

- Call \$display with literal values:

```
call \$display {"top2.i=%0d top2.r=%0b"} top2/i top2/r
```

Returns:

```
# top2.i=5 top2.r=110
```

### Calling a System Function

In the following examples \$pow is a user defined function that raises the 1<sup>st</sup> argument to the power of the 2<sup>nd</sup> (for example, \$pow(a, b) => ab)

- Call \$pow with literal values:

```
call \$pow 2 1
```

Returns:

```
# 2
```

```
call \$pow 3 2
```

Returns:

```
# 9
```

```
call \$pow [call \$pow 2 1] [call \$pow 3 2]
```

Returns:

```
# 512
```

- Call \$pow with variable values:

```
call -env /top/u1 display r1
```

Returns:

```
# 7
```

**call –env /top/u2 display r2**

Returns:

# 2

**call pow /top/u1/r1 /top/u2/r2**

Returns:

# 49

## cd

This command changes the ModelSim local directory to the specified directory.

This command cannot be executed while a simulation is in progress. Also, executing a **cd** command will close the current project.

### Syntax

```
cd [<dir>]
```

### Arguments

- **<dir>**  
(optional) Specifies a full or relative directory path for ModelSim to use as the local directory. If you do not specify a directory, the command changes to your home directory.

# change

This command modifies the value of a: VHDL constant, generic, or variable; Verilog register or variable

## Syntax

```
change <variable> <value>
```

## Description

For VHDL constants, The change command may not affect any and all uses of deferred (or other) constants. Refer to the following technote for more information:

<http://supportnet.mentor.com/portal?do=reference.technote&id=MG588185>

## Arguments

- <variable>

(required) A string that specifies the name of an object. The name can be a full hierarchical name or a relative name, where a relative name is relative to the current environment.

Wildcards are not permitted.

The following sections list supported objects:

- VHDL
  - Scalar variable, constant, or generics of all types except FILE.  
Generates a warning when changing a VHDL constant or generic. You can suppress this warning by setting the TCL variable WarnConstantChange to 0 or in the [vsim] section of the *modelsim.ini* file.
  - Scalar subelement of composite variable, constant, and generic of all types except FILE.
  - One-dimensional array of enumerated character types, including slices.
  - Access type. An access type pointer can be set to "null"; the value that an access type points to can be changed as specified above.
- Verilog
  - Parameter.
  - Register or memory.
  - Integer, real, realtime, time, and local variables in tasks and functions.
  - Subelements of register, integer, real, realtime, and time multi-dimensional arrays (all dimensions must be specified).

- Bit-selects and part-selects of the above except for objects whose basic type is real.

The name can be a full hierarchical name or a relative name. A relative name is relative to the current environment. Wildcards cannot be used.

- <value>

(required) Defines a value for <variable>. The specified value must be appropriate for the type of the variable. You must place <value> within quotation marks (""). If the string contains spaces, the quoted string must be placed inside curly braces ({}).

---

**Note**



The initial type of <variable> determines the type of value that it can be given. For example, if <variable> is initially equal to 3.14 then only real values can be set on it. Also note that changing the value of a parameter or generic will not modify any design elements that depended on the parameter or generic during elaboration (for example, sizes of arrays).

---

## Examples

- Change the value of the variable *count* to the hexadecimal value FFFF.  
**change count 16#FFFF**
- Change the value of the element of *regA* that is specified by the index (for example, 16).  
**change {regA[16]} 0**
- Change the value of the set of elements of *foo* that is specified by the slice (for example, 20:22).  
**change {foo[20:22]} 011**
- Set the Verilog register *file\_name* to "test2.txt". Note that the quote marks are escaped with '\'.  
**change file\_name \"test2.txt\"**
- Set the time value of the mytimegeneric variable to 500 ps. The time value is enclosed by curly braces because of the space between the value and the units.  
**change mytimegeneric {"500 ps"}**

# classinfo ancestry

This command returns class inheritance hierarchy for a named class type.

## Syntax

```
classinfo ancestry [-dataset <name>] [-n] [-o <outfile>] [-tcl] <class_type>
```

## Arguments

- <class\_type>  
(required) Name of the class type or the full path to the class type.
- -dataset <name>  
(optional) Specifies an open dataset to search for class information. The default is to search the currently active dataset.  
  
    <name> — The name of an open dataset.
- -n  
(optional) Returns class type names only. Does not include the path unless required to resolve name ambiguity.
- -o <outfile>  
(optional) Sends the results of the command to <outfile> instead of the transcript.  
  
    <outfile> — Specifies the name of the file where the output will be written.
- -tcl  
(optional) Returns a tcl list instead of formatted output.

## Related Topics

- [ClassDebug](#)
- [classinfo descriptive](#)
- [classinfo find](#)
- [classinfo instances](#)
- [classinfo isa](#)
- [classinfo report](#)
- [classinfo stats](#)
- [classinfo trace](#)
- [classinfo types](#)

# classinfo descriptive

This command returns the descriptive class name for the specified authoritative class name.

## Syntax

```
classinfo descriptive [-dataset <name>] [-exact | -glob | -regexp] [-tcl] [-o <outfile>]  
    <class_type>
```

## Arguments

- <class\_type>  
(required) Treats <class\_type> as a glob-style expression and returns all matches to the transcript. Wildcard characters asterisk ( \* ) and question mark ( ? ) are permitted.
- -dataset <name>  
(optional) Specifies an open dataset to search for class information. The default is to search the currently active dataset.  
    <name> — The name of an open dataset.
- -exact  
(optional) Returns results that match <class\_type> exactly.
- -glob  
(optional, default) Treats <class\_type> as a glob-style expression. Wildcard characters asterisk ( \* ) and question mark ( ? ) are permitted.
- -o <outfile>  
(optional) Sends the results of the command to <outfile> instead of the transcript.  
    <outfile> — Specifies the name of the file where the output will be written.
- -regexp  
(optional) Treats <class\_name> as a regular expression.
- -tcl  
(optional) Returns a tcl list instead of formatted output.

## Related Topics

[ClassDebug](#)

[Logging Class Types and Class Instances](#)

[Working with Class Types](#)

[Analyzing Class Types](#)

[classinfo ancestry](#)

[classinfo find](#)

[classinfo instances](#)

[classinfo isa](#)

[classinfo report](#)

[classinfo stats](#)

[classinfo trace](#)

[classinfo types](#)

## classinfo find

This command reports on the current state of a specified class instance, whether it exists, has not yet been created, or has been destroyed.

### Syntax

```
classinfo find [-dataset <name>] [-tcl] [-o <outfile>] <class_instance_identifier>
```

### Arguments

- <class\_instance\_identifier>  
(required) Class instance identifier of the specific class instance to find.
- -dataset <name>  
(optional) Specifies an open dataset to search for class information. The default is to search the currently active dataset.  
    <name> — The name of an open dataset.
- -o <outfile>  
(optional) Sends the results of the command to <outfile> instead of the transcript.  
    <outfile> — Specifies the name of the file where the output will be written.
- -tcl  
(optional) Returns a tcl list instead of formatted output.

### Examples

- Find the class instance @mem\_item@87

```
VSIM> classinfo find @mem_item@87
```

Returns:

```
# @mem_item@87 has been destroyed
```

- Find the class instance @mem\_item@200

```
VSIM> classinfo find @mem_item@200
```

Returns:

```
# @mem_item@200 not yet created
```

### Related Topics

- [ClassDebug](#)
- [classinfo ancestry](#)
- [classinfo descriptive](#)
- [classinfo instances](#)

[classinfo isa](#)  
[classinfo report](#)  
[classinfo stats](#)  
[classinfo trace](#)  
[classinfo types](#)

# classinfo implements

This command displays a list of which classes implement SystemVerilog interface classes. The type of the class argument affects the contents of this list.

## Syntax

```
classinfo implements [-dataset <name>] [-tcl] [-o <outfile>] <class_type>
```

## Arguments

- <class\_type>

(required) Name of the SystemVerilog class that you want to use to generate the output listing. The type of this class determines the type of classes listed, as follows:

- If <class\_type> is not an interface class, the output indicates which interface classes that class implements.
- If <class\_type> is an interface class, the output indicates which classes implement that interface class.

- -dataset <name>

(optional) Specifies an open dataset to search for class information. The default is to search the currently active dataset.

    <name> — The name of an open dataset.

- -o <outfile>

(optional) Sends the results of the command to <outfile> instead of the transcript.

    <outfile> — Specifies the name of the file where the output will be written.

- -tcl

(optional) Returns a tcl list instead of formatted output.

## Examples

The following module defines classes labeled A, B, C1, C2, M, N, X, Y:

```
module test8;
    interface class M; endclass
    interface class N; endclass
    interface class X extends M, N; endclass
    interface class Y extends M; endclass
    class A implements M; endclass
    class B extends A implements X; endclass
    class C1 extends B implements Y; endclass
    class C2 extends B; endclass
endmodule
```

- Use interface class M as argument:

```
vsim> classinfo implements M
```

Output list:

```
# /test8/A implements /test8/M
# /test8/B implements /test8/M
# /test8/C1 implements /test8/M
# /test8/C2 implements /test8/M
```

- Use class A as argument:

**vsim> classinfo implements A**

Output list:

```
# /test8/A implements /test8/M
```

- Use class B as argument to access extended classes defined in test8:

**vsim> classinfo implements B**

Output list:

```
# /test8/B implements /test8/M
# /test8/B implements /test8/N
# /test8/B implements /test8/X
```

## Related Topics

[ClassDebug](#)

[classinfo descriptive](#)

[classinfo find](#)

[classinfo instances](#)

[classinfo interfaces](#)

[classinfo isa](#)

[classinfo report](#)

[classinfo stats](#)

[classinfo trace](#)

[classinfo types](#)

# classinfo instances

This command reports the list of existing class instances of a specific class type. You can use this to determine what class instances to log or examine. It can also help in debugging problems where class instances are not being cleaned up as they should be, resulting in excessive memory usage.

## Syntax

```
classinfo instances [-dataset <name>] [-tcl] [-verbose] [-o <outfile>] <class_type>
```

## Arguments

- <class\_type>  
(required) Name of the class type or the full path of the class type. If this is an interface class, the output lists all instances that implement that interface class.
- -dataset <name>  
(optional) Specifies an open dataset to search for class information. The default is to search the currently active dataset.  
  
    <name> — The name of an open dataset.
- -tcl  
(optional) Returns a tcl list instead of formatted output.
- -verbose  
(optional) Includes the classname in the output along with the instance name.
- -o <outfile>  
(optional) Sends the results of the command to <outfile> instead of the transcript.  
  
    <outfile> — Specifies the name of the file where the output will be written.

## Examples

- List the current instances for the class type mem\_item.

```
vsim> classinfo instances mem_item
```

Returns:

```
# @mem_item@140
# @mem_item@139
# @mem_item@138
# @mem_item@80
# @mem_item@76
# @mem_item@72
# @mem_item@68
# @mem_item@64
```

- The following module defines classes labeled A, B, C1, C2, M, N, X, Y:

```
module test8;
  interface class M;  endclass
  interface class N;  endclass
  interface class X extends M, N; endclass
  interface class Y extends M;  endclass
  class A implements M; endclass
  class B extends A implements X; endclass
  class C1 extends B implements Y; endclass
  class C2 extends B; endclass
endmodule
```

The following commands show the difference between using and omitting the -verbose argument.

**vsim> classinfo instances -verbose M**

Returns:

```
# @A@1 /test8/A
# @B@1 /test8/B
```

**vsim> classinfo instances -verbose A**

Returns:

```
# @A@1 /test8/A
```

**vsim> classinfo instances M**

Returns:

```
# @A@1
# @B@1
```

**vsim> classinfo instances A**

Returns:

```
# @A@1
```

## Related Topics

[ClassDebug](#)

[classinfo ancestry](#)

[classinfo descriptive](#)

[classinfo find](#)

[classinfo implements](#)

[classinfo interfaces](#)

[classinfo isa](#)

[classinfo report](#)

[classinfo stats](#)

[classinfo trace](#)

[classinfo types](#)

# classinfo interfaces

This command lists the interface class types that match or do not match a specified pattern. Finds all interface classes that match a regular expression and determines the full path of interface class types.

## Syntax

```
classinfo interfaces [-dataset <name>] [-tcl] [-o <outfile>] [<class_type>]
```

## Arguments

- <class\_type>  
(optional) Name of the interface class type or the full path to the interface class type. If omitted, all interface classes are listed.
- -dataset <name>  
(optional) Specifies an open dataset to search for interface class information. The default is to search the currently active dataset.  
    <name> — The name of an open dataset.
- -o <outfile>  
(optional) Sends the results of the command to <outfile> instead of the transcript.  
    <outfile> — Specifies the name of the file where the output will be written.
- -tcl  
(optional) Returns a tcl list instead of formatted output.

## Examples

The following module defines classes labeled A, B, C1, C2, M, N, X, Y:

```
module test8;
    interface class M; endclass
    interface class N; endclass
    interface class X extends M, N; endclass
    interface class Y extends M; endclass
    class A implements M; endclass
    class B extends A implements X; endclass
    class C1 extends B implements Y; endclass
    class C2 extends B; endclass
endmodule
```

- Use with no argument, which will return the names of all interface classes:

```
vsim> classinfo interfaces
```

Output list:

```
# /test8/M
# /test8/N
# /test8/X
# /test8/Y
```

## Related Topics

[ClassDebug](#)

[classinfo descriptive](#)

[classinfo find](#)

[classinfo implements](#)

[classinfo instances](#)

[classinfo isa](#)

[classinfo report](#)

[classinfo stats](#)

[classinfo trace](#)

[classinfo types](#)

# classinfo isa

This command returns to the transcript a list of all classes extended from the specified class type.

## Syntax

```
classinfo isa [-dataset <name>] [-n] [-o <outfile>] [-tcl] <class_type>
```

## Arguments

- <class\_type>  
(required) Name of the class type or the full path of the class type.
- -dataset <name>  
(optional) Specifies an open dataset to search for class information. The default is to search the currently active dataset.  
    <name> — The name of an open dataset.
- -n  
(optional) Returns class names only. Does not include the path unless required to resolve name ambiguity.
- -o <outfile>  
(optional) Sends the results of the command to <outfile> instead of the transcript.  
    <outfile> — Specifies the name of the file where the output will be written.
- -tcl  
(optional) Returns a tcl list instead of formatted output.

## Related Topics

- [ClassDebug](#)
- [classinfo ancestry](#)
- [classinfo descriptive](#)
- [classinfo find](#)
- [classinfo instances](#)
- [classinfo report](#)
- [classinfo stats](#)
- [classinfo trace](#)
- [classinfo types](#)

## classinfo report

This command prints detailed reports on class instance usage. The command displays columns for class type names and their current, peak and total class instance counts. The columns may be arranged, sorted, or eliminated using the command arguments.

### Syntax

```
classinfo report [-c [fntpc]] [-dataset <name>] [-m <maxout>] [-o <outfile>]  
[-sort [a | d] [f | n | t | p | c]] [-tcl] [-z]
```

### Arguments

- **-c [fntpc]**  
(optional) Display the report columns in the specified order in a report. The default is to display all columns in the following order: Full Path, Class Name, Total, Peak, Current. You can specify one or more columns in any order.
  - f — The Full Path column displays the full relative path name.
  - n — The Class Name column displays the name of the class instance.
  - t — The Total column displays the total number of instances of the named class.
  - p — The Peak column displays the maximum number of instances of the named class that existed simultaneously at any time in the simulation.
  - c — The Current column displays the current number of instances of the named class.
- **-dataset <name>**  
(optional) Specifies an open dataset to search for class information. The default is to search the currently active dataset.
  - <name> — The name of an open dataset.
- **-m <maxout>**  
(optional) Display the specified number of lines of the report.
  - <maxout> — Any non-negative integer.
- **-o <outfile>**  
(optional) Sends the results of the command to <outfile> instead of the transcript.
  - <outfile> — Specifies the name of the file where the output will be written.
- **-sort [a | d] [f | n | t | p | c]**  
(optional) Specifies whether the report information is sorted in ascending or descending order and which column to sort by. Only one column can be specified for sorting.
  - a — Sort the entries in ascending order.
  - d — Sort the entries in descending order.
  - f — Sort by the Full Path column

n— Sort by the Class Name column  
t— Sort by the Total column  
p— Sort by the Peak column  
c— Sort by the Current column

- -tcl  
(optional) Returns a tcl list instead of formatted output.
- -z  
(optional) Remove all items from the report with a total instance count of zero.

## Examples

- Create a report of all class instances in descending order in the Total column. Print the Class Names, Total, Peak, and Current columns. List only the first six lines of that report.

```
vsim> classinfo report -s dt -c ntpc -m 6
```

Returns:

# Class Name	Total	Peak	Current
# uvm_pool_11	318	315	315
# uvm_event	286	55	52
# uvm_callback_iter_1	273	3	2
# uvm_queue_3	197	13	10
# uvm_object_string_pool_1	175	60	58
# mem_item	140	25	23

## Related Topics

[ClassDebug](#)  
[classinfo ancestry](#)  
[classinfo descriptive](#)  
[classinfo find](#)  
[classinfo instances](#)  
[classinfo isa](#)  
[classinfo stats](#)  
[classinfo trace](#)  
[classinfo types](#)

## classinfo stats

This command prints statistics about the total number of class types and total, peak, and current class instance counts during the simulation.

### Syntax

```
classinfo stats [-dataset <name>] [-tcl] [-o <outfile>]
```

### Arguments

- **-dataset <name>**  
(optional) Specifies an open dataset to search for class information. The default is to search the currently active dataset.  
    <name> — The name of an open dataset.
- **-o <outfile>**  
(optional) Sends the results of the command to <outfile> instead of the transcript.  
    <outfile> — Specifies the name of the file where the output will be written.
- **-tcl**  
(optional) Returns a tcl list instead of formatted output.

### Examples

- Display the current number of class types, the maximum number, peak number and current number of all class instances.

```
vsim> classinfo stats
```

Returns:

```
# class type count          451
# class instance count (total) 2070
# class instance count (peak) 1075
# class instance count (current) 1058
```

### Related Topics

- [ClassDebug](#)
- [classinfo ancestry](#)
- [classinfo descriptive](#)
- [classinfo find](#)
- [classinfo instances](#)
- [classinfo isa](#)
- [classinfo report](#)
- [classinfo trace](#)

[classinfo types](#)

## classinfo trace

This command displays the active references to the specified class instance. This is very useful in debugging situations where class instances are not being destroyed as expected because something in the design is still referencing them. Finding those references may lead to uncovering bugs in managing these class references which often lead to large memory savings.

### Syntax

```
classinfo trace [-dataset <name>] [-m <maxout>] [-tcl] [-o <outfile>] <class_instance_name>
```

### Arguments

- <class\_instance\_name>  
(required) Name of the class item in the following format @<name>@#.
- -dataset <name>  
(optional) Specifies an open dataset to search for class information. The default is to search the currently active dataset.  
    <name> — The name of an open dataset.
- -m <maxout>  
(optional) Display the specified number of lines of the report.  
    <maxout> — Any non-negative integer.
- -o <outfile>  
(optional) Sends the results of the command to <outfile> instead of the transcript.  
    <outfile> — Specifies the name of the file where the output will be written.
- -tcl  
(optional) Returns a tcl list instead of formatted output.

### Examples

- Report the active references to @mem\_item@200

```
VSIM> classinfo trace @uvm_resource_14@2
```

Returns:

```
# {uvm_pkg::uvm_resources.rtab["mem_interface"].queue[15] }
# {uvm_pkg::uvm_config_db::uvm_config_db_12::m_rsc[@uvm_root@1].
pool["uvm_test_topmem_interface"] }
```

### Related Topics

[ClassDebug](#)

[classinfo ancestry](#)

[classinfo descriptive](#)

[classinfo find](#)

[classinfo instances](#)

[classinfo isa](#)

[classinfo report](#)

[classinfo stats](#)

[classinfo types](#)

## classinfo types

This command lists the class types that match or do not match a specified pattern. Finds all classes that match a regular expression and determines the full path of class types.

### Syntax

```
classinfo types [-dataset <name>] [-exact | -glob | -regexp] [-n] [-o <outfile>] [-tcl] [-x] <pattern>
```

### Arguments

- <pattern>  
(required) A standard TCL glob expression used as a search string.
- -dataset <name>  
(optional) Specifies an open dataset to search for class information. The default is to search the currently active dataset.  
    <name> — The name of an open dataset.
- -exact  
(optional) Returns results that match <pattern> exactly.
- -glob  
(optional) Returns glob styles matches for <pattern>.
- -n  
(optional) Returns class names only. Does not include the path unless required to resolve name ambiguity.
- -o <outfile>  
(optional) Sends the results of the command to <outfile> instead of the transcript.  
    <outfile> — Specifies the name of the file where the output will be written.
- -regexp  
(optional) Returns regular expressions that match <pattern>.
- -tcl  
(optional) Returns a tcl list instead of formatted output.
- -x  
(optional) Display classes that do not match the pattern.

### Examples

- List the full path of the class types that do not match the pattern \*uvm\*.

```
vsim> classinfo types -x *uvm*
```

Returns:

```
# /environment_pkg::test_predictor
# /environment_pkg::threaded_scoreboard
# /mem_agent_pkg::mem_agent
# /mem_agent_pkg::mem_config
# /mem_agent_pkg::mem_driver
```

## Related Topics

[ClassDebug](#)

[classinfo ancestry](#)

[classinfo descriptive](#)

[classinfo find](#)

[classinfo implements](#)

[classinfo instances](#)

[classinfo interfaces](#)

[classinfo isa](#)

[classinfo report](#)

[classinfo stats](#)

[classinfo trace](#)

# configure

The configure command invokes the List or Wave widget configure command for the current default List or Wave window. Some arguments to this command are order-dependent. Please read through the arguments for further information.

## Syntax

### Base Command Usage

```
configure list | wave [<option> <value>]
```

### List Window Arguments

```
[-delta [all | collapse | events | none]] [-gateduration [<duration_open>]]  
[-gateexpr [<expression>]] [-usegating [off | on]] [-strobeperiod [<period>[<unit>]]]  
[-strobestart [<start_time>[<unit>]]] [-usesignaltriggers [0 | 1]] [-usestrobe [0 | 1]]
```

### Wave Window Arguments

```
[-childrowmargin [<pixels>]] [-cursorlockcolor [<color>]] [-gridauto [off | on]]  
[-gridcolor [<color>]] [-griddelta [<pixels>]] [-gridoffset [<time>[<unit>]]]  
[-gridperiod [<time>[<unit>]]] [-namecolwidth [<width>]] [-rowmargin [<pixels>]]  
[-signalnamewidth [<value>]] [-timecolor [<color>]] [-timeline [0 | 1]]  
[-timelineunits [fs | ps | ns | us | ms | sec | min | hr]] [-valuecolwidth [<width>]]  
[-vectorcolor [<color>]] [-waveselectcolor [<color>]] [-waveselectenable [0 | 1]]
```

## Description

The command works in three modes:

- without options or values it returns a list of all attributes and their current values
- with just an option argument (without a value) it returns the current value of that attribute
- with one or more option-value pairs it changes the values of the specified attributes to the new values

The returned information has five fields for each attribute: the command-line switch, the Tk widget resource name, the Tk class name, the default value, and the current value.

## Arguments

- list | wave

(required) Controls the widget to configure. Must be specified as the first argument to the configure command.

list — Specifies the List widget.

wave — Specifies the Wave widget.

- <option> <value>

-bg <color> — (optional) Specifies the window background color.

- fg <color> — (optional) Specifies the window foreground color.
- selectbackground <color> — (optional) Specifies the window background color when selected.
- selectforeground <color> — (optional) Specifies the window foreground color when selected.
- font <font> — (optional) Specifies the font used in the widget.
- height <pixels> — (optional) Specifies the height in pixels of each row. .

**Arguments, List window only**

- -delta [all | collapse | events | none]
  - (optional) Specifies how information is displayed in the delta column. To use -delta, -usesignaltriggers must be set to 1 (on).
    - all — Displays a new line for each time step on which objects change.
    - collapse — Displays the final value for each time step.
    - events — Displays an "event" column rather than a "delta" column and sorts List window data by event.
    - none — Turns off the display of the delta column.
- -gateduration [<duration\_open>]
  - (optional) Extends gating beyond the back edge (the last list row in which the expression evaluates to true). The duration for gating to remain open beyond when -gateexpr (below) becomes false, expressed in x number of timescale units. The default value for normal synchronous gating is zero. If -gateduration is set to a non-zero value, a simulation value will be displayed after the gate expression becomes false (if you don't want the values displayed, set -gateduration to zero).
    - <duration\_open> — Any non-negative integer where the default is 0 (values are not displayed).
- -gateexpr [<expression>]
  - (optional) Specifies the expression for trigger gating. (Use the -usegating argument to enable trigger gating.) The expression is evaluated when the List window would normally have displayed a row of data.
    - <expression> — An expression.
- -usegating [off | on]
  - (optional) Enables triggers to be gated on or off by an overriding expression. (Use the -gatexpr argument to specify the expression.) Refer to “[Using Gating Expressions to Control Triggering](#)” for additional information on using gating with triggers.
    - off — (default) Triggers are gated off (a value of 0).
    - on — Triggers are gated on (a value of 1).

- **-strobeperiod** [<period>[<unit>]]  
(optional) Specifies the period of the list strobe.  
    <period> — Any non-negative integer.  
    <unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <delay> and <unit> within curly braces ({}).
- **-strobestart** [<start\_time>[<unit>]]  
(optional) Specifies the start time of the list strobe.  
    <start\_time> — Any non-negative integer.  
    <unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <delay> and <unit> within curly braces ({}).
- **-usesignaltriggers** [0 | 1]  
(optional) Specifies whether or not signals are to be used as triggers.  
    0 — Signals are not used as triggers  
    1 — Signals are used as triggers
- **-usestrobe** [0 | 1]  
(optional) Specifies whether or not a strobe is used as a trigger.  
    0 — Strobe is not used to trigger.  
    1 — Strobe is used to trigger.

**Arguments, Wave window only**

- **-childrowmargin** [<pixels>]  
(optional) Specifies the distance in pixels between child signals. Related Tcl variable is PrefWave(childRowMargin).  
    <pixels> — Any non-negative integer where the default is 2.
- **-cursorlockcolor** [<color>]  
(optional) Specifies the color of a locked cursor. Related Tcl variable is PrefWave(cursorLockColor).  
    <color> — Any Tcl color where the default is red.
- **-gridauto** [off | on]  
(optional) Controls the grid period when in simulation time mode.  
    off — (default) user-specified grid period is used.  
    on — grid period is determined by the major tick marks in the time line.

- **-gridcolor [<color>]**  
(optional) Specifies the background grid color. Related Tcl variable is PrefWave(gridColor).  
    <color> — Any color where the default is grey50.
- **-griddelta [<pixels>]**  
(optional) Specifies the closest (in pixels) two grid lines can be drawn before intermediate lines will be removed. Related Tcl variable is PrefWave(gridDelta).  
    <pixels> — Any non-negative integer where the default is 40.
- **-gridoffset [<time>[<unit>]]**  
(optional) Specifies the time (in user time units) of the first grid line. Related Tcl variable is PrefWave(gridOffset).  
    <time> — Any non-negative integer where the default is 0.  
    <unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <delay> and <unit> within curly braces ({}).
- **-gridperiod [<time>[<unit>]]**  
(optional) Specifies the time (in user time units) between subsequent grid lines. Related Tcl variable is PrefWave(gridPeriod).  
    <time> — Any non-negative integer where the default is 1.  
    <unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <delay> and <unit> within curly braces ({}).
- **-namecolwidth [<width>]**  
(optional) Specifies the width of the name column in pixels. Related Tcl variable is PrefWave(nameColWidth).  
    <width> — Any non-negative integer where the default is 150.
- **-rowmargin [<pixels>]**  
(optional) Specifies the distance between top-level signals in pixels. Related Tcl variable is PrefWave(rowMargin).  
    <pixels> — Any non-negative integer where the default is 4.
- **-signalnamewidth [<value>]**  
(optional) Controls the number of hierarchical regions displayed as part of a signal name shown in the pathname pane. Related Tcl variable is PrefWave(SignalNameWidth). Can also be set with the WaveSignalNameWidth variable in the *modelsim.ini* file.

**<value>** — Any non-negative integer where the default is 0 (display the full path. For example, 1 displays only the leaf path element, 2 displays the last two path elements, and so on.

- **-timecolor [<color>]**

(optional) Specifies the time axis color. Related Tcl variable is PrefWave(timeColor).

**<color>** — Any color where the default is green.

- **-timeline [0 | 1]**

(optional) Specifies whether the horizontal axis displays simulation time or grid period count. Related Tcl variable is PrefWave(timeline).

0 — (default) Simulation time is displayed.

1 — Grid period count is displayed.

- **-timelineunits [fs | ps | ns | us | ms | sec | min | hr]**

(optional) Specifies units for timeline display. Does not affect the currently-defined simulation time.

fs — femtosecond ( $10^{-15}$  seconds)

ps — picosecond ( $10^{-12}$  seconds)

ns — nanosecond ( $10^{-9}$  seconds) (default)

us — microsecond ( $10^{-6}$  seconds)

ms — millisecond ( $10^{-3}$  seconds)

sec — second

min — minute (60 seconds)

hr — hour (3600 seconds)

- **-valuecolwidth [<width>]**

(optional) Specifies the width of the value column, in pixels. Related Tcl variable is PrefWave(valueColWidth).

**<width>** — Any non-negative integer where the default is 100.

- **-vectorcolor [<color>]**

(optional) Specifies the vector waveform color. Default is #b3ffb3. Related Tcl variable is PrefWave(vectorColor).

**<color>** — Any color where the default is #b3ffb3.

- **-waveselectcolor [<color>]**

(optional) Specifies the background highlight color of a selected waveform. Related Tcl variable is PrefWave(waveSelectColor).

**<color>** — Any color where the default is grey30.

- **-waveselectenable [0 | 1]**

(optional) Specifies whether the waveform background highlights when an object is selected. Related Tcl variable is PrefWave(waveSelectEnabled).

0 — (default) Highlighting is disabled.

1 — Highlighting is enabled.

There are more options than are listed here. See the output of a **configure list** or **configure wave** command for all options.

## Examples

- Display the current value of the **strobeperiod** attribute.

```
config list -strobeperiod
```

- Set the period of the list strobe and turns it on.

```
config list -strobeperiod {50 ns} -strobestart 0 -usestrobe 1
```

- Set the wave vector color to blue.

```
config wave -vectorcolor blue
```

- Set the display in the current Wave window to show only the leaf path of each signal.

```
config wave -signalnamewidth 1
```

## Related Topics

[Setting GUI Preferences](#)

## dataset alias

This command maps an alternate name (alias) to an open dataset. A dataset can have any number of aliases, but all dataset names and aliases must be unique even when more than one dataset is open. Aliases are not saved to the *.wlf* file and must be remapped if the dataset is closed and then re-opened.

Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

### Syntax

**dataset alias <dataset\_name> [<alias\_name>]**

### Arguments

- **<dataset\_name>**  
(required) Specifies a dataset name or currently assigned dataset alias. Must be specified as the first argument to the dataset alias command. Returns a list of all aliases mapped to the specified dataset file when specified without **<alias\_name>**.
- **<alias\_name>**  
(optional) Specifies string to assign to the dataset as an alias. Wildcard characters are permitted.

### Examples

Assign the alias name “bar” to the dataset named “gold.”

**dataset alias gold bar**

### Related Topics

- [dataset clear](#)
- [dataset close](#)
- [dataset config](#)
- [dataset info](#)
- [dataset list](#)
- [dataset open](#)
- [dataset rename](#)
- [dataset restart](#)
- [dataset save](#)
- [dataset snapshot](#)

# dataset clear

All event data is removed from the current simulation WLF file, while retaining all currently logged signals. Subsequent run commands will continue to accumulate data in the WLF file.

---

## Note

---

 This command applies only to WLF-based simulation datasets.

---

## Syntax

dataset clear

## Description

If you run this command when no design is loaded, then the error: “Dataset not found:sim” is returned. If you run the command when a design is loaded, then the “sim:” dataset is cleared, regardless of which dataset is currently set. Clearing the dataset will clear any open Wave window based on the “sim:” dataset.

## Arguments

None

## Examples

Clear data in the WLF file from time 0ns to 100000ns, then log data into the WLF file from time 100000ns to 200000ns.

```
add wave *
run 100000ns
dataset clear
run 100000ns
```

## Related Topics

[dataset alias](#)  
[dataset close](#)  
[dataset config](#)  
[dataset info](#)  
[dataset list](#)  
[dataset open](#)  
[dataset rename](#)  
[dataset restart](#)  
[dataset save](#)  
[dataset snapshot](#)

`log`

[Recording Simulation Results With Datasets](#)

# dataset close

This command closes an active dataset. To open a dataset, use the dataset open command.

## Syntax

```
dataset close {<dataset_name> | -all}
```

## Arguments

- <dataset\_name> | -all

(required) Closes active dataset(s).

<dataset\_name> — Specifies the name of the dataset or alias you wish to close.

-all — Closes all open datasets and the simulation.

## Related Topics

[dataset alias](#)

[dataset clear](#)

[dataset config](#)

[dataset info](#)

[dataset list](#)

[dataset open](#)

[dataset rename](#)

[dataset restart](#)

[dataset save](#)

[dataset snapshot](#)

## dataset config

This command configures WLF parameters for an open dataset and all aliases mapped to that dataset. Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

### Syntax

```
dataset config <dataset_name> [-wlfcachesize [<n>]] [-wlfdelteonquit [0 | 1]] [-wlfopt [0 | 1]]
```

### Arguments

- <dataset\_name>  
(required) Specifies a open dataset or dataset alias you wish to configure. Must be specified as the first argument to the dataset config command.
- -wlfcachesize [<n>]  
(optional) Sets the size, in megabytes, of the WLF reader cache. Does not affect the WLF write cache.  
  
    <n> — Any non-negative integer, in MB where the default is 256.  
  
    If you do not specify a value for <n>, this switch returns the size, in megabytes, of the WLF reader cache.
- -wlfdelteonquit [0 | 1]  
(optional) Deletes the WLF file automatically when the simulation exits. Valid for the current simulation dataset only.  
  
    0 — Disabled (default)  
    1 — Enabled  
  
    If you do not specify an argument, this switch returns the current setting for the switch.
- -wlfopt [0 | 1]  
(optional) Optimizes the display of waveforms in the Wave window.  
  
    0 — Disabled  
    1 — Enabled (default)  
  
    If you do not specify an argument, this switch returns the current setting for the switch.

### Examples

Set the size of the WLF reader cache for the dataset “gold” to 512 MB.

```
dataset config gold -wlfcachesize 512
```

### Related Topics

[dataset alias](#)

[dataset clear](#)

[dataset close](#)

[dataset info](#)

[dataset list](#)

[dataset open](#)

[dataset rename](#)

[dataset restart](#)

[dataset save](#)

[dataset snapshot](#)

[WLF File Parameter Overview](#)

## dataset current

This command activates the specified dataset and sets the GUI context to the last selected context of the specified dataset. All context dependent GUI data is updated and all context dependent CLI commands start working with respect to the new context.

### Syntax

dataset current [<dataset\_name>]

### Arguments

- <dataset\_name>  
(optional) Specifies the dataset name or dataset alias you want to activate. If no dataset name or alias is specified, the command returns the name of the currently active dataset.

### Related Topics

[dataset alias](#)

[dataset clear](#)

[dataset close](#)

[dataset info](#)

[dataset list](#)

[dataset open](#)

[dataset rename](#)

[dataset restart](#)

[dataset save](#)

[dataset snapshot](#)

[WLF File Parameter Overview](#)

# dataset info

This command reports a variety of information about a dataset. Arguments to this command are order dependent. Please read through the argument descriptions for more information.

## Syntax

```
dataset info {name | file | exists} <dataset_name>
```

## Arguments

- {name | file | exists}

(required) Identifies what type of information you want reported.

Only one option per command is allowed. The current options include:

name — Returns the name of the dataset. Useful for identifying the real dataset name of an alias.

file — Returns the name of the file associated with the dataset.

exists — Returns "1" if the dataset is currently open, "0" if it does not.

Must be specified as the first argument to the dataset info command.

- <dataset\_name>

(optional) Specifies the name of the dataset or alias for which you want information. If you do not specify a dataset name, ModelSim uses the dataset of the current environment.

## Related Topics

[dataset alias](#)

[dataset clear](#)

[dataset close](#)

[dataset config](#)

[dataset list](#)

[dataset open](#)

[dataset rename](#)

[dataset restart](#)

[dataset save](#)

[dataset snapshot](#)

[environment](#)

## dataset list

This command lists all active datasets.

### Syntax

`dataset list [-long]`

### Arguments

- `-long`

(optional) Lists the dataset name followed by the `.wlf` file to which the dataset name is mapped.

### Related Topics

[dataset alias](#)

[dataset clear](#)

[dataset close](#)

[dataset config](#)

[dataset info](#)

[dataset open](#)

[dataset rename](#)

[dataset restart](#)

[dataset save](#)

[dataset snapshot](#)

# dataset open

This command opens a WLF file (either the currently running *vsim.wlf* or a saved WLF file) and assigns it the logical name that you specify.

## Syntax

```
dataset open <file_name> [<dataset_name>]
```

## Description

The file can be the existing WLF file for a currently running simulation. To close a dataset, use the [dataset close](#) command.

Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

## Arguments

- <file\_name>  
(required) Specifies the file to open as a view-mode dataset. Must be specified as the first argument to the dataset open command. Specify *vsim.wlf* to open the currently running WLF file.
- <dataset\_name>  
(optional) Specifies a name for the open dataset. This is a name that will identify the dataset in the current session. By default the dataset prefix will be the name of the specified file.

## Examples

Open the dataset file *last.wlf* and assign it the name *test*.

```
dataset open last.wlf test
```

## Related Topics

- [dataset alias](#)
- [dataset clear](#)
- [dataset close](#)
- [dataset config](#)
- [dataset info](#)
- [dataset list](#)
- [dataset rename](#)
- [dataset restart](#)
- [dataset save](#)

[dataset snapshot](#)

# dataset rename

This command changes the name of a dataset to the new name you specify. Arguments to this command are order dependent. Follow the order specified in the Syntax section.

## Syntax

```
dataset rename <dataset_name> <new_dataset_name>
```

## Arguments

- <dataset\_name>  
Specifies the existing name of the dataset.
- <new\_dataset\_name>  
Specifies the new name for the dataset.

## Examples

Rename the dataset file "test" to "test2".

```
dataset rename test test2
```

## Related Topics

[dataset alias](#)  
[dataset clear](#)  
[dataset close](#)  
[dataset config](#)  
[dataset info](#)  
[dataset list](#)  
[dataset open](#)  
[dataset restart](#)  
[dataset save](#)  
[dataset snapshot](#)

## dataset restart

This command unloads the specified dataset or currently active dataset and reloads the dataset using the same dataset name.

### Syntax

```
dataset restart [<file_name>]
```

### Arguments

- <file\_name>

(optional) Specifies the file to open as a dataset. If <filename> is not specified, the currently active dataset is restarted.

### Related Topics

[dataset alias](#)

[dataset clear](#)

[dataset close](#)

[dataset config](#)

[dataset info](#)

[dataset list](#)

[dataset open](#)

[dataset rename](#)

[dataset save](#)

[dataset snapshot](#)

# dataset save

This command writes data from the current simulation to the specified file. This lets you save simulation data while the simulation is still in progress.

## Syntax

```
dataset save <dataset_name> <file_name>
```

## Description

Arguments to this command are order dependent. Follow the order specified in the Syntax section.

## Arguments

- <dataset\_name>  
(required) Specifies the name of the dataset you want to save.
- <file\_name>  
(required) Specifies the name of the file to save.

## Examples

Save all current log data in the sim dataset to the file *gold.wlf*.

```
dataset save sim gold.wlf
```

## Related Topics

- [dataset alias](#)
- [dataset clear](#)
- [dataset close](#)
- [dataset config](#)
- [dataset info](#)
- [dataset list](#)
- [dataset open](#)
- [dataset rename](#)
- [dataset restart](#)
- [dataset snapshot](#)

## dataset snapshot

This command saves data from the current WLF file (*vsim.wlf* by default) at a specified interval. It provides you with sequential or cumulative "snapshots" of your simulation data.

Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

### Syntax

```
dataset snapshot [-dir <directory>] [-disable] [-enable] [-file <file_name>]
  [-filemode {overwrite | increment}] [-mode {cumulative | sequential}] [-report] [-reset]
  {-size <file_size> | -time <n> [<unit>]}
```

### Arguments

- **-dir <directory>**  
(optional) Specifies a directory into which the files should be saved. Either absolute or relative paths may be used. Default is to save to the current working directory.
- **-disable**  
(optional) Turns snapshotting off. All dataset snapshot settings from the current simulation are stored in memory. All other options are ignored after you specify -disable.
- **-enable**  
(optional) Turns snapshotting on. Restores dataset snapshot settings from memory or from a saved dataset. (default)
- **-file <file\_name>**  
(optional) Specifies the name of the file to save snapshot data.  
  
*<file\_name>* — A specified file name where the default is *vsim\_snapshot.wlf*. The suffix *.wlf* will be appended to specified filename and, possibly, an incrementing suffix.  
  
When the duration of the simulation run is not a multiple of the interval specified by -size or -time, the incomplete portion is saved in the file *vsim.wlf*.
- **-filemode {overwrite | increment}**  
(optional) Specifies whether to overwrite the snapshot file each time a snapshot occurs.  
  
overwrite — (default)  
increment — A new file is created for each snapshot. An incrementing suffix (1 to n) is added to each new file (for example, *vsim\_snapshot\_1.wlf*).
- **-mode {cumulative | sequential}**  
(optional) Specifies whether to keep all data from the time signals are first logged.  
  
cumulative — (default)  
sequential — The current WLF file is cleared every time a snapshot is taken.

- **-report**  
(optional) Lists current snapshot settings in the Transcript window. All other options are ignored if you specify **-report**.
- **-reset**  
(optional) Resets values back to defaults. The behavior is to reset to the default, then apply the remainder of the arguments on the command line. See examples below. If specified by itself without any other arguments, **-reset** disables dataset snapshot and resets the values.
- **-size <file\_size>**  
(Required if **-time** is not specified.) Specifies that a snapshot occurs based on WLF file size. Must be specified as the final argument to the dataset snapshot command.  
    <file\_size> — Size of WLF file in MB.
- **-time <n> [<unit>]**  
(Required if **-size** is not specified.) Specifies that a snapshot occurs based on simulation time. Must be specified as the final argument to the dataset snapshot command.  
    <n> — Any positive integer.  
    <unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <limit> and <unit> within braces ({}).

## Examples

- Create the file *vsim\_snapshot\_<n>.wlf* that is written to every time the current WLF file reaches a multiple of 10 MB (i.e., at 10 MB, 20 MB, 30 MB, etc.).

**dataset snapshot -size 10**

- Similar to the previous example, but in this case the current WLF file is cleared every time it reaches 10 MB.

**dataset snapshot -size 10 -mode sequential**

- Assuming simulator time units are ps, this command saves a file called *gold\_<n>.wlf* every 1000000 ps. If you run the simulation for 3000000 ps, three files are saved: *gold\_1.wlf* with data from 0 to 1000000 ps, *gold\_2.wlf* with data from 1000000 to 2000000, and *gold\_3.wlf* with data from 2000000 to 3000000.

**dataset snapshot -time 1000000 -file gold.wlf -mode sequential  
-filemode increment**

Because this example sets the time interval to 1000000 ps, if you run the simulation for 3500000 ps, a file containing the data from 3000000 to 3500000 ps is saved as *vsim.wlf* (default).

- Enable snapshotting with time=10000 and default mode (cumulative) and default filemode (overwrite).

**dataset snapshot -reset -time 10000**

## Related Topics

[dataset alias](#)  
[dataset clear](#)  
[dataset close](#)  
[dataset config](#)  
[dataset info](#)  
[dataset list](#)  
[dataset open](#)  
[dataset rename](#)  
[dataset restart](#)  
[dataset save](#)

# delete

This command removes objects from either the List or Wave window. Arguments to this command are order dependent.

## Syntax

```
delete list [-window <wname>] <object_name>...
delete wave [-window <wname>] <object_name>...
```

## Arguments

- **list**  
Specifies the target is a list window.
- **wave**  
Specifies the target is a wave window.
- **-window <wname>**  
(optional) Specifies the name of the List or Wave window to target for the delete command. (The [view](#) command allows you to create more than one List or Wave window.) If no window is specified, the default window is used; the default window is determined by the most recent invocation of the view command and has “ -Default” appended to the name.
- **<object\_name>...**  
(required) Specifies the name of an object. Must match an object name used in an [add list](#) or [add wave](#) command. Multiple object names are specified as a space separated list. Wildcard characters are allowed. Must be specified as the final argument to the delete list and delete wave commands.

## Examples

- Remove the object *vec2* from the list2 window.

```
delete list -window list2 vec2
```

- Remove all objects beginning with the string */test* from the Wave window.

```
delete wave /test*
```

## describe

This command displays information about simulation objects and design regions in the Transcript window.

### Syntax

**describe <name>...**

### Description

This command displays information about the following types of simulation objects and design regions in the Transcript window:

- **VHDL** — signals, variables, constants, and FILE objects.
- **Verilog** — nets and registers
- Design region

VHDL signals, Verilog nets and registers, can be specified as hierarchical names.

### Arguments

- <name>...

(required) The name of an HDL object for which you want a description.

Multiple object names are specified as a space separated list. Wildcard characters are allowed. HDL object names can be relative or full hierarchical names.

### Examples

- Print the types of the three specified signals.

**describe clk prw prdy**

- Return information about /textio/INPUT.

**describe /textio/INPUT**

produces:

```
# File of
#   Unconstrained Array of
#     VHDL standard type CHARACTER
```

# disablebp

This command turns off breakpoints and when commands. To turn on breakpoints or when commands again, use the [enablebp](#) command.

## Syntax

`disablebp [<id#> | <label>]`

## Arguments

- `<id#>`  
(optional) Specifies the ID number of a breakpoint or when statement to disable.
- `<label>`  
(optional) Specifies the label name of a breakpoint or when statement to disable.

If you do not specify either of these arguments, all breakpoints and when statements are disabled.

Use the `bp` command with no arguments to find labels and ID numbers for all breakpoints in the current simulation. Use the `when` command with no arguments to find labels and ID numbers of all when statements in the current simulation.

---

### **Note**



Id numbers for breakpoints and when statements are assigned from the same pool.  
Even if you have not specified a given id number for a breakpoint, that number may still be used for a when command.

---

## Related Topics

[enablebp](#)

[onbreak](#)

## do

This command executes the commands contained in a DO file.

### Syntax

do <filename> [<parameter\_value>...]

### Description

A DO file can have any name and extension. An error encountered during the execution of a DO file script causes its execution to be interrupted, unless an [onerror](#) command or the [OnErrorHandler](#) Tcl variable is specified with the [resume](#) command. The [onbreak](#) command is used to take action with source code breakpoint cases.

Arguments to this command are order dependent. Please read through the argument descriptions for more information.

### Arguments

- <filename>  
(required) Specifies the name of the DO file to be executed. The name can be a pathname or a relative file name. Pathnames are relative to the current working directory. Must be specified as the first argument to the do command.

If the do command is executed from another DO file, pathnames are relative to the directory of the calling DO file. This allows groups of DO files to be stored in a separate sub-directory.

- <parameter\_value>...  
(optional) Specifies values that are to be passed to the corresponding parameters \$1 through \$9 in the DO file. Multiple parameter values must be separated by spaces.

If you want to make the parameters optional (for example, specify fewer parameter values than the number of parameters actually used in the file), you must use the argc simulator state variable in the DO file script. Refer to “[Simulator State Variables](#)” and “[Making Script Parameters Optional](#)”.

---

#### **Note**

 While there is no limit on the number of parameters that can be passed to macros, only nine values are visible at one time. Use the [shift](#) command to see the other parameters.

---

### Examples

- Execute the file *macros/stimulus* and pass the parameter value 100 to \$1 in the DO file.

**do macros/stimulus 100**

Where the DO file *testfile* contains the line

```
bp $1 $2
```

place a breakpoint in the source file named *design.vhd* at line 127.

```
do testfile design.vhd 127
```

## Related Topics

[Tcl and DO Files](#)

[General Modes of Operation](#)

[Using a Startup File](#)

[DOPATH](#)

[Saving a Transcript File as a DO File](#)

## drivers

This command displays the names and strength of all drivers of the specified object.

### Syntax

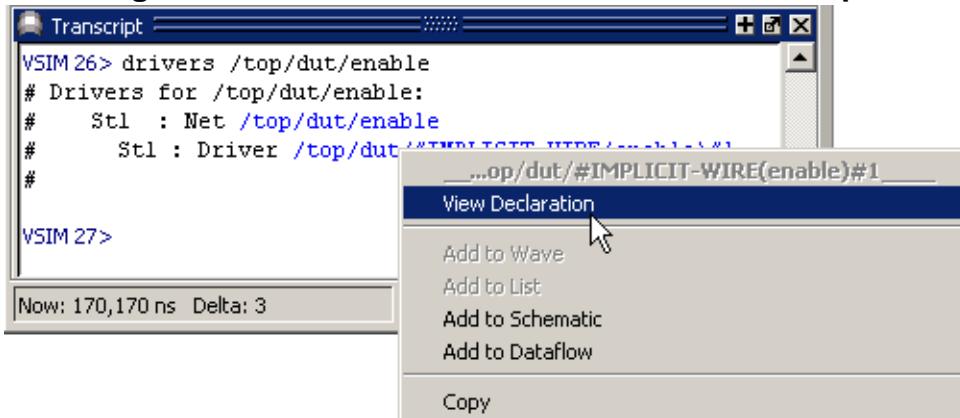
```
drivers <object_name> [-source]
```

### Description

The driver list is expressed relative to the top-most design signal/net connected to the specified object. If the object is a record or array, each sub-element is displayed individually.

The output from the drivers command, which is displayed in the Transcript window as a hypertext link, allowing you to right-click to open a drop-down menu and quickly add signals to various windows. It includes a "View Declaration" item to open the source definition of the signal.

**Figure 2-1. drivers Command Results in Transcript**



### Arguments

- <object\_name>  
(required) Specifies the name of the signal or net whose drivers are to be shown. All signal or net types are valid. Multiple names and wildcards are accepted.
- -source  
(optional) Returns the source file name and line number for each driver of the specified signal or net. If the source location cannot be determined, the value n/a is returned for that driver.

### Examples

```
drivers /top/dut/pkt_cnt(4)
```

```
# Drivers for /top/dut/pkt_cnt(4):
#   St0 : Net /top/dut/pkt_cnt[4]
#       St0 : Driver /top/dut/pkt_counter/#IMPLICIT-WIRE(cnt_out)#6
```

In some cases, the output may supply a strength value similar to 630 or 52x, which indicates an ambiguous Verilog strength.

## **dumplog64**

This command dumps the contents of the specified WLF file in a readable format to stdout. The WLF file cannot be opened for writing in a simulation when you use this command. This command cannot be used in a DO file.

### **Syntax**

`dumplog64 <filename>`

### **Arguments**

- `<filename>`  
(required) The name of the WLF file to be read.

# echo

This command displays a specified message in the Transcript window.

## Syntax

```
echo [<text_string>]
```

## Arguments

- <text\_string>

(required) Specifies the message text to be displayed. If the text string is surrounded by quotation marks, blank spaces are displayed as entered. If quotation marks are omitted, two or more adjacent blank spaces are compressed into one space.

## Examples

- If the current time is 1000 ns, this command:

```
echo "The time is $now ns."
```

returns the message:

```
The time is 1000 ns.
```

- If the quotes are omitted:

```
echo The time is $now ns.
```

all blank spaces of two or more are compressed into one space.

```
The time is $now ns."
```

- **echo** can also use command substitution, such as:

```
echo The hex value of counter is [examine -hex counter].
```

If the current value of counter is 21 (15 hex), this command returns:

```
The hex value of counter is 15.
```

## edit

This command invokes the editor specified by the EDITOR environment variable. By default, the specified filename will open in the Source window.

### Syntax

`edit [<filename>]`

### Arguments

- `<filename>`

(optional) Specifies the name of the file to edit. If the `<filename>` argument is omitted, the editor opens the current source file. If you specify a non-existent filename, it will open a new file. Either absolute or relative paths may be used.

### Related Topics

[notepad](#)

[EDITOR](#)

# enablebp

This command turns on breakpoints and when commands that were previously disabled.

## Syntax

`enablebp [<id#> | <label>]`

## Arguments

- `<id#>`  
(optional) Specifies a breakpoint ID number or when statement to enable.
- `<label>`  
(optional) Specifies the label name of a breakpoint or when statement to enable.

If you do not specify either of these arguments, all breakpoints are enabled.

Use the `bp` command with no arguments to find labels and ID numbers for all breakpoints in the current simulation. Use the `when` command with no arguments to find labels and ID numbers of all when statements in the current simulation.

## Related Topics

[disablebp](#)

[onbreak](#)

# encoding

These related commands translate between the 16-bit Unicode characters used in Tcl strings and a named encoding, such as Shift-JIS.

## Syntax

```
encoding convertfrom <encoding_name> <string>
encoding convertto <encoding_name> <string>
encoding names
encoding system <encoding_name>
```

## Description

The following encoding commands work with the encoding of your character representations in the GUI.

- `encoding convertfrom` — Converts a string from the named encoding to Unicode.
- `encoding convertto` — Converts a string to the named encoding from Unicode.
- `encoding names` — Returns a list of all valid encoding names (takes no arguments).
- `encoding system` — Changes the current system encoding to a named encoding. If a new encoding is omitted the command returns the current system encoding. The system encoding is used whenever Tcl passes strings to system calls.

## Arguments

- `string`  
Specifies a string to be converted.
- `encoding_name`  
The name of the encoding to use.

# environment

This command has two forms, environment and env. It allows you to display or change the current dataset and region/signal environment.

## Syntax

```
environment [-dataset | -nodataset] [<pathname> | -forward | -back]
```

## Arguments

- **-dataset**  
(optional) Displays the specified environment pathname *with* a dataset prefix. Dataset prefixes are displayed by default.
- **-nodataset**  
(optional) Displays the specified environment pathname *without* a dataset prefix.
- **<pathname>**  
(optional) Specifies a new pathname for the region/signal environment.  
If omitted the command causes the pathname of the current region/signal environment to be displayed.
- **-forward**  
(optional) Displays the next environment in your history of visited environments.
- **-back**  
(optional) Displays the previous environment in your history of visited environments.

## Examples

- Display the pathname of the current region/signal environment.

```
env
```

- Change to another dataset but retain the currently selected context.

```
env test:
```

- Change all unlocked windows to the context "test:/top/foo".

```
env test:/top/foo
```

- Move down two levels in the design hierarchy.

```
env blk1/u2
```

- Move to the top level of the design hierarchy.

```
env /
```

## Related Topics

[Setting your Context by Navigating Source Files](#)

## examine

This command has two forms, examine and exa. It examines one or more objects and displays current values (or the values at a specified previous time) in the Transcript window.

### Syntax

```
examine <name>... [-delta <delta>] [-env <path>] [-event <time>] [-handle] {[[-in] [-out]  
[-inout] | [-ports]} [-internal] [-maxlen <integer>] [-expr <expression>] [-name]  
[-<radix_type>] [-radix [<radix_type>][,<radix_flag>][,...]]  
[-radixenumnumeric | -radixenumsymbolic] [-showbase] [-time <time>] [-value]
```

### Description

It can also compute the value of an expression of one or more objects.

The following objects can be examined:

- **VHDL** — signals, shared variables, process variables, constants, generics, and FILE objects
- **Verilog** — nets, registers, parameters, and variables

To display a previous value, specify the desired time using the -time option.

To compute an expression, use the -expr option. The -expr and the -time options may be used together.

Virtual signals and functions may also be examined within the GUI (actual signals are examined in the kernel).

The following rules are used by the examine command to locate an HDL object:

- If the name does not include a dataset name, then the current dataset is used.
- If the name does not start with a path separator, then the current context is used.
- If the name is a path separator followed by a name that is not the name of a top-level design unit, then the first top-level design unit in the design is used.
- For a relative name containing a hierarchical path, if the first object name cannot be found in the current context, then an upward search is done up to the top of the design hierarchy to look for a matching object name.
- If no objects of the specified name can be found in the specified context, then an upward search is done to look for a matching object in any visible enclosing scope up to an instance boundary. If at least one match is found within a given context, no (more) upward searching is done; therefore, some objects that may be visible from a given context will not be found when wildcards are used if they are within a higher enclosing scope.

- The wildcards '\*' and '?' can be used at any level of a name except in the dataset name and inside of a slice specification.
- A wildcard character will never match a path separator. For example, `/dut/*` will match `/dut/siga` and `/dut/clk`. However, `/dut*` will not match either of those.

If no radix is specified with the examine command, the default radix and radix flags are used.

- Set the default radix with the `radix` command or by editing the `DefaultRadix` variable in the modelsim.ini file.
- Set the default radix flags value with the `radix` command or by editing the `DefaultRadixFlags` variable in the modelsim.ini file.
- Specifying `examine -<radix_type>` returns the value of the object in the specified radix and default radix flags value.
- Specifying `examine -radix <radix_type>` returns the value of the object in the specified radix.
- Specifying `examine -radix [<radix_type>][, [<radix_flag>]]` returns the value of the object in the specified radix and radix flags.
- Specifying `examine -radix <radix_flag>[,<radix_flag>]` returns the value of the object in the default radix and specified radix flags.

For example, assume a default of hexadecimal + showbase:

```
examine d
# 16'h0009

examine -binary d
# 16'b0000000000001001

examine -radix binary d
# 0000000000001001

examine -radix binary,showbase d
# 16'b0000000000001001

examine -radix hex,enumsymbolic nxt_state
# send5

examine -radix hex,enumnumeric nxt_state
# 000000d
```

Refer to [Design Object Names](#) for more information on specifying names.

## Arguments

- <name>...  
(required except when specifying -expr.) Specifies the name of any HDL object. All object types are allowed, except those of the type file. Multiple names and wildcards are accepted. Spaces, square brackets, and extended identifiers require braces; see examples below for more details. To examine a VHDL variable you can add a process label to the name. For example, (make certain to use two underscore characters):

```
exa line_36/i
```
- -delta <delta>  
(optional) Specifies a simulation cycle at the specified time step from which to fetch the value, where the default is to use the last delta of the time step. You must log the objects to be examined using the [add list](#), [add wave](#), or [log](#) command for the examine command to be able return a value for a requested delta.  
    <delta> — Any non-negative integer.
- -env <path>  
(optional) Specifies a path in which to look for an object name.  
    <path> — The specified path to a object.
- -event <time>  
(optional) Specifies a simulation cycle at the specified event time from which to fetch the value. The event <time> refers to the event time relative to events for all signals in the objects dataset at the specified time. You must log the objects to be examined using the [add list](#), [add wave](#), or [log](#) command for the examine command to be able return a value for a requested event.
- -expr <expression>  
(optional) Specifies an expression to be examined. You must log the expression using the [add list](#), [add wave](#), or [log](#) command for the examine command to return a value for a specified expression. The expression is evaluated at the current time simulation. If you also specify the **-time** argument, the expression will be evaluated at the specified time. It is not necessary to specify <name> when using this argument. See [GUI\\_expression\\_format](#) for the format of the expression.  
    <expression> — Specifies an expression enclosed in braces ({}).
- -handle  
(optional) Returns the memory address of the specified <name>. You can use this value as a tag when analyzing the simulation. This value also appears as the title of a box in the Watch window. This option will not return any value if you are in -view mode.
- -in  
(optional) Specifies that <name> include ports of mode IN.

- **-out**  
(optional) Specifies that <name> include ports of mode OUT.
- **-inout**  
(optional) Specifies that <name> include ports of mode INOUT.
- **-internal**  
(optional) Specifies that <name> include internal (non-port) signals.
- **-maxlen <integer>**  
(optional) Specifies the maximum number of characters in the output of the command.  
    <integer> — Any non-negative integer where 0 is unlimited.
- **-ports**  
(optional) Specifies that <name> include all ports. Has the same effect as specifying -in, -inout, and -out together.
- **-name**  
(optional) Displays object name(s) and value(s). Related switch is [-value](#).
- **-<radix\_type>**  
(optional) Specifies the radix type for the objects that follow in the command. Retains the current flag value for the objects that follow in the command. Valid entries (or any unique abbreviations) are: ascii, binary, decimal, fpoint, hexadecimal, octal, sfixed, symbolic, time, ufixed, unsigned, and default.  
This option overrides the global setting of the default radix (the [DefaultRadix](#) variable in the modelsim.ini file).
- **-radix [<radix\_type>][,<radix\_flag>][,...]**  
(optional) Specifies the radix and/or the radix flags to be used by the examine command.  
The -radix <radix\_type> switch can be used in place of examine -<radix\_type>.  
    <radix\_type> — (required unless specifying <radix\_flag>) Specifies a radix and clears the radix flags for the objects that follow in the command. Valid values are: ascii, binary, decimal, fpoint, hexadecimal, octal, sfixed, symbolic, time, ufixed, unsigned, default, and user-defined radix names (refer to the radix define command).  
    <radix\_flag> — (optional) Sets one or more radix flags on the objects that follow in the command. Multiple flags specified as a comma separated list. Must follow -<radix\_type> when the two are specified together.

Valid radix flags:

**Table 2-3. Radix flag Arguments to the Examine Command**

Argument	Description
enumeric	enumeric — Displays Verilog as numbers (formatted by the current radix). This overrides the default behavior of always showing enums symbolically.
enumsymbolic	enumsymbolic — Restores the default behavior of displaying Verilog enums as symbols by reversing the action of the -radixenumnumeric option.
showbase	showbase — Displays the number of bits of the vector and the radix used, where:
d	decimal
b	binary
h	hexadecimal
a	ASCII
t	time

For example, instead of simply displaying a vector value of “31”, a value of “16'h31” may be displayed to show that the vector is 16 bits wide, with a hexadecimal radix.

This option overrides the global default settings for the radix and the radix flag (the *DefaultRadix* and the *DefaultRadixFlags* in the modelsim.ini file).

- -radixenumnumeric
  - (optional) Causes Verilog enums to be displayed as numbers (formatted by the current radix). This overrides the default behavior of always showing enums symbolically.
- -radixenumsymbolic
  - (optional) Restores the default behavior of displaying Verilog and enums as symbols by reversing the action of the -radixenumnumeric option.
- -showbase
  - (optional) Display the number of bits of the vector and the radix used (binary = b, decimal = d, hexadecimal = h, ASCII = a, and time = t).

For example, instead of simply displaying a vector value of “31”, a value of “16'h31” may be displayed to show that the vector is 16 bits wide, with a hexadecimal radix.

- -time <time>
  - (optional) Specifies the time value between 0 and \$now for which to examine the objects.

**<time>** — A non negative integer where the default unit is the current time unit. If the **<time>** field uses a unit other than the current unit, the value and unit must be placed in braces. For example, the following are equivalent for ps resolution:

```
exa -time {3.6 ns} signal_a
exa -time 3600 signal_a
```

If an expression is specified it will be evaluated at that time. The objects to be examined must be logged via the add list, add wave, or log command in order for the examine command to be able to return a value for a requested time.

- **-value**

(default) Returns value(s) as a curly-braces separated Tcl list. Use to toggle off a previous use of **-name**.

## Examples

- Return the value of **/top/bus1**.

```
examine /top/bus1
```

- Return the value of the subelement of *reg1* that is specified by the index (16). Note that you must use braces when examining subelements.

```
{reg1[16]}
```

- Return information about **/textio/OUTPUT**

```
examine /textio/OUTPUT
```

returns the following:

```
# { STD_OUTPUT {stdout NOTPIPE} WRITE_MODE N/A }
```

The output is a Tcl list with up to four elements. There are three scenarios of results:

- If the file has not been elaborated, the result will be the following one-element list:

```
# { "NOT ELABORATED" }
```

- If the file is closed, the result will be the following one-element list.

```
# { "CLOSED" }
```

- In all other cases, the result will be a four-element list, following the format:

```
# {<file_path> {<descriptor> PIPE | NOTPIPE} <file_mode>
  <file_position>}
```

where,

- **<file\_path>** — references either the FILE declaration or corresponding **file\_open()** call

- <descriptor> — one of the following:
  - N — operating system file descriptor resource number.
  - stdin — identifying the file as stdin.
  - stdout — identifying the file as stdout.
  - <file\_mode> — identifying the file mode, of which the file was opened, as defined in std.standard package; either READ\_MODE, WRITE\_MODE, or APPEND\_MODE.
  - <file\_position> — value in bytes. For stdin or stdout files, the value will be “N/A”.
- Return the value of the contiguous subelements of *foo* specified by the slice (i.e., 20:22). Note the use of braces.

**examine {foo[20:22]}**

- Note that when specifying an object that contains an extended identifier as the last part of the name, there must be a space after the closing ‘\’ and before the closing ‘}’.

**examine {/top\My extended id\ }**

- In this example, the **-expr** option specifies a signal path and user-defined Tcl variable. The expression will be evaluated at 3450us.

**examine -time {3450 us} -expr {/top/bus and \$bit\_mask}**

- Using the \${fifo} syntax limits the variable to the simple name fifo, instead of interpreting the parenthesis as part of the variable. Quotation marks (“ “) are needed when spaces are involved; by using quotation marks instead of braces, the Tcl interpreter will expand variables before calling the command.

**examine -time \$t -name \$fifo "\${fifo}(1 to 3)" \${fifo}(1)**

- Because **-time** is not specified, this expression will be evaluated at the current simulation time. Note the signal attribute and array constant specified in the expression.

**examine -expr {clk'event && (/top/xyz == 16'hffae)}**

Commands like **find** and **examine** return their results as a Tcl list (just a blank-separated list of strings). You can do things like:

```
foreach sig [find sig ABC*] {echo "Signal $sig is [exa $sig]" ...}
```

```
if {[examine -bin signal_12] == "11101111XXXZ"} {...}
```

```
examine -hex [find *]
```

## Related Topics

[DefaultRadix](#)

# exit

This command exits the simulator and the ModelSim application.

## Syntax

```
exit [-force] [-code <integer>]
```

## Description

If you want to stop the simulation using a [when](#) command, use a [stop](#) command within your when statement, do not use an exit command or a [quit](#) command. The stop command acts like a breakpoint at the time it is evaluated.

## Arguments

- `-force`  
(optional) Quits without asking for confirmation. If this argument is omitted, ModelSim asks you for confirmation before exiting. You can also use `-f` as an alias for this switch.
- `-code <integer>`  
(optional) Quits the simulation and issues an exit code.  
`<integer>` — This is the value of the exit code. You should not specify an exit code that already exists in the tool. Refer to the section "[Exit Codes](#)" in the User's Manual for a list of existing exit codes. You can also specify a variable in place of `<integer>`.

You should always print a message before executing the exit -code command to explicitly state the reason for exiting.

## Examples

You can use exit -code to instruct a [vmake](#) command to exit when it encounters an assertion error. The [onbreak](#) command can specify commands to be executed upon an assert failure of sufficient severity, after which the simulator can be made to return an exit status. This is shown in the following example:

```
set broken 0
onbreak {
    set broken 88
    resume
}
run -all
if { $broken } {
    puts "failure -- exit status $broken"
    exit -code $broken} else {
    puts "success"
}
quit -f
```

The [resume](#) command gives control back to the commands following the run -all to handle the condition appropriately.

## find

This command locates objects by type and name. Arguments to the command are grouped by object type.

### Syntax

```
find nets | signals <object_name> ... [-internal] [-nofilter] {[[-in] [-inout] [-out] | [-ports]]} [-recursive]  
find instances | blocks {<object_name> ... | -bydu <design_unit> | -file <file_name>} [-arch] [-recursive] [-nodu]  
find virtuals <object_name> ... [-kind <kind>] [-unsaved] [-recursive]  
find classes [<class_name>]  
find objects [-class <class_name>] [-isa <class_name>] [<object_name>]
```

### Description

The following rules are used by the find command to locate an object:

- If the name does not include a dataset name, then the current dataset is used.
- If the name does not start with a path separator, then the current context is used.
- If the name is a path separator followed by a name that is not the name of a top-level design unit, then the first top-level design unit in the design is used.
- For a relative name containing a hierarchical path, if the first object name cannot be found in the current context, then an upward search is done up to the top of the design hierarchy to look for a matching object name.
- If no objects of the specified name can be found in the specified context, then an upward search is done to look for a matching object in any visible enclosing scope up to an instance boundary. If at least one match is found within a given context, no (more) upward searching is done; therefore, some objects that may be visible from a given context will not be found when wildcards are used if they are within a higher enclosing scope.
- The wildcards '\*' and '?' can be used at any level of a name except in the dataset name and inside of a slice specification. Square bracket ([]) wildcards can also be used.
- A wildcard character will never match a path separator. For example, `/dut/*` will match `/dut/siga` and `/dut/clk`. However, `/dut*` will not match either of those.
- Because square brackets are wildcards in the find command, only parentheses (()) can be used to index or slice arrays.
- The *WildcardFilter* Tcl preference variable is used by the find command to exclude the specified types of objects when performing the search.

See [Design Object Names](#) for more information on specifying names.

## Arguments

Arguments to the command are grouped by object type.

### Arguments for nets and signals

When searching for nets and signals, the find command returns the full pathname of all nets, signals, registers, variables, and named events that match the name specification.

- `<object_name> ...`  
(required) Specifies the net or signal for which you want to search. Multiple nets and signals and wildcard characters are allowed. Wildcards cannot be used inside of a slice specification. Spaces, square brackets, and extended identifiers require special syntax; see the examples below for more details.
- `-in`  
(optional) Specifies that the scope of the search is to include ports of mode IN.
- `-inout`  
(optional) Specifies that the scope of the search is to include ports of mode INOUT.
- `-internal`  
(optional) Specifies that the scope of the search is to include internal (non-port) objects.
- `-nofilter`  
(optional) Specifies that the *WildcardFilter* Tcl preference variable be ignored when finding signals or nets.
- `-out`  
(optional) Specifies that the scope of the search is to include ports of mode OUT.
- `-ports`  
(optional) Specifies that the scope of the search is to include all ports. Has the same effect as specifying `-in`, `-out`, and `-inout` together.
- `-recursive`  
(optional) Specifies that the scope of the search is to descend recursively into subregions. If omitted, the search is limited to the selected region.

### Arguments for instances and blocks

When searching for instances, the find command returns the primary design unit name.

- `-arch`  
Used with “instances” only: Lists the corresponding architecture name along with the entity name for any VHDL design unit names returned by the find command.
- `-bydu <design_unit>`  
Searches for a design unit. Mutually exclusive with `-file` and `<object_name>`.

<design\_unit> — Name of a single design unit to search for. This argument matches the pattern specified by primary <design\_unit> of the instance only. Library and Secondary names are not supported.

- **-file <file\_name>**  
Writes a complete list of the instances in a design to a file. Mutually exclusive with -bydu and <object\_name>.  
    <file\_name> — A string specifying the name for a file.
- **<object\_name> ...**  
Specifies the name of an instance or block for which you want to search. Multiple instances and wildcard characters are allowed. Mutually exclusive with -file and -bydu.
- **-recursive**  
(optional) Specifies that the scope of the search is to descend recursively into subregions. If omitted, the search is limited to the selected region.
- **-nodu**  
(optional) Removes the "du" string from the names of design units found with -bydu argument.

#### Arguments for **virtuals**

When searching for **virtuals**, all optional arguments must be specified before any object names.

- **<object\_name> ...**  
(required) Specifies the virtual object for which you want to search. Multiple **virtuals** and wildcard characters are allowed.
- **-kind <kind>**  
(optional) Specifies the kind of virtual object for which you want to search.  
    <kind> — A virtual object of one of the following kinds:
  - designs
  - explicits
  - functions
  - implicits
  - signals.
- **-unsaved**  
Specifies that ModelSim find only **virtuals** that have not been saved to a format file.

#### Arguments for **classes**

- **<class\_name>**  
(optional) Specifies the incrTcl class for which you want to search. Wildcard characters are allowed. The options for **class\_name** include nets, objects, signals, and **virtuals**. If you do

not specify a class name, the command returns all classes in the current namespace context. See incrTcl commands in the Tcl Man Pages (Help > Tcl Man Pages) for more information.

#### Arguments for objects

- **-class <class\_name>**  
(optional) Restricts the search to objects whose most-specific class is `class_name`.
- **-isa <class\_name>**  
(optional) Restricts the search to those objects that have `class_name` anywhere in their heritage.
- **<object\_name>**  
(optional) Specifies the incrTcl object for which you want to search. Wildcard characters are allowed. If you do not specify an object name, the command returns all objects in the current namespace context. See incrTcl commands in the Tcl Man Pages (Help > Tcl Man Pages) for more information.

#### Examples

- Find all signals in the entire design.  
**find signals -r /\***
- Find all instances in the entire design and save the list in the file `instancelist.txt`.  
**find instances -file instancelist.txt -r /\***
- Find all input signals in region `/top` that begin with the letters "xy".  
**find nets -in /top/xy\***
- Find all signals in the design hierarchy at or below the region `<current_context>/u1/u2` whose names begin with "cl".  
**find signals -r u1/u2/cl\***
- Find a signal named `s1`. Note that you must enclose the object in braces because of the square bracket wildcard characters.  
**find signals {s[1]}**
- Find signals `s1`, `s2`, or `s3`.  
**find signals {s[123]}**
- Find the element of signal `s` that is indexed by the value 1. Note that the find command uses parentheses (()), not square brackets ([]), to specify a subelement index.  
**find signals s(1)**
- Find a 4-bit array named `data`. Note that you must use braces ({} ) due to the spaces in the array slice specification.  
**find signals {/top/data(3 downto 0)}**

- Note that when specifying an object that contains an extended identifier as the last part of the name, there must be a space after the closing '\' and before the closing '}'.

```
find signals {/top\My extended id\}
```

- If */dut/core/pclk* exists, prints the message "pclk does exist" in the transcript. This would typically be run in a Tcl script.

```
if {[find signals /dut/core/pclk] != ""} {  
  echo "pclk does exist"
```

- Find instances based on their names using wildcards. Send search results to a text file that lists instance names, including the hierarchy path, on separate lines.

```
# Search for all instances with u1 in path  
set pattern_match "*u1*" ;  
# Get the list of instance paths  
set inst_list [find instances -r *] ;  
# Initialize an empty list to strip off the architecture names  
set ilist [list] ;  
foreach inst $inst_list {  
  set ipath [lindex $inst 0]  
  if {[string match $pattern_match $ipath]} {  
    lappend ilist $ipath  
  }  
}  
# At this point, ilist contains the list of instances only--  
# no architecture names  
#  
# Begin sorting list  
set ilist [lsort -dictionary $ilist]  
# Open a file to write out the list  
set fhandle [open "instancelist.txt" w]  
foreach inst $ilist {  
  # Print instance path, one per line  
  puts $fhandle $inst  
}  
  
# Close the file, done.  
close $fhandle ;
```

# find connections

This command returns the set of nets that are electrically equivalent to a specified net. It is only available during a live simulation.

## Syntax

**find connections <net>**

## Arguments

- <net>

(required) A net in the design. Returns a list of nets connected to the specified net.

## Examples

**find connections /top/p/strb**

returns:

```
# Connected nets for strb
#   output : /top/p/strb
#   internal : /top/pstrb
#   input : /top/c/pstrb
```

## find infiles

This command searches for a string in the specified file(s) and prints the results to the Transcript window. The results are individually hotlinked and will open the file and display the location of the string.

### Syntax

`find infiles <string_pattern> <file>...`

### Description

When you run this command in command-line mode from outside of the GUI, the results are sent to stdout with no hotlinks.

Arguments to this command are order-dependent. Follow the order specified in the Syntax section.

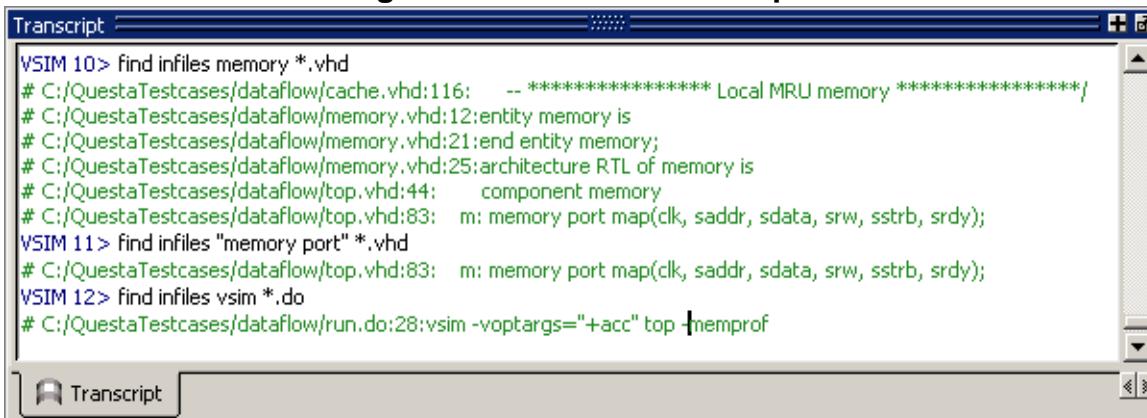
### Arguments

- `<string_pattern>`  
(required) The string you are searching for. You can use Tcl regular expression wildcards to further restrict the search capability.
- `<file>...`  
(required) The file(s) to search. You can use Tcl regular expression wildcards to further restrict the search capability.

### Examples

Figure 2-2 shows a screen capture containing a few examples of the `find infiles` command and the results.

**Figure 2-2. `find infiles` Example**



The screenshot shows the ModelSim Transcript window with the following content:

```
VSIM 10> find infiles memory *.vhd
# C:/QuestaTestcases/dataflow/cache.vhd:116:  -- ***** Local MRU memory *****/
# C:/QuestaTestcases/dataflow/memory.vhd:12:entity memory is
# C:/QuestaTestcases/dataflow/memory.vhd:21:end entity memory;
# C:/QuestaTestcases/dataflow/memory.vhd:25:architecture RTL of memory is
# C:/QuestaTestcases/dataflow/top.vhd:44:  component memory
# C:/QuestaTestcases/dataflow/top.vhd:83:  m: memory port map(clk, saddr, sdata, srw, sstrb, srdy);
VSIM 11> find infiles "memory port" *.vhd
# C:/QuestaTestcases/dataflow/top.vhd:83:  m: memory port map(clk, saddr, sdata, srw, sstrb, srdy);
VSIM 12> find infiles vsim *.do
# C:/QuestaTestcases/dataflow/run.do:28:vsim -voptargs="+acc" top |memprof
```

# find insource

This command searches for a string in the source files for the current design and prints the results to the Transcript window. The results are hotlinked individually and will open the file and display the location of the string. When you execute this command in command-line mode from outside of the GUI, the results are sent to stdout with no hotlinks.

## Syntax

```
find insource <pattern> [-exact | -glob | -regexp] [-inline] [-nocase]
```

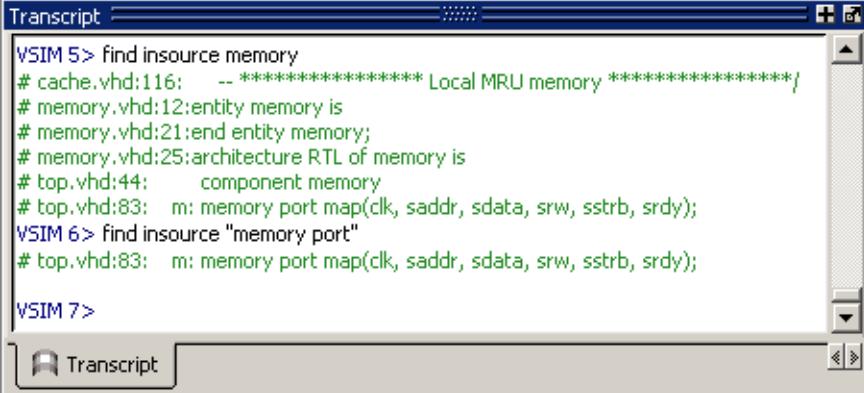
## Arguments

- <pattern>  
(required) The string you are searching for. You can use regular expression wildcards to further restrict the search. You must enclose <pattern> in quotation marks (" ") if it includes spaces. You must specify the <pattern> at the end of the command line; any switches specified after <pattern> will not be registered.
- -exact | -glob | -regexp  
(optional) Defines the style of regular expression used in the <pattern>
  - exact — Indicates that no characters have special meaning, thus disabling wildcard features.
  - glob — (default) Allows glob-style wildcard characters. For more information refer to the Tcl documentation:
    - Help > Tcl Man Pages
    - Select “Tcl Commands”, then “string”, then “string match”
  - regexp — Allows Tcl regular expressions. For more information refer to the Tcl documentation:
    - Help > Tcl Man Pages
    - Select “Tcl Commands”, then “re\_syntax”.
- -inline  
(optional) Returns the matches in the form of a Tcl list, which disables the hotlink feature but allows for easier post-processing.
- -nocase  
(optional) Treats <pattern> as case-insensitive.

## Examples

- [Figure 2-3](#) shows a couple of examples of the find insource command and the results.

**Figure 2-3. find insource Example**



The screenshot shows the ModelSim Transcript window with the following content:

```
VSIM 5> find insource memory
# cache.vhd:116:  .. **** Local MRU memory ****
# memory.vhd:12:entity memory is
# memory.vhd:21:end entity memory;
# memory.vhd:25:architecture RTL of memory is
# top.vhd:44:  component memory
# top.vhd:83:  m: memory port map(clk, saddr, sdata, srw, sstrb, srdy);
VSIM 6> find insource "memory port"
# top.vhd:83:  m: memory port map(clk, saddr, sdata, srw, sstrb, srdy);

VSIM 7>
```

- Searching for two keywords with whitespace between them:

**find insource -regexp {top\_dut\s+dut}**

returns:

```
# top.sv:20:          top_dut          dut (
```

- Searching for string starting with 'dut' and ending with 'o':

**find insource -regexp {dut.\*o}**

returns:

```
# top.sv:17:          DUT_io dut_io(.clock(tb_clk), .reset(tb_reset));
```

- Searching for string irrespective of case:

**find insource -regexp -nocase {DUT}**

returns:

```
# test.sv:10:          virtual DUT_io dut_io;
# test.sv:27:          this.dut_io = dut_io;
```

## Related Topics

[DISABLE\\_ELAB\\_DEBUG](#)

# force

This command allows you to apply stimulus interactively to VHDL signals, Verilog nets and registers.

## Syntax

**Forcing values, driver type, repetition time or stop time on an object**

```
force {<object_name> <value> [[@]<time_info>][, <value> [@]<time_info>]...
      [-deposit | -drive | -freeze] [-cancel [@]<time_info>] [-repeat [@]<time_info>]
```

### Reporting all force commands

If you specify this command without arguments, it returns a list of the most recently applied force commands and a list of forces coming from the Signal Spy signal\_force() and \$signal\_force() calls from within VHDL, Verilog.

For example, after entering:

```
force -freeze /top/p/addr 0 100, 1 150 -r 200 -cancel 2000
```

the times specified are relative to the current simulation time, in this case 2820 ns

Entering:

```
force
```

Returns:

```
# force -freeze /top/p/addr 0 {@2920 ns} , 1 {@2970 ns}
  -repeat {@3020 ns} -cancel {@4820 ns}
```

---

### Note

 When you run the force command, the simulator translates the relative time you specify into absolute time.

---

## Description

It is possible to create a complex sequence of stimuli when the force command is included in a DO file.

There are a number of constraints on what you can and cannot force.

You can force:

- VHDL signals or parts of signals.
- Verilog nets and registers, bit-selects, part-selects, and field-selects. Refer to “[Force and Release Statements in Verilog](#)” for more information.
- “[Virtual Signals](#)” if the number of bits corresponds to the signal value.
- An alias of a VHDL signal.

- An input port that is mapped at a higher level in VHDL and mixed models.

You cannot force:

- Virtual functions.
- VHDL variables. Refer to the [change](#) command for information on working with VHDL variables.
- An input port that has a conversion function on the input or on the path up the network mapped from the input.

This command provides additional information with the `-help` switch.

Arguments to this command are order dependent. Please read through the argument descriptions for more information.

## Arguments

- `<object_name>`

(required when forcing a value change) Specifies the name of the HDL object to be forced. A wildcard is permitted only if it matches one object. Refer to [Design Object Names](#) and [Tcl Syntax and Specification of Array Bits and Slices](#) for the full syntax of an object name. The object name must specify a scalar type or a one-dimensional array of character enumeration. You may also specify a record sub-element, an indexed array, or a sliced array, as long as the type is one of the above. Must be specified as the first argument to the **force** command.

- `<value>`

(required when forcing a value change) Specifies the value to which the object is to be forced. The specified value must be appropriate for the type. Must be specified as the second argument to the force command.

A one-dimensional array of character enumeration can be forced as a sequence of character literals or as a based number with a radix of 2, 8, 10 or 16. For example, the following values are equivalent for a signal of type `bit_vector(0 to 3)`:

Description	VHDL Value	Verilog Value
character literal sequence	F	F
binary radix	2#1111	'b1111
octal radix	8#17	'o17
decimal radix	10#15	'd15
hexadecimal radix	16#F	'hF

---

**Note**

 For based numbers in VHDL, ModelSim translates each 1 or 0 to the appropriate value for the number's enumerated type. The translation is controlled by the translation table in the *pref.tcl* file. If ModelSim cannot find a translation for 0 or 1, it uses the left bound of the signal type (type'left) for that value.

---

You can create a sequence of forced values on an object by specifying <value>[@]<time\_info> in a comma/space separated list.

For example:

```
force /top/p/addr 1 100ns, 0 200ns, 1 250ns
```

- -cancel [@]<time\_info>

(optional) Cancels the force command at the time specified by <time\_info>.

where:

<time\_info> is [@]<time\_value>[<time\_unit>]

Refer to [\[@\]<time\\_info>](#) for more information about specifying time values.

- -drive

(optional) Attaches a driver to the object and drives the specified <value> until the object is forced again or until it is unforced with the *noforce* command.

This option is illegal for unresolved signals.

- -deposit

(optional) Sets the object to the specified <value>. The <value> remains until the object is forced again, there is a subsequent driver transaction, or it is unforced with a *noforce* command. When used for registers, it behaves like the [change](#) command.

---

**Note**

 If the -freeze, -drive, or -deposit options are not used, then -freeze is the default for unresolved objects, and -drive is the default for resolved objects. If you prefer -freeze as the default for resolved and unresolved VHDL signals, change the [DefaultForceKind](#) variable in the *modelsim.ini* file.

---

- -freeze

(optional) Freezes the object at the specified <value> until it is forced again or until it is unforced with the *noforce* command.

---

**Note**

 If you prefer -freeze as the default for resolved and unresolved VHDL signals, change the [DefaultForceKind](#) variable in the *modelsim.ini* file.

---

- -repeat [@]<time\_info>

(optional) Repeats a series of forced values and times at the time specified.

where:

<time\_info> is [@]<time\_value>[<time\_unit>]

Refer to [\[@\]<time\\_info>](#) for more information about specifying time values.

You must specify at least two <value> <time\_info> pairs on the forced object before specifying -repeat, for example:

```
force top/dut/p 1 0, 0 100 -repeat 200 -cancel 1000
```

A repeating force command will force a value before other non-repeating force commands that occur in the same time step.

- [@]<time\_info>

(optional) Specifies the relative or absolute simulation time at which the <value> is to be applied.

where:

<time\_info> is [@]<time\_value>[<time\_unit>]

@ — A prefix applied to <time\_value> to specify an absolute time. By default, the specified time units are assumed to be relative to the current time unless the value is preceded by the character "at" ( @ ). Omit the "at" (@) character to specify relative time. For example:

```
-cancel {520 ns}      \\ Relative Time
```

```
-cancel {@ 520 ns}   \\ Absolute Time
```

<time\_value> — The time (either relative or absolute) to apply to <value>. Any non-negative integer. A value of zero cancels the force at the end of the current time period.

<time\_unit> — An optional suffix specifying a time unit where the default is to use the current simulator time by omitting <time\_unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr.

<time\_value> and <time\_unit> can be formatted in any of the following ways:

```
10ns
```

```
10 ns
```

```
{10 ns}
```

```
"10 ns"
```

---

**Note**

---

 If you specify a sequence of forces and use braces ({} ) surrounding a <time\_value> and <time\_unit> pair, you must place a space in front of the comma (,) separating the two value/time pairs. For example:

```
force foo 1 {10 ns} , 0 {20 ns}
```

---

**Examples**

- Reporting all recently applied force commands

If you specify this command with no arguments, it returns a list of all forced objects and the changes applied. For example, after executing:

```
force -freeze /top/p/addr 0 100, 1 150 -r 200 -cancel 2000
```

where the times specified are relative to the current simulation time, in this case 2820 ns.

Entering:

```
force
```

returns:

```
# force -freeze /top/p/addr 0 {@2920 ns} , 1 {@2970 ns}
  -repeat {@3020 ns} -cancel {@4820 ns}
```

---

**Note**

---

 Executing the force command translates the relative time you specified into absolute time.

---

- Force *input1* to 0 at the current simulator time.

```
force input1 0
```

- Force the fourth element of the array *bus1* to 1 at the current simulator time.

```
force bus1(4) 1
```

- Force *bus1* to 01XZ at 100 nanoseconds after the current simulator time.

```
force bus1 2#01XZ 100 ns
```

- Force *bus1* to 16#F at the absolute time 200 measured in the resolution units selected at simulation start-up.

```
force bus1 16#f @200
```

- Force *input1* to 1 at 10 time units after the current simulation time and to 0 at 20 time units after the current simulation time. Repeat this cycle every 100 time units after the current simulation time. If the current simulation time is 100 ns, the next transition is to 1 at 110 ns and 0 at 120 ns, this pattern to start repeating at 200 ns.

**force input1 1 10, 0 20 -r 100**

- Similar to the previous example, but also specifies the time units.

**force input1 1 10 ns, 0 20 ns -r 100 ns**

- Force signal *s* to alternate between values 1 and 0 every 100 time units until 1000 time units have occurred, starting from time Now. Cancellation occurs at the last simulation delta cycle of a time unit.

**force s 1 0, 0 100 -repeat 200 -cancel 1000**

So,

**force s 1 0 -cancel 0**

will force signal *s* to 1 for the duration of the current time period.

- Force *siga* to decimal value 85 whenever the value on the signal is 1.

```
when {/mydut/siga = 10#1} {
    force -deposit /mydut/siga 10#85
}
```

- Force one bit of a record containing an array.

**force struct1.bus1(4) 1**

- Force a slice of an array.

**force {bus1[2:5]} 'hF**

## Related Topics

[DefaultForceKind](#)

[Force and Release Statements in Verilog](#)

[Force Command Defaults](#)

[noforce](#)

[Virtual Signals](#)

# formatTime

This command provides global format control for all time values displayed in the GUI. When specified without arguments, this command returns the current state of the three arguments.

## Syntax

```
formatTime [[+|-]commas] [[+|-]nodefunits] [[+|-]bestunits]
```

## Arguments

- **[+|-]commas**  
(optional) Insert commas into the time value.
  - + prefix — On
  - prefix — Off. (default)
- **[+|-]nodefunits**  
(optional) Do not include default unit in the time.
  - + prefix — On
  - prefix — Off. (default)
- **[+|-]bestunits**  
(optional) Use the largest unit value possible.
  - + prefix — On
  - prefix — Off. (default)

## Examples

- Display commas in time values.

```
formatTime +commas
```

Instead of displaying 6458131 ps, the GUI will display 6,458,131 ps.

- Use largest unit value possible.

```
formatTime +bestunits
```

Displays 8 us instead of 8,000 ns.

## gc configure

Prerequisite:

Before using this command, do one of the following:

- Regular simulation — Simulate with the vsim command, but omit the -classdebug argument.
- Interactive class debugging — Simulate with the vsim -classdebug command.

This command specifies when the System Verilog garbage collector will run. The garbage collector may be configured to run after a memory threshold has been reached, after each simulation run command completes, and/or after each simulation step command. The default settings are optimized to balance performance and memory usage for either regular simulation or class debugging (vsim -classdebug). Returns the current settings when specified without arguments.

### Syntax

```
gc config [-onrun 0 | 1] [-onstep 0 | 1] [-threshold <n>]
```

### Arguments

- **-onrun 0 | 1**  
(optional) Enables/disables garbage collector execution after each simulation run command completes.
  - 0 — Off, default for regular simulation
  - 1 — On, default for interactive class debugging
- **-onstep 0 | 1**  
(optional) Enables or disables garbage collector execution after each step when stepping through your simulation.
  - 0 — Off, default for both regular simulation and interactive class debugging.
  - 1 — On
- **-threshold <n>**  
(optional) Sets the maximum amount of memory in megabytes allocated for storage of class objects before the garbage collector runs.
  - <n> — Any positive integer where <n> is the number of megabytes.
    - Regular simulation default =100 megabytes
    - Interactive class debugging default = 5 megabytes

### Related Topics

[SystemVerilog Class Debugging](#)

[Class Instance Garbage Collection](#)

[GCThreshold](#)

[GCThresholdClassDebug](#)

## gc run

This command runs the SystemVerilog garbage collector.

### Syntax

gc run

### Arguments

None

### Related Topics

[gc configure](#)

[SystemVerilog Class Debugging](#)

[Class Instance Garbage Collection](#)

[GCThreshold](#)

[GCThresholdClassDebug](#)

# help

This command displays in the Transcript window a brief description and syntax for the specified command.

## Syntax

```
help [<command> | <topic>]
```

## Arguments

- <command>

(optional) Specifies the command for which you want help. The entry is case and space sensitive.

- <topic>

(optional) Specifies a topic for which you want help. The entry is case and space sensitive. Specify one of the following six topics:

Topic	Description
commands	Lists all available commands and topics
debugging	Lists debugging commands
execution	Lists commands that control execution of your simulation.
Tcl	Lists all available Tcl commands.
Tk	Lists all available Tk commands
incrTCL	Lists all available incrTCL commands

# history

This command lists the commands you have executed during the current session. History is a Tcl command. For more information, consult the Tcl Man Pages (Help > Tcl Man Pages).

## Syntax

history [[clear](#)] [[keep <value>](#)]

## Arguments

- **clear**  
(optional) Clears the history buffer.
- **keep <value>**  
(optional) Specifies the number of executed commands to keep in the history buffer.  
*<value>* — Any positive integer where the default is 50.

# layout

This command allows you to perform a number of editing operations on custom GUI layouts, such as loading, saving, maximizing, and deleting.

## Syntax

```
layout active
layout current
layout delete <name>
layout load <name>
layout names
layout normal
layout maximized
layout restoretpe <window>
layout save <name>
layout showsuppresstypes
layout suppresstype <window>
layout togglezoom
layout zoomactive
layout zoomwindow <window>
```

## Description

The command options include:

- layout active – returns the current active window
- layout current – lists the current layout
- layout delete – removes the current layout from the Registry (Windows)
- layout load – opens the specified layout
- layout names – lists all known layouts
- layout normal – minimizes the current maximized window
- layout maximized – return a 1 if the current layout is maximized, or a 0 if minimized
- layout restoretpe — removes the list of window type(s) that will not be restored when a new layout is loaded.
- layout save – saves the current layout to the specified name

- layout showsuppresstypes — lists the window types that will not be restored when a new layout is loaded.
- layout suppresstype — adds the specified window type(s) to the list of types that will not be restored when a layout is reloaded.
- layout togglezoom – toggles the current zoom state of the active window (from minimized to maximized or maximized to minimized)
- layout zoomactive – maximizes the current active window
- layout zoomwindow – maximizes the specified window

## Arguments

- <name>  
(required) Specifies the name of the layout.
- <window>  
(required) The window specification can be any format accepted by the [view](#) command. The window can be specified by its type (such as wave, list, objects), by the windowobj name (such as main\_pane.wave, .main\_pain.library), or by the tab name (such as wave1, list3)

## Related Topics

[Simulator GUI Layout Customization](#)

[Configuring Default Windows for Restored Layouts](#)

# log

This command creates a wave log format (WLF) file containing simulation data for all HDL objects whose names match the provided specifications. Objects that are displayed using the add list and add wave commands are automatically recorded in the WLF file. By default the file is named vsim.wlf and stored in the current working directory. You can change the default name using the vsim -wlf option of the vsim command or by setting the WLFFilename variable in the *modelsim.ini* file.

## Syntax

```
log [-howmany] [-filter <f> | -nofilter <f>]
  {[-in] [-inout] [-out] | [-ports]} [-internal]
  [-recursive [-depth <level>]] <object_name> ...
log -flush [<object>]
```

## Description

If no port mode is specified, the WLF file contains data for all objects in the selected region whose names match the object name specification.

The WLF file contains a record of all data generated for the list and wave windows during a simulation run. Reloading the WLF file restores all objects and waveforms and their complete history from the start of the logged simulation run. See [dataset open](#) for more information.

For all transaction streams created through the SCV or Verilog APIs, logging is enabled by default. A transaction is logged to the WLF file if logging is enabled at the beginning of a simulation run when the design calls `::begin_transaction()` or `$begin_transaction`. The effective start time of the transaction (the time passed by the design as a parameter to `::begin_transaction`) is irrelevant. For example, a stream could have logging disabled between T1 and T2 and still record a transaction in that period, through retroactive logging after time T2. A transaction is always either entirely logged or entirely ignored.

Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

---

### Note

 The log command is also known as the "add log" command.

---

## Arguments

- `-depth <level>`  
(optional) Restricts a recursive search (specified with the `-recursive` argument) to a certain level of hierarchy.  
`<level>` — Any non-negative integer. For example, if you specify `-depth 1`, the command descends only one level in the hierarchy.

- **-filter <f> | -nofilter <f>**  
(optional) Allows a one-time modification of the WildcardFilter in the command invocation. The add list command can take as many [-filter <f>] and [-nofilter <f>] arguments as you would like to specify. Valid filters, <f>, are exactly the same set of words that can be applied to the WildcardFilter. The filter(s) used during a command starts with the WildcardFilter and then applies the user specified filters, if any. The -filter values are added to the filter, the -nofilter values are removed from the filter. They are applied in the order specified so conflicts are resolved with the last specified wins.
- **-flush [<object>]**  
(optional) Forces the WLF file to write all buffered region and event data to the WLF file. By default, the region and event data is buffered and periodically written to the file, as appropriate. If <object> is specified, that object is first logged and then the file is flushed.
- **-howmany**  
(optional) Returns an integer indicating the number of signals found.
- **-in**  
(optional) Specifies that the WLF file is to include data for ports of mode IN whose names match the specification.
- **-inout**  
(optional) Specifies that the WLF file is to include data for ports of mode INOUT whose names match the specification.
- **-internal**  
(optional) Specifies that the WLF file is to include data for internal (non-port) objects whose names match the specification.
- **-out**  
(optional) Specifies that the WLF file is to include data for ports of mode OUT whose names match the specification.
- **-ports**  
(optional) Specifies that the scope of the search is to include all ports, IN, INOUT, and OUT.
- **-recursive**  
(optional) Specifies that the scope of the search is to descend recursively into subregions. If omitted, the search is limited to the selected region. You can use the -depth argument to specify how far down the hierarchy to descend.
- **<object\_name>**  
(required) Specifies the object name that you want to log. Must be specified as the final argument to the **log** command. Multiple object names are specified as a space separated list. Wildcard characters are allowed. Note that the *WildcardFilter* Tcl preference variable identifies types to ignore when matching objects with wildcard patterns.

By default, wildcard card logging does not log the internals of cells. Refer to the `+libcell` | `+nolibcell` argument of the [vlog](#) command for more information.

## Examples

- Log all objects in the design.

```
log -r /*
```

- Log all output ports in the current design unit.

```
log -out *
```

## Related Topics

[dataset alias](#)

[dataset clear](#)

[dataset close](#)

[dataset config](#)

[dataset info](#)

[dataset list](#)

[dataset open](#)

[dataset restart](#)

[dataset rename](#)

[dataset save](#)

[dataset snapshot](#)

[nolog](#)

[Recording Simulation Results With Datasets](#)

[WLFFilename](#)

## **lshift**

This command takes a Tcl list as an argument and shifts it in-place, one place to the left, eliminating the left-most element.

### **Syntax**

`lshift <list> [<amount>]`

### **Description**

The number of shift places may also be specified. Returns nothing.

Arguments to this command are order dependent. Please read through the argument descriptions for more information.

### **Arguments**

- `<list>`  
(required) Specifies the Tcl list to target with lshift. Must be specified as the first argument to the lshift command.
- `<amount>`  
(optional) Specifies the number of places to shift where the default is 1.

### **Examples**

```
proc myfunc args {  
    # throws away the first two arguments  
    lshift args 2  
    ...  
}
```

# **lsublist**

This command returns a sublist of the specified Tcl list that matches the specified Tcl glob pattern. Arguments to this command are order dependent. Follow the order specified in the Syntax section.

## Syntax

```
lsublist <list> <pattern>
```

## Arguments

- <list>  
(required) Specifies the Tcl list to target with lsublist.
- <pattern>  
(required) Specifies the pattern to match within the <list> using Tcl glob-style matching.

## Examples

- In the example below, variable 't' returns "structure signals source".

```
set window_names "structure signals variables process source wave list"  
set t [lsublist $window_names s*]
```

## mem compare

This command compares a selected memory to a reference memory or file. Must have the "diff" utility installed and visible in your search path in order to run this command. Arguments to this command are order dependent. Please read through the argument descriptions for more information.

### Syntax

```
mem compare {[ -mem <ref_mem> ] | [ -file <ref_file> ]} [actual_mem]
```

### Arguments

- **-mem <ref\_mem>**  
(optional) Specifies a reference memory to be compared with actual\_mem.  
    <ref\_mem> — A memory record.
- **-file <ref\_file>**  
(optional) Specifies a reference file to be compared with actual\_mem.  
    <ref\_file> — A saved memory file.
- **actual\_mem**  
(required) Specifies the name of the memory to be compared against the reference data.  
Must be specified as the final argument to the mem compare command.

# mem display

This command prints to the Transcript window the memory contents of the specified instance. If the given instance path contains only a single array signal or variable, the signal or variable name need not be specified.

## Syntax

```
mem display [-addressradix [d | h]] [-compress] [-dataradix <radix_type>]
[-endaddress <end>][-format [bin | hex | mti]] [-noaddress] [-startaddress <st>]
[-wordsperline <n>] [<path>]
```

## Description

You can redirect the output of the mem display command into a file for later use with the mem load command. The output file can also be read by the Verilog \$readmem system tasks if the memory module is a Verilog module and Verilog memory format (hex or binary) is specified.

Address radix, data radix, and address range for the output can also be specified, as well as special output formats.

By default, identical data lines are printed. To replace identical lines with a single line containing the asterisk character, you can enable compression with the -compress argument.

### Note

 The format settings are stored at the top of this file as a pseudo comment so that subsequent mem load commands can correctly interpret the data. Do not edit this data when manipulating a saved file.

Arguments to this command are order dependent. Please read through the argument descriptions for more information.

## Arguments

- **-addressradix [d | h]**  
(optional) Specifies the address radix for the default (MTI) formatted files.
  - d — Decimal radix. (default if -format is specified as mti.)
  - h — Hex radix.
- **-compress**  
(optional) Specifies that identical lines not be printed. Reduces the file size by replacing exact matches with a single line containing an asterisk. These compressed files are automatically expanded during a mem load operation.
- **-dataradix <radix\_type>**  
(optional) Specifies the data radix for the default (MTI) formatted files. If unspecified, the global default radix is used.

**<radix\_type>** A specified radix type. Valid entries (or any unique abbreviations) are: binary, decimal, unsigned, octal, hex, symbolic, and default. If no radix is specified for an enumerated type, the symbolic representation is used.

You can change the default radix type for the current simulation using the [radix](#) command or make the default radix permanent by editing the [DefaultRadix](#) variable in the modelsim.ini file.

- **-endaddress <end>**

(optional) Specifies the end address for a range of addresses to be displayed.

**<end>** — Any valid address in the memory. If unspecified, the default is the end of the memory.

- **-format [bin | hex | mti]**

(optional) Specifies the output format of the contents.

**bin** — Specifies a binary output.

**hex** — Specifies a hex output.

**mti** — MTI format. (default).

- **-noaddress**

(optional) Specifies that addresses not be printed.

- **-startaddress <st>**

(optional) Specifies the start address for a range of addresses to be displayed.

**<st>** — Any valid address in the memory. If unspecified, the default is the start of the memory.

- **-wordsperline <n>**

(optional) Specifies how many words are to be printed on each line.

**<n>** — Any positive integer where the default is an 80 column display width.

- **<path>**

(required) Specifies the full path to the memory instance. The default is the current context, as shown in the Structure window. Indexes can be specified. Must be specified as the final argument to the mem display command.

## Examples

- This command displays the memory contents of instance */top/c/mru\_mem*, addresses 5 to 10:

```
mem display -startaddress 5 -endaddress 10 /top/c/mru_mem
```

- returns:

```
# 5: 110 110 110 110 110 000
```

- Display the memory contents of the same instance to the screen in hex format, as follows:

```
mem display -format hex -startaddress 5 -endaddress 10 /top/c/mru_mem
```

- returns:

```
# 5: 6 6 6 6 6 0
```

## Related Topics

[mem load](#)

## mem list

This command displays a flattened list of all memory instances in the current or specified context after a design has been elaborated.

### Syntax

**mem list [-r] [<path>]**

### Description

Each instance line is prefixed by "VHDL:" or "Verilog:", depending on the type of model.

Returns the signal/variable name, address range, depth, and width of the memory.

### Arguments

- **-r**  
(optional) Recursively descends into sub-modules when listing memories.
- **<path>**  
(optional) The hierarchical path to the location the search should start where the default is the current context, as shown in the Structure window.

### Examples

- Recursively list all memories at the top level of the design.

**mem list -r /**

Returns:

```
# Verilog: /top/m/mem[0:255] (256d x 16w)
#
```

- Recursively list all memories in */top2/uut*.

**mem list /top2/uut -r**

Returns:

```
# Verilog: /top2/uut/mem[0:255] x 16w
```

# mem load

This command updates the simulation memory contents of a specified instance. You can upload contents either from a memory data file, a memory pattern, or both. If both are specified, the pattern is applied only to memory locations not contained in the file.

## Syntax

```
mem load {-infile <infile> | -filldata <data_word> [-infile <infile>]} [-endaddress <end>]
[-fillradix <radix_type>] [-filltype {dec | inc | rand | value}] [-format [bin | hex | mti]]
[<path>] [-skip <Nwords>] [-startaddress <st>] [-truncate]
```

## Description

A relocatable memory file is one that has been saved without address information. You can load a relocatable memory file into the instance of a memory core by specifying an address range on the mem load command line. If no address range (starting and ending address) is specified, the memory is loaded starting at the first location.

The order in which the data is placed into the memory depends on the format specified by the -format option. If you choose bin or hex format, the memory is filled low to high, to be compatible with \$readmem commands. This is in contrast to the default MTI format, which fills the memory according to the memory declaration, from left index to right index.

For Verilog objects and VHDL integers and std\_logic types: if the word width in a file is wider than the word width of the memory, the leftmost bits (msb) in the data words are ignored. To allow wide words use the -truncate option which will ignore the msb bits that exceed the memory word size. If the word width in the file is less than the width of the memory, and the leftmost digit of the file data is not 'X', then the leftmost bits are zero filled. Otherwise, they are X-filled.

The type of data required for the -filldata argument is dependent on the -filltype specified: a fixed value, or one that governs an incrementing, decrementing, or random sequence.

- For fixed pattern values, the fill pattern is repeatedly tiled to initialize the memory block specified. The pattern can contain multiple word values for this option.
- For incrementing or decrementing patterns, each memory word is treated as an unsigned quantity, and each successive memory location is filled in with a value one higher or lower than the previous value. The initial value must be specified.
- For a random pattern, a random data sequence will be generated to fill in the memory values. The data type in the sequence will match the type stored in the memory. For std\_logic and associated types, unsigned integer sequences are generated. A seed value may be specified on the command line. For any given seed, the generated sequence is identical.

The interpretation of the pattern data is performed according to the default system radix setting. However, this can be overridden with a standard Verilog-style ‘<radix\_char><data>’ specification.

## Arguments

- **-infile <infile>**  
(required unless the -filldata argument is used) Updates memory data from the specified file.  
    <infile> — The name of a memory file.
- **-endaddress <end>**  
(optional) Specifies the end address for a range of addresses to be loaded.  
    <end> — Specified as any valid address in the memory.
- **-filltype {dec | inc | rand | value}**  
(optional, use with the -filldata argument) Fills in memory addresses in an algorithmic pattern starting with the data word specified in -filldata. If a fill pattern is used without a file option, the entire memory or specified address range is filled with the specified pattern. If both are specified, the pattern is applied only to memory locations not contained in the file.  
    dec — Decrement each succeeding memory word by one digit.  
    inc — Increment each succeeding memory word by one digit.  
    rand — Randomly generate each succeeding memory word starting with the word specified by -filldata as the seed.  
    value — Value (default) Substitute each memory word in the range with the value specified in -filldata.
- **-filldata <data\_word>**  
(required unless -infile is used) Specifies a data word used to fill memory addresses in the pattern specified by -filltype.  
    <data\_word> — Specifies a data word. Must be in the same format as specified by the -fillradix switch.
- **-fillradix <radix\_type>**  
(optional, use with -filldata) Specifies radix of the data specified by the -filldata switch.  
    <radix\_type> — Valid entries (or any unique abbreviations) are: binary, decimal, unsigned, octal, hex, symbolic, and default.
- **-format [bin | hex | mti]**  
(optional, use with -infile) Specifies the format of the file to be loaded.  
    bin — Specifies binary data format.  
    hex — Specifies hex format.  
    mti — MTI format. (default).

Specifies the format of the file to be loaded. The bin and hex values are the standard Verilog hex and binary memory pattern file formats. These can be used with Verilog memories, and with VHDL memories composed of std\_logic types.

In the MTI memory data file format, internal file address and data radix settings are stored within the file itself. Thus, there is no need to specify these settings on the mem load command line. If a format specified on the command line and the format signature stored internally within the file do not agree, the file cannot be loaded.

- <path>

(optional) The hierarchical path to the memory instance. If the memory instance name is unique, shorthand instance names can be used. The default is the current context, as shown in the Structure window.

Memory address indexes can be specified in the instance name also. If addresses are specified both in the instance name and the file, only the intersection of the two address ranges is populated with memory data.

- -skip <Nwords>

(optional) Specifies the number of words to be skipped between each fill pattern value. Used with -filltype and -filldata.

<Nwords> — Specified as an unsigned integer.

- -startaddress <st>

(optional) Specifies the start address for a range of addresses to be loaded.

<st> — Any valid address in the memory.

- -truncate

(optional) Ignores any most significant bits (msb) in a memory word which exceed the memory word size. By default, when memory word size is exceeded, an error results.

## Examples

- Load the memory pattern from the file *vals.mem* to the memory instance */top/m/mem*, filling the rest of the memory with the fixed-value 1'b0.

**mem load -infile vals.mem -format bin -filltype value -filldata 1'b0 /top/m/mem**

When you enter the **mem display** command on memory addresses 0 through 12, you see the following:

```
mem display -startaddress 0 -endaddress 12 /top/m/mem
# 0: 0000000000000000 0000000000000001 0000000000000010
0000000000000011
# 4: 000000000000100 000000000000101 000000000000110
000000000000111
# 8: 0000000000001000 0000000000001001 0000000000000000
0000000000000000
# 12: 0000000000000000
```

- Load the memory pattern from the file *vals.mem* to the memory instance */top/m/mru\_mem*, filling the rest of the memory with the fixed-value 16'Hbeef.  
**mem load -infile vals.mem -format hex -st 0 -end 12 -filltype value -filldata 16'Hbeef /top/m/mru\_mem**
- Load memory instance */top/mem2* with two words of memory data using the Verilog Hex format, skipping 3 words after each fill pattern sequence.  
**mem load -filltype value -filldata "16'hab 16'hcd" /top/mem2 -skip 3**
- Load memory instance */top/mem* with zeros (0).  
**mem load -filldata 0 /top/mem**
- Truncate the msb bits that exceed the maximum word size (specified in HDL code).  
**mem load -format h -truncate -infile data\_files/data.out /top/m\_reg\_inc/mem**

## Related Topics

[mem save](#)

## mem save

This command saves the contents of a memory instance to a file in any of the supported formats: Verilog binary, Verilog hex, and MTI memory pattern data.

### Syntax

```
mem save -outfile <filename> [-addressradix {dec | hex}] [-dataradix <radix_type>]  
[-format {bin | hex | mti}] [-compress | -noaddress] [<path>]  
[-startaddress <st> -endaddress <end>] [-wordsperline <Nwords>]
```

### Description

This command works identically to the mem display command, except that its output is written to a file rather than a display.

The order in which the data is placed into the saved file depends on the format specified by the -format argument. If you choose bin or hex format, the file is populated from low to high, to be compatible with \$readmem commands. This is in contrast to the default mti format, which populates the file according to the memory declaration, from left index to right index.

You can use the mem save command to generate relocatable memory data files. The -noaddress option omits the address information from the memory data file. You can later load the generated memory data file using the memory load command.

### Arguments

- **-outfile <filename>**  
(required) Specifies that the memory contents are to be stored in a file.  
    <filename> — The name of the file where the specified memory contents are to be stored.
- **-addressradix {dec | hex}**  
(optional) Specifies the address radix for the default mti formatted files.  
    dec — Decimal (default).  
    hex — Hexadecimal.
- **-compress**  
(optional) Specifies that only unique lines are printed, identical lines are not printed.  
    Mutually exclusive with the -noaddress switch.
- **-dataradix <radix\_type>**  
(optional) Specifies the data radix for the default mti formatted files.  
    <radix\_type> — Valid entries (or any unique abbreviations) are: binary, decimal, unsigned, octal, hex, and symbolic.

You can change the default radix for the current simulation using the [radix](#) command. You can change the default radix permanently by editing the [DefaultRadix](#) variable in the modelsim.ini file.

- **-endaddress <end>**

(optional) Specifies the end address for a range of addresses to be saved.

    <end> — Any valid address in the memory.

- **-format {bin | hex | mti}**

(optional) Specifies the format of the output file.

    bin — Binary data format.

    hex — Hexadecimal format.

    mti — MTI format. (default).

The bin and hex values are the standard Verilog hex and binary memory pattern file formats. These can be used with Verilog memories, and with VHDL memories composed of std\_logic types.

In the MTI memory data file format, internal file address and data radix settings are stored within the file itself.

- **-noaddress**

(optional) Prevents addresses from being printed. Mutually exclusive with the -compress switch.

- **<path>**

(optional) The hierarchical path to the location of the memory instance. The default is the current context, as shown in the Structure window.

- **-startaddress <st>**

(optional) Specifies the start address for a range of addresses to be saved.

    <st> — Any valid address in the memory.

- **-wordsperline <Nwords>**

(optional) Specifies how many memory values are to be printed on each line.

    <Nwords> — Any unsigned integer where the default assumes an 80 character display width.

## Examples

- Save the memory contents of the instance */top/m/mem(0:10)* to *memfile*, written in the mti radix.

```
mem save -format mti -outfile memfile -start 0 -end 10 /top/m/mem
```

The contents of *memfile* are as follows:

```
// memory data file (do not edit the following line - required for
mem load use)
// format=mti addressradix=d dataradix=s version = 1.0
0: 0000000000000000 0000000000000001 0000000000000010
0000000000000011
4: 000000000000100 000000000000101 000000000000110
000000000000111
8: 0000000000001000 0000000000001001 xxxxxxxxxxxxxxxxx
```

## Related Topics

[mem display](#)

[mem load](#)

## mem search

This command finds and prints to the screen the first occurring match of a specified memory pattern in the specified memory instance. Shorthand instance names are accepted. Optionally, you can instruct the command to print all occurrences. The search pattern can be one word or a sequence of words.

### Syntax

```
mem search {-glob <word> [<word>...] | -regexp <word> [<word>...]}  
[-addressradix {dec | hex}] [-dataradix <radix_type>] [-all] [-replace <word> [<word>...]]  
[-startaddress <address>] [-endaddress <address>] [<path>]
```

### Arguments

- **-glob <word> [<word>...]**

(required unless using -regexp) Specifies the value of the pattern, accepting glob pattern syntax for the search.

**<word>** — Any word pattern. Multiple word patterns are specified as a space separated list. Wildcards are accepted in the pattern.

This argument and -regexp are mutually exclusive arguments.

- **-regexp <word> [<word>...]**

(required unless using -glob) Specifies the value of the pattern, accepting regular expression syntax for the search.

**<word>** — Any word pattern. Wildcards are accepted in the pattern. Multiple word patterns are specified as a space separated list.

This argument and -glob are mutually exclusive arguments.

- **-addressradix { dec | hex }**

(optional) Specifies the radix for the address being displayed.

**dec** — Decimal (default).

**hex** — Hexadecimal.

- **-all**

(optional) Searches the specified memory range and returns all matching occurrences to the transcript. By default only the first matching occurrence is printed.

- **-dataradix <radix\_type>**

(optional) Specifies the radix for the memory data being displayed.

**<radix\_type>** — Can be specified as symbolic, binary, octal, decimal, unsigned, or hex. By default the radix displayed is the system default.

You can change the default radix for the current simulation using the [radix](#) command. You can change the default radix permanently by editing the [DefaultRadix](#) variable in the modelsim.ini file.

- **-endaddress <address>**  
(optional) Specifies the end address for a range of addresses to search.  
    <address> — Any valid address in the memory.
- **<path>**  
(optional) Specifies the hierarchical path to the location of the memory instance. The default is the current context, as shown in the Structure window.
- **-replace <word> [<word>...]**  
(optional) Replaces the found patterns with a designated pattern.  
    <word> — A word pattern Multiple word patterns are accepted, separated by a single space. No wildcards are allowed in the replaced pattern.
- **-startaddress <address>**  
(optional) Specifies the start address for a range of addresses to search.  
    <address> — Any valid address in the memory.

## Examples

- Search for and print to the screen all occurrences of the pattern 16'Hbeef in /uut/u0/mem3:

```
mem search -glob 16'Hbeef -dataradix hex /uut/u0/mem3
```

returns:

```
#7845: beef
#7846: beef
#100223: beef
```

- Search for and print only the first occurrence of 16'Hbeef in the address range 7845:150000, replacing it with 16'Hcafe in /uut/u1/mem3:

```
mem search -glob 16'Hbeef -d hex -replace 16'Hcafe -st 7846 -end 150000
/uut/u1/mem3
```

returns:

```
#7846: cafe
```

- Replace all occurrences of 16'Hbeef with 16'Habe in /uut/u1/mem3:

```
mem search -glob 16'Hbeef -r 16'Habe -addressadix hex -all /uut/u1/mem3
```

returns:

```
#1ea5: 2750
#1ea6: 2750
#1877f: 2750
```

- Search for and print the first occurrence any pattern ending in f:

```
mem search -glob "*f"
```

- Search for and print the first occurrence of this multiple word pattern:

```
mem search -glob "abe cafe" /uut/u1/mem3
```

- Search for patterns "0000 0000" or "0001 0000" in *m/mem*:

```
mem search -data hex -regexp {000[0|1] 0{4}} m/mem -all
```

- Search for a pattern that has any number of 0s followed by any number of 1s as a memory location, and which has a memory location containing digits as the value:

```
mem search -regexp {^0+1+$ \d+} m/mem -all
```

- Search for any initialized location in a VHDL memory:

```
mem search -regexp {[^U]} -all <vhdl_memory>
```

# modelsim

The **modelsim** command starts the ModelSim GUI without prompting you to load a design.

## Syntax

```
modelsim [-do <macrofile>] [<license_option>] [-nosplash]
```

## Description

This command is valid only for Windows platforms and may be invoked in one of three ways:

- from the DOS prompt
- from a ModelSim shortcut
- from the Windows Start > Run menu

To use `modelsim` arguments with a shortcut, add them to the target line of the properties of that shortcut. (As expected, arguments also work on the DOS command line.)

You can invoke the simulator from either the ModelSim> prompt after the GUI starts or from a DO file called by `modelsim`.

## Arguments

- `-do <macrofile>`  
(optional) Executes a DO file when `modelsim` is invoked.  
`<macrofile>` — The name of a DO file

---

### **Note**

 In addition to the script called by this argument, if a DO file is specified by the STARTUP variable in `modelsim.ini`, it will be called when the `vsim` command is invoked.

---

- `<license_option>`  
(optional) Restricts the search of the license manager.
- `-nosplash`  
(optional) Disables the splash screen.

## Related Topics

[do](#)

[Using a Startup File](#)

## **noforce**

This command removes the effect of any active force commands on the selected HDL objects, and also causes the object's value to be re-evaluated.

### **Syntax**

noforce [`<object\_name>`](#) ...

### **Arguments**

- [`<object\_name>`](#)

(required) Specifies the name of an object. Must match an object name used in a previous [`force`](#) command. Multiple object names may be specified as a space separated list. Wildcard characters are allowed.

### **Related Topics**

[`force`](#)

## nolog

This command suspends writing of data to the wave log format (WLF) file for the specified signals.

### Syntax

```
nolog [-all] [-depth <level>] [-howmany] [-in] [-inout] [-internal] [-out] [-ports] [-recursive]  
[-reset] [<object_name>...]
```

### Description

A flag is written into the WLF file for each signal turned off, and the GUI displays "-No Data-" for the signal(s) until logging (for the signal(s)) is turned back on. Logging can be turned back on by issuing another [log](#) command or by doing a nolog -reset.

Because use of the nolog command adds new information to the WLF file, WLF files created when using the nolog command cannot be read by older versions of the simulator.

Transactions written in SCV or Verilog are logged automatically, and can be removed with the nolog command. A transaction is logged into the .wlf file if logging is enabled (in other words, if no nolog command has disabled it) for that stream at the time when the transaction was begun. An entire span of a transaction is either logged or not logged, regardless of the begin and end times specified for that transaction.

### Arguments

- **-all**  
(optional) Turns off logging for all signals currently logged.
- **-depth <level>**  
(optional) Restricts a recursive search (specified with the -recursive argument) to a certain level of hierarchy.  
  
*<level>* — An integer greater than or equal to zero. For example, if you specify -depth 1, the command descends only one level in the hierarchy.
- **-howmany**  
(optional) Returns an integer indicating the number of signals found.
- **-in**  
(optional) Turns off logging only for ports of mode IN whose names match the specification.
- **-inout**  
(optional) Turns off logging only for ports of mode INOUT whose names match the specification.

- **-internal**  
(optional) Turns off logging only for internal (non-port) objects whose names match the specification.
- **-out**  
(optional) Turns off logging only for ports of mode OUT whose names match the specification.
- **-ports**  
(optional) Specifies that the scope of the search is to include all ports.
- **-recursive**  
(optional) Specifies that the scope of the search is to descend recursively into subregions. If omitted, the search is limited to the selected region. You can use the **-depth** argument to specify how many levels of the hierarchy to descend.
- **-reset**  
(optional) Turns logging back on for all unlogged signals.
- **<object\_name>...**  
(optional) Specifies the object name which you want to unlog. Multiple object names may be specified as a space separated list. Wildcard characters are allowed.

## Examples

- Unlog all objects in the design.  
**nolog -r /\***
- Turn logging back on for all unlogged signals.  
**nolog -reset**

## Related Topics

[log](#)

# notepad

This command opens a simple text editor. It may be used to view and edit ASCII files or create new files.

## Syntax

`notepad [<filename>] [-r | -edit]`

## Description

This mode can be changed from the Notepad Edit menu.

Returns nothing.

## Arguments

- `<filename>`  
(optional) Name of the file to be displayed.
- `-r`  
(optional) Specifies read-only mode.
- `-edit`  
(optional) Specifies editing mode. Will not save changes to an existing file that has the Read-only attribute turned on. (default)

## **noview**

This command closes a window in the ModelSim GUI. To open a window, use the view command.

### **Syntax**

**noview <window\_name>...**

### **Arguments**

- <window\_name>...

(required) Specifies the window(s) to close. Multiple window types may be specified in a space separated list. Wildcards permitted. At least one type (or wildcard) is required.

Refer to the view command for a complete list of possible arguments.

You can also close Source windows using the tab or file name.

### **Examples**

- Close the Wave window named "wave1".

**noview wave1**

- Close all List windows.

**noview List**

## nowhen

This command deactivates selected when commands.

### Syntax

**nowhen** [<label>]

### Arguments

- <label>  
(optional) Specifies an individual when command. Wildcards may be used to select more than one when command.

### Examples

- Deactivate the **when** command labeled 99.

**nowhen** 99

- Deactivate all **when** commands.

**nowhen** \*

## onbreak

This command is used within a DO file and specifies one or more scripts to be executed when running a script that encounters a breakpoint in the source code.

### Syntax

```
onbreak <script>; <script>...
```

### Description

The **onbreak** setting will affect any **run** commands that follow the **onbreak** statement until another **onbreak** command is issued. If a DO file is executed from within the script, the DO file script will inherit the **onbreak** setting specified prior to execution unless and until another **onbreak** command is given, in which case, that **onbreak** setting will be in effect until the DO file script completes at which point execution will return to the calling script and the calling script's **onbreak** setting is restored. For more information, refer to [Breakpoint Flow Control in Nested DO files](#). The script must be followed by a **run** command to take effect.

Default behavior when there is no **onbreak** setting in effect is defined by the **OnBreakDefaultAction** Tcl variable. If the **OnBreakDefaultAction** variable is not defined, the simulator default is to resume execution.

Use an empty string to change the **onbreak** command back to the default behavior:

```
onbreak ""
```

In this case, the script will resume after a breakpoint occurs (after any associated **bp** command string is executed).

If you specify this command in a DO file with an empty script (" "), the default behavior is to resume execution of the DO file. The **onbreak** command with no argument will return the current user specified script or empty if the default **onbreak** is in effect.

### Arguments

- <script>

(optional) Any command or script can be used as an argument to **onbreak**. If you want to use more than one command or script, use a semicolon to separate them or place them on multiple lines and enclose the entire script in braces ({} ) or quotation marks (""). You must use the **onbreak** command before a **run**, **run -continue**, or **step** command. If a **run** or **step** command is issued within an **onbreak** script, the script will return immediately and any following commands will not be executed. It is an error to execute any commands within an **onbreak** command string following any of the **run** commands. This restriction applies to any macros or Tcl procedures used in the **onbreak** command string.

## Examples

- Examine the value of the HDL object data when a breakpoint is encountered. Then continue the run command.

**onbreak {exa data ; cont}**

- Resume execution of the DO file on encountering a breakpoint.

**onbreak resume**

- This set of commands test for assertions. Assertions are treated as breakpoints if the severity level is greater than or equal to the current BreakOnAssertion variable setting (refer to [modelsim.ini Variables](#)). By default a severity level of failure or above causes a breakpoint; a severity level of error or below does not.

```
set broken 0
onbreak {
    lassign [runStatus -full] status fullstat
    if {$status eq "error"} {
        # Unexpected error, report info and force an error exit
        echo "Error: $fullstat"
        set broken 1
        resume
    } elseif {$status eq "break"} {
        # If this is a user break, then
        # issue a prompt to give interactive
        # control to the user
        if {[string match "user_*" $fullstat]} {
            pause
        } else {
            # Assertion or other break condition
            set broken 2
            resume
        }
    } else {
        resume
    }
}
run -all
if {$broken == 1} {
    # Unexpected condition.  Exit with bad status.
    echo "failure"
    quit -force -code 3
} elseif {$broken == 2} {
    # Assertion or other break condition
    echo "error"
    quit -force -code 1
} else {
    echo "success!"
}
quit -force
```

## Related Topics

[do](#)

[onerror](#)

[Useful Commands for Handling Breakpoints and Errors](#)

[DO Files](#)

## onElabError

This command specifies one or more commands to be executed when an error is encountered during the elaboration portion of a vsim command. The command is used by placing it within a DO file script. Use the onElabError command without arguments to return to a prompt.

### Syntax

```
onElabError {[<command> [<command>] ...]}
```

### Arguments

- <command>

(optional) Any command can be used as an argument to onElabError. If you want to use more than one command, use a semicolon to separate the commands, or place them on multiple lines. The entire command string must be placed in braces ({}).

### Related Topics

[do](#)

## onerror

This command is used within a DO file script before a run command; it specifies one or more commands to be executed when a running script encounters an error.

### Syntax

```
onerror {[<command> [<command>] ...]}
```

### Description

Using the onerror command without arguments will return the current onerror command string. Use an empty string (onerror "") to change the onerror command back to its default behavior. Use onerror with a [resume](#) command to allow an error message to be printed without halting the execution of the DO file.

When your onerror command is successful, the DO file script will continue normally, unless your command instructs the simulator to quit. For example:

```
onerror{quit -f}
```

or

```
onerror {break}
```

However, if your onerror command is not successful, the simulator will quit. For example:

```
onerror {add wave b}
```

when you do not have a signal named b.

The onerror command is executed when a Tcl command (such as break) encounters an error in the DO file that contains the onerror command (note that a run command does not necessarily need to be in process). Conversely, OnErrorDefaultAction will run even if the DO file does not contain a local onerror command. This can be useful when you run a series of DO files from one script, and you want the same behavior across all DO files.

### Arguments

- <command>  
(optional) Any command can be used as an argument to onerror. If you want to use more than one command, use a semicolon to separate the commands, or place them on multiple lines. The entire command string must be placed in braces ({}).

### Examples

- Force the simulator to quit if an error is encountered while the DO file is running.

```
onerror {quit -f}
```

## Related Topics

[do](#)

[onbreak](#)

[Useful Commands for Handling Breakpoints and Errors](#)

[DO Files](#)

## onfinish

This command controls simulator behavior when encountering \$finish or sc\_stop() in the design code. When you specify this command without an argument, it returns the current setting.

### Syntax

`onfinish [ask | exit | final | stop | default]`

### Arguments

- `ask`  
(optional) In batch mode, the simulation will exit; in GUI mode, the user is prompted for action.
- `exit`  
(optional) The simulation exits without asking for any confirmation.
- `final`  
(optional) The simulation executes all finish blocks before exiting.
- `stop`  
(optional) The simulation ends but remains loaded in memory, allowing for easier post-simulation tasks.
- `default`  
(optional) Uses the current setting for the `OnFinish` variable in the `modelsim.ini` file.

### Related Topics

[OnFinish](#)

## pause

This command interrupts the execution of a macro and allows you to perform interactive debugging of a macro file. Insert this command within the macro to be debugged.

### Syntax

`pause`

### Description

When a macro is interrupted during execution, the macro returns the prompt:

```
VSIM(paused) >
```

This “pause” prompt notifies you that a macro has been interrupted.

When a macro is paused, you can invoke another macro. If the second macro is interrupted, you can continue invoking macros up to a nesting level of 50 macros.

The [status](#) command lists summary information about all interrupted macros.

Use the [resume](#) command to resume execution of the macro. Use the [abort](#) command to stop execution of some or all of the macros.

### Arguments

None.

### Related Topics

[resume](#)

[run](#)

[status](#)

# precision

This command determines how real numbers display in the graphic interface (such as the Objects, Wave, Locals, and List windows). It does not affect the internal representation of a real number and therefore precision values over 17 are not allowed. Executing the precision command without any arguments returns the current precision setting.

## Syntax

**precision** [[\*\*<digits>\[#\]\*\*](#)]

## Arguments

- **<digits>[#]**  
(optional) Specifies the number of digits to display where the default is 6.  
# — A suffix that forces the display of trailing zeros. See examples for more details.

## Examples

- Results in 4 digits of precision.

**precision 4**

For example:

1.234 or 6543

- Results in 8 digits of precision including trailing zeros.

**precision 8#**

For example:

1.2345600 or 6543.2100

- Results in 8 digits of precision but doesn't print trailing zeros.

**precision 8**

For example:

1.23456 or 6543.21

# printenv

This command prints to the Transcript window the current names and values of all environment variables. If variable names are given as arguments, returns only the names and values of the specified variables.

## Syntax

```
printenv [<var>...]
```

## Arguments

- <var>...  
(optional) Specifies the name(s) of the environment variable(s) to print.

## Examples

- Print all environment variable names and their current values.

```
printenv
```

Returns:

```
# CC = gcc
# DISPLAY = srl:0.0
...
```

- Print the specified environment variables:

```
printenv USER HOME
```

Returns:

```
# USER = vince
# HOME = /scratch/srl/vince
```

## process report

This command creates a textual report of all processes displayed in the Process Window.

### Syntax

```
process report [-file <filename>] [-append]
```

### Arguments

- **-file <filename>**  
(optional) Creates an external file where raw process data will be saved. If -file is not specified, then the output is redirected to stdout.  
    <filename> — A user-specified name for the file.
- **-append**  
(optional) Specifies that process data is to be appended to the current process report file. If this option is not used, the process data will overwrite the existing process report file.

# project

Prerequisites:

Some arguments to this command require a project to be opened with either the project new or project open command. Some arguments must be used outside of a simulation session. Refer to the argument descriptions for more information.

This command is used to perform common operations on projects.

## Syntax

```
project [addfile <filename> [<file_type>] [<folder_name>]] | [addfolder <foldername> [<folder_parent>]] | [calculateorder] | [close] | [compileall [-n]] | [compileorder] | [compileoutofdate [-n]] | [delete <filename>] | [filenames] | [env] | [history] | [new <home_dir> <proj_name> [<defaultlibrary>] [<intialini>] [0 | 1]] | [open <project>] | [removefile <filename>]
```

## Arguments

- **addfile <filename> [<file\_type>] [<folder\_name>]**  
(optional) Adds the specified file to the current project. Requires a project to be open.  
    <filename> — (required) The name of an existing file.  
    <file\_type> — (optional) The HDL file type of the file being added. For example do for a *.do* file.  
    <folder\_name> — (optional) Places the file in an existing folder created with project addfolder command. If no folder name is specified the file will be placed in the top level folder.
- **addfolder <foldername> [<folder\_parent>]**  
(optional) Creates a project folder within the project. Requires a project to be open.  
    <foldername> — (required) Any string.  
    <folder\_parent> — (optional) Places <foldername> in an existing parent folder. If <folder\_parent> is unspecified, <foldername> is placed at the top level.
- **calculateorder**  
(optional) Determines the compile order for the project by compiling each file, then moving any compiles that fail to the end of the list. This is repeated until there are no more compile errors.
- **close**  
(optional) Closes the current project.
- **compileall [-n]**  
(optional) Compiles all files in the project using the defined compile order.  
    -n — (optional) Returns a list of the compile commands this command would execute, without actually executing the compiles.

- **compileorder**  
(optional) Returns the current compile order list.
- **compileoutofdate [-n]**  
(optional) Compiles all files that have a newer date/time stamp than the last time the file was compiled.  
-n — Returns a list of the compile commands this command would execute, without actually executing the compiles.
- **delete <filename>**  
(optional) Deletes a project file.  
<filename> — Any *.mpf* file.
- **filenames**  
Returns the absolute pathnames of all files contained in the currently open project.
- **env**  
(optional) Returns the current project file and path.
- **history**  
(optional) Lists a history of manipulated projects. Must be used outside of a simulation session.
- **new <home\_dir> <proj\_name> [<defaultlibrary>] [<initialini>] [0 | 1]**  
(optional) Creates a new project under a specified home directory with a specified name and optionally a default library. The name of the work library will default to "work" unless specified. A new project cannot be created while a project is currently open or a simulation is in progress.  
<home\_dir> — The path to the new project directory within the current working directory.  
<proj\_name> — Specifies a name for the new project. The file will be saved as an *.mpf* file  
<defaultlibrary> — Specifies a name for the default library.  
<initialini> — An optional *modelsim.ini* file can be specified as a seed for the project file by using the initialini option. If initialini is an empty string, then ModelSim uses the current *modelsim.ini* file when creating the project. You must specify a default library if you want to specify initialini.  
0 — (default) Copies all library mappings from the specified <initialini> file into the new project.  
1 — Copies library mappings referenced in an "others" clause in the initial *.ini* file.

- **open <project>**  
(optional) Closes any currently opened project and opens a specified project file (must be a valid *.mpf* file), making it the current project. Changes the current working directory to the project's directory. Must be used outside of a simulation session.
- **removefile <filename>**  
(optional) Removes the specified file from the current project.

## Examples

- Make */user/george/design/test3/test3.mpf* the current project and changes the current working directory to */user/george/design/test3*.

**project open /user/george/design/test3/test3.mpf**

- Execute current project library build scripts.

**project compileall**

## pwd

This Tcl command displays the current directory path in the Transcript window.

### Syntax

`pwd`

### Arguments

None

# quietly

This command turns off transcript echoing for the specified command.

## Syntax

`quietly <command>`

## Arguments

- `<command>`

(required) Specifies the command for which to disable transcript echoing. Any results normally echoed by the specified command will not be written to the Transcript window. To disable echoing for all commands use the [transcript](#) command with the -quietly option.

## Related Topics

[transcript](#)

## quit

This command exits the simulator.

### Syntax

```
quit [-f | -force] [-sim] [-code <integer>]
```

### Description

If you want to stop the simulation using a [when](#) command, you must use a [stop](#) command within your when statement, you must not use an [exit](#) or a quit command. The stop command acts like a breakpoint at the time it is evaluated.

### Arguments

- **-f | -force**  
(optional) Quits without asking for confirmation. If omitted, ModelSim asks you for confirmation before exiting. (The -f and -force arguments are equivalent.)
- **-sim**  
(optional) Unloads the current design in the simulator without exiting ModelSim. All files opened by the simulation will be closed including the WLF file (*vsim.wlf*).
- **-code <integer>**  
(optional) Quits the simulation and issues an exit code.  
  
*<integer>* — This is the value of the exit code. You should not specify an exit code that already exists in ModelSim. Refer to the section "[Exit Codes](#)" in the User's Manual for a list of existing exit codes. You can also specify a variable in place of *<integer>*.

You should always print a message before running the quit -code command to explicitly state the reason for exiting.

### Examples

Refer to the Examples section of the [exit](#) command for an example of using the -code argument. The quit and exit commands behave similarly in this regard.

# radix

This command specifies the default radix to be used for the current simulation. Specifying the command with no argument returns the current radix setting.

## Syntax

```
radix [-binary | -octal | -decimal | -hexadecimal | -unsigned | -ascii | -time]
      [-enumnumeric | -enumsymbolic | -showbase | -symbolic]
```

## Description

The command can be used at any time.

The specified radix is used for all commands ([force](#), [examine](#), [change](#), etc.) as well as for displayed values in the Objects, Locals, Dataflow, List, and Wave windows, as well as the Source window in the source annotation view.

Alternate methods for changing the default radix:

- In the *modelsim.ini* file, edit the [DefaultRadix](#) variable.
- Choose **Simulate > Runtime Options** from the main menu, click the **Defaults** tab, make your selection in the **Default Radix** box.

Numeric radix other than symbolic are defined to translate bits to a 4 state representation as part of the numeric conversion. Groups of bits are then converted to a number in the correct radix, or to 'x' or 'z' if the value is not numeric (that is, contains only '0's and '1's). There is no 'U' in the 4 state representation, nor 'W' or '-'. All of the odd values are converted to 'x'.

Alternatives to changing the default radix for the simulation session include:

- use [examine](#) <name> -<radix\_type> to transcript the current value of <name> using the specified radix
- use [radix](#) [signal](#) to set a signal-specific radix

## Arguments

You can abbreviate the following arguments to any length. For example, -dec is equivalent to -decimal.

- -ascii  
(optional) Display a Verilog object as a string equivalent using 8-bit character encoding.
- -binary  
(optional) Displays values in binary format.

- **-enumnumeric**  
(optional) Causes Verilog enums to be displayed as numbers (formatted by the current radix). This overrides the default behavior of always showing enums symbolically.
- **-enumsymbolic**  
(optional) Restores the default behavior of displaying Verilog enums as symbols by reversing the action of the **-enumnumeric** option.
- **-decimal**  
(optional) Displays values in decimal format. You can specify **-signed** as an alias for this argument.
- **-hexadecimal**  
(optional) Displays values in hexadecimal format.
- **-octal**  
(optional) Displays values in octal format.
- **-time**  
(optional) Displays values of time for register-based types in Verilog.
- **-showbase**  
(optional) Display the number of bits of the vector and the radix used (binary = b, decimal = d, hexadecimal = h, ASCII = a, and time = t)  
For example, instead of simply displaying a vector value of “31”, a value of “16’h31” may be displayed to show that the vector is 16 bits wide, with a hexadecimal radix.
- **-symbolic**  
(optional) Displays values in a form closest to their natural form.
- **-unsigned**  
(optional) Displays values in unsigned decimal format.

## Related Topics

[User-Defined Radices](#)

[radix define](#)

[radix delete](#)

[radix names](#)

[radix list](#)

[radix signal](#)

# radix define

This command is used to create or modify a user-defined radix. A user definable radix is used to map bit patterns to a set of enumeration labels or setup a fixed or floating point radix. User-defined radices are available for use in most windows and with the examine command.

## Syntax

### User Custom Radix

```
radix define <name> <definition_body> [-color <value>]
```

### Fixed or Floating Point Number Radix

```
radix define <name> [[-fixed [-signed] | -float] -fraction <n>] [-base <base>] [-precision <p>]
```

## Arguments

- <name>  
(required) User-specified name for the radix.
- <definition\_body>  
(required for custom radix) A list of number pattern, label pairs. The definition body has the form:

```
{  
  <numeric-value> <enum-label> [-color <color>],  
  <numeric-value> <enum-label>  
  -default <radix_type>  
  -defaultcolor <color>  
}
```

A <numeric-value> is any legitimate HDL integer numeric literal. To be more specific:

```
<integer>  
<base>#<value>#      --- <base> is 2, 8, 10, or 16  
<base>"value"        --- <base> is B, O, or X  
<size>'<base><value>  --- <size> is an integer,  
                           <base> is b, d, o, or h.
```

The question mark (?) wildcard character may be specified for bits or characters of the value. For example:

```
radix define bus-state {  
  6'b01??00 "Write" -color orange,  
  6'b10??00 "Read" -color green  
}
```

In this example, the first pattern will match "010000", "010100", "011000", and "011100". In case of overlaps, the first matching pattern is used, going from top to bottom.

The comma (,) in the definition body is optional. The <enum-label> is any arbitrary string. It should be surrounded by quotation marks (""), especially if it contains spaces.

The -default entry is optional. If present, it defines the radix to use if a match is not found for a given value. The -default entry can appear anywhere in the list, it does not have to be at the

end. The optional -color argument in the definition body will set the color for the specific value when the value is displayed in the Wave window.

Refer to the Verilog and VHDL Language Reference Manuals for exact definitions of these numeric literals.

- **-base <base>**

(optional for fixed and floating point radices) Specifies the base for a fixed or floating point radix.

**<base>** — Any valid radix type: binary, ascii, unsigned, decimal, octal, hex, symbolic, time, and default.

- **-color <value>**

(optional for custom radices) Designates a color for the waveform and text in the Wave window.

**<value>** — The color value may be a color name or its hex value (see example below).

- **-fixed**

(required for fixed point radix) Specifies a fixed number radix.

- **-float**

(required for floating point radix) Specifies a floating point number radix.

- **-fraction <n>**

(required for fixed and floating point radices) Specifies the location of the decimal point in a vector.

**<n>** — Any integer between 3 and the full bit value of the vector. For example, if you specify -fraction 3 for the eight bit vector “10001001”, the decimal is placed two bits away from the least significant bit on the right so the vector becomes “10001.001”.

- **-precision <p>**

(optional for fixed and floating point radices) Specifies how many numbers after the decimal point or significant digits of a floating point or fixed number in symbolic format.

**<p>** — A number, less than or equal to 17, and optional format specification taking the form “<width>[efg],” for example 3g, 4f, or 6.

- **-signed**

(optional for fixed point radix) Treats fixed numbers as signed where the most significant bit is the sign bit. The default is an unsigned number.

## Examples

- The radix define command used to create a radix called “States,” which will display state values in the List, Watch, and Wave windows instead of numeric values.

```

radix define States {
  11'b000000000001 "IDLE",
  11'b000000000010 "CTRL",
  11'b000000000100 "WT_WD_1",
  11'b000000001000 "WT_WD_2",
  11'b000000010000 "WT_BLK_1",
  11'b000000100000 "WT_BLK_2",
  11'b000010000000 "WT_BLK_3",
  11'b000100000000 "WT_BLK_4",
  11'b001000000000 "WT_BLK_5",
  11'b010000000000 "RD_WD_1",
  11'b100000000000 "RD_WD_2",
  11'bzzzzzzzzzzzz "UNCONNECTED",
  11'bxxxxxxxxxxx "ERROR",
  -default hex
}

```

**Note**

 The 'z' and 'x' values must be lower case.

- The following example illustrates how to specify the radix color:

```

radix define States {
  11'b000000000001 "IDLE" -color yellow,
  11'b000000000010 "CTRL" -color #ffee00,
  11'b000000000100 "WT_WD_1" -color orange,
  11'b000000001000 "WT_WD_2" -color orange,
  11'b000000010000 "WT_BLK_1",
  11'b000000100000 "WT_BLK_2",
  11'b000010000000 "WT_BLK_3",
  11'b000100000000 "WT_BLK_4",
  11'b001000000000 "WT_BLK_5",
  11'b010000000000 "RD_WD_1" -color green,
  11'b100000000000 "RD_WD_2" -color green,
  -default hex
  -defaultcolor white
}

```

If a pattern/label pair does not specify a color, the normal wave window colors will be used. If the value of the waveform does not match any pattern, then the -default radix and -defaultcolor will be used.

- Create a fixed point radix named fx5 and apply that radix to the signal checksf.

Entering

**VSIM> radix define fx5 -fixed -fraction 3 -base decimal -signed**

returns

#fx5

Entering

**VSIM> radix signal checksf**

returns

#fx5

Entering

**VSIM> examine -name cecksf**

returns

#/test\_fixed/basictest/checksf -15.8750

## Related Topics

[User-Defined Radices](#)

[precision](#)

[radix](#)

[radix delete](#)

[radix names](#)

[radix list](#)

[radix signal](#)

## radix delete

This command will remove the radix definition from the named radix.

### Syntax

`radix delete <name>`

### Arguments

- `<name>`  
(required) Removes the radix definition from the named radix.

### Related Topics

[User-Defined Radices](#)

[radix](#)

[radix define](#)

[radix list](#)

[radix names](#)

[radix signal](#)

## radix list

This command will return the complete definition of a radix, if a name is given. If no name is given, it will list all the defined radices.

### Syntax

`radix list [<name>]`

### Arguments

- `<name>`  
(optional) Returns the complete definition of the named radix.

### Related Topics

[User-Defined Radices](#)

[radix](#)

[radix define](#)

[radix delete](#)

[radix names](#)

[radix signal](#)

# radix names

This command returns a list of currently defined radix names.

## Syntax

`radix names`

## Arguments

None

## Related Topics

[User-Defined Radices](#)

[radix](#)

[radix define](#)

[radix delete](#)

[radix list](#)

[radix signal](#)

## radix signal

This command sets or inspects radix values for the specified signal in the Objects, Locals, Schematic, and Wave windows. When no argument is used, the radix signal command returns a list of all signals with a radix.

---

### Note

 The intent is for this command to be used for a small number of signals. If the majority of signals in a design are to use a particular radix value, then set that value as the default radix with the [radix](#) command, and use the radix signal command for the rest.

---

### Syntax

```
radix signal [<signal_name> [<radix_value>]] [-showbase]
```

### Arguments

- <signal\_name>  
(optional) Name of the signal for which the radix will be set (if <radix\_value> is specified) or inspected.
- <radix\_value>  
(optional) Value of the radix to be set for the specified signal. Use empty quotation marks ("") to unset the radix for the specified signal.
- -showbase  
(optional) Display the number of bits of the vector and the radix used (binary = b, decimal = d, hexadecimal = h, ASCII = a, and time = t).

For example, instead of simply displaying a vector value of “31”, a value of “16'h31” may be displayed to show that the vector is 16 bits wide, with a hexadecimal radix.

### Related Topics

[User-Defined Radices](#)

[radix](#)

[radix define](#)

[radix list](#)

[radix delete](#)

# readers

This command displays the names of all readers of the specified object.

## Syntax

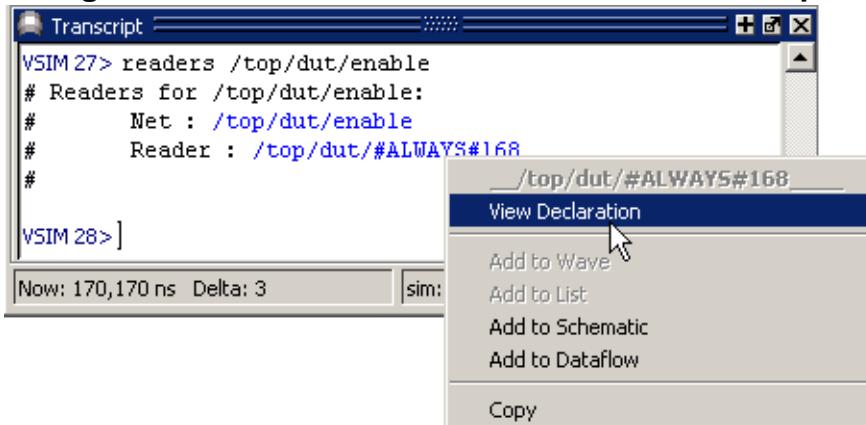
```
readers <object_name> [-source]
```

## Description

The reader list is expressed relative to the top-most design signal/net connected to the specified object.

The output from the readers command, which is displayed in the Transcript window as a hypertext link, allows you to right-click to open a drop-down menu and to quickly add signals to various windows. It includes a "View Declaration" item to open the source definition of the signal.

**Figure 2-4. readers Command Results in Transcript**



## Arguments

- <object\_name>

(required) Specifies the name of the signal or net whose readers are to be shown. All signal or net types are valid. Multiple names and wildcards are accepted.

- -source

(optional) Returns the source file name and line number for each driver of the specified signal or net. If the source location cannot be determined, the value n/a is returned for that driver.

## report

This command displays information relevant to the current simulation.

---

### Note

---

 Viewing preference variables: preference variables have more to do with the way things look (but not entirely) rather than controlling the simulator. You can view preference variables from the Preferences dialog box. Select **Tools > Edit Preferences** (Main window).

---

## Syntax

```
report files
report where [ini] [pwd] [transcript] [wlf] [project]
report simulator control
report simulator state
```

## Arguments

- **files**  
Returns a list of all source files used in the loaded design. This information is also available in the Specified Path column of the Files window.
- **where [ini] [pwd] [transcript] [wlf] [project]**  
Returns a list of configuration files where the arguments limit the list to those files specified. If specified without arguments, returns a list of all configuration files in the current simulation.
  - ini — (optional) Returns the location of the *modesim.ini* file.
  - pwd — (optional) Returns the current working directory.
  - transcript — (optional) Returns the location for saving the transcript file.
  - wlf — (optional) Returns the current location for saving the *.wlf* file.
  - project — (optional) Returns the current location of the project file.
- **simulator control**  
Displays the current values for all simulator control variables.
- **simulator state**  
Displays the simulator state variables relevant to the current simulation.

## Examples

- Display configuration file information

```
report where
```

Returns:

```
#INI {modelsim.ini}
#PWD ./Testcases/
#Transcript transcript
#WLF vsim.wlf
#Project {}
```

- Display all simulator control variables.

**report simulator control**

Returns:

```
#UserTimeUnit = ns
#RunLength =
#IterationLimit = 5000
#BreakOnAssertion = 3
#DefaultForceKind = default
#IgnoreNote = 0
#IgnoreWarning = 0
#IgnoreError = 0
#IgnoreFailure = 0
#IgnoreSVAInfo = 0
#IgnoreSVAWarning = 0
#IgnoreSVAError = 0
#IgnoreSVAFatal = 0
#CheckpointCompressMode = 1
#NumericStdNoWarnings = 0
#StdArithNoWarnings = 0
#PathSeparator = /
#DefaultRadix = symbolic
#DelayFileOpen = 1
#WLFFilename = vsim.wlf
#WLFTimeLimit = 0
#WLFSizeLimit = 0
```

- Display all simulator state variables. Only the variables that relate to the design being simulated are displayed:

**report simulator state**

Returns:

```
#now = 0.0
#delta = 0
#library = work
#entity = type_clocks
#architecture = full
#resolution = 1ns
```

## Related Topics

[modelsim.ini Variables](#)

[Setting GUI Preferences](#)

## restart

This command reloads the design elements and resets the simulation time to zero. Only design elements that have changed are reloaded. (Note that SDF files are always reread during a restart.)

### Syntax

```
restart [-force] [-nobreakpoint] [-nolist] [-nolog] [-nowave]
```

### Description

- If no design is loaded, the restart command produces a message to that effect and takes no further action.
- If a simulation is loaded, the restart command restarts the simulation.
- If multiple datasets are open, including a simulation, the environment is changed to the simulation context and the simulation is restarted.

Shared libraries are handled as follows during a restart:

- Shared libraries that implement VHDL foreign architectures only are reloaded at each restart when the architecture is elaborated (unless `vsim -keeploaded` is used).
- Shared libraries loaded from the command line (-foreign and -pli options) and from the Veriuser entry in the *modelsim.ini* file are reloaded (unless `vsim -keeploaded` is used).
- Shared libraries that implement VHDL foreign subprograms remain loaded (they are not reloaded) even if they also contain code for a foreign architecture.

You can configure defaults for the restart command by setting the `DefaultRestartOptions` variable in the *modelsim.ini* file. Refer to “[Restart Command Defaults](#)”.

To handle restarts with Verilog PLI applications, you need to define a Verilog user-defined task or function, and register a `misctf` class of callback.

Refer to “[Verilog Interfaces to C](#)” for more information on the Verilog HDL interfaces.

### Arguments

- `-force`  
(optional) Specifies that the simulation will be restarted without requiring confirmation in a popup window.
- `-nobreakpoint`  
(optional) Specifies that all breakpoints will be removed when the simulation is restarted where the default is for all breakpoints to be reinstalled after the simulation is restarted.

- **-nolist**

(optional) Specifies that the current List window environment will not be maintained after the simulation is restarted where the default is for all currently listed HDL objects and their formats to be maintained.

- **-nolog**

(optional) Specifies that the current logging environment will not be maintained after the simulation is restarted where the default is for all currently logged objects to continue to be logged.

- **-nowave**

(optional) Specifies that the current Wave window environment will not be maintained after the simulation is restarted where the default is for all objects displayed in the Wave window to remain in the window with the same format.

## resume

This command is used to resume execution of a macro (DO) file after a pause command or a breakpoint.

### Syntax

resume

### Description

This command may be input manually or placed in an [onbreak](#) command string. (Placing a **resume** command in a [bp](#) command string does not have this effect.) The **resume** command can also be used in an [onerror](#) command string to allow an error message to be printed without halting the execution of the macro file.

### Arguments

None

### Related Topics

[pause](#)

[Useful Commands for Handling Breakpoints and Errors](#)

# run

This command advances the simulation by the specified number of timesteps.

## Syntax

```
run {[<timesteps>[<time_units>]] | -all | -continue | -init | -next } |  
{-step [-current] [<n>] [-out] [-over [<n>]] [-this]}
```

## Description

You can control any return values after the run operation completes with the following preference variables:

- noRunTimeMsg — Set this variable to 0 to display simulation time and delta information or set it to 1 to disable the display of this information.
- noRunStatusMsg — Set this variable to 0 to display run status information or set it to 1 to disable the display of this information.

The following is an example that shows a series of run commands and how the output changes with the preference variable settings:

```
VSIM 1> run 105
VSIM 2> set PrefMain(noRunTimeMsg) 0
# 0

VSIM 3> run 112
# Time: @217 ns 0

VSIM 4> set PrefMain(noRunStatusMsg) 0
# 0

VSIM 5> run 100
# Time: @317 ns 0
# Status: ready end

VSIM 6> set PrefMain(noRunTimeMsg) 1
# 1

VSIM 7> run 50
# Status: ready end

VSIM 8> set PrefMain(noRunStatusMsg) 1
# 1

VSIM 9> run 55

VSIM 10>
```

## Arguments

- No arguments

Runs the simulation for the default time (100 ns).

You can change the default <timesteps> and <time\_units> in the GUI with the Run Length toolbar box in the Simulate toolbar or from the *modelsim.ini* file: [RunLength](#) and [UserTimeUnit](#) variables.

- <timesteps>[<time\_units>]

(optional) Specifies the number of timesteps for the simulation to run. The number may be fractional, or may be specified as absolute by preceding the value with the character @.

<time\_units> — Any valid time unit: fs, ps, ns, us, ms, or sec where the default is to use the current time unit.

- -all

(optional) Causes the simulator to run the current simulation forever, or until it hits a breakpoint or specified break event.

- -continue

(optional) Continues the last simulation run after a run -step, run -step -over command or a breakpoint. A run -continue command may be input manually or used as the last command in a [bp](#) command string.

- -final

(optional) Instructs the simulator to run all final blocks then exit the simulation.

- -init

(optional) Initializes non-trivial static SystemVerilog variables before beginning the simulation, for example, expressions involving other variables and function calls,. This could be useful for when you want to initialize values before executing any [force](#), [examine](#), or [bp](#) commands.

You cannot use run -init after any other run commands or if you have specified [vsim](#) -runinit on the command line because all variables would have been initialized by that point.

- -next

(optional) Causes the simulator to run to the next event time.

- -step

(optional) Steps the simulator to the next HDL.

Current values of local HDL variables may be observed at this time using the Locals window. You can specify the following arguments when you use -step:

-current

(optional) Instructs the simulation to step into an instance, process, or thread and stay in the current thread. Prevents stepping into a different thread.

<n>

(optional) Moves the simulator <n> steps ahead. Moves the debugger <n> lines ahead when you are using C Debug. Specified as a positive integer value.

-out

(optional) Instructs the simulation to step out of the current function or procedure and return to the caller.

**-over**

(optional) Directs ModelSim to run VHDL procedures and functions, Verilog tasks and functions but to treat them as simple statements instead of entering and tracing them line by line.

You can use the **-over** argument to skip over a VHDL procedures or functions, Verilog task or functions.

When a wait statement or end of process is encountered, time advances to the next scheduled activity. ModelSim then updates the Process and Source windows to reflect the next activity.

**-this "this==<class\_handle>"**

(optional) Instructs the simulation to step into a method of a SystemVerilog class when “this” refers to the specified class handle. To obtain the handle of the class, use the [examine -handle](#) command.

**<class\_handle>** — Specifies a SystemVerilog class. Note that you must use quotation marks (" ") with this argument.

## Examples

- Advance the simulator 1000 timesteps.

**run 1000**

- Advance the simulator the appropriate number of timesteps corresponding to 10.4 milliseconds.

**run 10.4 ms**

- Advance the simulator to timestep 8000.

**run @8000**

- Advance the simulator into the instance /top/p.

**run -step -current /top/p**

## Related Topics

[Simulate Toolbar](#)

[step](#)

## runStatus

This command returns the current state of your simulation to stdout after issuing a run or step command.

### Syntax

`runStatus [-full]`

### Arguments

- `-full`

(optional) Appends additional information to the output of the runStatus command.

### Return Values

[Table 2-4 \(runStatus Command States\)](#) and [Table 2-5 \(runStatus -full Command Information\)](#) show outputs of the runStatus command.

**Table 2-4. runStatus Command States**

State	Description
ready	The design is loaded and is ready to run.
break	The simulation stopped before completing the requested run.
error	The simulation stopped due to an error condition.
loading	The simulation is currently elaborating.
nodesign	There is no design loaded.
checkpoint	A checkpoint is being created, do not interrupt this process.
cready	The design is loaded and is ready to run in C debug mode.
initializing	The user interface initialization is in progress.

**Table 2-5. runStatus -full Command Information**

-full Information	Description
bkpt	stopped at breakpoint
bkpt_builtin	stopped at breakpoint on builtin process
end	reached end of requested run
fatal_error	encountered fatal error (such as, divide by 0)
iteration_limit	iteration limit reached, possible feedback loop
silent_halt	mti_BreakSilent() called,

**Table 2-5. runStatus -full Command Information (cont.)**

<b>-full Information</b>	<b>Description</b>
step	run -step completed
step_builtin	run -step completed on builtin process
step_wait_suspend	run -step completed, time advanced.
user_break	run interrupted do to break-key or ^C (SIGINT)
user_halt	mti_Break() called.
user_stop	stop or finish requested from stop command, etc.
gate_oscillation	Verilog gate iteration limit reached.
simulation_stop	pli stop_simulation() called.

## searchlog

This command searches one or more of the currently open logfiles for a specified condition.

### Syntax

```
searchlog [-command <cmd>] [-count <n>] [-deltas] [-endtime <time> [<unit>]] [-env <path>]
           [-event <time>] [-expr {<expr>}] [-reverse] [-rising | -falling | -anyedge]
           [-startDelta <num>] [-value <string>] <startTime> [<unit>] <pattern>
```

### Description

It can be used to search for rising or falling edges, for signals equal to a specified value, or for when a generalized expression becomes true.

Arguments to this command are order dependent. Please read the argument descriptions for more information.

If at least one match is found, it returns the time (and, optionally, delta) at which the last match occurred and the number of matches found, in a Tcl list:

```
{ {<time>} <matchCount> }
```

where <time> is in the format <number> <unit>. If the -deltas option is specified, the delta of the last match is also returned:

```
{ {<time>} <delta> <matchCount> }
```

If no matches are found, a TCL\_ERROR is returned. If one or more matches are found, but less than the number requested, it is not considered an error condition, and the time of the farthest match is returned, with the count of the matches found.

### Arguments

- **-command <cmd>**

(optional) Specifies a Tcl command that will be called for each event on the specified signal.

<cmd> — A Tcl command that receives four arguments and returns one of three values: "continue", "stop", or "" (empty).

The command will be passed four arguments: the reason for the call, the time of the event, the delta for the event, and the value. The reason value will be one of WLF\_STARTLOG, WLF\_ENDLOG, WLF\_EVENT, or WLF\_WAKEUP. The function is expected to return "continue", "stop", or "" (empty). If "continue" or "" (empty) is returned, the search continues, making additional callbacks as necessary. If "stop" is returned, the search stops and control is returned to the caller of the searchlog command.

Only searching for a single signal is supported.

- **-count <n>**

(optional) Specifies to search for the nth occurrence of the match condition.

**<n>** — Any positive integer.

- **-deltas**

(optional) Indicates to test for a match on simulation delta cycles. Otherwise, matches are only tested for at the end of each simulation time step.

- **-endtime <time> [<unit>]**

(optional) Specifies the simulation time at which to end the search. By default no end time is specified. Must be specified after **-reverse** when searching backwards.

**<time>** — Specified as an integer or decimal number. Current simulation units are the default unless specifying **<unit>**.

**<unit>** — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting **<unit>**. Valid VHDL time units are: fs, ps, ns, us, ms, sec, min, and hr. If **<unit>** is specified, you must enclose **<time>** and **<unit>** within curly braces ({}).

For **-reverse** searches the specified end time must be earlier than the specified **<startTime>**. For **-reverse** searches the default **<endTime>** is 0.

- **-env <path>**

(optional) Specifies a path to a design region. Wildcards not allowed.

- **-event <time>**

(optional) Indicates to test for a match on a simulation event. Otherwise, matches are only tested for at the end of each simulation time step.

- **-expr {<expr>}**

(optional) Specifies a search for a general expression of signal values and simulation time. **searchlog** will search until the expression evaluates to true.

{<expr>} — An expression that evaluates to a boolean true. See “[GUI\\_expression\\_format](#)” on page 32 for the format of the expression.

- **-reverse**

(optional) Specifies to search backwards in time from **<startTime>**. You can limit the time span for the reverse search by including the **-endtime <time>** argument.

- **-rising**

(optional) Specifies a search for rising edge on a scalar signal. This option is ignored for compound signals.

- **-falling**

(optional) Specifies a search for falling edge on a scalar signal. This option is ignored for compound signals.

- **-anyedge**

(optional) Specifies a search for a rising or falling edge on a scalar signal. This option is ignored for compound signals. (default)

- **-startDelta <num>**  
(optional) Indicates a simulation delta cycle on which to start.  
*<num>* — Any positive integer.
- **-value <string>**  
(optional) Specifies a match of a single scalar or compound signal against a specified string.  
*<string>* — Specifies a string to be matched.
- **<startTime> [<unit>]**  
(required) Specifies the simulation time at which to start the search. The time is specified as an integer or decimal number. Must be placed immediately before the *<pattern>* argument.  
*<unit>* — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting *<unit>*. Valid VHDL time units are: fs, ps, ns, us, ms, sec, min, and hr. If *<unit>* is specified, you must enclose *<time>* and *<unit>* within curly braces ({}).
- **<pattern>**  
(Required unless the -expr argument is used.) Specifies one or more signal names or wildcard patterns of signal names to search on. Must be specified as the final argument to the searchlog command.

## see

This command displays the specified number of source file lines around the current execution line and places a marker to indicate the current execution line. If specified without arguments, five lines will be displayed before and four lines after.

### Syntax

see [<n> | <pre> <post>]

### Arguments

- <n>  
(optional) Designates the number of lines to display before and after the current execution line.
- <pre>  
(optional) Designates the number of lines to display before the current execution line.
- <post>  
(optional) Designates the number of lines to display after the current execution line.

### Examples

- Display 2 lines before and 5 lines after the current execution line.

**see 2 5**

```
# 92 :
# 93 :           if (verbose) $display("Read/Write test done");
# ->94 :           $stop(1);
# 95 :           end
# 96 :           end
# 97 :
# 98 :           or2 i1 (
# 99 :           .y(t_set),
```

## **setenv**

This command changes or reports the current value of an environment variable. The setting is valid only for the current ModelSim session. Arguments to this command are order dependent. Please read the argument descriptions for more information.

### **Syntax**

`setenv <varname> [<value>]`

### **Arguments**

- `<varname>`  
(required) The name of the environment variable you wish to set or check. Must be specified as the first argument to the setenv command.
- `<value>`  
(optional) The new value for `<varname>`. If you do not specify a value, ModelSim reports the variable's current value.

### **Related Topics**

[unsetenv](#)

[printenv](#)

# shift

This command shifts macro parameter values left one place, so that the value of parameter  $\$2$  is assigned to parameter  $\$1$ , the value of parameter  $\$3$  is assigned to  $\$2$ , and so on. The previous value of  $\$1$  is discarded.

## Syntax

shift

## Description

The shift command and macro parameters are used in macro files. If a macro file requires more than nine parameters, they can be accessed using the shift command.

To determine the current number of macro parameters, use the `argc` simulator state variable. Refer to “[Simulator State Variables](#)” for more information.

For a macro file containing nine macro parameters defined as  $\$1$  to  $\$9$ , one shift command shifts all parameter values one place to the left. If more than nine parameters are named, the value of the tenth parameter becomes the value of  $\$9$  and can be accessed from within the macro file.

## Arguments

None

# show

This command lists HDL objects and subregions visible from the current environment.

## Syntax

```
show [-all] [<pathname>]
```

## Description

The objects listed include:

- VHDL — signals, processes, constants, variables, and instances.
- Verilog — nets, registers, tasks, functions, instances, variables, and memories.

The show command returns formatted results to stdout. To eliminate formatting (to use the output in a Tcl script), use the Show form of the command instead.

## Arguments

- **-all**  
(optional) Displays all names at and below the specified path recursively.
- **<pathname>**  
(optional) Specifies the pathname of the environment for which you want the objects and subregions to be listed; if omitted, the current environment is assumed.

## Examples

- List the names of all the objects and subregion environments visible in the current environment.

**show**

- List the names of all the objects and subregions visible in the environment named /uut.

**show /uut**

- List the names of all the objects and subregions visible in the environment named sub\_region which is directly visible in the current environment.

**show sub\_region**

- List the names of all the objects and subregions visible in all top-level environments.

**show -all /**

# simstats

This command returns performance-related statistics about elaboration and simulation with the data for each statistic on a separate line. The statistics measure the simulation kernel process (vsimk) for a single invocation of vsim. If you invoke vsim a second time, or restart the simulation, the current statistics are discarded and new values are collected.

## Syntax

```
simstats [ elabcpu | elabmemory | elabtime | license | logcpu | logtime | simcpu | simmemory |  
          simtime | tlcmdcpu | tlcmdtime | totalcpu | totaltime | verbose ] [kb]
```

## Description

If executed without arguments, the command returns a list of statistics and their related units on separate lines. For example:

```
# Memory Statistics  
#     mem: size after elab (VSZ)          88.89 Mb  
#     mem: size during sim (VSZ)           97.17 Mb  
# Elaboration Time  
#     elab: wall time                   0.41 s  
#     elab: cpu time                   0.23 s  
# Simulation Time  
#     sim: wall time                  1.18 s  
#     sim: cpu time                  0.66 s  
# Tcl Command Time  
#     cmd: wall time                356.39 s  
#     cmd: cpu time                  0.80 s  
# Total Time  
#     total: wall time              357.98 s  
#     total: cpu time              1.68 s
```

You can use the [simstatslist](#) command to provide this output as a continuous display (without line breaks).

All statistics are measured at the time you invoke simstats. See the arguments below for descriptions of each statistic.

Units for time values are in seconds. Units for memory values are auto-scaled.

---

### Note

 Different operating systems report these numbers differently.

---

## Arguments

- **elabcpu**  
(optional) Returns cpu time consumed by vsim elaboration.
- **elabmemory**  
(optional) Returns memory consumed during vsim elaboration.

- **elabtime**  
(optional) Returns wall clock time consumed by vsim elaboration.
- **kb**  
(optional) Returns statistics in kilobyte units with no auto-scaling.
- **license**  
(optional) Returns a ‘License Statistics’ section that includes license statistics for checkout time and checked-out license feature names.
- **logcpu**  
(optional) Returns cpu time consumed by WLF logging.
- **logtime**  
(optional) Returns wall clock time consumed by WLF logging.
- **simcpu**  
(optional) Returns cumulative cpu time consumed by all run commands
- **simmemory**  
(optional) Returns memory consumed during the whole simulation, including the elaboration memory.
- **simtime**  
(optional) Returns cumulative wall clock time consumed by all run commands.
- **tclcmdcpu**  
(optional) Returns cpu time consumed by all TCL commands, excluding run commands.
- **tclcmdtime**  
(optional) Returns wall clock time consumed by all TCL commands, excluding run commands.
- **totalcpu**  
(optional) Returns total cpu time consumed by vsim command.
- **totaltime**  
(optional) Returns total wall clock time consumed by vsim command.
- **verbose**  
(optional) Displays verbose performance statistics, including an ‘elab’ report for checked-out license feature names.

## simstatslist

This command returns performance-related statistics about elaboration and simulation as a continuous list (without line breaks).

### Syntax

```
simstatslist [elabcpu | elabmemory | elabtime | logcpu | logtime | simcpu | simmemory | simtime  
| tclcmdcpu | tclcmdtime | totalcpu | totaltime ]
```

### Description

Use this command in place of the [simstats](#) command to display the original statistics in a continuous format (without line breaks). For example:

```
 {{elab memory} 105348} {{sim memory} 171492} {{elab time} 0.410009}  
 {{elab cpu time} 0.234002} {{sim time} 1.18003} {{sim cpu time} 0.655204}  
 {{tclcmd time} 411.601} {{tclcmd cpu time} 0.795605}  
 {{total time} 413.191} {{total cpu time} 1.68481}
```

All statistics are measured at the time you invoke `simstatslist`. See the arguments below for descriptions of each statistic.

Units for time values are in seconds. Units for memory values are in kilobytes.

---

#### **Note**

 Different operating systems report these numbers differently.

---

### Arguments

- `elabcpu`  
(optional) Returns cpu time consumed by `vsim` elaboration.
- `elabmemory`  
(optional) Returns memory consumed during `vsim` elaboration.
- `elabtime`  
(optional) Returns wall clock time consumed by `vsim` elaboration.
- `logcpu`  
(optional) Returns cpu time consumed by WLF logging.
- `logtime`  
(optional) Returns wall clock time consumed by WLF logging.
- `simcpu`  
(optional) Returns cumulative cpu time consumed by all run commands

- **simmemory**  
(optional) Returns memory consumed during the whole simulation, including the elaboration memory.
- **simtime**  
(optional) Returns cumulative wall clock time consumed by all run commands.
- **tclcmdcpu**  
(optional) Returns cpu time consumed by all TCL commands, excluding run commands.
- **tclcmdtime**  
(optional) Returns wall clock time consumed by all TCL commands, excluding run commands.
- **totalcpu**  
(optional) Returns total cpu time consumed by vsim command.
- **totaltime**  
(optional) Returns total wall clock time consumed by vsim command

# stack down

This command moves down the call stack.

## Syntax

`stack down [n]`

## Description

If invoked without arguments, the command moves down the call stack by 1 level. The Locals window displays local variables at the level.

## Arguments

- `n`

(optional) Moves down the call stack by *n* levels. The default value is 1 level.

## Related Topics

[stack frame](#)

[stack level](#)

[stack tb](#)

[stack up](#)

## stack frame

This command selects the specified call frame.

### Syntax

stack frame n

### Arguments

- <n>

Selects call frame number *n*. The currently executing frame is zero (also called the innermost) frame, frame one is the frame that called the innermost, and so on. The highest numbered frame is that of *main*.

### Related Topics

[stack down](#)

[stack level](#)

[stack tb](#)

[stack up](#)

## stack level

This command reports the current call frame number.

### Syntax

`stack level`

### Arguments

None

### Related Topics

[stack down](#)

[stack frame](#)

[stack tb](#)

[stack up](#)

## **stack tb**

This command displays a stack trace for the current process in the Transcript window. This lists the sequence of HDL function calls that have been entered to arrive at the current state for the active process. The tb command is an alias for the stack tb command.

### **Syntax**

`tb`

### **Description**

None

### **Arguments**

None

# stack up

This command moves up the call stack.

## Syntax

`stack up [n]`

## Description

If invoked without arguments, the command moves up the call stack by 1 level. The Locals window displays local variables at the level.

## Arguments

- `n`

(optional) Moves up the call stack by *n* levels. The default value is 1 level.

## Related Topics

[stack down](#)

[stack frame](#)

[stack level](#)

[stack tb](#)

## status

This command lists summary information about currently interrupted macros.

### Syntax

status [[file](#) | [line](#)]

### Description

If invoked without arguments, the command lists the filename of each interrupted macro, the line number at which it was interrupted, and prints the command itself. It also displays any [onbreak](#) or [onerror](#) commands that have been defined for each interrupted macro.

### Arguments

- [file](#)  
(optional) Reports the file pathname of the current macro.
- [line](#)  
(optional) Reports the line number of the current macro.

### Examples

The transcript below contains examples of [resume](#), and status commands.

```
VSIM(paused) > status
# Macro resume_test.do at line 3 (Current macro)
#   command executing: "pause"
#   is Interrupted
#   ONBREAK commands: "resume"
# Macro startup.do at line 34
#   command executing: "run 1000"
#   processing BREAKPOINT
#   is Interrupted
#   ONBREAK commands: "resume"
VSIM(paused) > resume
# Resuming execution of macro resume_test.do at line 4
```

### Related Topics

[pause](#)

[resume](#)

# step

The step command is an alias for the run command with the -step switch. Steps the simulator to the next HDL.

## Syntax

```
step [-current] [<n>] [-out] [-over [<n>]] [-this "this==<class_handle>"]
```

## Description

Current values of local HDL variables may be observed at this time using the Locals window.

You can control any return values after the step operation completes with the following preference variables:

- noRunTimeMsg — Set this variable to 0 to display simulation time and delta information or set it to 1 to disable the display of this information.
- noRunStatusMsg — Set this variable to 0 to display run status information or set it to 1 to disable the display of this information.

The following is an example that shows a series of run commands (the step command behaves similarly) and how the output changes with the preference variable settings:

```
VSIM 1> run 105
VSIM 2> set PrefMain(noRunTimeMsg) 0
# 0

VSIM 3> run 112
# Time: @217 ns 0

VSIM 4> set PrefMain(noRunStatusMsg) 0
# 0

VSIM 5> run 100
# Time: @317 ns 0
# Status: ready end

VSIM 6> set PrefMain(noRunTimeMsg) 1
# 1

VSIM 7> run 50
# Status: ready end

VSIM 8> set PrefMain(noRunStatusMsg) 1
# 1

VSIM 9> run 55

VSIM 10>
```

## Arguments

- **-current**  
(optional) Instructs the simulation to step into an instance, process, or thread and stay in the current thread. Prevents stepping into a different thread.
- **<n>**  
Moves the simulator **<n>** steps ahead. Specified as a positive integer value.
- **-out**  
(optional) Instructs the simulation to step out of the current function or procedure and return to the caller.
- **-over**  
(optional) Directs ModelSim to run VHDL procedures and functions, Verilog tasks and functions but to treat them as simple statements instead of entering and tracing them line by line.

You can use the **-over** argument to skip over a VHDL procedure or function, Verilog task or function.

When a wait statement or end of process is encountered, time advances to the next scheduled activity. ModelSim then updates the Process and Source windows to reflect the next activity.

- **-this "this==<class\_handle>"**  
(optional) Instructs the simulation to step into a method of a SystemVerilog class when "this" refers to the specified class handle. To obtain the handle of the class, use the [examine -handle](#) command.

**<class\_handle>** — Specifies a SystemVerilog class. Note that you must use quotation marks (" ") with this argument.

## Related Topics

[run](#)

[Stepping Through Your Design](#)

# stop

This command is used with the when command to stop simulation in batch files. The stop command has the same effect as hitting a breakpoint. The stop command may be placed anywhere within the body of the when command.

## Syntax

`stop [-sync]`

## Description

Use `run -continue` to continue the simulation run, or the `resume` command to continue macro execution. If you want macro execution to resume automatically, put the `resume` command at the top of your macro file:

`onbreak {resume}`

---

### Note

---

 If you want to stop the simulation using a `when` command, you must use a `stop` command within your `when` statement. DO NOT use an `exit` command or a `quit` command. The `stop` command acts like a breakpoint at the time it is evaluated.

---

## Arguments

- `-sync`  
(optional) Stops the currently running simulation at the next time step.

## Related Topics

[resume](#)

[run](#)

## suppress

This command prevents one or more specified messages from displaying. You cannot suppress Fatal or Internal messages. The suppress command used without arguments returns the message numbers of all suppressed messages.

---

### Note

---

 To use the suppress command, you must have a design loaded in ModelSim. Otherwise, ModelSim will display an error message without running the command.

---

### Syntax

```
suppress [-clear <msg_number>[,<msg_number>,...]] [<msg_number>[,<msg_number>,...]]  
[<code_string>[,<code_string>,...]]
```

### Description

Edit the `suppress` variable in the *modelsim.ini* file to set a permanent default. Refer to “[Message Severity Level](#)” for more information.

### Arguments

- `-clear <msg_number>[,<msg_number>,...]`  
(optional) Clears suppression of one or more messages identified by message number.  
`<msg_number>` — A number identifying the message. Multiple message numbers are specified as a comma separated list.
- `<msg_number>[,<msg_number>,...]`  
(optional) A number identifying the message to be suppressed. Multiple message numbers are specified as a comma separated list.
- `<code_string>[,<code_string>,...]`  
(optional) A string identifier of the message to be suppressed. Disables warning messages in the category specified by `<code_string>`. Warnings that can be disabled include the `<code_string>` name in square brackets in the warning message.

### Examples

- Return the message numbers of all suppressed messages:

```
suppress
```

- Suppress messages by message number:

```
suppress 8241,8242,8243,8446,8447
```

- Suppress messages by numbers and code categories:

```
suppress 8241,TFMPC,CNNODP,8446,8447
```

- Clear message suppression for the designated messages:

**suppress -clear 8241,8242**

## tb

This (traceback) command displays a stack trace for the current process in the Transcript window. It lists the sequence of HDL function calls that have been entered to arrive at the current state for the active process. The tb command is an alias for the stack tb command.

### Syntax

tb

### Description

None

### Arguments

None

# Time

The string “Time” is used as the suffix for a collection of related commands that allow you to perform comparisons between, operations on, and conversions of, time values for simulation. Arguments for each command are order-dependent, as shown in the Syntax section below.

## Syntax

**eqTime <time1>[unit] <time2>[unit]**

Returns a 1 (true) or 0 (false) if <time1> and <time2> are equal.

**neqTime <time1>[unit] <time2>[unit]**

Returns a 1 (true) or 0 (false) if <time1> and <time2> are not equal.

**ltTime <time1>[unit] <time2>[unit]**

Returns a 1 (true) or 0 (false) if <time1> is less than <time2>.

**gtTime <time1>[unit] <time2>[unit]**

Returns a 1 (true) or 0 (false) if <time1> is greater than <time2>.

**lteTime <time1>[unit] <time2>[unit]**

Returns a 1 (true) or 0 (false) if <time1> is less than or equal to <time2>.

**gteTime <time1>[unit] <time2>[unit]**

Returns a 1 (true) or 0 (false) if <time1> is greater than or equal to <time2>.

**addTime <time1>[unit] <time2>[unit]**

Returns the sum of adding <time1> to <time2>

**subTime <time1>[unit] <time2>[unit]**

Returns the value of subtracting <time2> from <time1>

**mulTime <time1>[unit] <integer>**

Returns the value of multiplying <time1> by an <integer>

**divTime <time1>[unit] <time2>[unit]**

Returns an integer, that is the value of dividing <time1> by <time2>.

Specifying 0 for <time2> results in an error.

**intToTime <high\_32bit\_int> <low\_32bit\_int>**

Returns a 64-bit time value based on two 32-bit parts of a 64-bit integer.

This command is useful when you’ve performed an integer calculation that results in a 64-bit value and need to convert it to a time unit.

**scaleTime <time1>[unit] <scale\_factor>**

Returns a time value scaled by a real number and truncated to the current time resolution.

### RealToTime <real>

Returns a time value equivalent to the specified real number and truncated to the current time resolution.

### validTime <time>

Returns a 1 (true) or 0 (false) if the given string is a valid time for use with any of these Time calculations.

### formatTime {+ | -} commas | {+ | -}nodefunit | {+ | -}bestunits

Sets display properties for time values.

## Description

When [unit] is left unspecified, each of these commands assumes the current simulation time unit, as specified by the [Resolution](#) variable in the *modelsim.ini* file or by using the [vsim -t](#) command. For most commands, units of time (such as ns, us, ps) can be specified independently for each <time[1 | 2]>. See the description of each command and examples for more information.

## Arguments

- <time1>[unit]

<time> — Specified as an integer or decimal number. Current simulation units are the default unless specifying <unit>.

<unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid VHDL time units are:

fs — femtosecond ( $10^{-15}$  seconds)

ps — picosecond ( $10^{-12}$  seconds)

ns — nanosecond ( $10^{-9}$  seconds)

us — microsecond ( $10^{-6}$  seconds)

ms — millisecond ( $10^{-3}$  seconds)

sec — second

min — minute (60 seconds)

hr — hour (3600 seconds)

Note that if you put a space between the values, you must enclose the argument in braces ({ }) or quotation marks ("").

- <high\_32bit\_int> | <low\_32bit\_int>

<high\_32bit\_int> — The "high" part of the 64-bit integer.

<low\_32bit\_int> — The "low" part of the 64-bit integer.

- <scale\_factor> — a real number to be used as scaling factor. Common values can include:  
0.25, 0.5, 1.5, 2, 10, 100
- {+ | -} commas — controls whether commas are displayed in time values.
  - +commas — time values include commas
  - commas — time values do not include commas
- {+ | -}nodefunit — controls whether time values display time units
  - +nodefunit — time values do not include time units and will be in current time resolution
  - nodefunit — time values may include time units
- {+ | -}bestunits — controls whether time values display the largest possible time unit. For example, 8 us instead of 8,000 ns.
  - +bestunits — time values display the largest possible time unit
  - bestunits — time values display the default time unit

## Examples

- Entering the command:

**>ltTime 100ns 1ms**

Returns:

# 1

- Entering the command:

**>addTime {1545 ns} {455 ns}**

Returns:

# 2 us

- Entering the command:

**>gteTime "1000 ns" "1 us"**

Returns:

# 1

- Entering the command:

**>divTime 1us 10ns**

Returns:

# 100

- Entering the command:

**>formatTime +bestunit**

Returns:

# -commas -nodefunit +bestunits

- Entering the command:

**>scaleTime 3ms 1000**

Returns:

# 3 sec

- Entering the command:

**>RealToTime 1.345e04**

Returns:

# 13450 ns

# transcript

This command controls echoing of commands executed in a macro file. If no option is specified, the current setting is reported.

## Syntax

```
transcript [on | off | -q | quietly]
```

## Arguments

- **on**  
(optional) Specifies that commands in a macro file will be echoed to the Transcript window as they are executed.
- **off**  
(optional) Specifies that commands in a macro file will not be echoed to the Transcript window as they are executed.
- **-q**  
(optional) Returns "0" if transcripting is turned off or "1" if transcripting is turned on. Useful in a Tcl conditional expression.
- **quietly**  
(optional) Turns off the transcript echo for all commands. To turn off echoing for individual commands see the [quietly](#) command.

## Examples

- Commands within a macro file will be echoed to the Transcript window as they are executed.

```
transcript on
```

- If issued immediately after the previous example, the command:

```
transcript
```

returns

```
Macro transcripting is turned ON.
```

## Related Topics

[Transcript Window](#)

## transcript file

This command sets or queries the current PrefMain(file) Tcl preference variable. You can use this command to clear a transcript in batch mode or to limit the size of a transcript file. It offers an alternative to setting the PrefMain(file) Tcl preference variable through the GUI.

### Syntax

**transcript file [<filename>]**

### Arguments

- <filename>

(optional) Specifies a name for the transcript file. Wildcard characters are allowed, and “stdout” or “stderr” are valid file names. If you specify a new file, the existing transcript file is closed and a new transcript file opened. If you specify an empty string (“”), the existing file is closed and no new file is opened. If you don’t specify this argument, the current filename is returned.

---

#### **Note**

 You can prevent overwriting older transcript files by including a pound sign (#) in <filename> when <filename> is a repeated string. The simulator replaces the pound character (#) with the next available sequence number when saving a new transcript file.

---

### Examples

- Close the current transcript file and stops writing data to the file. This is a method for reducing the size of your transcript.

**transcript file ""**

- Close the current transcript file named *trans1.txt* and open a new transcript file, incrementing the file name by 1.

**transcript file trans#.txt**

Closes *trans1.txt* and opens *trans2.txt*.

- This series of commands results in the transcript containing only data from the second millisecond of the simulation. The first transcript file command closes the transcript so no data is being written to it. The second transcript file command opens a new transcript and records data from 1 ms to 2 ms.

```
transcript file ""  
run 1 ms  
transcript file transcript  
run 1 ms
```

### Related Topics

[Creating a Transcript File](#)

[Setting GUI Preferences](#)

[Transcript Window](#)

[transcript path](#)

[transcript sizelimit](#)

# transcript path

This command returns the full pathname to the current transcript file.

## Syntax

`transcript path`

## Arguments

None

## Related Topics

[Creating a Transcript File](#)

[Setting GUI Preferences](#)

[Transcript Window](#)

[transcript file](#)

# transcript sizelimit

This command sets or queries the current preference value for the transcript fileSizeLimit value. If the size limit is reached, the transcript file is saved and simulation continues.

## Syntax

```
transcript sizelimit [<size>]
```

## Arguments

- <size>

(optional) Specifies the size, in KB, of the transcript file where the default is 0 or unlimited. The actual file size may be larger by as much as one buffer size (usually about 4k), depending on the operating system default buffer size and the size of the messages written to the transcript.

---

### **Note**

 You can set the size of the transcript file with the \$PrefMain (fileSizeLimit) Tcl variable in the Preferences dialog. Refer to “[Setting GUI Preferences](#)” for more information.

---

## Related Topics

[Creating a Transcript File](#)

[Setting GUI Preferences](#)

[Transcript Window](#)

[transcript file](#)

## transcript wrapcolumn

This command defines the column width when wrapping output lines in the transcript file.

### Syntax

```
transcript wrapcolumn <integer>
```

### Arguments

- `<integer>`

An integer that defines the width, in characters, before forcing a line break. The default value is 30000.

### Description

This column is somewhat soft; the wrap will occur at the first white-space character after reaching the transcript wrapwscolumn value or at exactly the column width if no white-space is found.

# transcript wrapmode

This command controls wrapping of output lines in the transcript file.

## Syntax

`transcript wrapmode [0 | 1 | 2]`

## Arguments

- 0  
(default) Disables wrapping.
- 1  
Enables wrapping, based on the value of the transcript wrapcolumn command, which defaults to 30,000 characters.
- 2  
Enables wrapping and adds a continuation character (\) at the end of every wrapped line, except for the last.

## transcript wrapwscolumn

This variable defines the column width when wrapping output lines in the transcript file.

### Usage

```
transcript wrapwscolumn <integer>
```

### Arguments

- `<integer>`

An integer that specifies that the wrap will occur at the first white-space character after reaching the specified number of characters. If there is no white-space, the wrap will occur at the transcript wrapcolumn value. The default value is 27000.

## tssi2mti

This command is used to convert a vector file in TSSI Format into a sequence of force and run commands.

### Syntax

```
tssi2mti <signal_definition_file> [<sef_vector_file>]
```

### Description

The stimulus is written to the standard output.

The source code for tssi2mti is provided in the *examples* directory as:

```
<install_dir>/examples/tssi2mti/tssi2mti.c
```

### Arguments

- <signal\_definition\_file>  
(required) Specifies the name of the TSSI signal definition file describing the format and content of the vectors.
- <sef\_vector\_file>  
(optional) Specifies the name of the file containing vectors to be converted. If none is specified, standard input is used.

### Examples

- The command will produce a do file named *trigger.do* from the signal definition file *trigger.def* and the vector file *trigger.sef*.

```
tssi2mti trigger.def trigger.sef > trigger.do
```

- This example is the same as the previous one, but uses the standard input instead.

```
tssi2mti trigger.def < trigger.sef > trigger.do
```

### Related Topics

[run](#)

## ui\_VVMode

This command specifies behavior when encountering UI registration calls used by verification packages, such as AVM or OVM. Returns the current setting when specifies without an argument.

### Syntax

```
ui_VVMode [full | logclass | logobj | nolog | off ]
```

### Description

UI registration calls, Verilog system tasks specific to this product, are typically included in verification packages such as AVM and OVM so that key information about the packages is available when debugging the simulation. The UI registration calls include:

- `$ui_VVInstallInst()` — Defines a region in the context tree, which will appear in the Structure window.
- `$ui_VVInstallObj()` — Adds an object to the defined parent, which will appear in the Objects window when the parent instance is selected in the Structure window.
- `$ui_VVInstallPort()` — Adds a port that is an object that connects to another component, which will appear in the Objects window when the parent instance is selected in the Structure window.
- `$ui_VVSetFilter()` — Specifies which class properties should not be shown in the GUI.
- `$ui_VVSetAllow()` — Specifies which class properties should be retained that were filtered out with `$ui_VVSetFilter`.

### Arguments

- `full`  
(optional) Enables the context registration of the UI registration call and automatically logs both the class type and the registered object to the WLF file.
- `logclass`  
(optional) Enables the context registration of the UI registration call and automatically logs the class type of the registered object to the WLF file.
- `logobj`  
(optional) Enables the context registration of the UI registration call and automatically logs the registered object to the WLF file
- `nolog`  
(optional) Enables the context registration of the UI registration call, but does not automatically log the registration to the WLF file. (default)

- off

(optional) Disables context registration and automatic logging when encountering UI registration calls.

## unsetenv

This command deletes an environment variable. The deletion is not permanent – it is valid only for the current ModelSim session.

### Syntax

`unsetenv <varname>`

### Arguments

- `<varname>`

(required) The name of the environment variable you wish to delete.

### Related Topics

[setenv](#)

[printenv](#)

# vcd add

This command adds the specified objects to a VCD file.

## Syntax

```
vcd add [-dumports] [-file <filename>] [[-in] [-out] [-inout] | [-ports]] [-internal]  
[-r | -r -optcells] <object_name> ...
```

## Description

The allowed objects are Verilog nets and variables and VHDL signals of type bit, bit\_vector, std\_logic, and std\_logic\_vector (other types are silently ignored). The command works with mixed HDL designs.

All vcd add commands must be executed at the same simulation time. The specified objects are added to the VCD header and their subsequent value changes are recorded in the specified VCD file. By default all port driver changes and internal variable changes are captured in the file. You can filter the output using arguments detailed below.

Related Verilog tasks: \$dumpvars, \$fdumpvars

Arguments to this command are order dependent. Please read the argument descriptions for more information.

## Arguments

- **-dumports**  
(optional) Specifies port driver changes to be added to an extended VCD file. When the [vcd dumpports](#) command cannot specify all port driver changes that will appear within the VCD file, multiple vcd add -dumports commands can be used to specify additional port driver changes.
- **-file <filename>**  
(optional) Specifies the name of the VCD file. This option should be used only when you have created multiple VCD files using the [vcd files](#) command.  
    <filename> — A .vcd file.
- **-in**  
(optional) Includes only port driver changes from ports of mode IN.
- **-out**  
(optional) Includes only port driver changes from ports of mode OUT.
- **-inout**  
(optional) Includes only port driver changes from ports of mode INOUT.

- `-ports`  
(optional) Includes only port driver changes. Excludes internal variable or signal changes.
- `-internal`  
(optional) Includes only internal variable or signal changes. Excludes port driver changes.
- `-r | -r -optcells`  
(optional) Specifies that signal and port selection occurs recursively into subregions. If omitted, included signals and ports are limited to the current region. When `-r` is used with `-optcells`, it allows Verilog optimized cell ports to be visible when using wildcards. By default, Verilog optimized cell ports are not selected even if they match the specified wildcard pattern.
- `<object_name> ...`  
(required) Specifies the Verilog or VHDL object or objects to add to the VCD file. Multiple objects may be specified by separating names with spaces. Wildcards are accepted. Must be specified as the final argument to the `vcd add` command.

## Related Topics

[vcd checkpoint](#)

[vcd comment](#)

[vcd dumpports](#)

[vcd dumpportsall](#)

[vcd dumpportsflush](#)

[vcd dumpportslimit](#)

[vcd dumpportsoff](#)

[vcd dumpportson](#)

[vcd file](#)

[vcd files](#)

[vcd flush](#)

[vcd limit](#)

[vcd off](#)

[vcd on](#)

[vcd2wlf](#)

[DumpportsCollapse](#)

[Value Change Dump \(VCD\) Files](#)

# vcd checkpoint

This command dumps the current values of all VCD variables to the specified VCD file. While simulating, only value changes are dumped. Related Verilog tasks: \$dumpall, \$fdumpall

## Syntax

`vcd checkpoint [<filename>]`

## Arguments

- `<filename>`

(optional) Specifies the name of the VCD file. If omitted the command is executed on the file designated by the [vcd file](#) command or *dump.vcd* if vcd file was not invoked.

## Related Topics

[vcd add](#)

[vcd comment](#)

[vcd dumpports](#)

[vcd dumpportsall](#)

[vcd dumpportsflush](#)

[vcd dumpportslimit](#)

[vcd dumpportsoff](#)

[vcd dumpportson](#)

[vcd file](#)

[vcd files](#)

[vcd flush](#)

[vcd limit](#)

[vcd off](#)

[vcd on](#)

[vcd2wlf](#)

[DumpportsCollapse](#)

[Value Change Dump \(VCD\) Files](#)

## vcd comment

This command inserts the specified comment in the specified VCD file. Arguments to this command are order dependent. Please read the argument descriptions for more information.

### Syntax

`vcd comment <comment string> [<filename>]`

### Arguments

- `<comment string>`  
(required) Comment to be included in the VCD file. Must be enclosed by double quotation marks or curly braces. Must be specified as the first argument to the vcd comment command.
- `<filename>`  
(optional) Specifies the name of the VCD file. If omitted the command is executed on the file designated by the [vcd file](#) command or *dump.vcd* if vcd file was not invoked.

### Related Topics

[vcd add](#)  
[vcd checkpoint](#)  
[vcd dumports](#)  
[vcd dumportsall](#)  
[vcd dumportsflush](#)  
[vcd dumportslimit](#)  
[vcd dumpportsoff](#)  
[vcd dumpportson](#)  
[vcd file](#)  
[vcd files](#)  
[vcd flush](#)  
[vcd limit](#)  
[vcd off](#)  
[vcd on](#)  
[vcd2wlf](#)  
[DumportsCollapse](#)  
[Value Change Dump \(VCD\) Files](#)

# vcd dumpports

This command creates a VCD file that includes port driver data.

## Syntax

```
vcd dumpports [-compress] [-direction] [-file <filename>] [-force_direction] [-in] [-out] [-inout]  
[-no_strength_range] [-unique] [-vcdstim] <object_name> ...
```

## Description

By default all port driver changes are captured in the file. You can filter the output using arguments detailed below. Related Verilog task: \$dumpports

Arguments to this command are order dependent. Please read the argument descriptions for more information.

## Arguments

- **-compress**  
(optional) Produces a compressed VCD file. ModelSim uses the gzip compression algorithm. It is not necessary to specify -compress if you specify a *.gz* extension with the -file <filename> argument
- **-direction**  
(optional) Includes driver direction data in the VCD file.
- **-file <filename>**  
(optional) Creates a VCD file. Defaults to the current working directory and the filename *dumpports.vcd*. Multiple filenames can be opened during a single simulation.  
    <filename> — Specifies a filename. When specified with a *.gz* extension, the file is compressed.
- **-force\_direction**  
(optional) Causes vcd dumpports to use the specified port direction (instead of driver location) to determine whether the value being dumped is input or output. This argument overrides the default use of the location of drivers on the net to determine port direction (this is because Verilog port direction is not enforced by the language or by ModelSim).
- **-in**  
(optional) Includes ports of mode IN.
- **-out**  
(optional) Includes ports of mode OUT.
- **-inout**  
(optional) Includes ports of mode INOUT.

- **-no\_strength\_range**  
(optional) Ignores strength ranges when resolving driver values. This argument is an extension to the IEEE 1364 specification. Refer to [Resolving Values](#) for additional information.
- **-unique**  
(optional) Generates unique VCD variable names for ports even if those ports are connected to the same collapsed net.
- **-vcdstim**  
(optional) Ensures that port name order in the VCD file matches the declaration order in the instance module or entity declaration. Refer to [Port Order Issues](#) for further information.
- **<object\_name> ...**  
(required) Specifies one or more HDL objects to add to the VCD file. You can specify multiple objects by separating names with spaces. Wildcards are accepted. Must be specified as the final argument to the vcd dumports command.

## Examples

- Create a VCD file named *counter.vcd* of all IN ports in the region */test\_design/dut/*.  

```
vcd dumports -in -file counter.vcd /test_design/dut/*
```
- These two commands resimulate a design from a VCD file. Refer to [Simulating with Input Values from a VCD File](#) for further details.  

```
vcd dumports -file addern.vcd /testbench/uut/*
vsim -vcdstim addern.vcd addern -gn=8 -do "add wave /*; run 1000"
```
- This series of commands creates VCD files for the instances *proc* and *cache* and then resimulates the design using the VCD files in place of the instance source files. Refer to [Replacing Instances with Output Values from a VCD File](#) for more information.  

```
vcd dumports -vcdstim -file proc.vcd /top/p/*
vcd dumports -vcdstim -file cache.vcd /top/c/*
run 1000

vsim top -vcdstim /top/p=proc.vcd -vcdstim /top/c=cache.vcd
```

## Related Topics

- [vcd add](#)
- [vcd checkpoint](#)
- [vcd comment](#)
- [vcd dumportsall](#)
- [vcd dumportsflush](#)
- [vcd dumportslimit](#)

[vcd dumpportoff](#)

[vcd dumpportson](#)

[vcd file](#)

[vcd files](#)

[vcd flush](#)

[vcd limit](#)

[vcd off](#)

[vcd on](#)

[vcd2wlf](#)

[DumpportsCollapse](#)

[Value Change Dump \(VCD\) Files](#)

## vcd dumportsall

This command creates a checkpoint in the VCD file which shows the value of all selected ports at that time in the simulation, regardless of whether the port values have changed since the last timestep. Related Verilog task: \$dumportsall

### Syntax

vcd dumportsall [<filename>]

### Arguments

- <filename>  
(optional) Specifies the name of the VCD file. If omitted the command is executed on all open VCD files.

### Related Topics

[vcd add](#)  
[vcd checkpoint](#)  
[vcd comment](#)  
[vcd dumports](#)  
[vcd dumportsflush](#)  
[vcd dumportslimit](#)  
[vcd dumpportsoff](#)  
[vcd dumpportson](#)  
[vcd file](#)  
[vcd files](#)  
[vcd flush](#)  
[vcd limit](#)  
[vcd off](#)  
[vcd on](#)  
[vcd2wlf](#)  
[DumportsCollapse](#)  
[Value Change Dump \(VCD\) Files](#)

# vcd dumpportsflush

This command flushes the contents of the VCD file buffer to the specified VCD file. Related Verilog task: \$dumpportsflush

## Syntax

`vcd dumpportsflush [<filename>]`

## Arguments

- `<filename>`

(optional) Specifies the name of the VCD file. If omitted the command is executed on all open VCD files.

## Related Topics

[vcd add](#)

[vcd checkpoint](#)

[vcd comment](#)

[vcd dumpports](#)

[vcd dumpportsall](#)

[vcd dumpportslimit](#)

[vcd dumpportsoff](#)

[vcd dumpportson](#)

[vcd file](#)

[vcd files](#)

[vcd flush](#)

[vcd limit](#)

[vcd off](#)

[vcd on](#)

[vcd2wlf](#)

[DumpportsCollapse](#)

[Value Change Dump \(VCD\) Files](#)

## vcd dumpportslimit

This command specifies the maximum size of the VCD file (by default, limited to available disk space). When the size of the file exceeds the limit, a comment is appended to the file and VCD dumping is disabled.

### Syntax

`vcd dumpportslimit <dumplimit> [<filename>]`

### Description

Related Verilog task: `$dumpportslimit`

Arguments to this command are order dependent. Please read the argument descriptions for more information.

### Arguments

- `<dumplimit>`  
(required) Specifies the maximum VCD file size in bytes. Must be specified as the first argument to the `vcd dumpportslimit` command.
- `<filename>`  
(optional) Specifies the name of the VCD file. If omitted the command is executed on all open VCD files.

### Related Topics

[vcd add](#)  
[vcd checkpoint](#)  
[vcd comment](#)  
[vcd dumpports](#)  
[vcd dumpportsall](#)  
[vcd dumpportsflush](#)  
[vcd dumpportsoff](#)  
[vcd dumpportson](#)  
[vcd file](#)  
[vcd files](#)  
[vcd flush](#)  
[vcd limit](#)  
[vcd off](#)  
[vcd on](#)

[vcd2wlf](#)

[DumportsCollapse](#)

[Value Change Dump \(VCD\) Files](#)

## vcd dumpportoff

This command turns off VCD dumping and records all dumped port values as x.

### Syntax

vcd dumpportoff [<filename>]

### Description

Related Verilog task: \$dumpportoff

### Arguments

- <filename>  
(optional) Specifies the name of the VCD file. If omitted the command is executed on all open VCD files.

### Related Topics

[vcd add](#)

[vcd checkpoint](#)

[vcd comment](#)

[vcd dumpports](#)

[vcd dumpportsall](#)

[vcd dumpportsflush](#)

[vcd dumpportslimit](#)

[vcd dumpportson](#)

[vcd file](#)

[vcd files](#)

[vcd flush](#)

[vcd limit](#)

[vcd off](#)

[vcd on](#)

[vcd2wlf](#)

[DumpportsCollapse](#)

[Value Change Dump \(VCD\) Files](#)

# vcd dumpportson

This command turns on VCD dumping and records the current values of all selected ports. This command is typically used to resume dumping after invoking vcd dumpportsoff. Related Verilog task: \$dumpportson

## Syntax

`vcd dumpportson [<filename>]`

## Arguments

- `<filename>`  
(optional) Specifies the name of the VCD file. If omitted the command is executed on all open VCD files.

## Related Topics

[vcd add](#)  
[vcd checkpoint](#)  
[vcd comment](#)  
[vcd dumpports](#)  
[vcd dumpportsall](#)  
[vcd dumpportsflush](#)  
[vcd dumpportslimit](#)  
[vcd dumpportsoff](#)  
[vcd file](#)  
[vcd files](#)  
[vcd flush](#)  
[vcd limit](#)  
[vcd off](#)  
[vcd on](#)  
[vcd2wlf](#)  
[DumpportsCollapse](#)  
[Value Change Dump \(VCD\) Files](#)

## vcd file

This command specifies the filename and state mapping for the VCD file created by a vcd add command. The vcd file command is optional. If used, it must be issued before any vcd add commands.

### Syntax

```
vcd file [-dumports] [-direction] [<filename>] [-map <mapping pairs>] [-no_strength_range]  
[-nomap] [-unique]
```

### Description

Related Verilog task: \$dumpfile

### Arguments

- **-dumports**  
(optional) Capture detailed port driver data for Verilog ports and VHDL std\_logic ports. This option works only on ports, and any subsequent vcd add command will accept only qualifying ports (silently ignoring all other specified objects).
- **-direction**  
(optional) Includes driver direction data in the VCD file.
- **<filename>**  
(optional) Specifies the name of the VCD file that is created where the default is *dump.vcd*.
- **-map <mapping pairs>**  
(optional) Overrides the default mappings. Affects only VHDL signals of type std\_logic.  
  
<mapping pairs> — Specified as a list of character pairs. The first character in a pair must be one of the std\_logic characters UX01ZWLH- and the second character is the character you wish to be recorded in the VCD file. The Tcl convention for command strings that include spaces is to enclose them in quotation marks (" "). For example, to map L and H to z:  
  
`vcd file -map "L z H z"`

- **-no\_strength\_range**  
(optional) Ignores strength ranges when resolving driver values. This argument is an extension to the IEEE 1364 specification. Refer to [Resolving Values](#) for additional information.
- **-nomap**  
(optional) Affects only VHDL signals of type std\_logic. It specifies that the values recorded in the VCD file shall use the std\_logic enumeration characters of UX01ZWLH-. This option results in a non-standard VCD file because VCD values are limited to the four state character set of x01z. By default, the std\_logic characters are mapped as follows.

<b>VHDL</b>	<b>VCD</b>	<b>VHDL</b>	<b>VCD</b>
U	x	W	x
X	x	L	0
0	0	H	1
1	1	-	x
Z	z		

- **-unique**

(optional) Generates unique VCD variable names for ports even if those ports are connected to the same collapsed net.

## Related Topics

- [vcd add](#)
- [vcd checkpoint](#)
- [vcd comment](#)
- [vcd dumpports](#)
- [vcd dumpportsall](#)
- [vcd dumpportsflush](#)
- [vcd dumpportslimit](#)
- [vcd dumpportsoff](#)
- [vcd dumpportson](#)
- [vcd files](#)
- [vcd flush](#)
- [vcd limit](#)
- [vcd off](#)
- [vcd on](#)
- [vcd2wlf](#)
- [DumpportsCollapse](#)
- [Value Change Dump \(VCD\) Files](#)

## vcd files

This command specifies filenames and state mapping for VCD files created by the vcd add command. The vcd files command is optional. If used, it must be issued before any vcd add commands. Related Verilog task: \$fdumpfile

### Syntax

```
vcd files [-compress] [-direction] <filename> [-map <mapping pairs>] [-no_strength_range]  
[-nomap] [-unique]
```

### Arguments

- **-compress**  
(optional) Produces a compressed VCD file. ModelSim uses the gzip compression algorithm. If you specify a .gz extension on the -file <filename> argument, ModelSim compresses the file even if you don't use the -compress argument.
- **-direction**  
(optional) Includes driver direction data in the VCD file.
- **<filename>**  
(required) Specifies the name of a VCD file to create. Multiple files can be opened during a single simulation; however, you can create only one file at a time. If you want to create multiple files, invoke vcd files multiple times.
- **-map <mapping pairs>**  
(optional) Overrides the default mappings. Affects only VHDL signals of type std\_logic.  
  
<mapping pairs> — Specified as a list of character pairs. The first character in a pair must be one of the std\_logic characters UX01ZWLH- and the second character is the character you wish to be recorded in the VCD file. The Tcl convention for command strings that include spaces is to enclose them in quotation marks (""). For example, to map L and H to z:  
  

```
vcd file -map "L z H z"
```
- **-no\_strength\_range**  
(optional) Ignores strength ranges when resolving driver values. This argument is an extension to the IEEE 1364 specification. Refer to “[Resolving Values](#)” for additional information.
- **-nomap**  
(optional) Affects only VHDL signals of type std\_logic. It specifies that the values recorded in the VCD file shall use the std\_logic enumeration characters of UX01ZWLH-. This option results in a non-standard VCD file because VCD values are limited to the four state character set of x01z. By default, the std\_logic characters are mapped as follows.

<b>VHDL</b>	<b>VCD</b>	<b>VHDL</b>	<b>VCD</b>
U	x	W	x
X	x	L	0
0	0	H	1
1	1	-	x
Z	z		

- **-unique**

(optional) Generates unique VCD variable names for ports even if those ports are connected to the same collapsed net.

## Examples

The following example shows how to "mask" outputs from a VCD file until a certain time after the start of the simulation. The example uses two vcd files commands and the **vcd on** and **vcd off** commands to accomplish this task.

```
vcd files in inout.vcd
vcd files output.vcd
vcd add -in -inout -file in inout.vcd /*
vcd add -out -file output.vcd /*
vcd off output.vcd
run 1us
vcd on output.vcd
run -all
```

## Related Topics

[vcd add](#)  
[vcd checkpoint](#)  
[vcd comment](#)  
[vcd dumpports](#)  
[vcd dumpportsall](#)  
[vcd dumpportsflush](#)  
[vcd dumpportslimit](#)  
[vcd dumpportsoff](#)  
[vcd dumpportson](#)  
[vcd file](#)  
[vcd flush](#)  
[vcd limit](#)

[vcd off](#)

[vcd on](#)

[vcd2wlf](#)

[DumpportsCollapse](#)

[Value Change Dump \(VCD\) Files](#)

# vcd flush

This command flushes the contents of the VCD file buffer to the specified VCD file. This command is useful if you want to create a complete VCD file without ending your current simulation. Related Verilog tasks: \$dumpflush, \$fdumpflush

## Syntax

`vcd flush [<filename>]`

## Arguments

- `<filename>`  
(optional) Specifies the name of the VCD file. If omitted, the command is executed on the file designated by the [vcd file](#) command or *dump.vcd* if vcd file was not invoked.

## Related Topics

[vcd add](#)  
[vcd checkpoint](#)  
[vcd comment](#)  
[vcd dumports](#)  
[vcd dumportsall](#)  
[vcd dumportsflush](#)  
[vcd dumportslimit](#)  
[vcd dumpportsoff](#)  
[vcd dumpportson](#)  
[vcd file](#)  
[vcd files](#)  
[vcd limit](#)  
[vcd off](#)  
[vcd on](#)  
[vcd2wlf](#)  
[DumportsCollapse](#)  
[Value Change Dump \(VCD\) Files](#)

## vcd limit

This command specifies the maximum size of a VCD file (by default, limited to available disk space).

### Syntax

`vcd limit <filesize> [<filename>]`

### Description

When the size of the file exceeds the limit, a comment is appended to the file and VCD dumping is disabled.

Related Verilog tasks: `$dumplimit`, `$fdumplimit`

Arguments to this command are order dependent. Please read the argument descriptions for more information.

### Arguments

- `<filesize>`

(Required) Specifies the maximum VCD file size, in bytes. The numerical value of `<filesize>` can only be a whole number. Must be specified as the first argument to the `vcd limit` command.

You can specify a unit of Kb, Mb, or Gb with the numerical value (units are case insensitive). Do not insert a space between the numerical value and the unit (for example, 400Mb, not 400 Mb).

- `<filename>`

(Optional) Specifies the name of the VCD file. If omitted, the command is executed on the file designated by the `vcd file` command or `dump.vcd` if `vcd` file was not invoked.

### Examples

- Specify a maximum VCD file size of 6 gigabytes and a VCD file named `my_vcd_file.vcd`.

**`vcd limit 6gb my_vcd_file.vcd`**

### Related Topics

[vcd add](#)  
[vcd checkpoint](#)  
[vcd comment](#)  
[vcd dumports](#)  
[vcd dumportsall](#)  
[vcd dumportsflush](#)

[vcd dumpportslimit](#)  
[vcd dumpportoff](#)  
[vcd dumpportson](#)  
[vcd file](#)  
[vcd files](#)  
[vcd flush](#)  
[vcd off](#)  
[vcd on](#)  
[vcd2wlf](#)  
[DumpportsCollapse](#)  
[Value Change Dump \(VCD\) Files](#)

## vcd off

This command turns off VCD dumping to the specified file and records all VCD variable values as x. Related Verilog tasks: \$dumpoff, \$fdumpoff

### Syntax

vcd off [<filename>]

### Arguments

- <filename>

(optional) Specifies the name of the VCD file. If omitted, the command is executed on the file designated by the [vcd file](#) command or *dump.vcd* if vcd file was not invoked.

### Related Topics

[vcd add](#)  
[vcd checkpoint](#)  
[vcd comment](#)  
[vcd dumports](#)  
[vcd dumportsall](#)  
[vcd dumportsflush](#)  
[vcd dumportslimit](#)  
[vcd dumpportsoff](#)  
[vcd dumpportson](#)  
[vcd file](#)  
[vcd files](#)  
[vcd flush](#)  
[vcd limit](#)  
[vcd on](#)  
[vcd2wlf](#)  
[DumportsCollapse](#)  
[Value Change Dump \(VCD\) Files](#)

# vcd on

This command turns on VCD dumping to the specified file and records the current values of all VCD variables.

## Syntax

vcd on [<filename>]

## Description

By default, vcd on is automatically performed at the end of the simulation time that the [vcd add](#) command performed.

Related Verilog tasks: \$dumpon, \$fdumpon

## Arguments

- <filename>

(optional) Specifies the name of the VCD file. If omitted the command is executed on the file designated by the [vcd file](#) command or *dump.vcd* if vcd file was not invoked.

## Related Topics

[vcd add](#)  
[vcd checkpoint](#)  
[vcd comment](#)  
[vcd dumports](#)  
[vcd dumportsall](#)  
[vcd dumportsflush](#)  
[vcd dumportslimit](#)  
[vcd dumpportsoff](#)  
[vcd dumpportson](#)  
[vcd file](#)  
[vcd files](#)  
[vcd flush](#)  
[vcd limit](#)  
[vcd off](#)  
[vcd2wlf](#)  
[DumportsCollapse](#)  
[Value Change Dump \(VCD\) Files](#)

## vcd2wlf

This command is a utility that translates a VCD (Value Change Dump) file into a WLF file that you can display in ModelSim using the `vsim -view` argument. This command only works on VCD files containing positive time values.

### Syntax

```
vcd2wlf [-splitio] [-splitio_in_ext <extension>] [-splitio_out_ext <extension>] [-nocase]
{<vcd filename> | - } <wlf filename>
```

### Description

The `vcd2wlf` command functions as simple one-pass converter. If you are defining a bus in a VCD file, you must specify all bus bits before the next `$scope` or `$upscope` statement appears in the file. The best way to ensure that bits get converted together as a bus is to declare them on consecutive lines.

For example:

```
Line 21 : $var wire 1 $ in [2] $end
Line 22 : $var wire 1 $u in [1] $end
Line 23 : $var wire 1 # in [0] $end
```

Arguments to this command are order dependent. Please read the argument descriptions for more information.

### Arguments

- `-splitio`  
(optional) Specifies that extended VCD port values are to be split into their corresponding input and output components by creating two signals instead of just one in the resulting `.wlf` file. By default the new input-component signal keeps the same name as the original port name while the output-component name is the original name with `"_o"` appended to it.
- `-splitio_in_ext <extension>`  
(optional) Adds an extension to input-component signal names created by using `-splitio`.  
`<extension>` — Specifies a string.
- `-splitio_out_ext <extension>`  
(optional) Adds an extension to output-component signal names created by using `-splitio`.  
`<extension>` — Specifies a string.
- `-nocase`  
(optional) Converts all alphabetic identifiers to lowercase.

- {<vcd filename> | - }  
(required) Specifies the name of the VCD file, or standard input (-), you want to translate into a WLF file. Must be specified immediately preceding the <wlf filename> argument to the vcd2wlf command.
- <wlf filename>  
(required) Specifies the name of the output WLF file. Must be specified as the final argument to the vcd2wlf command.

## Examples

- Concatenate *my.vcd* file and pipe standard input to vcd2wlf and save output to *my.wlf* file.

```
cat my.vcd | vcd2wlf - my.wlf
```

- Redirect input from the file *my.vcd* file to vcd2wlf and save the output to *my.wlf* file.

```
vcd2wlf - my.wlf <my.vcd
```

## Related Topics

[vcd add](#)

[vcd checkpoint](#)

[vcd comment](#)

[vcd dumpports](#)

[vcd dumpportsall](#)

[vcd dumpportsflush](#)

[vcd dumpportslimit](#)

[vcd dumpportsoff](#)

[vcd dumpportson](#)

[vcd file](#)

[vcd files](#)

[vcd flush](#)

[vcd limit](#)

[vcd off](#)

[vcd on](#)

[DumpportsCollapse](#)

[Value Change Dump \(VCD\) Files](#)

## vcom

The vcom command compiles VHDL source code into a specified working library (or to the work library by default).

### Syntax

```
vcom [options] <filename> [<filename> ...]

[options]:
[-87 | -93 | -2002 | -2008]
[-addpragmaprefix <prefix>] [-allowProtectedBeforeBody] [-amsstd | -noamsstd]
[-bindAtCompile] [-bindAtLoad]
[-check_synthesis] [-nocreatelib]
[-defercheck] [-deferSubpgmCheck | -noDeferSubpgmCheck]
[-error <msg_number>[,<msg_number>,...]] [-explicit]
[(-F | -file | -f) <filename>] [-fatal <msg_number>[,<msg_number>,...]]
[-force_refresh <primary> [<secondary>]]
[-fsmimplicittrans | -nofsmimplicittrans] [-fsmresettrans | -nofsmresettrans] [-fsmsingle | -nofsmsingle]
[-fsmverbose [b | t | w]]
[-gen_xml <design_unit> <filename>]
[-ignoredefaultbinding] [-ignorepragmaprefix <prefix>] [ignoreStandardRealVector]
[-ignorevitalerrors] [-initoutcompositeparam | -noinitoutcompositeparam]
[-just abcep]
[-logfile <filename> | -l <filename>] [-line <number>] [-lint] [-lower]
[-lrmconfigvis]
[-mixedsvvh [b | l | r ][i]] [-modelsimini <path/modelsim.ini>]
[-msglimit [all, | none,] [-+]<msg_number>[,-+<msg_number>,...]]
[-msglimitcount <limit_value> -msglimit [all,none,] [-+]<msgNumber>[,-+<msgNumber>...]]
[-no1164] [-noaccel <package_name>] [-nocasestaticerror] [-nocheck]
[-nocreatelib] [-nodbgsym] [-nofprangecheck]
[-noFunctionInline] [-noindexcheck] [-nologo] [-nonstddriverinit]
[-noothersstaticerror] [-note <msg_number>[,<msg_number>,...]] [-novitalcheck] [-nowarn <category_number>]
[-oldconfigvis] [-optionset <optionset_name>]
[-outf <filename>]
[-pedanticerrors] [-performdefaultbinding] [-preserve] [-[w]prof=<filename>]
[-proftick=<integer>]
```

```
[ -quiet ]  
[ -rangecheck | -norangecheck ] [ -refresh <primary> [ <secondary> ] ]  
[ -s ] [ -separateConfigLibrary ] [ -showsubprograms | -noshowsubprograms ] [ -skip abcep ] [ -  
  skipsynthoffregion ] [ -smartdbgsym ] [ -source ]  
[ -stats [= [+ | -] <feature> [ , [+ | -] <mode> ] ] ]  
[ -suppress <msg_number> [ , <msg_number> , ... ] ]  
[ -version ] [ -vitalmemorycheck ] [ -vmake ]  
[ -warning <msg_number> [ , <msg_number> , ... ] ] [ -warning error ] [ -work <library_name> ]
```

## Description

You can invoke vcom either from within ModelSim or from the command prompt of your operating system. You can invoke this command during simulation.

Compiled libraries are dependent on the major version of ModelSim. When moving between major versions, you must refresh compiled libraries using the -refresh argument to vcom. This is not required for minor versions (letter releases).

All arguments to the vcom command are case-sensitive. For example, -WORK and -work are not equivalent.

This command provides additional information with the -help or -h switch.

## Arguments

- `-87 | -93 | -2002 | -2008`  
(optional) Specifies which LRM-specific compiler to use. You can also control this behavior with the VHDL93 variable in the *modelsim.ini* file. Refer to “[Differences Between Versions of VHDL](#)” for more information.
  - 87 — Enables support for VHDL 1076-1987.
  - 93 — Enables support for VHDL 1076-1993.
  - 2002 — Enables support for VHDL 1076-2002. (default)
  - 2008 — Enables support for VHDL 1076-2008.

- `-addpragmaprefix <prefix>`  
(optional) Enables recognition of pragmas with a user specified prefix. If this argument is not specified, pragmas are treated as comments.

All regular synthesis pragmas are honored.

`<prefix>` — Specifies a user defined string where the default is no sting, indicated by quotation marks.

You may also set this with the `AddPragmaPrefix` variable in the vcom section of the *modelsim.ini* file.

- **-allowProtectedBeforeBody**  
(optional) Allows a variable of a protected type to be created prior to declaring the body.
- **-amsstd | -noamsstd**  
(optional) Specifies whether vcom adds the declaration of REAL\_VECTOR to the STANDARD package. This is useful for designers using VHDL-AMS to test digital parts of their model.
  - amsstd** — REAL\_VECTOR is included in STANDARD.
  - noamsstd** — REAL\_VECTOR is not included in STANDARD (default).

You can also control this with the [AmsStandard](#) variable or the [MGC\\_AMS\\_HOME](#) environment variable.

- **-bindAtCompile**  
(optional) Forces ModelSim to perform default binding at compile time rather than at load time. Refer to “[Default Binding](#)” for more information. You can change the permanent default by editing the [BindAtCompile](#) variable in the *modelsim.ini*.
- **-bindAtLoad**  
(optional) Forces ModelSim to perform default binding at load time rather than at compile time. (Default)
- **-check\_synthesis**  
(optional) Turns on limited synthesis rule compliance checking. Specifically, it checks to see that signals read by a process are in the sensitivity list. The checks understand only combinational logic, not clocked logic. Edit the [CheckSynthesis](#) variable in the *modelsim.ini* file to set a permanent default.
- **-defercheck**  
(optional) Defers index checks until run time.
- **-deferSubpgmCheck**  
(optional) Forces the compiler to report array indexing and length errors as warnings (instead of as errors) when encountered within subprograms. Subprograms with indexing and length errors that are invoked during simulation cause the simulator to report errors, which can potentially slow down simulation because of additional checking.
- **-error <msg\_number>[,<msg\_number>,...]**  
(optional) Changes the severity level of the specified message(s) to “error.” Edit the [error](#) variable in the *modelsim.ini* file to set a permanent default. Refer to “[Message Severity Level](#)” for more information.
  - <msg\_number>** — A number identifying the message. Multiple message numbers are specified as a comma separated list.

- **-explicit**

(optional) Directs the compiler to resolve ambiguous function overloading by favoring the explicit function definition over the implicit function definition. Strictly speaking, this behavior does not match the VHDL standard. However, the majority of EDA tools choose explicit operators over implicit operators. Using this switch makes ModelSim compatible with common industry practice.

- **(-F | -file | -f) <filename>**

(optional) -f, -file and -F: each specifies an argument file with more command-line arguments, allowing complex argument strings to be reused without retyping. Nesting of -F, -f and -file commands is allowed. Allows gzipped input files.

With -F only: relative file names and paths within the arguments file <filename> are prefixed with the path of the arguments file when lookup with relative path fails. Refer to the section “[Argument Files](#)” on page 27 for more information.

- **-fatal <msg\_number>[,<msg\_number>,...]**

(optional) Changes the severity level of the specified message(s) to "fatal." Edit the **fatal** variable in the *modelsim.ini* file to set a permanent default. Refer to “[Message Severity Level](#)” for more information.

<msg\_number> — A number identifying the message. Multiple message numbers are specified as a comma separated list.

- **-force\_refresh <primary> [<secondary>]**

(optional) Forces the refresh of all specified design units. By default, the work library is updated; use -work <library\_name>, in conjunction with -force\_refresh, to update a different library (for example, vcom -work <your\_lib\_name> -force\_refresh).

<primary> [<secondary>] — Specifies the entity, package, configuration, or module to be deleted.

- If <primary> is an entity — only that entity, no related architectures, is refreshed.
- If <primary> is a package — the only legal value of <secondary> is “body”, and only the package is refreshed.
- If you specify both <primary> and <secondary> — Only the <secondary> architecture is updated, not the entity.

When the compiler refreshes a design unit, it checks each dependency to ensure its source has not been changed and recompiled. Sometimes the dependency checking algorithm changes from release to release. This can lead to false errors during the integrity checks performed by the -refresh argument. An example of such a message follows:

```
** Error: (vsim-13) Recompile /u/test/dware/
dware_6le_beta.dwpackages because /home/users/questasim/...
synopsys.attributes has changed.
```

---

The **-force\_refresh** argument forces the refresh of the design unit, overriding any dependency checking errors encountered by the **-refresh** argument.

A more conservative approach to working around **-refresh** dependency checks is to recompile the source code, if it is available.

You cannot specify the **<filename>** argument when specifying this argument.

- **-fsmimplicittrans** | **-nofsmimplicittrans**

(optional) Toggles recognition of implied same state transitions, which is off by default (**-nofsmimplicittrans**).

- **-fsmresettrans** | **-nofsmresettrans**

(optional) Toggles recognition of synchronous or asynchronous reset transitions, and is on by default (**-fsmresettrans**).

- **-fsmsingle** | **-nofsmsingle**

(optional) Toggles the recognition of VHDL FSMs where the current state variable of type `std_logic`, `bit`, `boolean`, or single-bit `std_logic_vector`/`bit_vector` and Verilog single-bit FSMs.

- **-fsmverbose** [**b** | **t** | **w**]

(optional) Provides information about FSMs detected, including state reachability analysis.

**b** — displays only basic information.

**t** — displays a transition table in addition to the basic information.

**w** — displays any warning messages in addition to the basic information.

When you do not specify an argument, this switch reports all information similar to:

```
# ** Note: (vcom-1947)  FSM RECOGNITION INFO
#      Fsm detected in : ../fpu/rtl/vhdl/serial_mul.vhd
#      Current State Variable : s_state : ../fpu/rtl/vhdl/serial_mul.vhd(76)
#      Clock : clk_i
#      Reset States are: { waiting , busy }
#      State Set is : { busy , waiting }
#      Transition table is
#
#      -----
#      busy      =>  waiting  Line : (114 => 114)
#      busy      =>  busy      Line : (111 => 111)
#      waiting   =>  waiting  Line : (120 => 120) (114 => 114)
#      waiting   =>  busy      Line : (111 => 111)
#      -----
```

When you do not specify this switch, you will receive a message similar to:

```
# ** Note: (vcom-143) Detected '1' FSM/s in design unit
'serial_mul.rtl'.
```

- `-gen_xml <design_unit> <filename>`

(optional) Produces an XML-tagged file containing the interface definition of the specified entity.

`<design_unit>` — The name of an entity or design unit in the Work library. Wildcards and multiple design unit names are not allowed.

`<filename>` — A user-specified name for the file.

For example:

This option requires a two-step process where you must:

- 1) compile `<filename>` into a library with vcom (without `-gen_xml`) then
- 2) execute vcom with the `-gen_xml` switch.

```
vlib work
vcom counter.vhd
vcom -gen_xml counter counter.xml
```

- `-ignoredefaultbinding`

(optional) Instructs the compiler not to generate a default binding during compilation. You must explicitly bind all components in the design through either configuration specifications or configurations. If an explicit binding is not fully specified, defaults for the architecture, port maps, and generic maps will be used as needed. Refer to [Default Binding](#) for more information. Edit the [RequireConfigForAllDefaultBinding](#) *modelsim.ini* variable to set a permanent default.

- `-ignorepragmaprefix <prefix>`

(optional) Directs vcom to ignore pragmas with the specified prefixname. All affected pragmas will be treated as regular comments. Edit the [IgnorePragmaPrefix](#) *modelsim.ini* variable to set a permanent default.

`<prefix>` — Specifies a user defined string.

- `ignoreStandardRealVector`

(optional) Instructs ModelSim to ignore the REAL\_VECTOR declaration in package STANDARD when compiling with vcom -2008. Edit the [ignoreStandardRealVector](#) *modelsim.ini* variable to set a permanent default. For more information refer to the REAL\_VECTOR section in [Help > Technotes > vhdl2008migration](#).

- `-ignorevitalerrors`

(optional) Directs the compiler to ignore VITAL compliance errors. The compiler still reports that VITAL errors exist, but it will not stop the compilation. You should exercise caution in using this switch; as part of accelerating VITAL packages, we assume that compliance checking has passed.

- `-initoutcompositeparam`

(optional) Causes initialization of subprogram parameters for array and record types when the subprogram is executed in designs compiled with LRM 1076-2002 and earlier. This

argument forces the output parameters to their default initial (“left”) values when entering a subprogram. By default, `-initoutcompositeparam` is enabled for designs compiled with vcom -2008 and later. You can also enable this by setting the [InitInOutCompositeParam](#) variable to 1 in the *modelsim.ini* file.

- `-noinitoutcompositeparam`

(optional) Disables initialization of subprogram parameters for array and record types when the subprogram is executed in designs compiled with LRM 1076-2002 and earlier. By default, designs compiled with LRM 1076-2008 and later do not initialize subprogram parameters for array and record types when the subprogram is executed. You can also disable initialization of subprogram parameters for array and record types by setting the [InitInOutCompositeParam](#) variable to 2 in the *modelsim.ini* file.

- `-just abcep`

(optional) Directs the compiler to include only the following:

- `a` — architectures
- `b` — bodies
- `c` — configurations
- `e` — entities
- `p` — packages

Any combination in any order can be used, but you must specify at least one choice if you use this switch.

- `-logfile <filename> | -l <filename>`

(optional) Generates a log file of the compile.

`-logfile <filename>` — Saves transcript data to `<filename>`. Can be abbreviated to `-l <filename>`. Overrides the default transcript file creation set with the [TranscriptFile](#) or [BatchTranscriptFile](#) *modelsim.ini* variables. You can also specify “`stdout`” or “`stderr`” for `<filename>`.

- `-line <number>`

(optional) Starts the compiler on the specified line in the VHDL source file. By default, the compiler starts at the beginning of the file.

`<number>` —

- `-lint`

(optional) Performs additional static checks on case statement rules and enables warning messages for the following situations:

- The result of the built-in concatenation operator (“`&`”) is the actual for a subprogram formal parameter of an unconstrained array type.
- If you specify the `-BindAtCompile` switch with vcom, the entity to which a component instantiation is bound has a port that is not on the component, and for which there is no error otherwise.

- A direct recursive subprogram call.
- In cases involving class SIGNAL formal parameters, as described in the IEEE Standard VHDL Language Reference Manual entitled "Signal parameters". This check only applies to designs compiled using -87. If you compile using -93, it would be flagged as a warning or error, even without the -lint argument. Can also be enabled using the [Show\\_Lint](#) variable in the *modelsim.ini* file.
- -lower
  - (optional) Forces vcom to convert uppercase letters in object identifiers to lowercase. You can also enable this by setting the [PreserveCase](#) variable to 0 in the *modelsim.ini* file.
- -lrmconfigvis
  - (optional, default) Forces vcom to use visibility rules that comply with the Language Reference Manual when processing VHDL configurations. Refer to vcom [-oldconfigvis](#) or the [oldVHDLConfigurationVisibility](#) variable in the *modelsim.ini* file for more information.
- -mixedsvvh [b | l | r ][i]
  - (optional) Facilitates using VHDL packages at the SystemVerilog-VHDL boundary of a mixed-language design. When you compile a VHDL package with -mixedsvvh, the package can be included in a SystemVerilog design as if it were defined in SystemVerilog itself.

Executing -mixedsvvh without arguments compiles VHDL vectors in the following ways:

  - VHDL bit\_vectors are treated as SystemVerilog bit vectors.
  - VHDL std\_logic\_vectors, std\_ulogic\_vectors, and vl\_logic\_vectors are treated as SystemVerilog logic vectors.

b — treats all scalars and vectors in the package as SystemVerilog bit type  
l — treats all scalars and vectors in the package as SystemVerilog logic type  
r — treats all scalars and vectors in the package as SystemVerilog reg type  
i — ignores the range specified with VHDL integer types. Can be specified together with b, l, or r, spaces are not allowed between arguments.
- -modelsimini <path/modelsim.ini>
  - (optional) Loads an alternate initialization file that replaces the current initialization file. Overrides the file path specified in the MODELSIM environment variable.

<path/modelsim.ini> — Specifies either an absolute or relative path to the initialization file, including the file itself. On Windows systems the path separator should be a forward slash (/).
- -msglimit [all, | none,] [-+]<msg\_number>[,-+<msg\_number>,...]
  - (optional) Limits the number of iterations of the specified message(s) to the default message limit count of five, then suppresses all new instances. Refer to [Suppression of Warning Messages](#) for more information.

all — Limits all messages to the default count except specified msgNumber(s).

none — Limits only the specified msgNumber(s) to the default count.

[-|+] — The ‘-’ argument is used *only* with “all” to specify exclusion of specified msgNumber(s). The ‘+’ argument is used *only* with “none” to specify inclusion of specific message IDs. If neither is used, the command works the same way.

**Note**

 The ‘-’ argument can *only* be used with the “all” argument and the ‘+’ argument can *only* be used with the “none” argument. Otherwise incorrect results may appear.

<msg\_number>[,<msg\_number>,...] — Specifies the message number(s) to limit to five iterations. Multiple messages are specified as a comma-separated list.

For example, the following limits all messages to the default count except *msgNumber1* and *msgNumber2*.

```
vsim -msglimit all, <msgNumber1>, <msgNumber2>
```

While the following, limits only *msgNumber1* and *msgNumber2* to the default count.

```
vsim -msglimit none, <msgNumber1>, <msgNumber2>
```

- **-msglimitcount <limit\_value> -msglimit [all|none,] [-|+]<msgNumber>[,-|+]<msgNumber>...]**  
(optional) Limits the reporting of listed messages to user-defined limit\_value. Overrides the **MsgLimitCount** variable in the modelsim.ini file.
- **-no1164**  
(optional) Causes the source files to be compiled without taking advantage of the built-in version of the IEEE std\_logic\_1164 package. This will typically result in longer simulation times for VHDL programs that use variables and signals of type std\_logic.
- **-noaccel <package\_name>**  
(optional) Turns off acceleration of the specified package in the source code using that package.  
    <package\_name> — A VHDL package name.
- **-nocasestaticerror**  
(optional) Suppresses case statement static warnings. VHDL standards require that case statement alternative choices be static at compile time. However, some expressions which are globally static are allowed. This switch prevents the compiler from warning on such expressions. If the -pedanticerrors switch is specified, this switch is ignored.
- **-nocheck**  
(optional) Disables index and range checks. You can disable these individually using the -noindexcheck and -norangecheck arguments, respectively.

- **-nocreatelib**  
(optional) Stops automatic creation of missing work libraries and reverts back to 10.3x and earlier version behavior. Overrides the [CreateLib](#) *modelsim.ini* variable.
- **-nodbgsym**  
(optional) Disables the generation of the symbols debugging database in the compiled library.

The symbols debugging database is the *.dbs* file in the compiled library that provides information to the GUI allowing you to view detailed information about design objects at the source level. Two major GUI features that use this database include source window annotation and textual dataflow.

You should only specify this switch if you know that anyone using the library will not require this information for design analysis purposes.

- **-noDefersubpgmCheck**  
(optional) Causes range and length violations detected within subprograms to be reported as errors (instead of as warnings). As an alternative to using this argument, you can set the [NoDefersubpgmCheck](#) variable in the *modelsim.ini* file to a value of 1.
- **-nofrangecheck**  
(optional) Disables range checks on floating type values only.
- **-noFunctionInline**  
(optional) Turns off VHDL subprogram inlining for design units using a local copy of a VHDL package. This may be needed in case the local package has the same name as an MTI supplied package.
- **-noindexcheck**  
(optional) Disables checking on indexing expressions to determine whether indexes are within declared array bounds.
- **-nologo**  
(optional) Disables display of the startup banner.
- **-nonstddriverinit**  
(optional) Forces ModelSim to match pre-5.7c behavior in initializing drivers in a particular case. Prior to 5.7c, VHDL ports of mode out or inout could have incorrectly initialized drivers if the port did not have an explicit initialization value and the actual signal connected to the port had explicit initial values. Depending on a number of factors, ModelSim could incorrectly use the actual signal's initial value when initializing lower level drivers. Note that the argument does not cause all lower-level drivers to use the actual signal's initial value. It does this only in the specific cases where older versions used the actual signal's initial value.

- **-noothersstaticerror**  
(optional) Disables warnings that result from array aggregates with multiple choices having "others" clauses that are not locally static. If **-pedanticerrors** is specified, this switch is ignored.
- **-norangecheck**  
(optional) Disables run time range checking. In some designs, this results in a 2X speed increase. Range checking is enabled by default or, once disabled, can be enabled using **-rangecheck**. If you run a simulation with range checking disabled, any scalar values that are out of range are indicated by showing the value in the following format: ?(N) where N is the current value.
- **-note <msg\_number> [<msg\_number>, ...]**  
(optional) Changes the severity level of the specified message(s) to "note. Edit the **note** variable in the *modelsim.ini* file to set a permanent default. Refer to "[Message Severity Level](#)" for more information.  
  
    <msg\_number> — A number identifying the message. Multiple message numbers are specified as a comma separated list.
- **-novitalcheck**  
(optional) Disables Vital level 1 and Vital level 0 checks defined in section 4 of the Vital-95 Spec (IEEE Std 1076.4-1995).
- **-nowarn <category\_number>**  
(optional) Selectively disables a category of warning messages. Warnings may be disabled for all compiles via the Main window **Compile > Compile Options** menu command or the *modelsim.ini* file (Refer to [modelsim.ini Variables](#)).  
  
    <category\_number> — Specifies one or more numbers corresponding to the categories in [Table 2-6](#). Multiple message categories are specified as a comma separated list.

**Table 2-6. Warning Message Categories for vcom -nowarn**

Category number	Description
1	unbound component
2	process without a wait statement
3	null range
4	no space in time literal
5	multiple drivers on unresolved signal
6	VITAL compliance checks ("VitalChecks" also works)
7	VITAL optimization messages

**Table 2-6. Warning Message Categories for vcom -nowarn (cont.)**

Category number	Description
8	lint checks
9	signal value dependency at elaboration
10	VHDL-1993 constructs in VHDL-1987 code
14	locally static error deferred until simulation run

Multiple message categories are specified as a comma separated list.

- **-oldconfigvis**  
(optional) Forces vcom to process visibility of VHDL component configurations consistent with prior releases. Default behavior is to comply with Language Reference Manual visibility rules. Refer to vcom -lrmconfigvis or the *modelsim.ini* variable [OldVHDLConfigurationVisibility](#) for more information.
- **-optionset <optionset\_name>**  
(optional) Calls an optionset as defined in the *modelsim.ini* file. Refer to the section “[Optionsets](#)” on page 26 for more information.
- **-outf <filename>**  
(optional) Specifies a file to which the final list of options is saved, after recursively expanding all -f, -file and -F files.
- **-pedanticerrors**  
(optional) Forces display of an error message (rather than a warning) on a variety of conditions. Refer to “[Enforcing Strict 1076 Compliance](#)” for a complete list of these conditions. This argument overrides -nocasestaticerror and -noothersstaticerror (refer above).

You can also view a complete list of errors by executing the command:

```
verror -kind vcom -pedanticerrors
```

- **-performdefaultbinding**  
(optional) Enables default binding when it has been disabled via the [RequireConfigForAllDefaultBinding](#) option in the *modelsim.ini* file.
- **-preserve**  
(optional) Forces vcom to preserve the case of letters in object identifiers. Can also be enabled by setting the [PreserveCase](#) variable to 1 in the *modelsim.ini* file.
- **-[w]prof=<filename>**  
(optional; -prof and -wprof are mutually exclusive) Enables CPU (-prof) or WALL (-wprof) time based profiling and saves the profile data to <filename>. Output from these arguments is used by Customer Support for debugging purposes.

- **-proftick=<integer>**  
(optional) Sets the time interval between the profile data collections. Default = 10.
- **-quiet**  
(optional) Disables ‘Loading’ messages.
- **-rangecheck**  
(default) Enables run time range checking. Range checking can be disabled using the **-norangecheck** argument.
- **-refresh <primary> [<secondary>]**  
(optional) Regenerates a library image. By default, the work library is updated. To update a different library, use **-work <library\_name>** with **-refresh** (for example, **vcom -work <your\_lib\_name> -refresh**).  
**<primary> [<secondary>]** — Specifies the entity, package, configuration, or module to be deleted.
  - If **<primary>** is an entity — only that entity, no related architectures, is refreshed.
  - If **<primary>** is a package — the only legal value of **<secondary>** is “body”, and only the package is refreshed.
  - If you specify both **<primary>** and **<secondary>** — Only the **<secondary>** architecture is updated, not the entity.

If a dependency checking error occurs which prevents the refresh, use the **vcom -force\_refresh** argument. Refer to the **vcom** Examples for more information. You may use a specific design name with **-refresh** to regenerate a library image for that design, but you may not use a file name.

You cannot specify the **<filename>** argument when specifying this argument.

- **-s**  
(optional) Instructs the compiler not to load the standard package. This argument should only be used if you are compiling the standard package itself.
- **-separateConfigLibrary**  
Allows the declaration of a VHDL configuration to occur in a different library than the entity being configured. Strict conformance to the VHDL standard (LRM) requires that they be in the same library.
- **-showsubprograms | -noshowsubprograms**  
(optional) Toggles viewing VHDL subprogram scopes on the command line and in GUI windows, for example, the Structure window. The default is not to show subprogram scopes.

- **-skip abcep**

(optional) Directs the compiler to skip all:

- a — architectures
- b — bodies
- c — configurations
- e — entities
- p — packages

Any combination in any order can be used, but one choice is required if you use this switch.

- **-skipsynthoffregion**

(optional) Ignore all constructs within synthesis\_off or translate\_off pragma regions.

- **-smartdbgsym**

(optional) Reduces the size of design libraries by minimizing the amount of debugging symbol files generated at compile time.

Edit the [SmartDbgSym](#) variable in the *modelsim.ini* file to set a permanent default.

- **-source**

(optional) Displays the associated line of source code before each error message that is generated during compilation. By default, only the error message is displayed.

- **-stats [=+ | -]<feature>[,[+ | -]<mode>]**

(optional) Controls display of compiler statistics sent to a logfile, stdout, or the transcript. Specifying -stats without options sets the default features (cmd, msg, and time).

Multiple features and modes for each instance of -stats are specified as a comma separated list. You can specify -stats multiple times on the command line, but only the last instance will take effect.

[+ | -] — Controls activation of the feature or mode where the plus character (+) enables the feature and the minus character (-) disables the feature. You can also enable a feature or mode by specifying a feature or mode without the plus (+) character. Setting this switch will add or subtract features and modes from the settings in the Stats *modelsim.ini* variable.

#### Features

all — Display all statistics features (cmd, msg, perf, time). Mutually exclusive with none option. When specified in a string with other options, all is applied first.

cmd — (default) Echo the command line.

msg — (default) Display error and warning summary at the end of command execution.

none — Disable all statistics features. Mutually exclusive with all option. When specified in a string with other options, none is applied first.

perf — Display time and memory performance statistics.

time — (default) Display Start, End, and Elapsed times.

#### Modes

Modes can be set for a specific feature or globally for all features. To add or subtract a mode for a specific feature, specify using the plus (+) or minus (-) character with the feature, for example, vcom -stats=cmd+verbose,perf+list. To add or subtract a mode globally for all features, specify the modes in a comma-separated list, for example, vcom -stats=time,perf,list,-verbose. You cannot specify global and feature specific modes together.

kb — Print performance statistics in kilobyte units with no auto-scaling.

list — Display statistics in a Tcl list format when available.

verbose — Display verbose statistics information when available.

---

#### Note

 You can disable all -stats features by specifying vcom -quiet.

---

- -suppress <msg\_number>[,<msg\_number>,...]

(optional) Prevents the specified message(s) from displaying. The <msg\_number> is the number preceding the message you wish to suppress. You cannot suppress Fatal or Internal messages. Edit the [suppress](#) variable in the *modelsim.ini* file to set a permanent default. Refer to “[Message Severity Level](#)” for more information.

<msg\_number> — A number identifying the message. Multiple message numbers are specified as a comma separated list.

- -version

(optional) Returns the version of the compiler as used by the licensing tools.

- -vitalmemorycheck

(optional) Enables VITAL level 1 checks.

- -vmake

(optional) Generates a complete record of all command line data and files accessed during the compile of a design. This data is then used by the [vmake](#) command to generate a comprehensive makefile for recompiling the design library. By default, vcom stores compile data needed for the -refresh switch and ignores compile data not needed for -refresh. The -vmake switch forces inclusion of all file dependencies and command line data accessed during a compile, whether they contribute data to the initial compile or not. Executing this switch can increase compile time in addition to increasing the accuracy of the compile. refer to the vmake command for more information.

- -warning <msg\_number>[,<msg\_number>,...]

(optional) Changes the severity level of the specified message(s) to "warning." Edit the [warning](#) variable in the *modelsim.ini* file to set a permanent default. Refer to “[Message Severity Level](#)” for more information.

**<msg\_number>** — A number identifying the message. Multiple message numbers are specified as a comma separated list.

- **-warning error**  
(optional) Reports all warnings as errors.
- **-work <library\_name>**  
(optional) Maps a library to the logical library work. By default, the compiled design units are added to the work library. The specified pathname overrides the pathname specified for work in the project file.

**<library\_name>** — A logical name or pathname of a library.

- **<filename>**  
(required, except for when you specify -refresh or -force\_refresh) Specifies the name of a file containing the VHDL source to be compiled. One filename is required; multiple filenames can be entered separated by spaces. Wildcards may be used, for example, *\*.vhd*.

If you don't specify a filename, and you are using the GUI, a dialog box pops up allowing you to select the options and enter a filename.

## Examples

- Compile the VHDL source code contained in the file *example.vhd*.

**vcom example.vhd**

- ModelSim supports designs that use elements conforming to the 1987, 1993, and 2002 standards. Compile the design units separately using the appropriate switches.

**vcom -87 o\_units1.vhd o\_units2.vhd**  
**vcom -93 n\_unit91.vhd n\_unit92.vhd**

- When compiling source that uses the numeric\_std package, this command turns off acceleration of the numeric\_std package, located in the ieee library.

**vcom -noaccel numeric\_std example.vhd**

- Although it is not obvious, the = operator is overloaded in the std\_logic\_1164 package. All enumeration data types in VHDL get an “implicit” definition for the = operator. So while there is no explicit = operator, there is an implicit one. This implicit declaration can be hidden by an explicit declaration of = in the same package (LRM Section 10.3). However, if another version of the = operator is declared in a different package than that containing the enumeration declaration, and both operators become visible through use clauses, neither can be used without explicit naming.

**vcom -explicit example.vhd**

To eliminate that inconvenience, the VCOM command has the -explicit option that allows the explicit = operator to hide the implicit one. Allowing the explicit declaration to hide the implicit declaration is what most VHDL users expect.

**ARITHMETIC."=" (left, right)**

---

- The -work option specifies mylib as the library to regenerate. -refresh rebuilds the library image without using source code, allowing models delivered as compiled libraries without source code to be rebuilt for a specific release of ModelSim (4.6 and later only).

**vcom -work mylib -refresh**

- Enable the display of Start, End, and Elapsed time as well as a message count summary. Echoing of the command line is disabled

**vcom -stats=time,-cmd,msg**

- The first -stats option is ignored. The none option disables all *modelsim.ini* settings and then enables the perf option.

**vcom -stats=time,cmd,msg -stats=none,perf**

# vdel

This command deletes a design unit from a specified library. This command provides additional information with the -help switch.

## Syntax

```
vdel [-lib <library_path>] [-modelsimini <path/modelsim.ini>] [-verbose]
      {-all | <primary> [<arch_name>]} -obj {<object_info>} | -dpiobj [<object_info>] }
```

## Arguments

- -all

(optional) Deletes an entire library.

### **Caution**



You cannot recover libraries once deleted. You are not prompted for confirmation.

- -dpiobj [<object\_info>]

(optional) Delete auto-compiled DPI object files.

<object\_info> — Specifies the type of object to remove, as reported by the output of the vdir -obj command. This will take the form of either:

<compiler> — a string identifying the compiler, such as gcc-3.3.1.

<platform> — a string identifying the platform.

<platform-compiler> — a string identifying a compiler/platform pair, such as linux\_gcc-3.2.3.

all — Specifies that all objects should be removed, as reported by the output of the vdir -obj command.

- -lib <library\_path>

(optional) Specifies location of the library that holds the design unit to be deleted. By default, the design unit is deleted from the work library.

<library\_path> — A logical name or pathname of the library.

- -modelsimini <path/modelsim.ini>

(optional) Loads an alternate initialization file that replaces the current initialization file. Overrides the file path specified in the MODELSIM environment variable.

<path/modelsim.ini> — Specifies either an absolute or relative path to the initialization file, including the file itself. On Windows systems the path separator should be a forward slash (/).

- -obj {<object\_info>}

(optional) removes directories containing DPI object files.

---

*<object\_info>* — Specifies the type of directory to remove, as reported by the output of the vdir -obj command. This will take the form of either:

*<compiler>* — a string identifying the compiler, such as gcc-3.3.1.

*<platform>* — a string identifying the platform.

*<platform-compiler>* — a string identifying a compiler/platform pair, such as linux\_gcc-3.2.3.

*all* — Specifies that all directories should be removed, as reported by the output of the vdir -obj command.

- *<primary> [<arch\_name>]*

(required unless -all is used) Specifies the entity, package, configuration, or module to be deleted.

*<arch\_name>* — Specifies the name of an architecture to be deleted. If omitted, all of the architectures for the specified entity are deleted. Invalid for a configuration or a package.

- *-verbose*

(optional) Displays progress messages.

## Examples

- Delete the work library.

**vdel -all**

- Delete the synopsys library.

**vdel -lib synopsys -all**

- Delete the entity named xor and all its architectures from the work library.

**vdel xor**

- Delete the architecture named behavior of the entity xor from the work library.

**vdel xor behavior**

- Delete the package named base from the work library.

**vdel base**

## vdir

This command lists the contents of a design library and checks the compatibility of a vendor library. If vdir cannot read a vendor-supplied library, the library may not be compatible with ModelSim.

### Syntax

```
vdir [-l | [-prop <prop>]] [-r] [-all | [-lib <library_name>]] [<design_unit>]  
[-modelsimini <path/modelsim.ini>]
```

### Description

This command provides additional information with the -help switch.

### Arguments

- **-all**  
(optional) Lists the contents of all libraries listed in the Library section of the active *modelsim.ini* file. Refer to [modelsim.ini Variables](#) for more information.
- **<design\_unit>**  
(optional) Indicates the design unit to search for within the specified library. If the design unit is a VHDL entity, its architectures are listed. By default all entities, configurations, modules, packages, and optimized design units in the specified library are listed.
- **-l**  
(optional) Prints the version of vcom/vlog with which each design unit was compiled, plus any compilation options used. Also prints the object-code version number that indicates which versions of vcom/vlog and ModelSim are compatible.
- **-lib <library\_name>**  
(optional) Specifies the logical name or the pathname of a library to be listed. By default, the contents of the work library are listed.  
  
*<library\_name>* — A logical name or pathname of a library.
- **-modelsimini <path/modelsim.ini>**  
(optional) Loads an alternate initialization file that replaces the current initialization file. Overrides the file path specified in the MODELSIM environment variable.  
  
*<path/modelsim.ini>* — Specifies either an absolute or relative path to the initialization file, including the file itself. On Windows systems the path separator should be a forward slash (/).
- **-prop <prop>**  
(optional) Reports on a specified design unit property.  
  
*<prop>* — Specifies a Design Unit Property, as listed in [Table 2-7](#). If you do not specify a value for *<prop>*, an error message is displayed.

**Table 2-7. Design Unit Properties**

<b>Value of &lt;prop&gt;</b>	<b>Description</b>
archcfg	configuration for arch
body	needs a body
cmpltime	compilation time
default	default options
dir	source directory
dpnd	depends on
entcfg	configuration for entity
fulloptions	Full compile options
inline	module inlined
lock	lock/unlock status
lrm	language standard
mtime	source modified time
name	short name
opcode	opcode format
options	compile options
pdu	preoptimized design unit
root	optimized Verilog design root
src	source file
top	top level model
ver	version string
vlogv	Verilog version

- **-r**

(optional) Prints architecture information for each entity in the output.

## Examples

- List the architectures associated with the module named and2 that reside in the default library work.

**vdir -l and2**

```
# Library vendor : Model Technology
# Maximum unnamed designs : 3
# MODULE and2
#     Verilog version: <XO@d;_mSdz@12Fz9b]_Z3
#     Version string: 3EdggZ>V3z51fE;>K[51?2
#     Source directory: C:\examples\dataflow_verilog
#     Source modified time: Tue Apr 28 22:48:56 2009
#     HDL source file: gates.v
#     Source file: gates.v
#     Start location: gates.v:18
#     Opcode format: 10.1a; VLOG SE Object version 51
#     Optimized Verilog design root: 1
#     VHDL language standard: 1
#     Compile options: -L mtiAvm -L mtiOvm -L mtiUvm -L mtiUPF
#     Debug Symbol file exists
```

## vencrypt

This command encrypts Verilog and SystemVerilog code contained within encryption envelopes. The code is not pre-processed before encryption, so macros and other `directives are unchanged. This allows IP vendors to deliver encrypted IP with undefined macros and `directives.

### Syntax

```
vencrypt <filename> [-d <dirname>] [-e <extension>] [-f <filename>]  
    [-hea <filename>] [-logfile <filename> | -l <filename>] [-o <filename>] [-p <prefix>]  
    [-quiet] [[-stats [=+ | -]<feature>[,[+ | -]<mode>]]]
```

### Description

Upon execution of this command, the filename extension will be changed to *.vp* for Verilog files (*.v* files) and *.svp* for SystemVerilog files (*.sv* files).

If the vencrypt utility processes the file (or files) and does not find any encryption directives it reprocesses the file using the following default encryption:

```
'pragma protect data_method = "aes128-cbc"  
'pragma protect key_keyowner = "MTI"  
'pragma protect key_keyname = "MGC-DVT-MTI"  
'pragma protect key_method = "rsa"  
'pragma protect key_block encoding = (enctype = "base64")  
'pragma protect begin
```

The vencrypt command must be followed by a compile command – such as [vlog](#) – for the design to be compiled.

This command provides additional information with the *-help* or *-hel* switch.

### Arguments

- <filename>  
(required) Specifies the name of the Verilog source code file to encrypt. One filename is required. Multiple filenames can be entered separated by spaces. Wildcards can be used. Default encryption pragmas will be used, as described above, if no encryption directives are found during processing.
- -d <dirname>  
(optional) Specifies where to save encrypted Verilog files. If no directory is specified, current working directory will be used.  
  
<dirname> — Specifies the directory to contain the encrypted Verilog or SystemVerilog files. The original file extension (*.v* for Verilog and *.sv* for SystemVerilog) will be preserved.

- **-e <extension>**  
(optional) Specifies a filename extension.  
    <extension> — Any alpha-numeric string.
- **-f <filename>**  
(optional) Specifies a file with more command line arguments. Allows complex arguments to be reused without retyping. Nesting of -f options is allowed.  
Refer to the section “[Argument Files](#)” on page 27 for more information.  
    <filename> — Specifies the name of a file containing command line arguments.
- **-hea <filename>**  
(optional) Concatenates header information into all design files listed with <filename>. Allows the user to pass a large number of files to the vencrypt utility that do not contain the `pragma protect or `protect information about how to encrypt the file. Saves the user from editing hundreds of files to add in the same `pragma protect to every file.  
    <filename> — Specifies an existing file.
- **-logfile <filename> | -l <filename>**  
(optional) Redirects log output to the file designated by <filename>.   
    <filename> — Specifies a file for saving output.
- **-o <filename>**  
(optional) Combines all encrypted output into a single file.  
    <filename> — Specifies a file for saving output.
- **-p <prefix>**  
(optional) Prepends file names with a prefix.  
    <prefix> — Any alpha-numeric string.
- **-quiet**  
(optional) Disables encryption messages.
- **-stats [=+ | -]<feature>[,[+ | -]<mode>]**  
(optional) Controls display of statistics sent to a logfile, stdout, or the transcript. Specifying -stats without options sets the default features (cmd and msg).  
Multiple features and modes for each instance of -stats are specified as a comma separated list. You can specify -stats multiple times on the command line, but only the last instance will take effect.  
    [+ | -] — Controls activation of the feature or mode where the plus character (+) enables the feature and the minus character (-) disables the feature. You can also enable a feature or mode by specifying a feature or mode without the plus (+) character. Setting this switch will add or subtract features and modes from the default settings "cmd,msg".

## Features

all — Display all statistics features (cmd, msg, perf). Mutually exclusive with none option. When specified in a string with other options, all is applied first.

cmd — (default) Echo the command line.

msg — (default) Display error and warning summary at the end of command execution.

none — Disable all statistics features. Mutually exclusive with all option. When specified in a string with other options, none is applied first.

perf — Display time and memory performance statistics.

time — Display Start, End, and Elapsed times. Has no effect and is ignored.

## Modes

Modes can be set for a specific feature or globally for all features. To add or subtract a mode for a specific feature, specify using the plus (+) or minus (-) character with the feature, for example, vencrypt -stats=cmd+verbose,perf+list. To add or subtract a mode globally for all features, specify the modes in a comma-separated list, for example, vencrypt -stats=time,perf,list,-verbose. You cannot specify global and feature specific modes together.

kb — Print performance statistics in kilobyte units with no auto-scaling.

list — Display statistics in a Tcl list format when available.

verbose — Display verbose statistics information when available.

---

### Note



vencrypt -quiet disables all default or user-specified -stats features.

---

## Examples

- Insert header information into all design files listed.

**vencrypt -h encrypt\_head top.v cache.v gates.v memory.v**

The *encrypt\_head* file may look like the following:

```
`pragma protect data_method = "aes128-cbc"
`pragma protect author = "IP Provider"
`pragma protect key_keyowner = "MTI", key_method = "rsa"
`pragma protect key_keyname = "MGC-DVT-MTI"
`pragma protect begin
```

There is no `pragma protect end expression in the header file, just the header block that starts the encryption. The `pragma protect end expression is implied by the end of the file. For more detailed examples, refer to "[Protecting Your Source Code](#)" in the User's Manual.

- Enable the display of message count summary. Echoing of the command line is disabled.

**vencrypt -stats=msg,-cmd,**

- The first -stats option is ignored. The none option disables all default settings and then enables the perf option.

**vencrypt -stats=msg,cmd -stats=none,perf**

## Related Topics

[Protecting Your Source Code](#)

[vhencrypt](#)

## verror

Returns a detailed description about a message number or a list of messages related to a specified portion of the product.

### Syntax

```
verror [-fmt | -full] <msgNum> ...
verror [-fmt | -full] [-kind <tool>] -all
verror [-kind <tool>] {-pedanticerrors | -permissive | -suppressibleerrors}
```

### Arguments

- **-fmt | -full**

(optional) Specifies the type and amount of information to return.

**-fmt**

Returns the format string used in the message.

**-full**

Returns the format string and complete text associated with the message.

- **[-kind <tool>] -all**

(required when not specifying <msgNum>) Returns information about all messages associated with a specified tool, where <tool> can be one of the following:

aid	hm_entity	mc2com	qverilog
sccom	scgenmod	sdfcomp	sm_entity
vcd2wlf	vcom	vcovkill	vdel
vdir	vencrypt	vgencomp	vish
vlib	vlog	vmake	vmap
vopt	vsim	wlf	wlf2log
wlfman	wlfrecover		

- **[-kind <tool>] {-pedanticerrors | -permissive | -suppressibleerrors}**

(optional) Specifies filtering for messages according to either or both of the following:

**<tool>**

Any of the values allowed for the -kind argument.

**-pedanticerrors**

Display messages that are reported as errors due to adhering to a more strict interpretation of the LRM.

**-permissive**

Display messages reported as warnings that would be displayed as errors if you use `vsim -pedanticerrors`.

**-suppressibleerrors**

Display messages that you can suppress from the command line or *modelsim.ini* file.

**• <msgNum>**

(required when not specifying -all) Specifies the message number(s) you would like more information about. You can find the message number in messages of the format:

```
** <Level>: ([<Tool>- [<Group>-] ]<MsgNum>) <FormattedMsg>
```

You can specify <msgNum> any number of times for one `verror` command in a space-separated list.

Optionally, you can specify the toolname prior to the message number, similar to how it appears in an error message. For example:

```
verror vsim-5003
```

## Examples

- If you receive the following message in the transcript:

```
** Error (vsim-3061) foo.v(22): Too many Verilog port connections.
```

and you would like more information about this message, you would type:

```
verror 3061
```

and receive the following output:

```
Message # 3061:  
Too many Verilog ports were specified in a mixed VHDL/Verilog instantiation. Verify that the correct VHDL/Verilog connection is being made and that the number of ports matches.  
[DOC: ModelSim User's Manual - Mixed VHDL and Verilog Designs Chapter]
```

## vgencomp

Once a Verilog module is compiled into a library, you can use this command to write its equivalent VHDL component declaration to standard output.

### Syntax

```
vgencomp [-lib <library_name>] [-b] [-bool] [-modelsimini <path/modelsim.ini>] [-s] [-v] [-work <name>] <module_name>
```

### Description

Optional switches allow you to generate bit or vl\_logic port types; std\_logic port types are generated by default.

This command provides additional information with the -help switch.

### Arguments

- **-lib <library\_name>**  
(optional) Specifies the working library where the default is to use the work library.  
    <library\_name> — Specifies the path and name of the working library.
- **-b**  
(optional) Causes vgencomp to generate bit port types.
- **-bool**  
(optional) Causes vgencomp to generate boolean port types.
- **-modelsimini <path/modelsim.ini>**  
(optional) Loads an alternate initialization file that replaces the current initialization file. Overrides the file path specified in the MODELSIM environment variable.  
    <path/modelsim.ini> — Specifies either an absolute or relative path to the initialization file, including the file itself. On Windows systems the path separator should be a forward slash (/).
- **-s**  
(optional) Used for the explicit declaration of default std\_logic port types.
- **-v**  
(optional) Causes vgencomp to generate vl\_logic port types.
- **-work <name>**  
(optional) Specifies the name of the work library, where the default is the library containing the module.
- **<module\_name>**  
(required) Specifies the name of the Verilog module to be accessed.

## Examples

- This example uses a Verilog module that is compiled into the work library. The module begins as Verilog source code:

```
module top(i1, o1, o2, io1);
    parameter width = 8;
    parameter delay = 4.5;
    parameter filename = "file.in";

    input i1;
    output [7:0] o1;
    output [4:7] o2;
    inout [width-1:0] io1;
endmodule
```

After compiling, vgencomp is invoked on the compiled module:

**vgencomp top**

and writes the following to stdout:

```
component top
    generic(
        width          : integer := 8;
        delay          : real    := 4.500000;
        filename       : string  := "file.in"
    );
    port(
        i1            : in     std_logic;
        o1            : out    std_logic_vector(7 downto 0);
        o2            : out    std_logic_vector(4 to 7);
        io1           : inout  std_logic_vector
    );
end component;
```

## vhencrypt

This command encrypts VHDL code contained within encryption envelopes. The code is not compiled before encryption, so dependent packages and design units do not have to exist before encryption.

### Syntax

```
vhencrypt <filename> [-d <dirname>] [-e <extension>] [-f <filename>]  
[-hea <filename>] [-logfile <filename> | -l <filename>] [-o <filename>] [-p <prefix>]  
[-quiet] [-stats [=+ | -]<feature>[,[+ | -]<mode>]]
```

### Description

Upon execution of this command, the *.vhd* filename extension is changed to *.vhdp* and the *.vhdl* filename extension is changed to *.vhdlp*.

If the vhencrypt utility does not find any encryption directives, no output file is produced.

The vhencrypt command must be followed by a compile command – such as vcom – for the design to be compiled.

This command provides additional information with the -help or -hel switch.

### Arguments

- <filename>  
(required) Specifies the name of the VHDL source code file to encrypt. One filename is required. Multiple filenames can be entered separated by spaces. Wildcards can be used.
- -d <dirname>  
(optional) Specifies where to save encrypted VHDL files. If no directory is specified, the current working directory will be used.  
    <dirname> — Specifies the directory to contain the encrypted VHDL files. The original file extension (*.vhd* or *.vhdl*) will be preserved.
- -e <extension>  
(optional) Specifies a filename extension to be applied to the encrypted file.  
    <extension> — Any alpha-numeric string.
- -f <filename>  
(optional) Specifies a file with more command line arguments. Allows complex arguments to be reused without retyping. Nesting of -f options is allowed.  
Refer to the section “[Argument Files](#)” on page 27 for more information.  
    <filename> — Specifies the name of a file containing command line arguments.

- **-hea <filename>**  
(optional) Concatenates header information into all design files listed with <filename>. Allows the user to pass a large number of files to the vhencrypt utility that do not contain the encryption information (between the `protect and `protect end directives) about how to encrypt the file. Saves the user from editing hundreds of files to add the same encryption information into every file.  
    <filename> — Specifies an existing file.
- **-logfile <filename> | -l <filename>**  
(optional) Redirects log output to the file designated by <filename>.   
    <filename> — Specifies a file for saving output.
- **-o <filename>**  
(optional) Combines all encrypted output into a single file.  
    <filename> — Specifies a file for saving output.
- **-p <prefix>**  
(optional) Prepends encrypted file names with a prefix.  
    <prefix> — Any alpha-numeric string.
- **-quiet**  
(optional) Disables encryption messages.
- **-stats [=+ | -]<feature>[,[+ | -]<mode>]**  
(optional) Controls display of statistics sent to a logfile, stdout, or the transcript. Specifying -stats without options sets the default features (cmd and msg).  
Multiple features and modes for each instance of -stats are specified as a comma separated list. You can specify -stats multiple times on the command line, but only the last instance will take effect.  
    [+ | -] — Controls activation of the feature or mode where the plus character (+) enables the feature and the minus character (-) disables the feature. You can also enable a feature or mode by specifying a feature or mode without the plus (+) character. Setting this switch will add or subtract features and modes from the default settings "cmd,msg".

#### Features

- all — Display all statistics features (cmd, msg, perf). Mutually exclusive with none option. When specified in a string with other options, all is applied first.
- cmd — (default) Echo the command line.
- msg — (default) Display error and warning summary at the end of command execution.
- none — Disable all statistics features. Mutually exclusive with all option. When specified in a string with other options, none is applied first.

perf — Display time and memory performance statistics.

time — Display Start, End, and Elapsed times. Has no effect and is ignored.

### Modes

Modes can be set for a specific feature or globally for all features. To add or subtract a mode for a specific feature, specify using the plus (+) or minus (-) character with the feature, for example, vhencrypt -stats=cmd+verbose,perf+list. To add or subtract a mode globally for all features, specify the modes in a comma-separated list, for example, vhencrypt -stats=time,perf,list,-verbose. You cannot specify global and feature specific modes together.

kb — Print performance statistics in kilobyte units with no auto-scaling.

list — Display statistics in a Tcl list format when available.

verbose — Display verbose statistics information when available.

---

#### Note

 vhencrypt -quiet disables all default or user-specified -stats features.

---

## Examples

- Enable the display of message count summary. Echoing of the command line is disabled.  
**vhencrypt -stats=msg,-cmd,**
- The first -stats option is ignored. The none option disables all default settings and then enables the perf option.  
**vhencrypt -stats=msg,cmd -stats=none,perf**

## Related Topics

[Protecting Your Source Code](#)

[vencrypt](#)

# view

This command opens the specified window. If you specify this command without arguments, it returns a list of all open windows in the current layout.

## Syntax

```
view <window_type>...[-aliases][-names] [-title {New Window Title}]  
      [-undock {[-icon] [-height <n>] [-width <n>] [-x <n>] [-y <n>]}] | -dock]
```

## Description

To remove a window, use the [noview](#) command.

The view command with one or more options and no window names specified applies the options to the currently open windows. Refer to examples for additional details.

## Arguments

- <window\_type>...  
(required) Specifies the window type to view. You do not need to type the full type name (see the examples below); implicit wildcards are accepted; multiple window types are accepted. Available window types are:

assertions	atv	browser	calltree
canalysis	capacity	classgraph	classtree
covergroups	dataflow	details	duranked
exclusions	fcovers	files	fsmlist
fsmview	instance	library	list
locals	memdata	memory	msgviewer
objects	process	profiledetails	project
ranked	runmgr	schematic	source
stackview	structural	structure	tracker
transaction	transcript	uvmdetails	watch
wave			

Not all windows are available with all variants of ModelSim and Questa SIM

- -aliases  
(optional) Returns a list of <window\_type> aliases.
- -height <n>  
(optional) Specifies the window height in pixels. Can only be used with the -undock switch.

<n> — Any non-negative integer.

- **-icon**  
(optional) Toggles the view between window and icon. Can only be used with the **-undock** switch.
- **-names**  
(optional) Returns a list of valid <window\_type> arguments.
- **-title {New Window Title}**  
(optional) Specifies the window title of the designated window.  
{New Window Title} — Any string. Curly braces are needed for a string containing spaces. Double quotes (" ") can be used in place of braces, for example "New Window Title."
- **-dock**  
(optional) Docks the specified standalone window into the Main window.
- **-undock**  
(optional) Opens the specified window as a standalone window, undocked from the Main window.
- **-width <n>**  
(optional) Specifies the window width in pixels. Can only be used with the **-undock** switch.  
<n> — Any non-negative integer.
- **-x <n>**  
(optional) Specifies the window upper-left-hand x-coordinate in pixels. Can only be used with the **-undock** switch.  
<n> — Any non-negative integer.
- **-y <n>**  
(optional) Specifies the window upper-left-hand y-coordinate in pixels. Can only be used with the **-undock** switch.  
<n> — Any non-negative integer.

## Examples

- Undock the Wave window from the Main window and makes it a standalone window.  
**view -undock wave**
- Display an undocked Processes window in the upper left-hand corner of the monitor with a window size of 300 pixels, square.  
**view process -undock -x 0 -y 0 -width 300 -height 300**
- Display the Watch and Wave windows.

**view w**

- Display the Objects and Processes windows.

**view ob pr**

- Open a new Wave window with My Wave Window as its title.

**view -title {My Wave Window} wave**

## virtual count

This command reports the number of currently defined virtuals that were not read in using a macro file.

### Syntax

`virtual count [-kind {implicits | explicits}] [-unsaved]`

### Arguments

- `-kind {implicits | explicits}`

(optional) Reports only a subset of virtuals.

    implicits — virtual signals created internally by the product.

    explicits — virtual signals explicitly created by a user, such as with the `virtual signal` command.

    Unique abbreviations are accepted.

- `-unsaved`

(optional) Reports the count of only those virtuals that have not been saved to a macro file.

### Related Topics

[virtual define](#)

[virtual save](#)

[virtual show](#)

[Virtual Objects](#)

# virtual define

This command prints to the transcript the definition of the virtual signals, functions, or regions in the form of a command that can be used to re-create the object.

## Syntax

```
virtual define [-kind {implicits | explicits}] <pathname>
```

## Arguments

- `-kind {implicits | explicits}`

(optional) Transcripts only a subset of virtuals.

    implicits — virtual signals created internally by the tool.

    explicits — virtual signals explicitly created by a user, such as with the `virtual signal` command.

Unique abbreviations are accepted.

- `<pathname>`

(required) Specifies the path to the virtual(s) for which you want definitions, where wildcards are allowed.

## Examples

- Show the definitions of all the virtuals you have explicitly created.

```
virtual define -kind explicits *
```

## Related Topics

[virtual describe](#)

[virtual show](#)

[Virtual Objects](#)

## virtual delete

This command removes the matching virtuals.

### Syntax

**virtual delete [-kind {implicits | explicits}] <pathname>**

### Arguments

- **-kind {implicits | explicits}**

(optional) Removes only a subset of virtuals.

    implicits — virtual signals created internally by the product.

    explicits — virtual signals explicitly created by a user, such as with the **virtual signal** command.

Unique abbreviations are accepted.

- **<pathname>**

(required) Specifies the path to the virtual(s) you want to delete, where wildcards are allowed.

### Examples

- Delete all of the virtuals you have explicitly created.

**virtual delete -kind explicits \***

### Related Topics

[virtual signal](#)

[virtual function](#)

[Virtual Objects](#)

# virtual describe

This command prints to the transcript a complete description of the data type of one or more virtual signals. Similar to the existing describe command.

## Syntax

```
virtual describe [-kind {implicits | explicits}] <pathname>
```

## Arguments

- `-kind {implicits | explicits}`

(optional) Transcripts only a subset of virtuals.

    implicits — virtual signals created internally by the product.

    explicits — virtual signals explicitly created by a user, such as with the virtual signal command.

Unique abbreviations are accepted.

- `<pathname>`

(required) Specifies the path to the virtual(s) for which you want descriptions, where wildcards are allowed.

## Examples

- Describe the data type of all virtuals you have explicitly created.

```
virtual describe -kind explicits *
```

## Related Topics

[virtual define](#)

[virtual show](#)

[Virtual Objects](#)

## virtual expand

This command prints to the transcript a list of all the non-virtual objects contained in the specified virtual signal(s). You can use this to create a list of arguments for a command that does not accept or understand virtual signals.

### Syntax

`virtual expand [-base] <pathname> ...`

### Arguments

- `-base`

(optional) Outputs the root signal parent in place of a subelement. For example:

```
vcd add [virtual expand -base myVirtualSignal]
```

the resulting command after substitution would be:

```
vcd add signala signalb signalc
```

- `<pathname>`

(required) Specifies the path to the signals and virtual signals to expand, where wildcards are allowed and you can specify any number of paths.

### Examples

- Add the elements of a virtual signal to the VCD file.

In the Tcl language, the square brackets specify that the enclosed command should be executed first ("virtual expand ..."), then the result substituted into the surrounding command.

```
vcd add [virtual expand myVirtualSignal]
```

Therefore, if *myVirtualSignal* is a concatenation of *signal<sub>a</sub>*, *signal<sub>b</sub>.rec1* and *signal<sub>c</sub>(5 downto 3)*, the resulting command after substitution would be:

```
vcd add signala signalb.rec1 {signalc(5 downto 3)}
```

The slice of *signal<sub>c</sub>* is enclosed in curly braces, because it contains spaces.

### Related Topics

[virtual signal](#)

[Virtual Objects](#)

# virtual function

This command creates a new signal, known only by the GUI (not the kernel), that consists of logical operations on existing signals and simulation time, as described in <expressionString>.

## Syntax

```
virtual function [-env <path>] [-install <path>] [-delay <time> <unit>] {<expressionString>}  
    <name>
```

## Description

It cannot handle bit selects and slices of Verilog registers. Please see “[Syntax and Conventions](#)” on page 13 for more details on syntax.

If the virtual function references more than a single scalar signal, it will display as an expandable object in the Wave and Objects windows. The children correspond to the inputs of the virtual function. This allows the function to be “expanded” in the Wave window to see the values of each of the input waveforms, which could be useful when using virtual functions to compare two signal values.

Virtual functions can also be used to gate the List window display.

---

### Note

 The virtual function and virtual signal commands are interchangeable. The product will keep track of whether you’ve created a signal or a function with the commands and maintain them appropriately. We document both commands because the virtual save, virtual describe, and virtual define commands will reference your virtual objects using the correct command.

---

## Arguments

Arguments for virtual function are the same as those for virtual signal, except for the contents of the expression string.

- **-env <path>**  
(optional) Specifies a hierarchical context for the signal names in <expressionString> so they don't all have to be full paths.  
    <path> — Specifies a relative path to the signal(s).
- **-install <path>**  
(optional) Causes the newly-created signal to become a child of the specified region. If -install is not specified, the newly-created signal becomes a child of the nearest common ancestor of all objects appearing in <expressionString>. If the expression references more than one WLF file (dataset), the virtual signal will automatically be placed in region virtuals:/Functions.  
    <path> — Specifies a relative path to the signal(s). On Windows systems the path separator should be a forward slash (/).

- **-delay <time> <unit>**  
(optional) Specifies a value by which the virtual function will be delayed. You can use negative values to look forward in time. Refer to the examples below for more details.  
  
**<time>** — Specified as an integer or decimal number. Current simulation units are the default unless specifying **<unit>**.  
  
**<unit>** — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting **<unit>**. Valid VHDL time units are: fs, ps, ns, us, ms, sec, min, and hr. You must enclose **<time>** and **<unit>** within curly braces ({}).
- **{<expressionString>}**  
(required) A text string expression, enclosed in curly braces ({} ) using the “[GUI\\_expression\\_format](#)” on page 32.
- **<name>**  
(required) The name you define for the virtual signal.  
Case is ignored unless installed in a Verilog region.  
Use alpha, numeric, and underscore characters only, unless you are using VHDL extended identifier notation.  
If using VHDL extended identifier notation, **<name>** needs to be quoted with double quotes (" ") or with curly braces ({} ).

## Examples

- Create a signal */chip/section1/clk\_n* that is the inverse of */chip/section1/clk*.  
**virtual function { not /chip/section1/clk } clk\_n**
- Create a std\_logic\_vector equivalent of a Verilog register *rega* and installs it as */chip/rega\_slv*.  
**virtual function -install /chip { (std\_logic\_vector) chip.vlog.rega } rega\_slv**
- Create a boolean signal */chip/addr\_eq\_fab* that is true when */chip/addr[11:0]* is equal to hex "fab", and false otherwise. It is acceptable to mix VHDL signal path notation with Verilog part-select notation.  
**virtual function { /chip/addr[11:0] == 0xfab } addr\_eq\_fab**
- Create a signal that is high only during times when signal */chip/siga* of the gate-level version of the design does not match */chip/siga* of the rtl version of the design. Because there is no common design region for the inputs to the expression, *siga\_diff* is installed in region *virtuals:/Functions*. The virtual function *siga\_diff* can be added to the Wave window, and when expanded will show the two original signals that are being compared.  
**virtual function { gate:/chip/siga XOR rtl:/chip/siga } siga\_diff**
- Create a virtual signal consisting of the logical "AND" function of */top signalA* with */top signalB*, and delays it by 10 ns.  
**virtual function { delay:10ns AND /top.signalA AND /top.signalB } signal\_and**

**virtual function -delay {10 ns} {/top/signalA AND /top/signalB} myDelayAandB**

- Create a one-bit signal *outbus\_diff* which is non-zero during times when any bit of */chip/outbus* in the gate-level version doesn't match the corresponding bit in the rtl version.

This expression uses the "OR-reduction" operator, which takes the logical OR of all the bits of the vector argument.

**virtual function { | (gate:/chip/outbus XOR rtl:/chip/outbus) } outbus\_diff**

Commands fully compatible with virtual functions

[add log](#) and [log](#)

[delete](#)

[describe](#)

[examine](#)

[find](#)

[restart](#)

[searchlog](#)

[show](#)

Commands not compatible with virtual functions

[drivers](#)

[force](#)

[noforce](#)

[vcd add](#)

[when](#)

## Related Topics

[virtual count](#)

[virtual define](#)

[virtual delete](#)

[virtual describe](#)

[virtual expand](#)

[virtual hide](#)

[virtual log](#)

[Virtual Objects](#)

[virtual nohide](#)

[virtual nolog](#)

[virtual region](#)

[virtual save](#)

[virtual show](#)

[virtual signal](#)

[virtual type](#)

## virtual hide

This command causes the specified real or virtual signals to not be displayed in the Objects window. This is used when you want to replace an expanded bus with a user-defined bus. You make the signals reappear using the virtual nohide command.

### Syntax

```
virtual hide {[[-kind {implicits | explicits}] | [-region <path>]} <pattern>
```

### Arguments

- **-kind {implicits | explicits}**  
(optional) Hides only a subset of virtuals.
  - implicits — virtual signals created internally by the tool.
  - explicits — virtual signals explicitly created by a user, such as with the virtual signal command.Unique abbreviations are accepted.
- **-region <path>**  
(optional) Specifies a region of design space in which to look for the signal names.  
<path> — Specifies an absolute or relative path to the signal(s). On Windows systems the path separator should be a forward slash (/).
- **<pattern>**  
(required) Indicates which signal names or wildcard patterns should be used in finding the signals to hide, where wildcards are allowed and you can specify any number of names or patterns.

### Related Topics

[virtual nohide](#)

[Virtual Objects](#)

# virtual log

This command causes the simulation-mode dependent signals of the specified virtual signals to be logged by the kernel. If wildcard patterns are used, it will also log any normal signals found, unless the **-only** option is used. You unlog the signals using the virtual **nolog** command.

## Syntax

```
virtual log {[ -kind {implicits | explicits} ] | [-region <path>]} [-recursive] [-only] [-in] [-out]  
[-inout] [-internal] [-ports] <pattern>
```

## Arguments

- **-kind {implicits | explicits}**  
(optional) Logs only a subset of virtuals.
  - implicits — virtual signals created internally by the tool.
  - explicits — virtual signals explicitly created by a user, such as with the **virtual signal** command.Unique abbreviations are accepted.
- **-region <path>**  
(optional) Specifies a region of design space in which to look for signals to log.  
<path> — Specifies an absolute or relative path to the signal(s). On Windows systems the path separator should be a forward slash (/).
- **-recursive**  
(optional) Specifies that the scope of the search is to descend recursively into subregions. If omitted, the search is limited to the selected region.
- **-only**  
(optional) Specify that only virtual signals (as opposed to all signals) found by a <pattern> containing a wildcard should be logged.
- **-in**  
(optional) Specifies that the kernel log data for ports of mode IN whose names match the specification.
- **-out**  
(optional) Specifies that the kernel log data for ports of mode OUT whose names match the specification.
- **-inout**  
(optional) Specifies that the kernel log data for ports of mode INOUT whose names match the specification.

- **-internal**  
(optional) Specifies that the kernel log data for internal (non-port) objects whose names match the specification.
- **-ports**  
(optional) Specifies that the kernel log data for all ports.
- **<pattern>**  
(required) Indicates which signal names or wildcard patterns should be used in finding the signals to log, where you can specify any number of names or wildcard patterns.

## Related Topics

[Virtual Objects](#)

[virtual nolog](#)

# virtual nohide

This command reverses the effect of a virtual hide command, causing the specified real or virtual signals to reappear the Objects window.

## Syntax

```
virtual nohide {[-kind {implicits | explicits}] | [-region <path>]} <pattern>
```

## Arguments

- **-kind {implicits | explicits}**

(optional) Unhides only a subset of virtuals.

    implicits — virtual signals created internally by the tool.

    explicits — virtual signals explicitly created by a user, such as with the virtual signal command.

Unique abbreviations are accepted.

- **-region <path>**

(optional) Specifies a region of design space in which to look for the signal names.

    <path> — Specifies an absolute or relative path to the signal(s). On Windows systems the path separator should be a forward slash (/).

- **<pattern>**

(required) Indicates which signal names or wildcard patterns should be used in finding the signals to hide, where wildcards are allowed and you can specify any number of names or patterns.

## Related Topics

[virtual hide](#)

[Virtual Objects](#)

## virtual nolog

This command reverses the effect of a virtual log command. It causes the simulation-dependent signals of the specified virtual signals to be excluded ("unlogged") by the kernel. If wildcard patterns are used, it will also unlog any normal signals found, unless the -only option is used.

### Syntax

```
virtual nolog {[ -kind {implicits | explicits} ] | [-region <path>]} [-recursive] [-only] [-in] [-out]  
[-inout] [-internal] [-ports] <pattern>
```

### Arguments

- **-kind {implicits | explicits}**  
(optional) Excludes only a subset of virtuals.
  - implicits — virtual signals created internally by the tool.
  - explicits — virtual signals explicitly created by a user, such as with the virtual signal command.Unique abbreviations are accepted.
- **-region <path>**  
(optional) Specifies a region of design space in which to look for signals to unlog.  
<path> — Specifies an absolute or relative path to the signal(s). On Windows systems the path separator should be a forward slash (/).
- **-recursive**  
(optional) Specifies that the scope of the search is to descend recursively into subregions. If omitted, the search is limited to the selected region.
- **-only**  
(optional) Specify that only virtual signals (as opposed to all signals) found by a <pattern> containing a wildcard should be unlogged.
- **-in**  
(optional) Specifies that the kernel exclude data for ports of mode IN whose names match the specification.
- **-out**  
(optional) Specifies that the kernel exclude data for ports of mode OUT whose names match the specification.
- **-inout**  
(optional) Specifies that the kernel exclude data for ports of mode INOUT whose names match the specification.

- **-internal**  
(optional) Specifies that the kernel exclude data for internal (non-port) objects whose names match the specification.
- **-ports**  
(optional) Specifies that the kernel exclude data for all ports.
- **<pattern>**  
(required) Indicates which signal names or wildcard patterns should be used in finding the signals to unlog, where wildcards are allowed and you can specify any number of names or patterns.

## Related Topics

[virtual log](#)

[Virtual Objects](#)

# virtual region

This command creates a new user-defined design hierarchy region.

---

## Note

---

 Virtual regions cannot be used in the [when](#) command.

---

## Syntax

`virtual region <parentPath> <regionName>`

## Arguments

- `<parentPath>`  
(required) The full path to the region that will become the parent of the new region.
- `<regionName>`  
(required) The name you want for the new region.

## Related Topics

[virtual function](#)

[virtual signal](#)

[Virtual Objects](#)

# virtual save

This command saves the definitions of virtuals to a file named virtual.do in the current directory.

## Syntax

```
virtual save [-kind {implicits | explicits}] [-append] [<filename>]
```

## Arguments

- `-kind {implicits | explicits}`

(optional) Saves only a subset of virtuals.

    implicits — virtual signals created internally by the tool.

    explicits — virtual signals explicitly created by a user, such as with the `virtual signal` command.

Unique abbreviations are accepted.

- `-append`

(optional) Specifies to save only virtuals that are not already saved or weren't read in from a macro file. These unsaved virtuals are then appended to the specified or default file.

- `<filename>`

(optional) The name of the file containing the definitions. If you don't specify `<filename>`, the default virtual filename (virtuals.do) will be used. You can specify a different default in the `pref.tcl` file.

## Related Topics

[virtual count](#)

[Virtual Objects](#)

## virtual show

This command lists the full path names of all explicitly defined virtuals.

### Syntax

`virtual show [-kind {implicits | explicits}]`

### Arguments

- `-kind {implicits | explicits}`

(optional) Lists only a subset of virtuals.

`implicits` — virtual signals created internally by the tool.

`explicits` — virtual signals explicitly created by a user, such as with the `virtual signal` command.

Unique abbreviations are accepted.

### Related Topics

[virtual define](#)

[virtual describe](#)

[Virtual Objects](#)

# virtual signal

This command creates a new signal, known only by the GUI (not the kernel), that consists of concatenations of signals and subelements as specified in <expressionString>.

## Syntax

```
virtual signal [-env <path>] [-install <path>] [-delay <time> <unit>] {<expressionString>}  
          <name>
```

## Description

It cannot handle bit selects and slices of Verilog registers. Please see “[Concatenation of Signals or Subelements](#)” on page 39 for more details on syntax.

### Note

 The virtual function and virtual signal commands are interchangeable. The product will keep track of whether you've created a signal or a function with the commands and maintain them appropriately. We document both commands because the virtual save, virtual describe, and virtual define commands will reference your virtual objects using the correct command.

## Arguments

- **-env <path>**  
(optional) Specifies a hierarchical context for the signal names in <expressionString> so they don't all have to be full paths.  
  
    <path> — Specifies a relative path to the signal(s). On Windows systems the path separator should be a forward slash (/).
- **-install <path>**  
(optional) Causes the newly-created signal to become a child of the specified region. If -install is not specified, the newly-created signal becomes a child of the nearest common ancestor of all objects appearing in <expressionString>. If the expression references more than one WLF file (dataset), the virtual signal will automatically be placed in region virtuals:/Signals.  
  
    <path> — Specifies a relative path to the signal(s). On Windows systems the path separator should be a forward slash (/).
- **-delay <time> <unit>**  
(optional) Specifies a value by which the virtual function will be delayed. You can use negative values to look forward in time. Refer to the examples below for more details.  
  
    <time> — Specified as an integer or decimal number. Current simulation units are the default unless specifying <unit>.  
  
    <unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid VHDL time units are: fs, ps,

ns, us, ms, sec, min, and hr. You must enclose <time> and <unit> within curly braces ({}).

- {<expressionString>}  
(required) A text string expression, enclosed in curly braces ({} ) using the “[GUI\\_expression\\_format](#)” on page 32.
- <name>  
(required) The name you define for the virtual signal.  
Case is ignored unless installed in a Verilog region.  
Use alpha, numeric, and underscore characters only, unless you are using VHDL extended identifier notation.  
If using VHDL extended identifier notation, <name> needs to be quoted with double quotes (" ") or with curly braces ({} ).

## Examples

- Reconstruct a bus *sim:/chip/alu/a(4 downto 0)*, using VHDL notation, assuming that *a<sub>ii</sub>* are all scalars of the same type.  

```
virtual signal -env sim:/chip/alu { (concat_range (4 downto 0))(a_04 & a_03 & a_02 & a_01 & a_00) } a
```
- Reconstruct a bus *sim:/chip.alu.a[4:0]*, using Verilog notation. Note that the concatenation notation starts with "&{" rather than "{".  

```
virtual signal -env sim:/chip.alu
{ (concat_range [4:0])&{a_04, a_03, a_02, a_01, a_00} } a
```
- Create a signal *sim:/testbench/stuff* which is a record type with three fields corresponding to the three specified signals. The example assumes */chipa/mode* is of type integer, */chipa/alu/a* is of type std\_logic\_vector, and */chipa/decode/inst* is a user-defined enumeration.  

```
virtual signal -install sim:/testbench
{ /chipa/alu/a(19 downto 13) & /chipa/decode/inst & /chipa/mode } stuff
```
- Create a virtual signal that is the same as */top/signalA* except it is delayed by 10 ps.  

```
virtual signal -delay {10 ps} {/top/signalA} myDelayedSignalA
```
- Create a three-bit signal, *chip.address\_mode*, as an alias to the specified bits.  

```
virtual signal { chip.instruction[23:21] } address_mode
```
- Concatenate signals *a*, *b*, and *c* with the literal constant ‘000’.  

```
virtual signal {a & b & c & 3'b000} myextendedbus
```
- Add three missing bits to the bus *num*, creates a virtual signal *fullbus*, and then adds that signal to the Wave window.  

```
virtual signal {num & 3'b000} fullbus
```

```
virtual signal {num & "000"} fullbus
add wave -unsigned fullbus
```

- Reconstruct a bus that was fragmented by synthesis and is missing the lower three bits. Note that you would have to type in the actual bit names (for example, num28, num27, and so on) represented by the ... in the syntax above.

```
virtual signal { num31 & num30 & num29 & ... & num4 & num3 & "000" } fullbus
add wave -unsigned fullbus
```

- Create a two-bit signal (with an enumerated type) based on the results of the subexpressions. For example, if *aold* equals *anew*, then the first bit is true (1). Alternatively, if *bold* does not equal *bnew*, the second bit is false (0). Each subexpression is evaluated independently.

```
virtual signal {(aold == anew) & (bold == bnew)} myequalityvector
```

- Create signal *newbus* that is a concatenation of bus1 (bit-reversed) and bus2[7:4] (bit-reversed). Assuming bus1 has indices running 7 downto 0, the result will be newbus[11:0] with the upper 8 bits being bus1[0:7] and the lower 4 bits being bus2[4:7]. See “Concatenation of Signals or Subelements” on page 39 for further details.

```
virtual signal {(concat_reverse)(bus1 & bus2[7:4])} newbus
```

### Commands fully compatible with virtual signals

add list	add log or log	add wave
delete	describe	examine
find	force and noforce	restart
searchlog	show	

### Commands compatible with virtual signals using [virtual expand <signal>]

drivers	vcd add
---------	---------

### Commands not currently compatible with virtual signals

- when

## Related Topics

virtual count
virtual describe
virtual log
virtual region

[virtual function](#)

[virtual define](#)

[virtual expand](#)

[virtual nohide](#)

[virtual save](#)

[virtual type](#)

[virtual delete](#)

[virtual hide](#)

[virtual nolog](#)

[virtual show](#)

[Virtual Objects](#)

# virtual type

This command creates a new enumerated type known only by the GUI, not the kernel. Virtual types are used to convert signal values to character strings. The command works with signed integer values up to 64 bits.

## Syntax

```
virtual type -delete <name> | {<list_of_strings>} <name>
```

## Description

Virtual types cannot be used in the [when](#) command.

### Note

 If you are using SystemVerilog, you can also convert signal values to character strings using associative arrays in your code. See the SystemVerilog LRM for more information.

## Arguments

- `-delete <name>`

(Required if not defining a type.) Deletes a previously defined virtual type.

`<name>` — The name you gave the virtual type when you originally defined it. .

- `{<list_of_strings>}`

(Required if `-delete` is not used.) A list of values and their associated character strings. Values can be expressed in decimal or based notation and can include "don't-cares" (see examples below). Three kinds of based notation are supported: Verilog, VHDL, and C-language styles. The values are interpreted without regard to the size of the bus to be mapped. Bus widths up to 64 bits are supported.

If the string contains spaces the string must be enclosed in quotation marks (" ") If they contain special characters square brackets, curly braces, backslashes...), they need to be quoted within curly braces.

See the examples below for further syntax.

- `<name>`

(Required if `-delete` is not used.) The user-defined name of the virtual type. Case is not ignored. Use alpha, numeric, and underscore characters only, unless you are using VHDL extended identifier notation. If using VHDL extended identifier notation, `<name>` needs to be quoted with double quotes (" ") or with curly braces ( { } ).

## Examples

- Using positional notation, associates each string with an enumeration index, starting at zero and increasing by one in the positive direction. When *myConvertedSignal* is displayed in the Wave, List, or Objects window, the string "state0" will appear when *mysignal* == 0, "state1" when *mysignal* == 1, "state2" when *mysignal* == 2, and so on.

```
virtual type {state0 state1 state2 state3} mystateType
virtual function {({mystateType})mysignal} myConvertedSignal
add wave myConvertedSignal
```

- Use sparse mapping of bus values to alphanumeric strings for an 8-bit, one-hot encoding. It shows the variety of syntax that can be used for values. The value "default" has special meaning and corresponds to any value not explicitly specified.

```
virtual type {{0 NULL_STATE} {1 st1} {2 st2} {0x04 st3} {16'h08 st4} \
             {'h10 st5} {16#20 st6} {0b01000000 st7} {0x80 st8} \
             {default BAD_STATE}} myMappedType
virtual function {({myMappedType})mybus} myConvertedBus
add wave myConvertedBus
```

- Delete the virtual type "mystateType".

```
virtual type -delete mystateType
```

- Create a virtual type that includes "don't-cares" (the '-' character).

```
virtual type {{0x01-- add}{0x02-- sub}{default bad}} mydecodetype
```

- Create a virtual type using a mask for "don't-cares." The middle field is the mask, and the mask should have bits set to 1 for the bits that are don't care.

```
virtual type {{0x0100 0xff add}{0x0200 0xff sub}{default bad}} mydecodetype
```

## Related Topics

[virtual function](#)

[Virtual Objects](#)

# vlib

This command creates a design library. You must use vlib rather than operating system commands to create a library directory or index file.

## Syntax

```
vlib -help  
vlib [-short | -dos | -long | -unix] [-dirpath <pathname>] [-format { 1 | 3 | 4 }]  
      [-type {directory | archive | flat}]  
      [{-lock | -unlock} <design_unit>] [-locklib | -unlocklib] [-compress | -nocompress]  
      <library_name>
```

## Description

If the specified library already exists as a valid ModelSim library, the vlib command will exit with a warning message without touching the library.

This command provides additional information with the -help switch.

## Arguments

- **-compress | -nocompress**  
(optional) Defines whether some compiled results are stored in the library in a compressed form.
  - compress — Compression occurs, producing smaller libraries. However, this can slow down your subsequent executions of the vopt command.
  - nocompress — (default) No compression is made on the libraries.
- **-dirpath <pathname>**  
(optional) Specifies the location of a working directory to be stored in the library in order to override the current working directory. This allows you hide the directory path information.

---

### Caution



Use of this argument is not recommended.

For example, if you use -dirpath to override the working directory information, then the ModelSim user interface will not be able to find the source files if the end user selects something in the design and asks to see the declaration.

- **-dos**  
(optional) Specifies that subdirectories in a library have names that are compatible with DOS. Not recommended if you use the **vmake** utility.  
On by default for ModelSim PE.

- `-format { 1 | 3 | 4 }`

(optional) Prepares a library for conversion to be compatible with a previous release, by altering the `_info` file.

1 — allows you to convert a library to be compatible with the 6.2 series and earlier.

3 — allows you to convert a library to be compatible with the 6.3 series and newer.

4 — allows you to convert a library to be compatible with the 10.2 series and newer.

The usage flow is:

```
\\"1) Using a current release of the simulator, run:
  vlib -format 1 current_lib
  vcom -refresh -work current_lib
  \\      to prepare current_lib for conversion back to a 6.2 release
  \\
\\"2) Using a 6.2 release of the simulator, run:
  vcom -refresh -work current_lib
  \\      to refresh current_lib for use with the previous release
```

- `-long`

(optional) Interchangeable with the `-unix` argument.

- `{-lock | -unlock} <design_unit>`

(optional) Locks an existing design unit so it cannot be recompiled or refreshed. The `-unlock` switch reverses this action. File permissions are not affected by these switches.

- `-locklib | -unlocklib`

(optional) Locks a complete library so that compilation cannot target the library and the library cannot be refreshed. The `-unlocklib` switch reverses this action. File permissions are not affected by these switches.

- `-short`

(optional) Interchangeable with the `-dos` argument.

- `-type {directory | archive | flat}`

(optional) Specifies the type of library you want to create.

`directory` — directory-based, legacy library. Use this option when working in a flow requiring the `vmake` command.

`archive` — archive library (replaces `vlib -archive` option).

`flat` — (default) condensed library without design unit directories.

- `-unix`

(optional) Specifies that subdirectories in a library may have long file names that are NOT compatible with DOS.

- `<library_name>`

(required) Specifies the pathname of the library to be created.

## Examples

- Create the design library *design*. You can define a logical name for the library using the [vmap](#) command or by adding a line to the library section of the *modelsim.ini* file that is located in the same directory.

**vlib design**

- Create the design library *uut* and specifies that any design units compiled into the library are created as archives.

**vlib -type archive uut**

## vlog

The vlog command compiles Verilog source code and SystemVerilog extensions into a specified working library (or to the work library by default). Compressed SystemVerilog source files (those compressed with zlib) are accepted.

### Syntax

```
vlog [options] <filename> [<filename> ...]  
[options]:  
[-93]  
[-addpragmaprefix <prefix>]  
[-compat] [-compile_uselibs[=<directory_name>]] [-convertallparams] [-cuname  
<package_name>] [-cuautoname[=file | du]]  
[+define+<macro_name>[=<macro_text>]] [-deglitchalways | -nodeglitchalways]  
[+delay_mode_distributed] [+delay_mode_path] [+delay_mode_unit]  
[+delay_mode_zero] [-dirpath <pathname>] [-dpiforceheader] [-dpiforceheader <filename>]  
[-E <filename>] [-Edebug <filename>] [-enumfirstinit] [-Epretty <filename>]  
[-error <msg_number>[,<msg_number>,...]]  
[(-F | -file | -f) <filename>] [-force_refresh <design_unit>]  
[-fsmimplicittrans | -nofsmimplicittrans] [-fsmresettrans | -nofsmresettrans] [-fsmsingle | -  
nofsmsingle]  
[-fsmverbose[b | t | w]] [-fsmxassign | -nofsmxassign]  
[-gen_xml <design_unit> <filename>]  
[-hazards]  
[-ignorepragmaprefix <prefix>] [+inmdir+<directory>] [-incr | -noincr]  
[-isymfile] [+iterevaluation]  
[+libcell | +nolibcell] [+libext+<suffix>]  
[-libmap <pathname>] [-libverbose=libmap] [-libmap_verbose] [+librescan] [-line  
<number>]  
[-lint]  
[-logfile <filename> | -l <filename>] [-lrmclassinit]  
[+maxdelays] [+mindelays] [-mixedansiports] [-mixedsvvh [b | s | v]]  
[-mfcu[=macro] | -sfcu] [-modelsimini <path/modelsim.ini>]  
[-msglimit [all, | none,] [-+<msg_number>[,-+<msg_number>,...]]]  
[-msglimitcount <limit_value> -msglimit [all,|none,] [-+<msgNumber>[,-  
+]<msgNumber>...]]  
[-nocreatelib] [-nodbgsym]  
[-noForceUnsignedToVhdlInteger] [-nologo] [-nooverrideundef] [+nospecify]  
[-note <msg_number>[,<msg_number>,...]] [+notimingchecks]
```

```
[-novtblfixup] [+nowarn<CODE>] [-nowarn <category_number>]

[-optionset <optionset_name>] [-outf <filename>] [-override_precision]
  [-override_timescale[=][ ]<time_unit> / <time_precision>] [-O0]
  [-pedanticerrors] [-permissive] [-permit_defunct_sv] [-printinfilenames[=<filename>]]

[-quiet]

[-R [<simargs>]] [-refresh]

[-s] [-sfcu] [-skipprotected] [-skipprotectedmodule]
  [-skipsynthoffregion] [-smartdbgsym] [-source]
  [-stats [=+ | -]<feature>[,[+ | -]<mode>]]
  [-suppress <msg_number>[,<msg_number>,...]] [-sv]
  [-svext=[+|-]<extension>[,[+|-]<extension>]...[sceq]]
  [-svfilesuffix=<extension>[,<extension>... ] <filename>]
  [-svinputport=net | var | relaxed] [-svpkgcasesens]
  [-sv05compat] [-sv09compat] [-sv12compat]

[-timescale[=][ ]<time_units>/<time_precision>] [+typdelays]

[-u]

[-v <library_file>] [-version] [-vlog01compat] [-vlog95compat] [-vmake]

[-warning <msg_number>[,<msg_number>,...]] [-warning error] [-warnrbw]
  [-work <library_name>] [-writetoplevels <fileName>]

[-y <library_directory>]
```

## Description

The vlog command may be invoked from within ModelSim or from the operating system command prompt. It may also be invoked during simulation.

Compiled libraries are major-version dependent. When moving between major versions, you have to refresh compiled libraries using the -refresh argument to vlog. This is not true for minor versions (letter releases).

All arguments to the vlog command are case sensitive: -WORK and -work are not equivalent.

SystemVerilog requires that the default behavior of the vlog command is to treat each Verilog design file listed on the command line as a separate compilation unit. To treat multiple files listed within a single command line as a single compilation unit, use either the vlog -mfcu argument or the [MultiFileCompilationUnit](#) *modelsim.ini* file variable.

This command provides additional information with the -help switch.

## Arguments

- **-93**  
(optional) Specifies that the VHDL interface to Verilog modules use VHDL 1076-1993 extended identifiers to preserve case in Verilog identifiers that contain uppercase letters.
- **-addpragmaprefix <prefix>**  
(optional) Enables recognition of pragmas with a user specified prefix. If this argument is not specified, pragmas are treated as comments.  
All regular synthesis pragmas are honored.  
**<prefix>** — Specifies a user defined string where the default is no string, indicated by quotation marks ("").  
You may also set this with the [AddPragmaPrefix](#) variable in the vlog section of the *modelsim.ini* file.
- **-compat**  
(optional) Disables optimizations that result in different event ordering than Verilog-XL. ModelSim Verilog generally duplicates Verilog-XL event ordering, but there are cases where it is inefficient to do so. Using this option does not help you find event order dependencies, but it allows you to ignore them. Keep in mind that this option does not account for all event order discrepancies, and that using this option may degrade performance. Refer to [“Event Ordering in Verilog Designs”](#) for additional information.
- **-compile\_uselibs[=<directory\_name>]**  
(optional) Locates source files specified in a `uselib directive (Refer to [“Verilog-XL uselib Compiler Directive”](#)), compiles those files into automatically created libraries, and updates the *modelsim.ini* file with the logical mappings to the new libraries. If a *directory\_name* is not specified, ModelSim uses the name specified in the MTI\_USELIB\_DIR environment variable. If that variable is not set, ModelSim creates the directory *mti\_uselibs* in the current working directory.
- **-convertallparams**  
(optional) Enables converting parameters not defined in ANSI style to VHDL generics of type std\_logic\_vector, bit\_vector, std\_logic, vl\_logic, vl\_logic\_vector, and bit.
- **-cuname <package\_name>**  
(optional) Used only in conjunction with -mfcu. The -cuname argument names the compilation unit (*package\_name*) being created by vlog. The named compilation unit can then be specified on the vsim command line, along with the <top> design unit. The purpose of doing so is to force elaboration of specified compilation unit package, thereby forcing elaboration of a necessary ‘bind’ statement within that compilation unit that would otherwise not be elaborated. An example of the necessary commands is:

```
vlog -cuname pkg_name -mfcu file1.sv file2.sv
vsim top pkg_name
```

You need to do this only in cases where you have a ‘bind’ statement in a module that might otherwise not be elaborated, because no module in the design depends on that compilation unit. In other words, if a module that depends on that compilation unit exists, you don’t need to force the elaboration, for it occurs automatically. Also, if you are using qverilog to compile and simulate the design, this binding issue is handled properly automatically.

- **-cuautoname=[file | du]**

(optional) Specifies the method for naming \$unit library entries.

file — (default) Base the name on first file in on the command line.

du — Base the name on the first design unit following items found in the \$unit scope.

This option is useful for cases where you have multiple vlog command lines that specify the same file as the first entry.

- **+define+<macro\_name>[=<macro\_text>]**

Allows you to define a macro from the command line that is equivalent to the following compiler directive:

```
`define <macro_name> <macro_text>
```

Optionally, you can specify more than one macro with a single +define. For example:

```
vlog +define+one=r1+two=r2+three=r3 test.v
```

A command line macro overrides a macro of the same name defined with the `define compiler directive. It will also override all `undef directives in the RTL code — i.e., `undef for that macro will be ignored. Use the **-nooverrideundef** option for backward compatibility with previous operation. If a macro is defined using +define command line option and the -nooverrideundef option is used, the `undef will be honored for that macro.

- **-deglitchalways | -nodeglitchalways**

Controls the behavior related to zero-delay oscillations among always\_comb and always @\* combinatorial logic blocks, as well as regular always blocks, that produce glitches on the variables they write.

-deglitchalways — (default) Reduces the incidents of zero delay oscillations among the affected blocks.

-nodeglitchalways — Disables the functionality. A side effect of this behavior is that time zero races involving the glitch-producing always blocks may resolve in a different order.

- **+delay\_mode\_distributed**

(optional) Disables path delays in favor of distributed delays. Refer to “[Delay Modes](#)” for details.

- **+delay\_mode\_path**

(optional) Sets distributed delays to zero in favor of using path delays.

- **+delay\_mode\_unit**  
(optional) Sets path delays to zero and non-zero distributed delays to one time unit.
- **+delay\_mode\_zero**  
(optional) Sets path delays and distributed delays to zero.
- **-dirpath <pathname>**  
(optional) Specifies the location of a working directory to be stored in the library in order to override the current working directory. This allows you hide the directory path information.

---

**Caution**

---



Use of this argument is not recommended.

For example, if you use **-dirpath** to override the working directory information, then the ModelSim user interface will not be able to find the source files if the end user selects something in the design and asks to see the declaration.

---

- **-dpiforceheader**  
(optional) Forces the generation of a DPI header file even if it will be empty of function prototypes.
- **-dpiheader <filename>**  
(optional) Generates a header file that may then be included in C source code for DPI import functions. Refer to “[DPI Use Flow](#)” for additional information.
- **-E <filename>**  
(optional) Captures text processed by the Verilog parser after preprocessing has occurred and copies that text to an output file. This includes text read from source files specified by using the **-v** or **-y** argument.

**<filename>** — Specifies a name for the debugging output file. Wildcards are not allowed.

Generally, preprocessing consists of the following compiler directives: ``ifdef`, ``else`, ``elsif`, ``endif`, ``ifndef`, ``define`, ``undef`, ``include`.

The ``line` directive attempts to preserve line numbers, file names, and level in the output file (per the 1800-2009 LRM). White space is usually preserved, but sometimes it may be deleted or added to the output file.

- **-Edebug <filename>**  
(optional) Captures text processed by the Verilog parser after preprocessing has occurred and copies that text to a debugging output file.

**<filename>** — Specifies a name for the debugging output file. Wildcards are not allowed.

Generally, preprocessing consists of the following compiler directives: ``ifdef`, ``else`, ``elsif`, ``endif`, ``ifndef`, ``define`, ``undef`, ``include`. The file is a concatenation of source files with

`include expanded. The file can be compiled and then used to find errors in the original source files. The `line directive attempts to preserve line numbers and file names in the output file. White space is usually preserved, but sometimes it may be deleted or added to the output file.

- **-enumfirstinit**

(optional) Initializes enum variables in SystemVerilog using the leftmost value as the default. You must also use the argument with the vsim command in order to implement this initialization behavior. Specify the EnumBaseInit variable as 0 in the *modelsim.ini* file to set this as a permanent default.

- **-Epretty <filename>**

(optional) Captures text processed by the Verilog parser after preprocessing has occurred, performs some formatting for better readability, and copies that text to an output file, <filename>.

- **-error <msg\_number>[,<msg\_number>,...]**

(optional) Changes the severity level of the specified message(s) to "error." Edit the **error** variable in the *modelsim.ini* file to set a permanent default. Refer to "["Message Severity Level"](#)" for more information.

- **(-F | -file | -f) <filename>**

(optional) -f, -file and -F: each specifies an argument file with more command-line arguments, allowing complex argument strings to be reused without retyping. Nesting of -F, -f and -file commands is allowed. Allows gzipped input files.

With -F only: relative file names and paths within the arguments file <filename> are prefixed with the path of the arguments file when lookup with relative path fails. Refer to the section "["Argument Files"](#) on page 27" for more information.

- **-force\_refresh <design\_unit>**

(optional) Forces the refresh of all specified design units. By default, the work library is updated; use -work <library\_name>, in conjunction with -force\_refresh, to update a different library (for example, vlog -work <your\_lib\_name> -force\_refresh).

When the compiler refreshes a design unit, it checks each dependency to ensure its source has not been changed and recompiled. Sometimes the dependency checking algorithm changes from release to release. This can lead to false errors during the integrity checks performed by the -refresh argument. An example of such a message follows:

```
** Error: (vsim-13) Recompile /u/test/dware/
dware_61e_beta.dwpackages because /home/users/questasim/...
synopsys.attributes has changed.
```

The -force\_refresh argument forces the refresh of the design unit, overriding any dependency checking errors encountered by the -refresh argument.

A more conservative approach to working around -refresh dependency checks is to recompile the source code, if it is available.

- **-fsmimplicittrans | -nofsmimplicittrans**  
(optional) Toggles recognition of implied same state transitions. This setting is off by default.
- **-fsmresettrans | -nofsmresettrans**  
(optional) Toggles recognition of synchronous or asynchronous reset transitions. This setting is on by default.
- **-fsmsingle | -nofsmsingle**  
(optional) Toggles the recognition of VHDL FSMs where the current state variable of type std\_logic, bit, boolean, or single-bit std\_logic\_vector/bit\_vector and Verilog single-bit FSMs. This setting is off by default.
- **-fsmverbose[b | t | w]**  
(optional) Provides information about FSMs detected, including state reachability analysis.
  - b — displays only basic information.
  - t — displays a transition table in addition to the basic information.
  - w — displays any warning messages in addition to the basic information.

When you do not specify an argument, this switch reports all information similar to:

```
# ** Note: (vlog-1947)      FSM RECOGNITION INFO
:#   Fsm detected in : ../fpu/rtl/vhdl/serial_mul.vhd
:#   Current State Variable : s_state : ../fpu/rtl/vhdl/serial_mul.vhd(76)
:#   Clock : clk_i
:#   Reset States are: { waiting , busy }
:#   State Set is : { busy , waiting }
:#   Transition table is
#
#   -----
#   busy      =>  waiting  Line : (114 => 114)
#   busy      =>  busy     Line : (111 => 111)
#   waiting  =>  waiting  Line : (120 => 120) (114 => 114)
#   waiting  =>  busy     Line : (111 => 111)
#   -----
```

When you do not specify this switch, you will receive a message similar to:

```
# ** Note: (vlog-143) Detected '1' FSM/s in design unit
'serial_mul.rtl'.
```

- **-fsmxassign | -nofsmxassign**  
(optional) Toggles recognition of finite state machines (FSMs) containing X assignment. This option is used to detect FSMs if current state variable or next state variable has been assigned "X" value in a "case" statement. FSMs containing X-assign are otherwise not detectable. This setting is on by default.
- **-gen\_xml <design\_unit> <filename>**  
(optional) Produces an XML-tagged file containing the interface definition of the specified module. This option requires a two-step process where you must 1) compile <filename> into

a library with vlog (without -gen\_xml) then 2) execute vlog with the -gen\_xml switch, for example:

```
vlib work
vlog counter.v
vlog -gen_xml counter counter.xml
```

- -hazards

(optional) Detects event order hazards involving simultaneous reading and writing of the same register in concurrently executing processes. You must also specify this argument when you simulate the design with **vsim**. Refer to “[Hazard Detection](#)” for more details.

---

**Note**



Enabling -hazards implicitly enables the -compat argument. As a result, using this argument may affect your simulation results.

---

- -ignorepragmaprefix <prefix>

(optional) Directs vlog to ignore pragmas with the specified prefixname. All affected pragmas will be treated as regular comments. Edit the [IgnorePragmaPrefix](#) *modelsim.ini* variable to set a permanent default.

<prefix> — Specifies a user defined string.

- +includer+<directory>

(optional) Specifies directories to search for files included with `include compiler directives. By default, the current directory is searched first and then the directories specified by the +includer options in the order they appear on the command line. You may specify multiple +includer options as well as multiple directories separated by "+" in a single +includer option.

- -incr

(optional) Performs an incremental compilation. Compiles only code that has changed. For example, if you change only one module in a file containing several modules, only the changed module will be recompiled. Note however that if the compile options change, all modules are recompiled, regardless of whether you use vlog -incr or not.

- -isymfile

Generates a complete list of all imported tasks and functions (TFs). Used with DPI to determine all imported TFs that are expected by ModelSim.

- +iterative

(default) Enable an iterative evaluation mechanism on optimized gate-level cells with feedback loops.

- -logfile <filename> | -l <filename>

(optional) Generates a log file of the compile.

-logfile <filename> — Saves transcript data to <filename>. Can be abbreviated to -l <filename>. Overrides the default transcript file creation set with the [TranscriptFile](#)

---

or [BatchTranscriptFile](#) *modelsim.ini* variables. You can also specify “stdout” or “stderr” for <filename>.

- `+libcell | +nolibcell`
  - `+libcell` — (optional) Treats all modules found and compiled by source library search as though they contained a ‘celldesign compiler directive, thus marking them as cells (refer to the `-v` and `-y` arguments of `vlog`, which enable source library search). Using the `+libcell` argument matches historical behavior of Verilog-XL with respect to source library search.
  - `+nolibcell` — (default) Disables treating all modules found and compiled by source library search as though they contained a ‘celldesign compiler directive. That is, this argument restores the default library search behavior if you have changed it using the `+libcell | +nolibcell` argument.

---

**Note**



- `+libext+<suffix>`  
(optional) Works in conjunction with the `-y` option. Specifies file extensions for the files in a source library directory. By default, the compiler searches for files without extensions. If you specify the `+libext` argument, then the compiler will search for a file with the suffix appended to an unresolved name. You may specify only one `+libext` option, but it may contain multiple suffixes separated by the plus character (+). The extensions are tried in the order you specify them with the `+libext` argument.
- `-libmap <pathname>`  
(optional) Specifies a Verilog 2001 library map file. You can omit this argument by placing the library map file as the first option on the `vlog` invocation (for example, `vlog top.map top.v top_cfg.v`). You can use the `vlog -mfcu` argument to compile macros for all files in a given testbench. Any macros already defined before the `-libmap` argument appears are still defined for use by the `-libmap` files.
- `-libverbose=libmap`  
(optional)  
Displays library map pattern matching information during compilation. Use this argument to troubleshoot problems with matching filename patterns in a library map file.  
For example, when a resolved module has a choice between two libraries, you want to know which one it selected (confirming that your config file worked).
- `-libmap_verbose`  
(optional) Displays library map pattern matching information during compilation.

---

**Note**



This argument is being deprecated—you should use the `-libverbose=libmap` argument instead. However, `-libmap_verbose` will continue to be supported indefinitely for compatibility reasons, so it is safe to use until further notice.

---

- **+librescan**  
(optional) Scans libraries in command-line order for all unresolved modules.
- **-line <number>**  
(optional) Starts the compiler on the specified line in the Verilog source file. By default, the compiler starts at the beginning of the file.
- **-lint**  
(optional) Issues warnings on the following lint-style static checks:
  - when Module ports are NULL.
  - when assigning to an input port.
  - when referencing undeclared variables/nets in an instantiation.

This switch generates additional array bounds-checking code, which can slow down simulation, to check for the following:

- index warnings for dynamic arrays
- when an index for a Verilog unpacked variable array reference is out of bounds.

The warnings are reported as **WARNING[8]**. You can also enable this option using the [Show\\_Lint](#) variable in the *modelsim.ini* file.

- **-lrmclassinit**  
Changes initialization behavior to match the SystemVerilog specification (per IEEE Std 1800-2007) where all superclass properties will be initialized before any subclass properties.
- **+maxdelays**  
(optional) Selects maximum delays from the "min:typ:max" expressions. If preferred, you can defer delay selection until simulation time by specifying the same option to the simulator.
- **+mindelays**  
(optional) Selects minimum delays from the "min:typ:max" expressions. If preferred, you can defer delay selection until simulation time by specifying the same option to the simulator.  
v — treats all scalars/vectors in the package as VHDL `vl_logic`/`vl_logic_vector`
- **-mfcu[=macro]**  
(optional) Instructs the compiler to treat all files within a compilation command line as a single compilation unit. The default behavior is to treat each file listed in a command as a separate compilation unit, per the SystemVerilog standard. Prior versions concatenated the contents of the multiple files into a single compilation unit by default. When specified, the `=macro` modifier enables the visibility of macro definitions across different files.

All global declarations present in both compile file and library files specified with the **-v** argument will be lumped together in a single \$unit scope.

You can use **-mfcu** to compile macros for all files in a given testbench. Any macros already defined before the **-libmap** argument appears are still defined for use by the **-libmap** files.

You can also enable this option (without the **=macro** functionality) using the [MultiFileCompilationUnit](#) variable in the `modelsim.ini` file.

- **-mixedansiports**

Use this switch only when your design files contain a combination of ANSI and non-ANSI port declarations and task/function declarations. For example:

```
module top (input reg [7:0] a,
            output b);
    reg [7:0] b;
endmodule
```

- **-mixedsvvh [b | s | v]**

(optional) Facilitates using SystemVerilog packages at the SystemVerilog-VHDL boundary of a mixed-language design. When you compile a SystemVerilog package with **-mixedsvvh**, the package can be included in a VHDL design as if it were defined in VHDL itself.

**b** — treats all scalars/vectors in the package as VHDL bit/bit\_vector

**s** — treats all scalars/vectors in the package as VHDL std\_logic/std\_logic\_vector

- **-modelsimini <path/modelsim.ini>**

Loads an alternate initialization file that replaces the current initialization file. Overrides the file path specified in the MODELSIM environment variable. Specifies either an absolute or relative path to the initialization file, including the file itself. On Windows systems the path separator should be a forward slash (/).

- **-msglimit [all, | none,] [-+]<msg\_number>[,-+<msg\_number>,...]**

(optional) Limits the number of iterations of the specified message(s) to the default message limit count of five, then suppresses all new instances. Refer to [Suppression of Warning Messages](#) for more information.

**all** — Limits all messages to the default count except specified msgNumber(s).

**none** — Limits only the specified msgNumber(s) to the default count.

**[-+]** — The ‘-’ argument is used *only* with “all” to specify exclusion of specified msgNumber(s). The ‘+’ argument is used *only* with “none” to specify inclusion of specific message IDs. If neither is used, the command works the same way.

---

#### **Note**

 The ‘-’ argument can *only* be used with the “all” argument and the ‘+’ argument can *only* be used with the “none” argument. Otherwise incorrect results may appear.

---

**<msg\_number>[,<msg\_number>,...]** — Specifies the message number(s) to limit to five iterations. Multiple messages are specified as a comma-separated list.

For example, the following limits all messages to the default count except *msgNumber1* and *msgNumber2*.

```
vsim -msglimit all, <msgNumber1>, <msgNumber2>
```

While the following, limits only *msgNumber1* and *msgNumber2* to the default count.

```
vsim -msglimit none, <msgNumber1>, <msgNumber2>
```

- **-msglimitcount <limit\_value>** -msglimit [all,|none,] [-|+]<msgNumber>[,-|+]<msgNumber>...]  
(optional) Limits the reporting of listed messages to user-defined limit\_value. Overrides the [MsgLimitCount](#) variable in the modelsim.ini file.
- **-nocreatelib**  
(optional) Stops automatic creation of missing work libraries and reverts back to 10.3x and earlier version behavior. Overrides the [CreateLib](#) modelsim.ini variable.
- **-nodbgsym**  
Disables the generation of the symbols debugging database in the compiled library.  
The symbols debugging database is the .dbs file in the compiled library that provides information to the GUI allowing you to view detailed information about design objects at the source level. Two major GUI features that use this database include source window annotation and textual dataflow.  
You should only specify this switch if you know that anyone using the library will not require this information for design analysis purposes.
- **-noForceUnsignedToVhdlInteger**  
Prevents untyped Verilog parameters in mixed-language designs that are initialized with unsigned values between 2\*31-1 and 2\*32 from being converted to a VHDL generic. By default, untyped Verilog parameters that are initialized with unsigned values between 2\*3 -1 and 2\*32 are converted to VHDL INTEGER generics. Because VHDL INTEGER parameters are signed numbers, the Verilog values 2\*31 -1 to 2\*32 are converted to negative VHDL values in the range from -2\*31 to -1 (the 2's complement value).
- **-noincr**  
(optional) Disables incremental compilation previously turned on with -incr argument.  
Default.
- **-nologo**  
(optional) Disables the startup banner.
- **-nooverrideundef**  
(optional) Prevents `undefs from being overridden by macros defined using the +define command line option. If a macro is defined using +define command line option, and -nooverrideundef is also passed as a compile option, the `undef will be honored for that macro.

- **+nospecify**  
(optional) Disables specify path delays and timing checks.
- **-note <msg\_number>[,<msg\_number>,...]**  
(optional) Changes the severity level of the specified message(s) to "note." Edit the [note](#) variable in the *modelsim.ini* file to set a permanent default. Refer to "[Message Severity Level](#)" for more information.
- **+notimingchecks**  
(optional) Removes all timing check entries from the design as it is parsed.
- **-novtblfixup**  
Causes virtual method calls in SystemVerilog class constructors to behave as they would in normal class methods, which prevents the type of a this reference from changing during construction.  
This overrides default behavior, where the type of a this reference is treated as if it is a handle to the type of the active new() method while a constructor is executing (which implies that virtual method calls resolve will not execute methods of an uninitialized class type).
- **+nowarn<CODE>**  
(optional) Disables warning messages in the category specified by <CODE>. Warnings that can be disabled include the <CODE> name in square brackets in the warning message. For example,

```
** Warning: test.v(15) : [RDGN] - Redundant digits in numeric
literal.
```

This warning message can be disabled by specifying +nowarnRDGN.

- **-nowarn <category\_number>**  
(optional) Prevents the specified message(s) from displaying. The <msg\_number> is the number preceding the message you wish to suppress. Multiple -nowarn switches are allowed. Warnings may be disabled for all compiles via the Main window **Compile** > **Compile Options** menu command or the *modelsim.ini* file (refer to [modelsim.ini Variables](#)).

The warning message categories are described in the following table:

**Table 2-8. Warning Message Categories for vlog -nowarn**

Category number	Description
12	non-LRM compliance in order to match Cadence behavior

- **-optionset <optionset\_name>**  
(optional) Calls an optionset as defined in the *modelsim.ini* file. Refer to the section “[Optionsets](#)” for more information.
- **-outf <filename>**  
(optional) Specifies a file to which the final list of options is saved, after recursively expanding all -f, -file and -F files.
- **-override\_precision**  
(optional) Used with the -timescale argument, this argument overrides the precision of `timescale specified in the source code.
- **-override\_timescale[=][ ]<time\_unit> / <time\_precision>**  
(optional) Specifies a timescale for all compiled design units. This timescale overrides all ‘timescale directives and all declarations of timeunit and timeprecision. An equal sign (=) or whitespace is accepted between option and arguments.  
  
time\_unit — unit of measurement for times and delays. This specification consists of one of three integers (1, 10, or 100) representing order of magnitude and one of six character strings representing units of measurement:  
  
 $\{1 \mid 10 \mid 100\} \{s \mid ms \mid us \mid ns \mid ps \mid fs\}$   
  
For example, 10 ns.  
  
time\_precision — unit of measurement for rounding delay values before being used in simulation. Allowable values are the same as for time\_unit.
- **-O0**  
(optional) Lower the optimization to a minimum with -O0 (capital oh zero). Use this to work around bugs, increase your debugging visibility on a specific cell, or when you want to place breakpoints on source lines that have been optimized out.
- **-pedanticerrors**  
(optional) Enforces strict compliance of the IEEE Std 1800-2005. The following are some of the cases:
  - Using "new" for queues is not legal. When strict compliance is not enforced, use of "new" creates a queue of the specified size where all elements are initialized to the default value of the queue element type.
  - Using underscore character (\_) in sized, based literals is not legal. When you specify this argument, an error will occur for literals such as 2'b\_01.
  - Omitting the grave accent mark (`) preceding the left brace ({) when writing structure literals is not legal. When you specify this argument, an error will occur for literals written without that mark.
  - Inserting the grave accent mark to precede quotation marks (`") that enclose string literals is not legal—only string literals within quotation marks ("") are allowed.

---

When you specify this argument, an error will occur for string literals using that mark.

- Using class extern method prototypes with lifetime (automatic/static) designations produces a compliance error (instead of a warning).
- Using “cover bool@clk” as a PSL statement.
- Using an unsized constant in a concatenation if it is the leftmost value in the list.
- Calling a virtual function in the constructor of the same class.
- Using integers to define macro names.

This argument also produces a report of mismatched ‘else’ directives.

You can produce a complete list by executing the command:

```
verror -kind vlog -pedanticerrors
```

- **-permissive**

(optional) Allows messages in the LRM group of error messages to be downgraded to a warning. Allows reserved keywords ‘config’ and ‘instance’ to be used outside of unit and configuration scopes. Also allows named port connections on bit-select and part-select ports, though only when multiple bit-select or part-select ports of same name are not present in the port list.

You can produce a complete list by executing the command:

```
verror -kind vlog -permissive
```

- **-permit\_defunct\_sv**

(optional) Allows using a selected set of constructs no longer supported by the SystemVerilog standard. Currently, the set supports only the use of the keyword “char.” This argument allows use of the keyword “char” to be interpreted as the SystemVerilog “byte” type.

- **-printfilenames[=<filename>]**

Prints the path names of all source files opened (including “include” files) during the compile. Specifies whether each file is a Verilog or SystemVerilog file. To write these path names to a text file in the current directory, add =<filename> to this argument. If you use this argument again with the same filename, you overwrite the contents of the previous version of the file.

- **-quiet**

(optional) Disables ‘Loading’ messages.

- **-R [<simargs>]**

Instructs the compiler to invoke **vsim** after compiling the design. The compiler automatically determines which top-level modules are to be simulated.

When using -R option, the log files for vlog and vsim need to be specified separately. The file specified before -R will capture the output of the vlog compiler and the one provided after -R will capture the vsim output.

For example, in the following vlog command, "log1.txt" will contain the vlog output and "log2.txt" will contain the vsim output.

```
vlog -l log1.txt top.sv -R -c -do "run -all;quit" -l log2.txt
```

The -R option is not a Verilog-XL option, but it is used by ModelSim to combine the compile and simulate phases together as you may be used to doing with Verilog-XL. It is not recommended that you regularly use this option because you will incur the unnecessary overhead of compiling your design for each simulation run. Mainly, it is provided to ease the transition to ModelSim.

- **-refresh**

(optional) Regenerates a library image. By default, the work library is updated. To update a different library, use -work <library\_name> with -refresh (for example, vlog -work <your\_lib\_name> -refresh). If a dependency checking error occurs which prevents the refresh, use the vlog -force\_refresh argument. See vlog examples for more information. You may use a specific design name with -refresh to regenerate a library image for that design, but you may not use a file name.

- **-s**

(optional) Instructs the compiler not to load the standard package. This argument should only be used when you are compiling the sv\_std package.

- **-sfcu**

Instructs the compiler to treat all files within a compilation command line as a separate compilation units. This is the default behavior and is the inverse of the behavior of [-mfcu\[=macro\]](#).

A local \$unit scope will be created for a library file passed through -v argument if this file has global declarations.

This switch will override the [MultiFileCompilationUnit](#) variable if it is set to "1" in the modelsim.ini file.

- **-skipprotected**

(optional) Ignores any 'protected'/'endprotected region contained in a module.

- **-skipprotectedmodule**

(optional) Prevents adding any module containing a 'protected'/'endprotected region to the library.

- **-skipsynthoffregion**

(optional) Ignore all constructs within synthesis\_off or translate\_off pragma regions.

- **-smartdbgsym**  
(optional) Reduces the size of design libraries by minimizing the amount of debugging symbol files generated at compile time.  
Edit the [SmartDbgSym](#) variable in the *modelsim.ini* file to set a permanent default.
- **-source**  
(optional) Displays the associated line of source code before each error message that is generated during compilation. By default, only the error message is displayed.
- **-stats [=+ | -]<feature>[,[+ | -]<mode>]**  
(optional) Controls display of compiler statistics sent to a logfile, stdout, or the transcript. Specifying -stats without options sets the default features (cmd, msg, and time).  
Multiple features and modes for each instance of -stats are specified as a comma separated list. You can specify -stats multiple times on the command line, but only the last instance will take effect.  
[+ | -] — Controls activation of the feature or mode where the plus character (+) enables the feature and the minus character (-) disables the feature. You can also enable a feature or mode by specifying a feature or mode without the plus (+) character. Setting this switch will add or subtract features and modes from the settings in the Stats *modelsim.ini* variable.

#### Features

**all** — Display all statistics features (cmd, msg, perf, time). Mutually exclusive with none option. When specified in a string with other options, all is applied first.

**cmd** — (default) Echo the command line.

**msg** — (default) Display error and warning summary at the end of command execution.

**none** — Disable all statistics features. Mutually exclusive with all option. When specified in a string with other options, none is applied first.

**perf** — Display time and memory performance statistics.

**time** — (default) Display Start, End, and Elapsed times.

#### Modes

Modes can be set for a specific feature or globally for all features. To add or subtract a mode for a specific feature, specify using the plus (+) or minus (-) character with the feature, for example, vlog -stats=cmd+verbose,perf+list. To add or subtract a mode globally for all features, specify the modes in a comma-separated list, for example, vlog -stats=time,perf,list,-verbose. You cannot specify global and feature specific modes together.

**kb** — Print performance statistics in kilobyte units with no auto-scaling.

**list** — Display statistics in a Tcl list format when available.

**verbose** — Display verbose statistics information when available.

---

**Note**

---

 vlog -quiet disables all default or user-specified -stats features.

---

- -suppress <msg\_number>[,<msg\_number>,...]

(optional) Prevents the specified message(s) from displaying. The <msg\_number> is the number preceding the message you wish to suppress. You cannot suppress Fatal or Internal messages. Edit the [suppress](#) variable in the *modelsim.ini* file to set a permanent default. Refer to “[Message Severity Level](#)” for more information.

- -sv

(optional) Enables SystemVerilog features and keywords. By default ModelSim follows the IEEE Std 1364-2001 and ignores SystemVerilog keywords. If a source file has a ".sv" extension, ModelSim will automatically parse SystemVerilog keywords.

- -svext=[+|-]<extension>[,[+|-]<extension>]...[sceq]

(optional) Enables SystemVerilog language extensions through a comma-separated list of arguments.

[+ | -] — controls activation of the *extension*. Remember that arguments on the command line control the precedence; any settings to this switch will override your settings of the [SvExtensions](#) *modelsim.ini* variable.

+ — activates the *extension*.

- — deactivates the *extension*.

If you do not specify either a “+” or “-”, the command assumes you are activating the specified *extension*.

<extension> —

---

**Note**

---

 Multiple extensions are specivlog -svext=+feci,-uslt,pae as a comma-separated list. For example:

vlog -svext=+feci,-uslt,pae

---

acum — Specifies that the get(), try\_get(), peek(), and try\_peek() methods on an untyped mailbox will return successfully if the argument passed is assignment-compatible with the entry in the mailbox. The LRM-compliant behavior is to return successfully only if the argument and entry are of equivalent types.

arif — Allow the use of refs in fork-join\_any or fork-join\_none blocks inside tasks.

atpi — Use type names as port identifiers. Disabled when compiling with -pedanticerrors.

ared — Allows use of array reduction methods on multi-dimensional unpacked arrays, without the need of using a 'with' clause. A multi-dimensional unpacked array will be treated as if it had a single dimension [0:total\_number\_of\_elements-1].

**catx** — Allow an assignment of a single unsized constant in a concat to be treated as an assignment of 'default:val'.

**daoa** — Allows the passing a dynamic array as the actual argument of DPI open array output port. Without this option, a runtime error, similar to the following, is generated, which is compliant with LRM requirement.

```
# ** Fatal: (vsim-2211) A dynamic array cannot be passed as an
argument to the DPI import function 'impcall' because the formal 'o'
is an unsized output.
#   Time: 0 ns  Iteration: 0  Process: /top/#INITIAL#56 File:
dynarray.sv
# Fatal error in Module dynarray_sv_unit at dynarray.sv line 2
```

**defervda** — SV variables having an initializer in the declaration will trigger top-blocking always blocks at time zero.

**ddup** — (Drive Default Unconnected Port) Reverts behavior to where explicit named unconnected ports are driven by the default value of the port.

**evdactor** — enables early variable declaration assignments during class construction. The default behavior is to perform all superclass initialization before initializing any fields in a subclass.

**evis** — Supports the expansion of environment variables within curly braces ( {} ) within `include string literals and in `include path names. For example, if MYPATH exists in the environment then it will be expanded in the following:

```
`include "$MYPATH/inc.svh"
```

**feci** — Treat constant expressions in a foreach loop variable index as constant.

**fin0** — Treats \$finish() system call as \$finish(0), which results in no diagnostic information being printed.

**ias** — Iterate on always @\* evaluations until inputs settle. Typically, an always @\* block is not sensitive to events generated by executing the block itself. This argument increases the sensitivity of the block so that it will re-trigger if any input has changed since the last iteration of the always block.

**idcl** — Allows passing of import DPI call locations as implicit scopes.

**iddp** — Ignore the DPI task disable protocol check.

**ncref** — A ref argument in the new operator of a covergroup will not be treated as a constant, unless specified.

**pae** — Automatically export all symbols imported and referenced in a package.

**sccts** — Process string concatenations converting the result to string type.

**spsl** — (default) Search for packages in source libraries specified with -y and +libext.

**stop0** — Treats \$stop and \$stop() as \$stop(0), which results in no diagnostic information being printed.

**substr1** — Allows one argument in the builtin function substr. A second argument will be treated as the end of the string.

This extension runs a top-blocking always @\* at time zero, as is done for an always\_comb.

udm0 — Expands any undefined macro with the text “1'b0”.

uslt — (default) Promote unused design units found in source library files specified with the -y option to top-level design units.

vmctor — Allows virtual method calls in class constructor. The default is to treat them as non-virtual during construction.

### sceq

Allow string comparison with SystemVerilog case equality operator (==).

- -svfilesuffix=<extension>[,<extension>... ] <filename>

Allows specification of filename extensions for SystemVerilog files. Overrides the **SVFileSuffixes** variable in the *modelsim.ini* file for specified <filename>.

- -svinputport=net | var | relaxed

(optional) Used in conjunction with -sv to determine the default data type assigned to an input port declaration.

net — declares the port to be a net. This value enforces strict compliance to the Verilog LRM (IEEE Std 1364-2005), where the port declaration defaults to wire.

var — declares the port to be a variable. This value enforces behavior from previous releases, where the port declaration defaults to variable.

relaxed — (default) declares the port to be a net only if the type is a 4-state scalar or 4-state single dimensional vector. Otherwise, the port is declared a variable.

- -svpkgcasesens

(optional) Requires case-sensitive matching between SystemVerilog package import statements and package names.

- -sv05compat

Used in conjunction with the -sv switch to ensure compatibility with the reserved keyword set of IEEE Std 1800-2005.

- -sv09compat

Used in conjunction with the -sv switch to ensure compatibility with the reserved keyword set of IEEE Std 1800-2009.

- -sv12compat

Used in conjunction with the -sv switch to ensure compatibility with the reserved keyword set of IEEE Std 1800-2012.

- -timescale[=][ ]<time\_units>/<time\_precision>

(optional) Specifies the default timescale for all design unit types (modules, interfaces, programs, packages, checkers, and so forth) not having an explicit timescale directive in effect during compilation.

The format of the `-timescale` argument is the same as that of the ``timescale` directive. An equal sign (`=`) or whitespace is accepted between option and arguments in which case `<time_units> / <time_precision>` must be enclosed in quotation marks (`"`). The format for `<time_units>` and `<time_precision>` is `<n><units>`. The value of `<n>` must be 1, 10, or 100. The value of `<units>` must be fs, ps, ns, us, ms, or s. In addition, the `<time_precision>` must be smaller than or equal to the `<time_units>`. Refer to “[Simulator Resolution Limit \(Verilog\)](#)” for more information.

- `+typdelays`

Selects typical delays from the “`min:typ:max`” expressions. Default. If preferred, you can defer delay selection until simulation time by specifying the same option to the simulator.

- `-u`

(optional) Converts regular Verilog identifiers to uppercase. Allows case insensitivity for module names.

- `-v <library_file>`

(optional) Specifies a source library file containing module and UDP definitions. Refer to “[Verilog-XL Compatible Compiler Arguments](#)” for more information.

After all explicit filenames on the `vlog` command line have been processed, the compiler uses the `-v` option to find and compile any modules that were referenced but not yet defined. Modules and UDPs within the file are compiled only if they match previously unresolved references. Multiple `-v` options are allowed. See additional discussion in the examples.

- `-version`

(optional) Returns the version of the compiler as used by the licensing tools.

- `-vlog01compat`

(default) Ensures compatibility with rules of IEEE Std 1364-2001.

- `-vlog95compat`

(optional) Disables Verilog 2001 keywords, which ensures that code that was valid according to the 1364-1995 spec can still be compiled. By default ModelSim follows the rules of IEEE Std 1364-2001. Some requirements in 1364-2001 conflict with requirements in 1364-1995. Edit the `vlog95compat` variable in the `modelsim.ini` file to set a permanent default.

- `-vmake`

Generates a complete record of all command line data and files accessed during the compile of a design. This data is then used by the `vmake` command to generate a comprehensive makefile for recompiling the design library. By default, vcom stores compile data needed for the `-refresh` switch and ignores compile data not needed for `-refresh`. The `-vmake` switch forces inclusion of all file dependencies and command line data accessed during a compile, whether they contribute data to the initial compile or not. Executing this switch can increase compile time in addition to increasing the accuracy of the compile. See the `vmake` command for more information.

- **-warning <msg\_number>[,<msg\_number>,...]**  
(optional) Changes the severity level of the specified message(s) to "warning." Edit the [warning](#) variable in the *modelsim.ini* file to set a permanent default. Refer to "[Message Severity Level](#)" for more information.
- **-warning error**  
(optional) Reports all warnings as errors.
- **-warnrbw**  
(optional) Displays a warning when a variable is read before written in an always @\* block.
- **-work <library\_name>**  
(optional) Specifies a logical name or pathname of a library that is to be mapped to the logical library work. By default, the compiled design units are added to the work library. The specified pathname overrides the pathname specified for work in the project file.
- **-writetoplevels <fileName>**  
(optional) Records the names of all top level module names in a specified file. Also records any compilation unit name specified with -cuname. May only be specified when compiling the top level modules.

<fileName> — Required. Specifies the name of the file where module names are to be recorded.

- **-y <library\_directory>**  
(optional) Specifies a source library directory containing definitions for modules, packages, interfaces, and user-defined primitives (UDPs). Usually, this is a directory of source files that you want to scan if the compiled versions do not already exist in a library. Refer to "[Verilog-XL Compatible Compiler Arguments](#)" for more information.

After all explicit filenames on the vlog command line have been processed, the compiler uses the -y option to find and compile any modules that were referenced but not yet defined. Files within this directory are compiled only if the file names match the names of previously unresolved references. Multiple -y options are allowed. You will need to specify a file suffix by using -y in conjunction with the +libext+<suffix> option. See additional discussion in the examples.

---

**Note**

 Any -y arguments that follow a -refresh argument on a vlog command line are ignored. Any -y arguments that come before the -refresh argument on a vlog command line are processed.

---

- **<filename>**  
Specifies the name of the Verilog source code file to compile. One filename is required. Multiple filenames can be entered separated by spaces. Wildcards can be used.

## Examples

- Compile the Verilog source code contained in the file *example.vlg*.

**vlog example.vlg**

- After compiling *top.v*, vlog will scan the file *und1* for modules or primitives referenced but undefined in *top.v*. Only referenced definitions will be compiled.

**vlog top.v -v und1**

- After compiling *top.v*, vlog will scan the *vlog\_lib* library for files with modules with the same name as primitives referenced, but undefined in *top.v*. The use of *+libext+.v+.u* implies filenames with a *.v* or *.u* suffix (any combination of suffixes may be used). Only referenced definitions will be compiled.

**vlog top.v +libext+.v+.u -y vlog\_lib**

The *-work* option specifies *mylib* as the library to regenerate. *-refresh* rebuilds the library image without using source code, allowing models delivered as compiled libraries without source code to be rebuilt for a specific release of ModelSim.

- If your library contains VHDL design units, be sure to regenerate the library with the [vcom](#) command using the *-refresh* option as well. Refer to “[Regenerating Your Design Libraries](#)” for more information.

**vlog -work mylib -refresh**

- The *-incr* option determines whether or not the module source or compile options have changed as *module1.v* is parsed. If no change is found, the code generation phase is skipped. Differences in compile options are determined by comparing the compiler options stored in the *\_info* file with the compiler options given. They must match exactly.

**vlog module1.v -u -O0 -incr**

- The *-timescale* option specifies the default timescale for *module1.v*, which did not have an explicit timescale directive in effect during compilation. Quotes (" ") are necessary because the argument contains white spaces.

**vlog module1.v -timescale "1 ns / 1 ps"**

- Enable the display of Start, End, and Elapsed time as well as a message count summary. Echoing of the command line is disabled

**vlog -stats=time,-cmd,msg**

- The first *-stats* option is ignored. The *none* option disables all default settings and then enables the *perf* option.

**vlog -stats=time,cmd,msg -stats=none,perf**

# vmake

Requirement: This command does not work with the default format of the output from the [vlib](#) command, therefore you must add the -type directory argument to the vlib command line.

The vmake utility allows you to use a MAKE program to maintain individual libraries. You run vmake on a compiled design library. This utility operates on multiple source files per design unit; it supports Verilog include files as well as Verilog and VHDL PSL vunit files.

## **Note**

 If a design is spread across multiple libraries, then each library must have its own makefile and you must build each one separately.

## Syntax

```
vmake [-du <design_unit_name> ...] [-f <filename>] [-fullsrcpath] [-ignore] [<library_name>]  
[-modelsimini <path/modelsim.ini>]
```

## Description

By default, the output of vmake is sent to stdout—however, you can send the output to a makefile by using the shell redirect operator (>) along with the name of the file. You can then run the makefile with a version of MAKE (not supplied with ModelSim) to reconstruct the library. *This command must be invoked from either the system prompt.*

A MAKE program is included with Microsoft Visual C/C++, as well as many other program development environments.

After running the vmake utility, MAKE recompiles only the design units (and their dependencies) that have changed. You run vmake only once; then you can simply run MAKE to rebuild your design. If you add new design units or delete old ones, you should re-run vmake to generate a new makefile.

The vmake utility ignores library objects compiled with -nodebug.

This command provides additional information with the -help switch.

## Arguments

- **-du <design\_unit\_name>**  
(optional) Specifies that a vmake file will be generated only for the specified design unit. You can specify this argument any number of times for a single vmake command.
- **-f <filename>**  
(optional) Specifies a file to read command line arguments from.  
Refer to the section "[“Argument Files”](#) on page 27" for more information

- **-fullsrcpath**  
(optional) Produces complete source file paths within generated makefiles. By default, source file paths are relative to the directory in which compilations originally occurred. Use this argument to copy and evaluate generated makefiles within directories that are different from where compilations originally occurred.
- **-ignore**  
(optional) Omits a make rule for the named primary design unit and its secondary design units.
- **<library\_name>**  
(optional) Specifies the library name; if none is specified, then work is assumed.
- **-modelsimini <path/modelsim.ini>**  
(optional) Loads an alternate initialization file that replaces the current initialization file. Overrides the file path specified by the MODELSIM environment variable. Specifies either an absolute or relative path to the initialization file, including the file itself. On Windows systems, the path separator should be a forward slash (/).

## Examples

- To produce a makefile for the work library:  
**vmake >mylib.mak**
- To run vmake on libraries other than work:  
**vmake mylib >mylib.mak**
- To rebuild mylib, specify its makefile when you run MAKE:  
**make -f mylib.mak**
- To use vmake and MAKE on your work library:  
**C:\MIXEDHDL> vmake >makefile**
- To edit an HDL source file within the work library:  
**C:\MIXEDHDL> make**  
Your design gets recompiled for you. You can change the design again and re-run MAKE to recompile additional changes.
- To run vmake on libraries other than work:  
**C:\MIXEDHDL> vmake mylib >mylib.mak**
- To rebuild mylib, specify its makefile when you run MAKE:  
**C:\MIXEDHDL> make -f mylib.mak**

# vmap

The vmap command defines a mapping between a logical library name and a directory by modifying the *modelsim.ini* file.

## Syntax

```
vmap [-c | -del <logical_name> ... | <logical_name> [<path>] ]  
      [-modelsimini <path/modelsim.ini>]
```

## Description

With no arguments, vmap reads the appropriate *modelsim.ini* file(s) and prints to the transcript the current logical library to physical directory mappings.

This command provides additional information with the -help switch.

## Arguments

- **-c**  
(optional) Copies the default *modelsim.ini* file from the ModelSim installation directory to the current directory.  
This argument is intended only for making a copy of the default *modelsim.ini* file to the current directory. Do not use it while making your library mappings or the mappings may end up in the incorrect copy of the *modelsim.ini*.
- **-del <logical\_name> ...**  
(optional) Deletes the mapping specified by <logical\_name> from the current project file. You can specify multiple logical name arguments to the -del switch to delete multiple library mappings.
- **<logical\_name> [<path>]**  
(optional) Maps a logical library name to the specified physical library.  
If you do not specify <path> the command returns the current mapping for <logical\_name>.
- **-modelsimini <path/modelsim.ini>**  
(optional) Loads an alternate initialization file that replaces the current initialization file. Overrides the file path specified in the MODELSIM environment variable. Specifies either an absolute or relative path to the initialization file, including the file itself. On Windows systems the path separator should be a forward slash (/).

## Examples

- Map two logical libraries to the physical library “work”:

```
vlib work  
vmap library1 work  
vmap library2 work
```

- Display information about the logical library “library1”:

**vmap library1**

- Delete the logical library mappings:

**vmap -del library1 library2**

# vsim

The vsim command invokes the VSIM simulator, which you can use to view the results of a previous simulation run (when invoked with the -view argument)

## Syntax

This section lists all arguments of the vsim command in alphabetical order.

The Arguments section groups the argument descriptions into the following categories:

---

### Note

---

#### Argument Groups

- [All languages](#)
- [VHDL Arguments](#)
- [Verilog Arguments](#)
- [Object Arguments](#)

---

vsim [options]

[options]:

[\[-accessobjdebug | -noaccessobjdebug\]](#) [\[+alt\\_path\\_delays\]](#) [\[-assertfile <filename>\]](#)

[\[-batch\]](#) [\[+bitblast\[=iopath | tcheck\]\]\]](#)

[\[-c\]](#) [\[-capacity\[=line\]\]](#) [\[-checkvifacedrivers\]](#) [\[-classdebug | -noclassdebug\]](#) [\[-colormap new\]](#)

[\[-default\\_radix <radix>\]](#) [\[-defaultstdlogicinit0z\]](#) [\[+delayed\\_timing\\_checks\]](#) [\[-display <display\\_spec>\]](#)

[\[-displaymsgmode both | tran | wlf\]](#) [\[-do "<command\\_string>"\]](#) [\[<do\\_file\\_name>\]](#)

[\[-donotcollapsepartiallydriven\]](#) [\[-dpiforceheader\]](#) [\[-dipiheader\]](#) [\[-dpilib <libname>\]](#)

[\[-dipioutoftheblue 0 | 1 | 2\]](#) [\[+dumports+collapse\]](#) [\[+dumports+nocollapse\]](#)

[\[+dumports+direction\]](#) [\[+dumports+no\\_strength\\_range\]](#) [\[+dumports+unique\]](#)

[\[-error <msg\\_number>\[,<msg\\_number>,...\]\]](#)

[\[-enumfirstinit\]](#)

[\[-f <filename>\]](#) [\[-fatal <msg\\_number>\[,<msg\\_number>,...\]\]](#) [\[-fsmdebug\]](#)

[\[-g <Name>=<Value> ...\]](#) [\[-G<Name>=<Value> ...\]](#) [\[-gblso <shared\\_obj>\[,<shared\\_obj>\]\]](#)

[\[-gconrun | -nogconrun\]](#) [\[-gconstep | -nogconstep\]](#) [\[-gcthreshold <n>\]](#)

[\[-geometry <geometry\\_spec>\]](#) [\[-gui\]](#)

[\[-hazards\]](#) [\[-help\]](#)

[\[-i\]](#) [\[-ignoreinilibs\]](#) [\[+initregNBA | +noinitregNBA\]](#)

[\[-installcolormap\]](#)

---

[-keeploaded] [-keeploadedrestart] [-keepstdout]  
 [-logfile <filename> | -l <filename> | -nolog] [-L <library\_name> ...]  
 [-lib <libname>] [<library\_name>.<design\_unit>]  
 [-Ldir <pathname> [<pathname> ...]]  
 [-Lf <library\_name> ...]  
  
 [-modelsimini <path/modelsim.ini>]  
 [-msgfile <filename>] [-msglimit [all, | none,] <msgNumber>[, <msgNumber>,...]]  
 [-msglimitcount <limit\_value> -msglimit [all, | none,] <msgNumber>[, <msgNumber>,...]]  
 [-msgmode both | tran | wlf]  
 [-multisource\_delay min | max | latest] [+multisource\_int\_delays]  
  
 [-name <name>] [-noappendclose] [+no\_autodtc] [-noautoldlibpath] [-nodpiexports]  
 [+no\_cancelled\_e\_msg] [+no\_glitch\_msg] [+no\_neg\_tchk] [+no\_notifier] [+no\_path\_edge]  
 [+no\_pulse\_msg] [-no\_risefall\_delaynets]  
 [+no\_show\_cancelled\_e] [+no\_tchk\_msg] [-nocollapse] [-nocapacity] [-nocompress]  
 [-nofileshare] [-noimmedca] [-noglitch][+nosdferror] [+nosdfwarn] [+nospecify] [-nostdout]  
 [-note <msg\_number>[, <msg\_number>,...]] [+notifier\_ondetect] [+notimingchecks |  
 +ntcnatchks] [-novhdlvariablelogging] [+nowarnBSOB] [+nowarn<CODE | number>]  
 [-nowiremodelforce] [+ntc\_warn] [+ntcnatchks]  
  
 [-oldvhdlforgennames] [-onfinish ask | stop | exit | final] [-optionset <optionset\_name>]  
  
 [-pduignore[=<instpath>]] [-pedanticerrors] [-permissive]  
 [-permit\_unmatched\_virtual\_intf] [-pli "<object list>"]  
 [+<plusarg>] [-postsimdataflow]  
 [-printsimstats[=<val>][v]] [+pulse\_e/<percent>] [+pulse\_e\_style\_ondetect]  
 [+pulse\_e\_style\_onevent] [+pulse\_r/<percent>] [+pulse\_int\_e/<percent>]  
 [+pulse\_int\_r/<percent>]  
  
 [-quiet]  
  
 [-runinit]  
  
 [+sdf\_iopath\_to\_prim\_ok] [+sdf\_nocheck\_celltype]  
 [-sdfmin | -sdftyp | -sdfmax[@<delayScale>] [<instance>=<sdf\_filename>]  
  
 [-sdfmaxerrors <n>] [-sdfnoerror] [-sdfnowarn] [-sdfreport=<filename>]  
 [+sdf\_report\_unannotated\_insts] [+sdf\_verbose] [-showlibsearchpath] [-stackcheck] [-  
 std\_input <filename>] [-std\_output <filename>] [+show\_cancelled\_e]  
 [-stats [=+ | -]<feature>[,[+ | -]<mode>]]  
 [-strictvital] [-suppress <msg\_number>[, <msg\_number>,...]] [-sv\_lib <shared\_obj>]  
 [-sv\_liblist <filename>] [-sv\_root <dirname>] [-sync]  
 [-syncio | -nosyncio]

```
[-t [<multiplier>]<time_unit>] [-tab <tabfile>] [-tag <string>] [-title <title>]
  [-trace_foreign <int>] [+transport_int_delays]
  [+transport_path_delays]

[-undefsyms={<args>}] [-uvmcontrol={<args>}]

[-v2k_int_delays][-vcdstim [<instance>=]<filename>]
  [-version] [-vhdlmergepdupackage] [-vhdlseparatepdupackage]
  [-vhdlvariablelogging] [-view [<alias_name>=]<WLF_filename>]
  [-visual <visual>][-vital2.2b]

[-warning <msg_number>[,<msg_number>,...]] [-warning error] [-wlf <file_name>]
  [-wlfcachesize <n>] [-wlfcollapsesdelta] [-wlfcollapsetime] [-nowlfcollapse]
  [-wlfcollapse] [-nowlfcollapse] [-wlfdelteonquit] [-nowlfdelteonquit]
  [-wlflock] [-nowlflock] [-nowlfopt] [-wlfsimcachesize <n>]
  [-wlfslim <size>] [-wlftlim <duration>] [-work <pathname>] [-wrealdefaultzero]
```

## Description

You can simulate a VHDL configuration or an entity/architecture pair, a Verilog module or configuration.

If you specify a VHDL configuration, it is invalid to specify an architecture. During elaboration, ModelSim determines if the source has been modified since the last compile.

You can use this command in batch mode from the Windows command prompt. Refer to “[Batch Mode](#)” for more information on the VSIM batch mode.

To manually interrupt design loading, use the Break key or press <Ctrl-C> from a shell.

You can invoke vsim from a command prompt or in the Transcript window of the Main window. You can also invoke it from the GUI by selecting Simulate > Start Simulation.

Package names may be used at the command line and will be treated as top-level design units.

All arguments to the vsim command are case-sensitive; for example, -g and -G are not equivalent.

## Arguments

### All languages

- **-assertfile <filename>**

(optional) Designates an alternative file for recording VHDL assertion messages.

An alternate file may also be specified by the [AssertFile](#) *modelsim.ini* variable. By default, assertion messages are output to the file specified by the [TranscriptFile](#) variable in the *modelsim.ini* file. Refer to “[Creating a Transcript File](#)” for more information.

- **-batch**

(optional) Runs scripted batch simulations via the `-do` argument to `vsim`. Must be specified from a Windows command prompt or a UNIX terminal. The simulator returns an error if `-batch` is used with the `-c`, the `-gui`, or the `-i` argument to `vsim`. You can edit the [BatchMode](#) `modelsim.ini` variable to automatically run in batch mode when none of `-c`, `-gui`, or `-i` are used.

By default, `vsim -batch` prevents automatic creation of a transcript file by disabling the [TranscriptFile](#) `modelsim.ini` variable and sending transcript data to `stdout`. You can create a transcript file by specifying the `-logfile <filename>` argument to `vsim` or by uncommenting the [BatchTranscriptFile](#) `modelsim.ini` variable. You can also disable sending transcript data to `stdout` by specifying `vsim -nostdout` however, you must then save transcript data to a file. Refer to “[Batch Mode](#)” for more information about saving transcript data.

- **+bitblast[=[iopath | tcheck]]**

(optional) Enables bit-blasting of specify block iopaths and timing checks (tchecks) with wide atomic ports. Without the optional qualifiers, this argument operates on both specify paths and tchecks. The qualifiers work as follows:

`+bitblast=iopath` — bit-blasts only specify paths with wide ports.

`+bitblast=tcheck` — bit-blasts only tchecks with wide ports.

This argument is intended for use with applications employing SDF annotation.

- **-c**

(optional) Specifies that the simulator is to be run in command-line mode. Refer to “[General Modes of Operation](#)” for more information.

- **-capacity[=line]**

(optional) Enables the fine-grain analysis display of memory capacity. (The default is a coarse-grain analysis display.) The “`=line`” option allows the point of allocation to be generated along with the point of declaration.

- **-colormap new**

(optional) Specifies that the window should have a new private colormap instead of using the default colormap for the screen.

- **-default\_radix <radix>**

(optional) Sets the default radix for the simulation and overrides the [DefaultRadix](#) preference variable. `<radix>` may be any of the following: ascii, binary, decimal, hexadecimal, octal, symbolic, unsigned.

- **-defaultstdlogicinittoz**

(optional) Sets the default VHDL initialization of `std_logic` to "Z" (high impedance) for ports of type OUT and INOUT. IEEE Std 1076-1987 VHDL Language Reference Manual (LRM) compliant behavior is for `std_logic` to initialize to "U" (uninitialized) which is incompatible with the behavior expected by synthesis and hardware.

- **-display <display\_spec>**

(optional) Specifies the name of the display to use. Does not apply to Windows platforms.

For example:

```
-display :0
```

- **-displaymsgmode both | tran | wlf**

(optional) Controls the transcription of \$display system task messages to the transcript and/or the Message Viewer. Refer to the section "[Message Viewer Window](#)" in the User's Manual for more information and the [displaymsgmode .ini](#) file variable.

both — outputs messages to both the transcript and the WLF file.

tran — outputs messages only to the transcript, therefore they are not available in the Message Viewer. Default behavior.

wlf — outputs messages only to the WLF file/Message Viewer, therefore they are not available in the transcript.

The display system tasks displayed with this functionality include: \$display, \$strobe, \$monitor, \$write as well as the analogous file I/O tasks that write to STDOUT, such as \$fwrite or \$fdisplay.

- **-do "<command\_string>" | <do\_file\_name>**

(optional) Instructs vsim to use the command(s) specified by <command\_string> or the DO file named by <do\_file\_name> rather than the startup file specified in the .ini file, if any. Multiple commands can be specified as a semi-colon (;) separated list. You can also specify multiple instances of -do "<command\_string>" on the same command line. The commands are joined together in the order specified.

For example:

```
vsim -do "force clk 0 0, 1 10 -r 20" top -wlf top.wlf /  
-do "testfile.do" -do "run -all"
```

will turn into the following script:

```
"force clk 0 0, 1 10 -r 20; do testfile.do; run -all"
```

You can include nested vsim-do operations. A vsim command do-file that contains another vsim command with its own do-file executes the nested do-file.

- **-donotcollapsepartiallydriven**

(optional) Prevents the collapse of partially driven and undriven output ports during optimization. Prevents incorrect values that can occur when collapsed.

- **+dumports+collapse | +dumports+nocollapse**

(optional) Determines whether vectors (VCD id entries) in dumports output are collapsed or not. The default behavior is collapsed, and can be changed by setting the [DumportsCollapse](#) variable in the *modelsim.ini* file.

- **+dumpports+direction**  
(optional) Modifies the format of extended VCD files to contain direction information.
- **+dumpports+no\_strength\_range**  
(optional) Ignores strength ranges when resolving driver values for an extended VCD file. This argument is an extension to the IEEE 1364 specification. Refer to “[Resolving Values](#)” for additional information.
- **+dumpports+unique**  
(optional) Generates unique VCD variable names for ports in a VCD file even if those ports are connected to the same collapsed net.
- **-enumfirstinit**  
(optional) Initializes enum variables in SystemVerilog using the leftmost value as the default. You must also use the argument with the vlog command in order to implement this initialization behavior. Specify the EnumBaseInit variable as 0 in the *modelsim.ini* file to set this as a permanent default.
- **-error <msg\_number>[,<msg\_number>,...]**  
(optional) Changes the severity level of the specified message(s) to "error." Edit the [error](#) variable in the *modelsim.ini* file to set a permanent default. Refer to “[Message Severity Level](#)” for more information.
- **-f <filename>**  
(optional) Specifies a file with more vsim command arguments. Allows complex argument strings to be reused without retyping.  
Refer to the section ““[Argument Files](#)” on page 27” for more information.
- **-fatal <msg\_number>[,<msg\_number>,...]**  
(optional) Changes the severity level of the specified message(s) to "fatal." Edit the [fatal](#) variable in the *modelsim.ini* file to set a permanent default. Refer to “[Message Severity Level](#)” for more information.
- **-fsmdebug**  
(optional) Enables visualization of FSMs in the GUI. You must specify this argument to view FSM information the GUI.
- **-g <Name>=<Value> ...**  
(optional) Assigns a value to all specified VHDL generics and Verilog parameters that have not received explicit values in generic maps, instantiations, or from defparams (such as top-level generics/parameters and generics/parameters that would otherwise receive their default values).  
Multiple -g options are allowed, one for each generic/parameter, specified as a space separated list.

**<Name>** — Name of a generic/parameter, exactly as it appears in the VHDL source (case is ignored) or Verilog source. Name may be prefixed with a relative or absolute hierarchical path to select generics in an instance-specific manner. For example, specifying `-g/top/u1/tpd=20ns` on the command line would affect only the *tpd* generic on the */top/u1* instance, assigning it a value of 20ns. Specifying `-gu1/tpd=20ns` affects the *tpd* generic on all instances named *u1*. Specifying `-gtpd=20ns` affects all generics named *tpd*.

**<Value>** — Specifies an appropriate value for the declared data type of a VHDL generic or any legal value for a Verilog parameter. Make sure the value you specify for a VHDL generic is appropriate for VHDL declared data types. Integers are treated as signed values. For example, `-gp=-10` overwrites the parameter *p* with the signed value of -10.

If more than one `-g` option selects a given generic the most explicit specification takes precedence. For example,

```
vsim -g/top/ram/u1/tpd_hl=10ns -gtpd_hl=15ns top
```

This command sets *tpd\_hl* to 10ns for the */top/ram/u1* instance. However, all other *tpd\_hl* generics on other instances will be set to 15ns.

**Limitation:** In general, generics/parameters of composite type (arrays and records) cannot be set from the command line. However, you can set string arrays, std\_logic vectors, and bit vectors if they can be set using a quoted string. For example,

```
-gstrgen="This is a string"  
-gslv="01001110"
```

The quotation marks (" ") must make it into vsim as part of the string because the type of the value must be determinable outside of any context. Therefore, when entering this command from a shell, add single quotes (' ') around the string. For example:

```
-gstrgen='This is a string'
```

If working within the ModelSim GUI, you would enter the command as follows:

```
{-gstrgen="This is a string"}
```

You can also enclose the value escaped quotes (\\"), for example:

```
-gstrgen=\\"This is a string\\\"
```

- **-G<Name>=<Value> ...**

(optional) Same as `-g` (see above) except that it will also override generics/parameters that received explicit values in generic maps, instantiations, or from defparams.

This argument is the only way for you to alter the generic/parameter, such as its length, (other than its value) after the design has been loaded.

**<Name>** — Name of a generic/parameter, exactly as it appears in the VHDL source (case is ignored) or Verilog source. Name may be prefixed with a relative or absolute hierarchical path to select generics in an instance-specific manner. For example, specifying `-G/top/u1/tpd=20ns` on the command line would affect only the *tpd* generic on the */top/u1* instance,

assigning it a value of 20ns. Specifying -Gu1/tpd=20ns affects the *tpd* generic on all instances named *u1*. Specifying -Gtpd=20ns affects all generics named *tpd*.

<Value> — Specifies an appropriate value for the declared data type of a VHDL generic or any legal value for a Verilog parameter. Make sure the value you specify for a VHDL generic is appropriate for VHDL declared data types. Integers are treated as signed values. For example, -Gp=-10 overwrites the parameter p with the signed value of -10.

- -gblso <shared\_obj>[,<shared\_obj>]

(optional) Open the specified shared object(s) with global symbol visibility. Essentially all data and functions are exported from the specified shared object and are available to be referenced and used by other shared objects. If you specify multiple, comma-separated, shared objects, they will be merged internally and then loaded as a single shared object. You can also specify this argument with the [GlobalSharedObjectsList](#) variable in the *modelsim.ini* file.

- -geometry <geometry\_spec>

(optional) Specifies the size and location of the main window. Where <geometry\_spec> is of the form:

WxH+X+Y

- -gui

(optional) Starts the ModelSim GUI without loading a design and redirects the standard output (stdout) to the GUI Transcript window.

- -help

(optional) Sends the arguments and syntax for vsim to the transcript.

- -i

(optional) Specifies that the simulator be run in interactive mode.

- +initregNBA | +noinitregNBA

(optional) Controls whether +initreg settings applied to registers of sequential UDPs should be non-blocking. This is useful when continuous assignments overwrite register initialization.

+initregNBA — (default) enables this functionality.

+noinitregNBA — disables this functionality.

- -installcolormap

(optional) For UNIX only. Causes vsim to use its own colormap so as not to hog all the colors on the display. This is similar to the -install argument on Netscape.

- -keeploaded

(optional) Prevents the simulator from unloading/reloading any HDL interface shared libraries when it restarts or loads a new design. The shared libraries will remain loaded at

their current positions. User application code in the shared libraries must reset its internal state during a restart in order for this to work effectively.

- **-keeploadedrestart**

(optional) Prevents the simulator from unloading/reloading any HDL interface shared libraries during a restart. The shared libraries will remain loaded at their current positions. User application code in the shared libraries must reset its internal state during a restart in order for this to work effectively.

We recommend using this option if you'll be doing warm restores after a restart and the user application code has set callbacks in the simulator. Otherwise, the callback function pointers might not be valid if the shared library is loaded into a new position.

- **-keepstdout**

(optional) For use with foreign programs. Instructs the simulator to not redirect the stdout stream to the Main window.

- **-logfile <filename> | -l <filename> | -nolog**

(optional) Controls saving of transcript data during batch and regular simulations.

**-logfile <filename>** — Saves transcript data to <filename>. Can be abbreviated to **-l <filename>**. Overrides the default transcript file creation set with the [TranscriptFile](#) or [BatchTranscriptFile](#) *modelsim.ini* variables. You can also specify “stdout” or “stderr” for <filename>.

**-nolog** — Disables transcript file creation. Overrides the [TranscriptFile](#) or [BatchTranscriptFile](#) variables set in the *modelsim.ini* file.

Refer to “[Batch Mode](#)” for more information about saving transcript data.

- **-L <library\_name> ...**

(optional) Specifies the library to search for top level design units instantiated from Verilog and for VHDL default component binding. Prints a list of all visible top level libraries if a top level design unit cannot be found. Refer to “[Library Usage](#)” for more information. If multiple libraries are specified, each must be preceded by the **-L** option. Libraries are searched in the order in which they appear on the command line.

- **-Ldir <pathname> [<pathname> ...]**

(optional) Passes one or more container folders for libraries specified by either `vsim -L` or `vsim -Lf`. Once you specify a container folder (pathname), the libraries contained in this folder can be directly referenced using their logical names. When you specify multiple values for pathname, ModelSim searches in the order in which these paths are specified on the command line.

---

**Note**

 The current working directory (\$cwd) is always searched before any pathnames you specify for **-Ldir**. That is, ModelSim searches as if there was an implicit `vsim -Ldir` specified first on the command line.

---

---

- **-Lf <library\_name> ...**  
(optional) Same as -L but libraries are searched before `uselib directives. Refer to “[Library Usage](#)” for more information.
- **-lib <libname>**  
(optional) Specifies the default working library where vsim will look for the design unit(s). Default is "work".
- **-libverbose[=prlib]**  
(optional) Enables verbose messaging about library search and resolution operations. The “=prlib” option will print out the -L or -Lf option that was used to locate each design unit loaded by vsim. This information is printed to the right of the existing “Loading design unit xyz...” messages.  
Libraries containing top design units that are not explicitly present in the set of -L/-Lf options are implicitly promoted to searchable libraries at the end of the library search order. They will appear as -Ltop in the output of the -libverbose option. To stop creation of -Ltop libraries, use the -noltop argument of vsim.
- **-modelsimini <path/modelsim.ini>**  
(optional) Loads an alternate initialization file that replaces the current initialization file. Overrides the file path specified in the MODELSIM environment variable. Specifies either an absolute or relative path to the initialization file, including the file itself. On Windows systems the path separator should be a forward slash (/).
- **-msgfile <filename>**  
(optional) Designates an alternative file for recording error messages. An alternate file may also be specified by the [ErrorFile](#) modelsim.ini variable. By default, error messages are output to the file specified by the [TranscriptFile](#) variable in the *modelsim.ini* file (refer to “[Creating a Transcript File](#)”).
- **-msglimit [all, | none,] <msgNumber>[, <msgNumber>,...]**  
(optional) Limits the number of iterations of the specified message(s) to five (the default count) then suppresses all new instances. Refer to “[Suppression of Warning Messages](#)” for more information.
  - all — Limits all messages to the default count except the specified msgNumber(s).
  - none — Limits only the specified msgNumber(s) to the default count.
  - <msgNumber>[, <msgNumber>,...] — Specifies the message number(s) to limit to five iterations. Multiple messages are specified as a comma-separated list.

For example:

`vsim -msglimit all, <msgID1>, <msgID2>`

Limits all messages to the default count except *msgID1* and *msgID2*.

`vsim -msglimit none, <msgID1>, <msgID2>`

Limits only *msgID1* and *msgID2* to the default count.

- `-msglimitcount <limit_value> -msglimit [all, | none,] <msgNumber>[, <msgNumber>...]`  
(optional) Limits the reporting of listed messages to user-defined limit\_value. Overrides the **MsgLimitCount** variable in the *modelsim.ini* file.
- `-msgmode both | tran | wlf`  
(optional) Specifies the location(s) for the simulator to output elaboration and runtime messages.
  - both — outputs messages to both the transcript and the WLF file.
  - tran — (default) outputs messages only to the transcript, therefore they are not available in the Message Viewer.
  - wlf — outputs messages only to the WLF file/Message Viewer windows, therefore they are not available in the transcript.

Refer to the section "[Message Viewer Window](#)" in the User's Manual for more information.

- `-multisource_delay min | max | latest`  
(optional) Controls the handling of multiple PORT or INTERCONNECT constructs that terminate at the same port. By default, the Module Input Port Delay (MIPD) is set to the max value encountered in the SDF file. Alternatively, you may choose the min or latest of the values. If you have a Verilog design and want to model multiple interconnect paths independently, use the `+multisource_int_delays` argument.
- `+multisource_int_delays`  
(optional) Enables multisource interconnect delay with pulse handling and transport delay behavior. Works for both Verilog and VITAL cells.  
Use this argument when you have interconnect data in your SDF file and you want the delay on each interconnect path modeled independently. Pulse handling is configured using the `+pulse_int_e` and `+pulse_int_r` arguments (described below).  
The `+multisource_int_delays` argument cannot be used if you compiled using the `-novital` argument to vcom. The `-novital` argument instructs vcom to implement VITAL functionality using VHDL code instead of accelerated code, and multisource interconnect delays cannot be implemented purely within VHDL.
- `-name <name>`  
(optional) Specifies the application name used by the interpreter for send commands. This does not affect the title of the window.
- `-noautoldlibpath`  
(optional) Disables the default internal setting of LD\_LIBRARY\_PATH, enabling you to set it yourself. Use this argument to make sure that LD\_LIBRARY\_PATH is not set automatically while you are using the GUI,
- `-nocapacity`  
(optional) Disables the display of both coarse-grain and fine-grain analysis of memory capacity.

- **-nocompress**  
(optional) Causes VSIM to create uncompressed checkpoint files. This option may also be specified with the [CheckpointCompressMode](#) variable in the *modelsim.ini* file.
- **-noimmedca**  
(optional) Causes Verilog event ordering to occur without enforced prioritization—continuous assignments and primitives are not run before other normal priority processes scheduled in the same iteration. Use this argument to prevent the default event ordering where continuous assignments and primitives are run with “immediate priority.” You may also set even ordering with the [ImmediateContinuousAssign](#) variable in the *modelsim.ini* file.
- **+no\_notifier**  
(optional) Disables the toggling of the notifier register argument of all timing check system tasks. By default, the notifier is toggled when there is a timing check violation, and the notifier usually causes a UDP to propagate an X. This argument suppresses X propagation in both Verilog and VITAL for the entire design.
- **+nospecify**  
(optional) Disables specify path delays and timing checks in Verilog.
- **-nostdout**  
(optional) Directs all output to the transcript only when in command line and batch mode. Prevents duplication of I/O between the shell and the transcript file. Has no affect on interactive GUI mode. Refer to “[Batch Mode](#)” for information about batch mode usage.
- **+no\_tchk\_msg**  
(optional) Disables error messages generated when timing checks are violated. For Verilog, it disables messages issued by timing check system tasks. For VITAL, it overrides the `MsgOn` arguments and generics.  
Notifier registers are still toggled and may result in the propagation of Xs for timing check violations.
- **-note <msg\_number>[,<msg\_number>,...]**  
(optional) Changes the severity level of the specified message(s) to “note.” Edit the `note` variable in the *modelsim.ini* file to set a permanent default. Refer to “[Message Severity Level](#)” for more information.
- **+notifier\_ondetect**  
(optional) Causes timing check notifier toggle generated X output values to be scheduled with zero delay.
- **+notimingchecks | +ntcnotchks**  
(optional) Disables Verilog timing checks. (This option sets the generic `TimingChecksOn` to FALSE for all VHDL Vital models with the `Vital_level0` or `Vital_level1` attribute. Generics with the name `TimingChecksOn` on non-VITAL models are unaffected.) By default, Verilog

timing check system tasks (\$setup, \$hold,...) in specify blocks are enabled. For VITAL, the timing check default is controlled by the ASIC or FPGA vendor, but most default to enabled.

Additionally, +ntcnotchks maintains the delay net delays necessitated by negative timing check limits. For this reason when using +ntcnotchks it is necessary to SDF annotate all timing check values.

- **-nowiremodelforce**  
(optional) Restores the **force** command to previous usage (prior to version 10.0b) where an input port cannot be forced directly if it is mapped at a higher level in VHDL and mixed models. Signals must be forced at the top of the hierarchy connected to the input port.
- **-optionset <optionset\_name>**  
(optional) Calls an optionset as defined in the *modelsim.ini* file. Refer to the section ““[Optionsets](#)” on page 26” in the Reference Manual for more information.
- **-pduignore[=<instpath>]**  
Ignore Preoptimized Design Unit (black-box). If <instpath> is not specified, all PDUs found in compiled libraries will be ignored. Otherwise, the PDU specified by <instpath> will be ignored. You can specify this argument multiple times using different values of <instpath>. Equivalent to the deprecated -ignore\_bbox argument.
- **-permissive**  
(optional) Allows messages in the LRM group of error messages to be downgraded to a warning.

You can produce a complete list of error messages by entering the following command:

```
verror -kind vsim -permissive
```

- **-postsimdataflow**  
(optional) Makes Dataflow window available for post simulation debug operations. By default, the Dataflow window is not available for post-sim debug.
- **-printsimstats[=<val>][v]**  
(optional) Prints the output of the **simstats** command to the transcript at the end of simulation before exiting. <val> is 0 - disables simstats, 1(default) - prints stats at the end of simulation,2 - prints out stats at the end of each run command and simulation. v- prints out verbose statistics, including the checkout time.

Each performance statistic is printed with its related units on a separate line. Edit the **PrintSimStats** variable in the *modelsim.ini* file to set the simulation to print the simstats data by default.

The command **vsim -printsimstats=v** is equivalent to **vsim -stats=perf+verbose**. The command **vsim -printsimstats=2v** is equivalent to **vsim -stats=perf+verbose+eor**.

---

- `+pulse_int_e/<percent>`

(optional) Controls how pulses are propagated through interconnect delays, where `<percent>` is a number between 0 and 100 that specifies the error limit as a percentage of the interconnect delay. Used in conjunction with `+multisource_int_delays` (see above). This option works for both Verilog and VITAL cells, though the destination of the interconnect must be a Verilog cell. The source may be VITAL or Verilog.

A pulse greater than or equal to the error limit propagates to the output in transport mode (transport mode allows multiple pending transitions on an output). A pulse less than the error limit and greater than or equal to the rejection limit (see `+pulse_int_r/<percent>` below) propagates to the output as an X. If the rejection limit is not specified, then it defaults to the error limit. For example, consider an interconnect delay of 10 along with a `+pulse_int_e/80` option. The error limit is 80% of 10 and the rejection limit defaults to 80% of 10. This results in the propagation of pulses greater than or equal to 8, while all other pulses are filtered.

- `+pulse_int_r/<percent>`

(optional) Controls how pulses are propagated through interconnect delays, where `<percent>` is a number between 0 and 100 that specifies the rejection limit as a percentage of the interconnect delay. This option works for both Verilog and VITAL cells, though the destination of the interconnect must be a Verilog cell. The source may be VITAL or Verilog.

A pulse less than the rejection limit is filtered. If the error limit is not specified by `+pulse_int_e` then it defaults to the rejection limit.

- `-quiet`

(optional) Disable 'Loading' messages during batch-mode simulation.

- `-runinit`

(optional) Initializes non-trivial static SystemVerilog variables, for example expressions involving other variables and function calls, before displaying the simulation prompt.

- `+sdf_iopath_to_prim_ok`

(optional) Prevents vsim from issuing an error when it cannot locate specify path delays to annotate. If you specify this argument, IOPATH statements are annotated to the primitive driving the destination port if a corresponding specify path is not found. Refer to “[SDF to Verilog Construct Matching](#)” for additional information.

- `-sdfmin | -sdftyp | -sdfmax[@<delayScale>] [<instance>=<sdf_filename>]`

(optional) Annotates VITAL or Verilog cells in the specified SDF file (a Standard Delay Format file) with minimum, typical, or maximum timing. Can also specify instances under VHDL generates as the SDF back-annotation point.

`@<delayScale>` — scales all values by the specified value. For example, if you specify `-sdfmax@1.5`, all maximum values in the SDF file are scaled to 150% of their original value.

`<instance>=` — specifies a specific instance for the associated SDF file. Use this when not performing backannotation at the top level.

`<sdf_filename>` — specifies the file containing the SDF information.

- `-sdfmaxerrors <n>`  
(optional) Controls the number of Verilog SDF missing instance messages to be generated before terminating vsim. `<n>` is the maximum number of missing instance error messages to be emitted. The default number is 5.
- `-sdfnoerror`  
(optional) Errors issued by the SDF annotator while loading the design prevent the simulation from continuing, whereas warnings do not. Changes SDF errors to warnings so that the simulation can continue.
- `-sdfnowarn`  
(optional) Disables warnings from the SDF reader. Refer to “[VHDL Simulation](#)” for an additional discussion of SDF.
- `-sdfoutreport=<filename>`  
(optional) Produces a report at the location of `<filename>` containing information about unannotated and partially-annotated specify path objects, specifically path delays and timing checks. Refer to the section “[Reporting Unannotated Specify Path Objects](#)” in the User’s Manual for more information.
- `+sdf_report_unannotated_insts`  
(optional) Enables error messages for any un-annotated Verilog instances with specify blocks or VHDL instances with VITAL timing generics that are under regions of SDF annotation.
- `+sdf_verbose`  
(optional) Turns on the verbose mode during SDF annotation. The Transcript window provides detailed warnings and summaries of the current annotation as well as information including the module name, source file name, and line number. When the `+multisource_int_delays` argument is also specified, the `+sdf_verbose` argument causes the output from the `write timing` command to contain more detail.
- `-stackcheck`  
(optional) Enables runtime stack usage sanity checking. This argument causes vsim to add additional instrumentation at runtime to monitor the system stack usage. If the usage exceeds the reserved stack limit, vsim will report a fatal error. An uncaught stack overflow will lead to a potentially mysterious and random downstream crash.
- `-stats [=+ | -]<feature>[,[+ | -]<mode>]`  
(optional) Controls display of statistics sent to a logfile, stdout, or the transcript. Specifying `-stats` without options sets the default features (cmd, msg, and time).

Multiple features and modes for each instance of -stats are specified as a comma separated list. You can specify -stats multiple times on the command line, but only the last instance takes effect.

You can specify -printsimstats and -stats on the same command line, however -stats will always override -printsimstats regardless of the order in which the options are specified.

[+ | -] — Controls activation of the feature or mode where the plus character (+) enables the feature and the minus character (-) disables the feature. You can also enable a feature or mode by specifying a feature or mode without the plus (+) character. Setting this argument will add or subtract features and modes from the default settings "cmd,msg,time".

#### Features

all — Display all statistics features (cmd, msg, perf, time). Mutually exclusive with none option. When specified in a string with other options, all is applied first.

cmd — (default) Echo the command line.

msg — (default) Display error and warning summary at the end of command execution.

none — Disable all statistics features. Mutually exclusive with all option. When specified in a string with other options, none is applied first.

perf — Display time and memory performance statistics.

time — (default) Display Start, End, and Elapsed times.

#### Modes

Modes can be set for a specific feature or globally for all features. To add or subtract a mode for a specific feature, specify using the plus (+) or minus (-) character with the feature, for example, vcover dump -stats=cmd+verbose,perf+list. To add or subtract a mode globally for all features, specify the modes in a comma-separated list, for example, vcover dump -stats=time,perf,list,-verbose. You cannot specify global and feature specific modes together.

eor — Print performance statistics at the end of each run command. Valid for use only with perf feature (-stats=perf+eor); it is invalid with other features.

kb — Print statistics in kilobyte units with no auto-scaling.

list — Display statistics in a Tcl list format when available.

verbose — Display verbose statistics information when available.

---

#### **Note**

 By default, vsim prints the command line with the '-f filename' option. Prior to 10.3c, behavior was to print the command line with expanded arguments from '-f filename'. To enable the previous behavior, specify -stats=cmd+verbose.

---

[Refer to Tool Statistics Messages for more information.](#)

- **-suppress <msg\_number>[,<msg\_number>,...]**  
(optional) Prevents the specified message(s) from displaying. You cannot suppress Fatal or Internal messages. Edit the **suppress** variable in the *modelsim.ini* file to set a permanent default. Refer to “[Message Severity Level](#)” for more information.
- **-sync**  
(optional) Executes all X server commands synchronously, so that errors are reported immediately. Does not apply to Windows platforms.
- **-syncio | -nosyncio**  
(optional) Controls buffering of text output to console and logfile. Both options are no-ops when vsim -batch is in effect.
  - syncio — (default) I/O is synchronous with simulation activity, always up-to-date.
  - nosyncio — Disables I/O synchronization. This allows vsim to run faster by buffering (delaying) output.
- **-t [<multiplier>]<time\_unit>**  
(optional) Specifies the simulator time resolution. <time\_unit> must be one of the following:  
  
`fs, ps, ns, us, ms, sec`  
  
The default is 1ps; the optional <multiplier> may be 1, 10 or 100.  
Note that there is no space between the multiplier and the unit (for example, 10fs, not 10 fs).  
If you omit the -t argument, the default simulator time resolution depends on design type:
  - In a VHDL design—the value specified for the **Resolution** variable in *modelsim.ini* is used.
  - In a Verilog design with ‘timescale directives—the minimum specified time precision of all directives is used.
  - In a Verilog design with no ‘timescale directives—the value specified for the **Resolution** variable in the *modelsim.ini* file is used.
  - In a mixed design with VHDL on top—the value specified for the **Resolution** variable in the *modelsim.ini* file is used.
  - In a mixed design with Verilog on top—
    - for Verilog modules not under a VHDL instance: the minimum value specified for their ‘timescale directives is used.
    - for Verilog modules under a VHDL instance: all their ‘timescale directives are ignored (the minimum value for ‘timescale directives in all modules not under a VHDL instance is used).

If there are no ‘timescale directives in the design, the value specified for the Resolution variable in *modelsim.ini* is used.

---

**Tip**

 After you have started a simulation, you can view the current simulator resolution by using the [report](#) command as follows: `report simulator state`

---

- **-tab <tabfile>**

(optional) Specifies the location of a Synopsys VCS “tab” file (.tab), which the simulator uses to automate the registration of PLI functions in the design.

<tabfile> — The location of a *.tab* file contains information about PLI functions. The tool expects the *.tab* file to be based on Synopsys VCS version 7.2 syntax. Because the format for this file is non-standard, changes to the format are outside of the control of Mentor Graphics.

- **-tag <string>**

(optional) Specifies a string tag to append to foreign trace filenames. Used with the `-trace_foreign <int>` option. Used when running multiple traces in the same directory.

- **-title <title>**

(optional) Specifies the title to appear for the ModelSim Main window. If omitted the current ModelSim version is the window title. Useful when running multiple simultaneous simulations. Text strings with spaces must be in quotes (for example, "my title").

- **-trace\_foreign <int>**

(optional) Creates two kinds of foreign interface traces: a log of what functions were called, with the value of the arguments, and the results returned; and a set of C-language files to replay what the foreign interface side did.

The purpose of the logfile is to aid the debugging of your PLI/VPI code. The primary purpose of the replay facility is to send the replay file to MTI support for debugging co-simulation problems, or debugging problems for which it is impractical to send the PLI/VPI code.

- **-undefsyms={<args>}**

(optional) Manages the undefined symbols in the shared libraries currently being loaded into the simulator. You can also manage undefined symbols with the [UDefS](#) *modelsim.ini* variable.

{<args>}

You must specify at least one argument.

on — Enables automatic generation of stub definitions for undefined symbols and permits loading of the shared libraries despite the undefined symbols.

off — (default) Disables loading of undefined symbols. Undefined symbols trigger an immediate shared library loading failure.

verbose — Permits loading to the shared libraries despite the undefined symbols and reports the undefined symbols for each shared library.

- **-uvmcontrol={<args>}**

(optional) Controls UVM-Aware debug features. These features work with either a standard Accelera-released open source toolkit or the pre-compiled UVM library package in ModelSim.

{<args>}

You must specify at least one argument. You can enable or disable some arguments by prefixing the argument with a dash (-). Refer to the argument descriptions for more information.

**all** — Enables all UVM-Aware functionality and debug options except disable and verbose. You must specify verbose separately.

**certe** — Enables the integration of the elaborated design in the Certe tool. Disables Certe features when specified as **-certe**.

**disable** — Prevents the UVM-Aware debug package from being loaded. Changes the results of randomized values in the simulator.

**msglog** — Enables messages logged in UVM to be integrated into the Message Viewer. You must also enable wlf message logging by specifying **tran** or **wlf** with **vsim** **-msgmode**. Disables message logging when specified as **-msglog**

**none** — Turns off all UVM-Aware debug features. Useful when multiple **-uvmcontrol** options are specified in a separate script, makefile or alias and you want to be sure all UVM debug features are turned off.

**reseed** — Disables behavior of UVM simulation, where if you reseed the simulation, the random sequences generated by UVM will change.

**struct** — (default) Enables UVM component instances to appear in the Structure window. UVM instances appear under “**uvm\_root**” in the Structure window. Disables Structure window support when specified as **-struct**.

**trlog** — Enables or disables UVM transaction logging. Logs UVM transactions for viewing in the Wave window. Disables transaction logging when specified as **-trlog**.

**verbose** — Sends UVM debug package information to the transcript. Does not affect functionality. Must be specified separately.

Arguments may be specified as multiple instances of **-uvmcontrol**. Multiple arguments are specified as a comma separated list without spaces. For example,

```
vsim -uvmcontrol=all,-trlog
```

enables all UVM features except UVM transaction logging. Where arguments are in conflict, the last argument will override earlier arguments and a warning is issued.

You can also control UVM-Aware debugging with the [UVMControl](#) modelsim.ini variable.

- **-vcdstim** [<instance>=<filename>]  
(optional) Specifies a VCD file from which to re-simulate the design. The VCD file must have been created in a previous ModelSim simulation, which was executed with the `+dumports+nocollapse` option, then using the [vcd dumports](#) command. Refer to “[Using Extended VCD as Stimulus](#)” for more information.
- **-version**  
(optional) Returns the version of the simulator as used by the licensing tools.
- **-view** [<alias\_name>=<WLF\_filename>]  
(optional) Specifies a wave log format (WLF) file for vsim to read. Allows you to use vsim to view the results from an open simulation (*vsim.wlf*) or an earlier saved simulation. The Structure, Objects, Wave, and List windows can be opened to look at the results stored in the WLF file (other ModelSim windows will not show any information when you are viewing a dataset).
  - <alias\_name> — Specifies an alias for <WLF\_file\_name> where the default is to use the prefix of the WLF\_filename. Wildcard characters are allowed.
  - <WLF\_file\_name> — Specifies the pathname of a saved WLF file.
- See additional discussion in the Examples.
- **-visual** <visual>  
(optional) Specifies the visual to use for the window. Does not apply to Windows platforms.  
Where <visual> may be:
  - <class> <depth> — One of the following:
    - {directcolor | grayscale | greyscale | pseudocolor | staticcolor | staticgray | staticgrey | truecolor}
  - followed by:
    - <depth> — Specifies how many bits per pixel are needed for the visual.
    - default — Instructs the tool to use the default visual for the screen
    - <number> — Specifies a visual X identifier.
    - best <depth> — Instructs the tool to choose the best possible visual for the specified <depth>, where:
      - <depth> — Specifies how many bits per pixel are needed for the visual.
- **-warning** <msg\_number>[,<msg\_number>,...]  
(optional) Changes the severity level of the specified message(s) to "warning." Edit the [warning](#) variable in the *modelsim.ini* file to set a permanent default. Refer to “[Message Severity Level](#)” for more information.
- **-warning error**  
(optional) Reports all warnings as errors.

- **-wlf <file\_name>**  
(optional) Specifies the name of the wave log format (WLF) file to create. The default file name is *vsim.wlf*. This option may also be specified with the [WLFFilename](#) variable in the *modelsim.ini* file.
- **-wlfcachesize <n>**  
(optional) Specifies the size in megabytes of the WLF reader cache. By default the cache size is set to zero. WLF reader caching caches blocks of the WLF file to reduce redundant file I/O. This should have significant benefit in slow network environments. This option may also be specified with the [WLFCacheSize](#) variable in the *modelsim.ini* file.
- **-wlfcollapsedelta**  
(default) Instructs ModelSim to record values in the WLF file only at the end of each simulator delta step. Any sub-delta values are ignored. May dramatically reduce WLF file size. This option may also be specified with the [WLFCollapseMode](#) variable in the *modelsim.ini* file.
- **-wlfcollapsetime**  
(optional) Instructs ModelSim to record values in the WLF file only at the end of each simulator time step. Any delta or sub-delta values are ignored. May dramatically reduce WLF file size. This option may also be specified with the [WLFCollapseMode](#) variable in the *modelsim.ini* file.
- **-nowlfcollapse**  
(optional) Instructs ModelSim to preserve all events for each logged signal and their event order to the WLF file. May result in relatively larger WLF files. This option may also be specified with the [WLFCollapseMode](#) variable in the *modelsim.ini* file.
- **-wlfcompress**  
(default) Creates compressed WLF files. Use **-nowlfcompress** to turn off compression. This option may also be specified with the [WLFCompress](#) variable in the *modelsim.ini* file.
- **-nowlfcompress**  
(optional) Causes vsim to create uncompressed WLF files. WLF files are compressed by default in order to reduce file size. This may slow simulation speed by one to two percent. You may want to disable compression to speed up simulation or if you are experiencing problems with faulty data in the resulting WLF file. This option may also be specified with the [WLFCompress](#) variable in the *modelsim.ini* file.
- **-wlfdeleteonquit**  
(optional) Deletes the current simulation WLF file (*vsim.wlf*) automatically when the simulator exits. This option may also be specified with the [WLFDeleteOnQuit](#) variable in the *modelsim.ini* file.

- **-nowlfdeleteonquit**  
(default) Preserves the current simulation WLF file (vsim.wlf) when the simulator exits. This option may also be specified with the [WLFDeleteOnQuit](#) variable in the *modelsim.ini* file.
- **-wlflock**  
(optional) Locks a WLF file. An invocation of ModelSim will not overwrite a WLF file that is being written by a different invocation.
- **-nowlflock**  
(optional) Disables WLF file locking. This will prevent vsim from checking whether a WLF file is locked prior to opening it as well as preventing vsim from attempting to lock a WLF once it has been opened.
- **-nowlfopt**  
(optional) Disables optimization of waveform display in the Wave window. This option may also be specified with the [WLFOptimize](#) variable in the *modelsim.ini* file.
- **-wlfsimcachesize <n>**  
(optional) Specifies the size in megabytes of the WLF reader cache for the current simulation dataset only. By default the cache size is set to zero. This makes it easier to set different sizes for the WLF reader cache used during simulation and those used during post-simulation debug. WLF reader caching caches blocks of the WLF file to reduce redundant file I/O. If neither the **-wlfsimcachesize** argument nor the [WLFSimCacheSize](#) *modelsim.ini* variable are specified, the **-wlfcachesize** argument or the [WLFCacheSize](#) *modelsim.ini* variable settings will be used.
- **-wlfslim <size>**  
(optional) Specifies a size restriction for the event portion of the WLF file.  
  
size — an integer, in megabytes, where the default is 0, which implies an unlimited size.

---

**Note**

 Note that a WLF file contains event, header, and symbol portions. The size restriction is placed on the event portion only. Consequently, the resulting file will be larger than the specified size.

---

If used in conjunction with **-wlftlim**, the more restrictive of the limits takes precedence.

This option may also be specified with the [WLFSizeLimit](#) variable in the *modelsim.ini* file. (See [Limiting the WLF File Size](#).)

- **-wlftlim <duration>**  
(optional) Specifies the duration of simulation time for WLF file recording. The default is infinite time (0). The <duration> is an integer of simulation time at the current resolution; you can optionally specify the resolution if you place curly braces around the specification. For example,

{ 5000 ns }

sets the duration at 5000 nanoseconds regardless of the current simulator resolution.

The time range begins at the current simulation time and moves back in simulation time for the specified duration. For example,

```
vsim -wlftlim 5000
```

writes at most the last 5000ns of the current simulation to the WLF file (the current simulation resolution in this case is ns).

If used in conjunction with -wlfslim, the more restrictive of the limits will take effect.

This option may also be specified with the [WLFTimeLimit](#) variable in the *modelsim.ini* file.

The -wlfslim and -wlftlim arguments were designed to help users limit WLF file sizes for long or heavily logged simulations. When small values are used for these arguments, the values may be overridden by the internal granularity limits of the WLF file format. (See [Limiting the WLF File Size](#).)

#### VHDL Arguments

- **-absentisempty**

(optional) Causes VHDL files opened for read that target non-existent files to be treated as empty, rather than ModelSim issuing fatal error messages.

- **-accessobjdebug**

(optional) Enables logging of VHDL access type variables—both the variable value and any access object that the variable points to during the simulation. Further, it changes the default form of display-only names (such as [10001]) to a different form that you can use as input to any command that expects an object name.

By default, logging is turned off. This means that while access variables themselves can be logged and displayed in the ModelSim display windows, any access objects that they point to will not be logged.

Overrides the setting for the [AccessObjDebug](#) variable in the *modelsim.ini* file.

- **-noaccessobjdebug**

(optional) Disables logging of VHDL access type variables, which is the default setting. This means that while access variables themselves can be logged and displayed in the ModelSim display windows, any access objects that they point to will not be logged.

Overrides the setting for the [AccessObjDebug](#) variable in the *modelsim.ini* file.

- **-nocollapse**

(optional) Disables the optimization of internal port map connections.

- **-nofileshare**

(optional) Turns off file descriptor sharing. By default ModelSim shares a file descriptor for all VHDL files opened for write or append that have identical names.

- **-noglitch**  
(optional) Disables VITAL glitch generation.  
Refer to “[VHDL Simulation](#)” for additional discussion of VITAL.
- **+no\_glitch\_msg**  
(optional) Disable VITAL glitch error messages.
- **-novhdlvariablelogging**  
(optional) This argument turns off the ability to log recursively or add process variables to the Wave or List windows. Refer to `-vhdlvariable` logging and [VhdlVariableLogging](#) `modelsim.ini` variable for more information.
- **-std\_input <filename>**  
(optional) Specifies the file to use for the VHDL TextIO STD\_INPUT file.
- **-std\_output <filename>**  
(optional) Specifies the file to use for the VHDL TextIO STD\_OUTPUT file.
- **-strictvital**  
(optional) Specifies to exactly match the VITAL package ordering for messages and delta cycles. Useful for eliminating delta cycle differences caused by optimizations not addressed in the VITAL LRM. Using this argument negatively impacts simulator performance.
- **+transport\_int\_delays**  
(optional) Selects transport mode with pulse control for single-source nets (one interconnect path). By default interconnect delays operate in inertial mode (pulses smaller than the delay are filtered). In transport mode, narrow pulses are propagated through interconnect delays.  
This option works for both Verilog and VITAL cells, though the destination of the interconnect must be a Verilog cell. The source may be VITAL or Verilog. This option works independently from `+multisource_int_delays`.
- **+transport\_path\_delays**  
(optional) Selects transport mode for path delays. By default, path delays operate in inertial mode (pulses smaller than the delay are filtered). In transport mode, narrow pulses are propagated through path delays. Note that this option affects path delays only, and not primitives. Primitives always operate in inertial delay mode.
- **-vhdlmergepdupackage**  
(optional) Turns off sharing of one package between two PDUs. Each PDU will have a separate copy of the package. This option may also be specified with the [VhdlSeparatePduPackage](#) variable in the `modelsim.ini` file.
- **-vhdlseparatepdupackage**  
(optional, default) Turns on sharing of packages between two or more PDUs.

- **-vhdlvariablelogging**

(optional) This argument makes it possible for process variables to be logged recursively or added to the Wave and List windows (process variables can still be logged or added to the Wave and List windows explicitly with or without this argument).

You can disable this argument with **-novhdlvariablelogging**. Refer to **-vhdlvariable logging** and [VhdlVariableLogging](#) modelsim.ini variable for more information.

---

**Note**

 Logging process variables inherently decreases simulation performance because of their nature. It is recommended that they not be logged or added to the Wave and List windows. However, if debugging does require logging them, then you can use this argument to minimize the performance decrease.

---

- **-vital2.2b**

(optional) Selects SDF mapping for VITAL 2.2b (default is VITAL 2000).

**Verilog Arguments**

- **+alt\_path\_delays**

(optional) Configures path delays to operate in inertial mode by default. In inertial mode, a pending output transition is canceled when a new output transition is scheduled. The result is that an output may have no more than one pending transition at a time, and that pulses narrower than the delay are filtered. The delay is selected based on the transition from the canceled pending value of the net to the new pending value. The **+alt\_path\_delays** option modifies the inertial mode such that a delay is based on a transition from the current output value rather than the canceled pending value of the net. This option has no effect in transport mode (see **+pulse\_e/<percent>** and **+pulse\_r/<percent>**).

- **-checkvifacedrivers**

(optional) Turns off/on checks for multiple-driver analysis in assignments made through virtual interfaces.

- **-classdebug | -noclassdebug**

(optional) Enables/disables visibility into class instances, and includes SystemVerilog queues, dynamic arrays, and associative arrays for class and UVM debugging. You can also enable visibility into class instances by setting the [ClassDebug](#) modelsim.ini variable to 1. Refer to the [classinfo find](#) command for more information.

- **+delayed\_timing\_checks**

(optional) Causes timing checks to be performed on the delayed versions of input ports (used when there are negative timing check limits). By default, ModelSim automatically detects and applies **+delayed\_timing\_checks** to cells with negative timing checks. To turn off this feature, specify **+no\_autodtc** with vsim.

- **-dpiforceheader**  
(optional) Forces the generation of a DPI header file even if it will be empty of function prototypes.
- **-dpiheader**  
(optional) Generates a header file that may then be included in C source code for DPI import functions. Simulation quits after header file is generated. Refer to “[DPI Use Flow](#)” for additional information.
- **-dpilib <libname>**  
(optional) Specifies the design library name that contains DPI exports and automatically compiled object files. If the -dpilib argument is not set, vsim loads export symbols from all libraries accessible via vsim options -L, -Lf, and -lib. Multiple occurrences of -dpilib are supported.
- **-dpoutoftheblue 0 | 1 | 2**  
(optional) Instructs vsim to allow DPI out-of-the-blue calls from C functions. The C functions must not be declared as import tasks or functions.
  - 0 — Support for DPI out-of-the-blue calls is disabled.
  - 1 — Support for DPI out-of-the-blue calls is enabled, but debugging support is not available.
- **-gconrun | -nogconrun**  
(optional) Enables/disables garbage collector execution after each simulation [run](#) command completes.
- **-gconstep | -nogconstep**  
(optional) Enables/disables garbage collector execution after each step when stepping through your simulation.
- **-gcthreshold <n>**  
(optional) Sets the maximum amount of memory in megabytes allocated for storage of class objects before the garbage collector runs to delete unreferenced objects.
  - <n> — Any positive integer where <n> is the number of megabytes. The default size is 100 megabytes.

Related *modelsim.ini* file variables are [GCThreshold](#) and [GCThresholdClassDebug](#).

- **-hazards**  
(optional) Enables event order hazard checking in Verilog modules (Verilog only). You must also specify this argument when you compile your design with [vlog](#). Refer to “[Hazard Detection](#)” for more details.

---

**Note**

Using -hazards implicitly enables the -compat argument. As a result, using this argument may affect your simulation results.

---

- `-ignoreinilibs`  
(optional) Ignore the libraries specified in the [LibrarySearchPath](#) variable in the vsim section of the *modelsim.ini* file.
- `-noappendclose`  
(optional) Simulator will **not** immediately close files opened in APPEND mode. Designed to override the [AppendClose](#) *modelsim.ini* variable when it is set to one (On). Subsequent calls to `file_open` in APPEND mode will therefore not require operating system interaction, resulting in faster performance.
- `+no_autodtc`  
(optional) Turns off auto-detection of optimized cells with negative timing checks and auto-application of `+delayed_timing_checks` to those cells.
- `+no_cancelled_e_msg`  
(optional) Disables negative pulse warning messages. By default vsim issues a warning and then filters negative pulses on specify path delays. You can drive an X for a negative pulse using `+show_cancelled_e`.
- `+no_neg_tchk`  
(optional) Disables negative timing check limits by setting them to zero. By default negative timing check limits are enabled. This is just the opposite of Verilog-XL, where negative timing check limits are disabled by default, and they are enabled with the `+neg_tchk` option.
- `+no_notifier`  
(optional) Disables the toggling of the notifier register argument of all timing check system tasks. By default, the notifier is toggled when there is a timing check violation, and the notifier usually causes a UDP to propagate an X. This argument suppresses X propagation on timing violations for the entire design.
- `+no_path_edge`  
(optional) Causes ModelSim to ignore the input edge specified in a path delay. The result of this argument is that all edges on the input are considered when selecting the output delay. Verilog-XL always ignores the input edges on path delays.
- `+no_pulse_msg`  
(optional) Disables the warning message for specify path pulse errors. A path pulse error occurs when a pulse propagated through a path delay falls between the pulse rejection limit and pulse error limit set with the `+pulse_r` and `+pulse_e` options. A path pulse error results in a warning message, and the pulse is propagated as an X. The `+no_pulse_msg` option disables the warning message, but the X is still propagated.
- `-no_risefall_delaynets`  
(optional) Disables the default rise/fall delay net delay negative timing check algorithm. This argument is provided to return ModelSim to older behavior where violation regions must overlap in order to find a delay net solution. Beginning with Release 6.0, you do not

need to use this argument because ModelSim uses separate rise/fall delays, so violation regions need not overlap for a delay solution to be found.

- `+no_show_cancelled_e`  
(optional) Filters negative pulses on specify path delays so they do not show on the output. Default. Use `+show_cancelled_e` to drive a pulse error state.
- `+no_tchk_msg`  
(optional) Disables error messages issued by timing check system tasks when timing check violations occur. Notifier registers are still toggled and may result in the propagation of Xs for timing check violations.
- `-nodpiexports`  
(optional) Instructs ModelSim to not generate C wrapper code for DPI export task and function routines found at elaboration time. More specifically, the command does not generate the *exportwrapper.so* shared object file.  
For a description on when you should use this argument, refer to the section “[Deprecated Legacy DPI Flows](#)” in the User’s Manual.
- `+nosdferror`  
(optional) Errors issued by the SDF annotator while loading the design prevent the simulation from continuing, whereas warnings do not. Changes SDF errors to warnings so that the simulation can continue.
- `+nosdfwarn`  
(optional) Disables warnings from the SDF annotator.
- `+nospecify`  
(optional) Disables specify path delays and timing checks.
- `+nowarnBSOB`  
(optional) Disables run-time warning messages for bit-selects in initial blocks that are out of bounds.
- `+nowarn<CODE | number>`  
(optional) Disables warning messages in the category specified by a warning code or number. Warnings that can be disabled include the code name in square brackets in the warning message. For example:

```
** Warning: (vsim-3017) test.v(2): [TFMPC] - Too few port
connections. Expected <m>, found <n>.
```

The warning code for this example is TFMPC, and the warning number is 3017. Therefore, this warning message can be disabled with `+nowarnTFMPC` or `+nowarn3017`.

- `+ntc_warn`  
(optional) Enables warning messages from the negative timing constraint algorithm. By default, these warnings are disabled.

This algorithm attempts to find a set of delays for the timing check delayed net arguments such that all negative limits can be converted to non-negative limits with respect to the delayed nets. If there is no solution for this set of limits, then the algorithm sets one of the negative limits to zero and recalculates the delays. This process is repeated until a solution is found. A warning message is issued for each negative limit set to zero.

- `+ntcnotchks`

(optional) Instructs vsim to not simulate timing checks but still consider negative timing check limits for the calculation of delayed input delays.

- `-oldvhdlforgennames`

(optional) Enables the use of a previous style of naming in VHDL for ... generate statement iteration names in the design hierarchy. The previous style is controlled by the value of the [GenerateFormat](#) value. The default behavior is to use the current style names, which is described in “[Naming Behavior of VHDL For Generate Blocks](#)” This argument duplicates the function of the [OldVhdlForGenNames](#) variable in *modelsim.ini* and will override the setting of that variable if it specifies the current style.

- `-onfinish ask | stop | exit | final`

(optional) Customizes the simulator shutdown behavior when it encounters \$finish in the design:

- `ask` —
  - In batch mode, the simulation exits.
  - In GUI mode, a dialog box pops up and asks for user confirmation on whether to quit the simulation.
- `stop` — stops simulation and leaves the simulation kernel running
- `exit` — exits out of the simulation without a prompt
- `final` — executes all final blocks then exits the simulation

By default, the simulator exits in batch mode; prompts you in GUI mode. Edit the [OnFinish](#) variable in the *modelsim.ini* file to set the default operation of \$finish.

- `-pedanticerrors`

(optional) Forces display of an error message (rather than a warning) on a variety of conditions.

You can view a complete list of errors by executing the command:

```
verror -kind vsim -pedanticerrors
```

- `-permit_unmatched_virtual_intf`

(optional) Permits vsim to elaborate designs containing virtual interface declarations for which no actual interface instances exist that are compatible with that declaration. Such virtual interface declarations are considered "unmatched" since there is no matching or

compatible interface instance. By default, unmatched virtual interfaces prevent vsim from elaborating the design. For further information on this design situation, see "Unmatched Virtual Interface Declarations".

- **-pli "<object list>"**

(optional) Loads a comma- or space-separated list of PLI shared objects. The list must be quoted if it contains more than one object. This is an alternative to specifying PLI objects in the Veriuser entry in the *modelsim.ini* file, refer to [modelsim.ini Variables](#). You can use environment variables as part of the path.

- **+<plusarg>**

(optional) Arguments preceded with "+" are accessible by the Verilog PLI routine `mc_scan_plusargs()`. All plusarg argument values can be overridden in -restore mode and will take effect when simulation resumes after restoring the design.

- **+pulse\_e/<percent>**

(optional) Controls how pulses are propagated through specify path delays, where <percent> is a number between 0 and 100 that specifies the error limit as a percentage of the path delay.

A pulse greater than or equal to the error limit propagates to the output in transport mode (transport mode allows multiple pending transitions on an output). A pulse less than the error limit and greater than or equal to the rejection limit (see **+pulse\_r/<percent>**) propagates to the output as an X. If the rejection limit is not specified, then it defaults to the error limit. For example, consider a path delay of 10 along with a **+pulse\_e/80** option. The error limit is 80% of 10 and the rejection limit defaults to 80% of 10. This results in the propagation of pulses greater than or equal to 8, while all other pulses are filtered. Note that you can force specify path delays to operate in transport mode by using the **+pulse\_e/0** option.

- **+pulse\_e\_style\_ondetect**

(optional) Selects the "on detect" style of propagating pulse errors (see **+pulse\_e**). A pulse error propagates to the output as an X, and the "on detect" style is to schedule the X immediately, as soon as it has been detected that a pulse error has occurred. "on event" style is the default for propagating pulse errors (see **+pulse\_e\_style\_onevent**).

- **+pulse\_e\_style\_onevent**

(optional) Selects the "on event" style of propagating pulse errors (see **+pulse\_e**). Default. A pulse error propagates to the output as an X, and the "on event" style is to schedule the X to occur at the same time and for the same duration that the pulse would have occurred if it had propagated through normally.

- **+pulse\_r/<percent>**

(optional) Controls how pulses are propagated through specify path delays, where <percent> is a number between 0 and 100 that specifies the rejection limit as a percentage of the path delay.

A pulse less than the rejection limit is suppressed from propagating to the output. If the error limit is not specified by `+pulse_e` then it defaults to the rejection limit.

- `+sdf_nocheck_celltype`  
(optional) Disables the error check a for mismatch between the `CELLTYPE` name in the SDF file and the module or primitive name for the `CELL` instance. It is an error if the names do not match.
- `+show_cancelled_e`  
(optional) Drives a pulse error state ('X') for the duration of a negative pulse on a specify path delay. By default ModelSim filters negative pulses.
- `-showlibsearchpath`  
(optional) Returns to the transcript all libraries that will be searched for precompiled modules.
- `-sv_lib <shared_obj>`  
(required for use with DPI import libraries) Specifies the name of the DPI shared object with no extension. Refer to “[DPI Use Flow](#)” for additional information.
- `-sv_liblist <filename>`  
(optional) Specifies the name of a bootstrap file containing names of DPI shared objects (libraries) to be loaded. Refer to “[DPI File Loading](#)” for format information.
- `-sv_root <dirname>`  
(optional) Specifies the directory name to be used as the prefix for DPI shared object lookups.
- `+transport_int_delays`  
(optional) Selects transport mode with pulse control for single-source nets (one interconnect path). By default interconnect delays operate in inertial mode (pulses smaller than the delay are filtered). In transport mode, narrow pulses are propagated through interconnect delays.  
This option works for both Verilog and VITAL cells, though the destination of the interconnect must be a Verilog cell. The source may be VITAL or Verilog. This option works independently from `+multisource_int_delays`.
- `+transport_path_delays`  
(optional) Selects transport mode for path delays. By default, path delays operate in inertial mode (pulses smaller than the delay are filtered). In transport mode, narrow pulses are propagated through path delays. Note that this option affects path delays only, and not primitives. Primitives always operate in inertial delay mode.
- `-v2k_int_delays`  
(optional) Causes interconnect delays to be visible at the load module port per the IEEE 1364-2001 spec. By default ModelSim annotates `INTERCONNECT` delays in a manner compatible with Verilog-XL. If you have `$sdf_annotation()` calls in your design that are not getting executed, add the Verilog task `$sdf_done()` after your last `$sdf_annotation()` to remove

any zero-delay MIPDs that may have been created. May be used in tandem with the `+multisource_int_delays` argument (see above).

- `-work <pathname>`  
(optional) When using a 2-step flow, this argument overrides the library in which `vsim` writes the optimized design generated by the internally invoked `vopt` command.
- `-wrealdefaultzero`  
(optional) For nets declared as `wreal`, sets the default value for an undriven `wreal` net to zero (0).

### Object Arguments

The object arguments may be a [`<library_name>`].`<design_unit>`, an `.mpf` file, a `.wlf` file, or a text file. Multiple design units may be specified for Verilog modules and mixed VHDL/Verilog configurations.

- `<library_name>.<design_unit>`  
(optional) Specifies a library and associated design unit; multiple library/design unit specifications can be made. If no library is specified, the work library is used. You cannot use the wildcard character (\*) for this argument. Environment variables can be used.  
`<design_unit>` may be one of the following:

<code>&lt;configuration&gt;</code>	Specifies the VHDL configuration to simulate.
<code>&lt;module&gt; ...</code>	(optional) Specifies the name of one or more top-level Verilog modules to be simulated.
<code>&lt;entity&gt; [(&lt;architecture&gt;)]</code>	(optional) Specifies the name of the top-level VHDL entity to be simulated. The entity may have an architecture optionally specified; if omitted the last architecture compiled for the specified entity is simulated. An entity is not valid if a configuration is specified. <sup>1</sup>

1. Most UNIX shells require arguments containing () to be single-quoted to prevent special parsing by the shell. See the examples below.

- `<MPF_file_name>`  
(optional) Opens the specified project.
- `<WLF_file_name>`  
(optional) Opens the specified dataset. When you open a WLF file using the following command:

```
vsim test.wlf
```

The default behavior is to not automatically load any signals into the Wave window. You can change this behavior, such that the Wave window contains all signals in the design, by setting the preference PrefWave(OpenLogAutoAddWave) to 1 (true).

- <text\_file\_name>  
(optional) Opens the specified text file in a Source window.

## Examples

- Invoke vsim on the entity *cpu* and assigns values to the generic parameters *edge* and *VCC*.

```
vsim -gedge="low high" -gVCC=4.75 cpu
```

If working within the ModelSim GUI, you would enter the command as follows:

```
vsim {-gedge="low high"} -gVCC=4.75 cpu
```

Instruct ModelSim to view the results of a previous simulation run stored in the WLF file *sim2.wlf*. The simulation is displayed as a dataset named *test*. Use the *-wlf* argument to specify the name of the WLF file to create if you plan to create many files for later viewing.

```
vsim -view test=sim2.wlf
```

For example:

```
vsim -wlf my_design.i01 my_asic structure
vsim -wlf my_design.i02 my_asic structure
```

Annotate instance */top/u1* using the minimum timing from the SDF file *myasic.sdf*.

```
vsim -sdfmin /top/u1=myasic.sdf
```

Use multiple arguments to annotate multiple instances:

```
vsim -sdfmin /top/u1=sdf1 -sdfmin /top/u2=sdf2 top
```

- This example searches the libraries *mylib* for *top(only)* and *gatelib* for *cache\_set*. If the design units are not found, the search continues to the work library. Specification of the architecture (*only*) is optional.

```
vsim 'mylib.top(only)' gatelib.cache_set
```

- Invoke vsim on *test\_counter* and run the simulation until a break event, then quit when it encounters a \$finish task.

```
vsim -do "set PrefMain(forceQuit) 1; run -all" work.test_counter
```

- Enable the display of Start, End, and Elapsed time as well as a message count summary. Echoing of the command line is disabled

```
vsim -stats=time,-cmd,msg
```

- The first *-stats* option is ignored. The *none* option disables all modelsim.ini settings and then enables the *perf* option.

```
vsim -stats=time,cmd,msg -stats=none,perf
```

## vsim<info>

Series of commands that return information about the current vsim executable.

### Syntax

**vsimAuth**

Returns the authorization level (PE/SE, VHDL/Verilog/PLUS).

**vsimDate**

Returns the date the executable was built, such as "Apr 10 2000".

**vsimId**

Returns the identifying string, such as "ModelSim 6.1".

**vsimVersion**

Returns the version as used by the licensing tools, such as "1999.04".

**vsimVersionString**

Returns the full vsim version string. You can obtain this same information using the -version argument of the **vsim** command.

### Arguments

none

## **vsim\_break**

Stop (interrupt) the current simulation before it runs to completion. To stop a simulation and then resume it, use this command in conjunction with `run -continue`.

### **Syntax**

`vsim_break`

### **Arguments**

None.

### **Examples**

- Interrupt a simulation, then restart it from the point of interruption.

```
vsim_break  
run -continue
```

## vsouce

This command specifies an alternative file to use for the current source file. This command is used when the current source file has been moved. The alternative source mapping exists for the current simulation only.

### Syntax

```
vsouce [<filename>]
```

### Arguments

- <filename>

(optional) Specifies a relative or full pathname. If filename is omitted, the source file for the current design context is displayed.

### Examples

```
vsouce design.vhd
vsouce /old/design.vhd
```

## wave

A collection of related commands that manipulate and report on the Wave window.

### Syntax

```
wave cursor active [-window <win>] [<cursor-num>]
wave cursor add [-window <win>] [-time <time>] [-name <name>] [-lock <0 |1>]
wave cursor configure [<cursor-num>] [-window <win>] [<option> [<value>]]
wave cursor delete [-window <win>] [<cursor-num>]
wave cursor see [-window <win>] [-at <percent>] [<cursor-num>]
wave cursor time [-window <win>] [-time <time>] [<cursor-num>]
wave collapse all [-window <win>]
wave collapse cursor [-window <win>] [<cursor-num>]
wave collapse range [-window <win>] <start-time> <end-time>
wave expand all [-window <win>]
wave expand cursor [-window <win>] [<cursor-num>]
wave expand mode [-window <win>] [off | deltas | events]
wave expand range [-window <win>] <start-time> <end-time>
wave interrupt [-window <win>]
wave refresh [-window <win>]
wave seetime [-window <win>] [-at <percent>] -time <time>
wave zoom in [-window <win>] [<factor>]
wave zoom out [-window <win>] [<factor>]
wave zoom full [-window <win>]
wave zoom last [-window <win>]
wave zoom range [-window <win>] [<start-time> <end-time>]
```

## Description

The following tables summarize the available options for manipulating cursors, for zooming, and for adjusting the wave display view in the Wave window:

**Table 2-9. Wave Window Commands for Cursor**

Cursor Commands	Description
wave cursor active	Sets the active cursor to the specified cursor or, if no cursor is specified, reports the active cursor
wave cursor add	Adds a new cursor at specified time and returns the number of the newly added cursor
wave cursor configure	Sets or reports values for the specified cursor
wave cursor delete	Deletes the specified cursor or, if no cursor is specified, the active cursor
wave cursor see	Positions the wave display such that the specified or active cursor appears at the specified percent from the left edge of the display – 0% is the left edge, 100% is the right edge.
wave cursor time	Moves or reports the time of the specified cursor or, if no cursor is specified, the time of the active cursor

**Table 2-10. Wave Window Commands for Expanded Time Display**

Display view Commands	Description
wave expand mode	Selects the expanded time display mode: Delta Time, Event Time, or off.
wave expand all	Expands simulation time into delta time steps if Delta Time mode is currently selected (WLFCollapseMode = 1) or into event time steps if Event Time mode is currently selected (WLFCollapseMode = 0) over the full range of the simulation from time 0 to the current time.
wave expand cursor	Expands simulation time into delta time steps if Delta Time mode is currently selected (WLFCollapseMode = 1) or into event time steps if Event Time mode is currently selected (WLFCollapseMode = 0) at the simulation time of the active cursor.
wave expand range	Expands simulation time into delta time steps if Delta Time mode is currently selected (WLFCollapseMode = 1) or into event time steps if Event Time mode is currently selected (WLFCollapseMode = 0) over a time range specified by a start time and an end time.

**Table 2-10. Wave Window Commands for Expanded Time Display (cont.)**

Display view Commands	Description
wave collapse all	Collapses simulation time over the full range of the simulation from time 0 to the current time.
wave collapse cursor	Collapses simulation time at the time of the active cursor.
wave collapse range	Collapses simulation time over a specific simulation time range.

**Table 2-11. Wave Window Commands for Controlling Display**

Display view Commands	Description
wave interrupt	Immediately stops wave window drawing
wave refresh	Cleans wave display and redraws waves
wave cursor see	Positions the wave display such that the specified or active cursor appears at the specified percent from the left edge of the display – 0% is the left edge, 100% is the right edge.
wave seetime	Positions the wave display such that the specified time appears at the specified percent from the left edge of the display – 0% is the left edge, 100% is the right edge.

**Table 2-12. Wave Window Commands for Zooming**

Zooming Commands	Description
wave zoom in	Zoom in the wave display by the specified factor. The default factor is 2.0.
wave zoom out	Zoom out the wave display by the specified factor. The default factor is 2.0.
wave zoom full	Zoom the wave display to show the full simulation time.
wave zoom last	Return to last zoom range.
wave zoom range	Sets left and right edge of wave display to the specified start time and end time. If times are not specified, reports left and right edge times.

## Arguments

- `-at <percent>`  
(optional) Positions the display such that the time or cursor is the specified `<percent>` from the left edge of the wave display.

<percent> — Any non-negative number where the default is 50. 0 is the left edge of the Wave window and 100 is the right edge.

- <cursor-num>  
(optional) Specifies a cursor number. If not specified, the active cursor is used.
- <factor>  
(optional) A number that specifies how much you want to zoom into or out of the wave display. Default value is 2.0.
- -lock <0 |1>  
(optional) Specify the lock state of the cursor.
  - 0 — (default) Unlocked
  - 1 — Locked
- -name <name>  
(optional) Specify the name of the cursor.  
<name> — Any string where the default is "Cursor <n>" where <n> is the cursor number.
- off | deltas | events  
(optional) Specifies the expanded time display mode for the Wave window. Default is off.
- <option> [<value>]  
(optional) Specify a value for the designated option. Currently supported options are -name, -time, and -lock. If no option is specified, current value of all options are reported.
- <start-time> <end-time>  
(optional) start-time and end-time are times that specify an expand, collapse, or zoom range. If neither number is specified, the command returns the current range.
- -time <time>  
(optional) Specifies a cursor time.  
<time> — Any positive integer.
- -window <win>  
(optional) All commands default to the active Wave window unless this argument is used to specify a different Wave window.  
<win> — Specifies the name of a Wave window other than the current active window.

## Examples

- Either of these commands creates a zoom range with a start time of 20 ns and an end time of 100 ns.

```
wave zoom range 20ns 100ns
wave zoom range 20 100
```

- Return the name of cursor 2:

**wave cursor configure 2 -name**

- Name cursor 2, "reference cursor" and return that name with:

**wave cursor configure 2 -name {reference cursor}**

- Return the values of all wave cursor configure options for cursor 2:

**wave cursor configure 2**

# wave create

This command generates a waveform known only to the GUI. You can then modify the waveform interactively or with the wave edit command and use the results to drive simulation.

## Syntax

### All waveforms

```
wave create [-driver {freeze | deposit | driver | expectedoutput}] [-initialvalue <value>]  
[-language {vhdl | verilog}] [-portmode {input | output | inout | internal}] [-range <msb  
lsb>]  
[-starttime {<time><unit>}] [-endtime {<time><unit>}] <object_name>
```

### Clock patterns

```
wave create -pattern clock [-dutycycle <value>] [-period {<time><unit>}] <object_name>
```

### Constant patterns

```
wave create -pattern constant [-initialvalue <value>] [-value <value>] <object_name>
```

### Random patterns

```
wave create -pattern random [-initialvalue <value>] [-period {<time><unit>}]  
[-random_type {normal | uniform | poisson | exponential}] [-seed <value>] <object_name>
```

### Repeater patterns

```
wave create -pattern repeater [-initialvalue <value>] [-period {<time><unit>}]  
[-repeat {forever | never | <n>}] [-sequence {<val1>} <val2> ...]
```

### Counter patterns

```
wave create -pattern counter [-direction {up | down | upthendown | downthenup}]  
[-initialvalue <value>] [-period {<time><unit>}] [-repeat {forever | never | <n>}]  
[-startvalue <value>] [-endvalue <value>] [-step <value>]  
[-type {binary | gray | johnson | onehot | range | zerohot}] <object_name>
```

### No pattern

```
wave create -pattern none <object_name>
```

## Description

Refer to “[Generating Stimulus with Waveform Editor](#)” for more information.

Some arguments to this command are order-dependent. Please read through the argument descriptions for more information. d

The following table summarizes the available waveform pattern options:

Command	Description
wave create -pattern clock	Generates a clock waveform. Recommended that you specify an initial value, duty cycle, and clock period for the waveform.

Command	Description
wave create -pattern constant	Generates a waveform with a constant value. It is suggested that you specify a value.
wave create -pattern random	Generates a random waveform based upon a seed value. Specify the type (normal or uniform), an initial value, and a seed value. If you don't specify a seed value, Questa uses a default value of 5.
wave create -pattern repeater	Generates a waveform that repeats. Specify an initial value and pattern that repeats. You can also specify how many times the pattern repeats.
wave create -pattern counter	Generates a waveform from a counting pattern. Specify start and end values, repeat, step count, time period, and type (Binary, Gray, Johnson, OneHot, Range, and ZeroHot).
wave create -pattern none	Creates a placeholder for a waveform. Specify an object name.

## Arguments

- **-pattern** `clock | constant | random | repeater | counter | none`  
(required) Specifies the waveform pattern. Refer to “[Accessing the Create Pattern Wizard](#)” for a description of the pattern types.
  - `clock` — Specifies a clock pattern.
  - `constant` — Specifies a constant pattern.
  - `random` — Specifies a random pattern.
  - `repeater` — Specifies a repeating pattern.
  - `counter` — Specifies a counting pattern.
  - `none` — Specifies a blank pattern.
- **-direction** `{up | down | upthendown | downthenup}`  
(optional, recommended when specifying -pattern counter) The direction in which the counter will increment or decrement.
  - `up` — (default) Increment only.
  - `down` — Decrement only.
  - `upthendown` — Increment then decrement.
  - `downthenup` — Decrement then increment.
- **-driver** `{freeze | deposit | driver | expectedoutput}`  
(optional) Specifies that the signal is a driver of the specified type. Applies to waveforms created with -portmode inout or -portmode internal.

- **-dutycycle <value>**  
(optional, recommended for -pattern clock) Specifies the duty cycle of the clock. Expressed as a percentage of the period that the clock is high.  
*<value>* — Any integer from 0 to 100 where the default is 50.
- **-endtime {<time><unit>}**  
(optional) The simulation time where the waveform will stop. If omitted, the waveform stops at 1000 simulation time units.  
*<time>* — Specified as an integer or decimal number.  
*<unit>* — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting *<unit>*. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If *<unit>* is specified, you must enclose *<time>* and *<unit>* within curly braces ({}).
- **-endvalue <value>**  
(optional, recommended when specifying -pattern counter) The end value for the counter. This option applies to patterns specifying -type Range only. All other counter patterns start from 0 and go to the maximum value for that particular signal (for example, for a 3-bit signal, the start value will be 000 and the end value will be 111).  
*<value>* — Value must be appropriate for the type of waveform you are creating.
- **-initialvalue <value>**  
(optional) The initial value for the waveform. Not applicable to counter patterns.  
*<value>* — Value must be appropriate for the type of waveform you are creating.
- **-language {vhdl | verilog}**  
(optional) Controls which language is used for the created wave.  
vhdl — (default) Specifies the VHDL language.  
verilog — Specifies the Verilog language.
- **-period {<time><unit>}**  
(optional, recommended for all patterns except -constant) Specifies the period of the signal.  
*<time>* — Specified as an integer or decimal number. Current simulation units are the default unless specifying *<unit>*.  
*<unit>* — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting *<unit>*. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If *<unit>* is specified, you must enclose *<time>* and *<unit>* within curly braces ({}).
- **-portmode {input | output | inout | internal}**  
(optional) The port type for the waveform. Useful for creating signals prior to loading a design.  
in — Ports of type IN. You can also specify “input” as an alias for in.

out — Ports of type OUT. You can also specify “output” as an alias for out.

inout — Ports of type INOUT.

internal — (default) Ports of type INTERNAL.

- **-random\_type {normal | uniform | poisson | exponential}**

(optional, recommended when specifying -pattern random) Specifies the type of algorithm used to generate a random waveform pattern.

normal — Normal or Gaussian distribution of waveform events.

uniform — (default) Uniform distribution of waveform events.

poisson — Poisson distribution of waveform events.

exponential — Exponential distribution of waveform events.

- **-range <msb lsb>**

(optional) Identifies bit significance in a counter pattern.

msb lsb — Most significant bit and least significant bit. Both must be specified.

- **-repeat {forever | never | <n>}**

(optional, recommended when specifying -pattern repeater or -pattern counter) Controls duration of pattern repetition.

forever — Repeat the pattern for as long as the simulation runs.

never — Never repeat the pattern during simulation.

<n> — Repeat the pattern <n> number of times where <n> is any positive integer.

- **-seed <value>**

(optional, recommended when specifying -pattern random) Specifies a seed value for a randomly generated waveform.

<value> — Any non-negative integer where the default is 5.

- **-sequence {<val1>} <val2> ...**

(optional, recommended when specifying pattern -repeater) The set of values that you want repeated.

<val1> — Value must be appropriate for the type of waveform you are creating.

Multiple values are entered as a space separated list and enclosed in curly braces ({}).

- **-starttime {<time><unit>}**

(optional) The simulation time at which the waveform should start. If omitted, the waveform starts at 0 simulation time units.

<time> — Specified as an integer or decimal number.

<unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <time> and <unit> within curly braces ({}).

- **-startvalue <value>**

(required when specifying -pattern counter) The initial value of the counter. This option applies to patterns specifying -type Range only. All other counter patterns start from 0 and go to the maximum value for that particular signal (e.g., for a 3-bit signal, the start value will be 000 and the end value will be 111).

<value> — Value must be appropriate for the type of waveform you are creating.

- **-step <value>**

(optional, recommended when specifying -pattern counter) The step by which the counter is incremented/decremented.

<value> — Value must be appropriate for the type of waveform you are creating.

- **-type {binary | gray | johnson | onehot | range | zerohot}**

(optional) Specifies a counter format.

**binary** — Specifies a binary counter.

**gray** — Specifies a binary counter where two successive values differ in only one bit.  
Also known as a reflected binary counter.

**johnson** — Specifies a twisted ring or Johnson counter.

**onehot** — Specifies a shift counter where only one bit at a time is set to “on” (1).

**range** — (default) Specifies a binary counter where the values range between -startvalue and -endvalue

**zerohot** — Specifies a shift counter where only one bit at a time is set to “off” (0).

- **-value <value>**

(optional, recommended when specifying -pattern constant) Specifies a value for the constant pattern.

<value> — Value must be appropriate for the type of waveform you are creating.

- **<object\_name>**

(required) User specified name for the waveform. Must be the final argument.

## Examples

- Create a clock signal with the following default values:

```
wave create -pattern clock -period 100 -dutycycle 50 -starttime 0 -endtime 1000  
-initialvalue 0 /counter/clk
```

- Create a constant 8-bit signal vector from 0 to 1000 ns with a value of 1111 and a drive type of freeze.

```
wave create -driver freeze -pattern constant -value 1111 -range 7 0 -starttime 0ns  
-endtime 1000ns sim:/andm/v_cont2
```

## Related Topics

[wave edit](#)

[wave modify](#)

[Generating Stimulus with Waveform Editor](#)

[Accessing the Create Pattern Wizard](#)

# wave edit

This command modifies waveforms created with the wave create command.

## Syntax

```
wave edit {cut | copy | paste | invert | mirror} -end {<time><unit>} -start {<time><unit>}  
    <object_name>  
wave edit insert_pulse [-duration {<time><unit>}] -start {<time><unit>} <object_name>  
wave edit delete -time {<time><unit>} <object_name>  
wave edit stretch | move {-backward {<time><unit>} | -forward {<time><unit>}}  
    -time {<time><unit>} <object_name>  
wave edit change_value -end {<time><unit>} -start {<time><unit>} <value> <object_name>  
wave edit extend -extend to | by -time {<time><unit>}  
wave edit driveType -driver freeze | deposit | driver | expectedoutput -end {<time><unit>}  
    -start {<time><unit>}  
wave edit undo <number>  
wave edit redo <number>
```

## Description

Some arguments to this command are order-dependent. Please read through the argument descriptions for more information.

The following table summarizes the available editing options:

Command	Description
wave edit cut	Cut part of a waveform to the clipboard
wave edit copy	Copy part of a waveform to the clipboard
wave edit paste	Paste the waveform from the clipboard
wave edit invert	Vertically flip part of a waveform
wave edit mirror	Mirror part of a waveform
wave edit insert_pulse	Insert a new edge on a waveform; doesn't affect waveform duration
wave edit delete	Delete an edge from a waveform; doesn't affect waveform duration
wave edit stretch	Move an edge by stretching the waveform
wave edit move	Move an edge without moving other edges
wave edit change_value	Change the value of part of a waveform
wave edit extend	Extend all waves

Command	Description
wave edit driveType	Change the driver type
wave edit undo	Undo an edit
wave edit redo	Redo a previously undone edit

## Arguments

- **-backward {<time><unit>}**  
(required if -forward <time> isn't specified) The amount to stretch or move the edge backwards in simulation time.  
  
*<time>* — Specified as an integer or decimal number.  
*<unit>* — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting *<unit>*. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If *<unit>* is specified, you must enclose *<time>* and *<unit>* within curly braces ({}).
- **cut | copy | paste | invert | mirror**  
(required) Specifies the type of edit to perform.  
  
*cut* — Deletes the specified portion of the waveform.  
*copy* — Saves a copy of the specified portion of the waveform.  
*paste* — Inserts the contents of the clipboard into the specified portion of the waveform.  
*invert* — Flips the specified portion of the waveform vertically.  
*mirror* — Flips the specified portion of the waveform horizontally.
- **-driver freeze | deposit | driver | expectedoutput**  
(required) Specifies the type of driver to which you want the specified section of the waveform changed. Applies to signals of type inout or internal.
- **-duration {<time><unit>}**  
(optional) The length of the pulse.  
  
*<time>* — Specified as an integer or decimal number where the default is 10 time units.  
*<unit>* — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting *<unit>*. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If *<unit>* is specified, you must enclose *<time>* and *<unit>* within curly braces ({}).
- **-end {<time><unit>}**  
(required unless specifying paste) The end of the section of waveform to perform the editing operation upon, denoted by a simulation time.  
  
*<time>* — Specified as an integer or decimal number.

**<unit>** — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting **<unit>**. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If **<unit>** is specified, you must enclose **<time>** and **<unit>** within curly braces ({}).

- **-extend to | by**

(required) Specifies the format for extending waves.

**to** — Extends the wave to the time specified by **-time <time>**.

**by** — Extends the wave by the amount of time specified by **-time <time>**.

- **-forward {<time><unit>}**

(required if **-backward <time>** is not specified) The amount to stretch or move the edge forwards in simulation time.

**<time>** — Specified as an integer or decimal number.

**<unit>** — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting **<unit>**. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If **<unit>** is specified, you must enclose **<time>** and **<unit>** within curly braces ({}).

- **<number>**

(optional) The number of editing operations to undo or redo. If omitted, only one editing operation is undone or redone.

- **<object\_name>**

(required) The pathname of the waveform to edit. Must be specified as the last argument to **wave edit**.

- **-start {<time><unit>}**

(required) The beginning of the section of waveform to perform the editing operation upon, denoted by a simulation time.

**<time>** — Specified as an integer or decimal number.

**<unit>** — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting **<unit>**. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If **<unit>** is specified, you must enclose **<time>** and **<unit>** within curly braces ({}).

- **-time {<time><unit>}**

(required) The amount of time to extend or stretch waves.

**<time>** — Specified as an integer or decimal number.

**<unit>** — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting **<unit>**. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If **<unit>** is specified, you must enclose **<time>** and **<unit>** within curly braces ({}).

- <value>

(required) The new value. Must match the type of the <object\_name>.

## Related Topics

[wave create](#)

[Generating Stimulus with Waveform Editor](#)

# wave export

This command creates a stimulus file from waveforms created with the wave create command.

## Syntax

```
wave export -designunit <name> -starttime {<time><unit>} -endtime {<time><unit>} -file <filename> {-format force | vcd | vhdl | verilog}
```

## Arguments

- **-designunit <name>**

(required) Specifies a design unit for which you want to export created waves. If omitted, the command exports waves from the active design unit.

    <name> — Specifies a design unit in the simulation.

- **-endtime {<time><unit>}**

(required) The simulation time at which you want to stop exporting.

    <time> — Specified as an integer or decimal number.

    <unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <time> and <unit> within curly braces ({}).

- **-file <filename>**

(required) The filename for the saved export file.

    <name> — Any user specified string.

- **-format force | vcd | vhdl | verilog**

(required) The format of the saved stimulus file. The format options include:

    force — A Tcl script that recreates the waveforms. The file should be sourced when reloading the simulation.

    vcd — An extended VCD file. Load using the -vcdstim argument to vsim.

    vhdl — A VHDL test bench. Compile and load the file as your top-level design unit.

    verilog — A Verilog test bench. Compile and load the file as your top-level design unit.

- **-starttime {<time><unit>}**

(required) The simulation time at which you want to start exporting.

    <time> — Specified as an integer or decimal number.

    <unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <time> and <unit> within curly braces ({}).

## Related Topics

[wave create](#)

[wave import](#)

[Generating Stimulus with Waveform Editor](#)

## wave import

This command imports an extended VCD file that was created with the wave export command. It cannot read extended VCD file created by software other than ModelSim. Use this command to apply a VCD file as stimulus to the current simulation.

### Syntax

`wave import <VCD_file>`

### Arguments

- `<VCD_file>`  
(required) The name of the extended VCD file to import.

### Related Topics

[wave create](#)

[wave export](#)

[Generating Stimulus with Waveform Editor](#)

## wave modify

This command modifies waveform parameters set by a previous wave create command.

### Syntax

#### All waveforms

```
wave modify [-driver freeze | deposit | driver | expectedoutput] [-endtime {<time><unit>}]  
[-initialvalue <value>] [-portmode {input | output | inout | internal}] [-range <msb lsb>]  
[-starttime {<time><unit>}] <wave_name>
```

#### Clock patterns only

```
wave modify -pattern clock -period <value> -dutycycle <value> <wave_name>
```

#### Constant patterns only

```
wave modify -pattern constant [-driver freeze | deposit | driver | expectedoutput]  
[-language {vhdl | verilog}] [-value <value>] <wave_name>
```

#### Counter patterns only

```
wave modify -pattern counter -period <value> -repeat forever | <n> | never -startvalue <value> -  
step <value> [-direction {up | down | upthendown | downthenup}]  
[-endvalue <value>] [-type {binary | gray | johnson | onehot | range | zerohot}]  
<wave_name>
```

#### Random patterns only

```
wave modify -pattern random -period <value>  
-random_type exponential | normal | poisson | uniform [-seed <value>] <wave_name>
```

#### Repeater patterns only

```
wave modify -pattern repeater -period <value> -repeat forever | <n> | never  
-sequence {val1 val2 val3 ...} <wave_name>
```

#### No pattern

```
wave create -pattern none <wave_name>
```

### Description

Some arguments to this command are order-dependent. Please read through the argument descriptions for more information.

The following table summarizes the available wave modification options:

Command	Description
wave modify -pattern clock	Generates a clock waveform. Specify an initial value, duty cycle, and clock period for the waveform.
wave modify -pattern constant	Generates a waveform with a constant value. Specify a value.

Command	Description
wave modify -pattern counter	Generates a waveform from a counting pattern. Specify start and end values, repeat, step count, time period, and type (Binary, Gray, Johnson, OneHot, Range, and ZeroHot).
wave modify -pattern random	Generates a random waveform based upon a seed value. Specify the type (normal or uniform), an initial value, and a seed value. If you don't specify a seed value, Questa uses a default value of 5.
wave modify -pattern repeater	Generates a waveform that repeats. Specify an initial value and pattern that repeats. You can also specify how many times the pattern repeats.
wave modify -pattern none	Creates a placeholder for a waveform. Specify an object name.

## Arguments

- **-direction {up | down | upthendown | downthenup}**  
(optional, recommended when specifying -pattern counter) The direction in which the counter will increment or decrement.
  - up — (default) Increment only.
  - down — Decrement only.
  - upthendown — Increment then decrement.
  - downthenup — Decrement then increment.
- **-driver freeze | deposit | driver | expectedoutput**  
(optional) Specifies that the signal is a driver of the specified type. Applies to signals of type inout or internal.
- **-dutycycle <value>**  
(required) The duty cycle of the clock, expressed as a percentage of the period that the clock is high.
  - <value> — Any integer from 0 to 100 where the default is 50.
- **-endtime {<time><unit>}**  
(optional) The simulation time that the waveform should stop. If omitted, the waveform stops at 1000 simulation time units.
  - <time> — Specified as an integer or decimal number.
  - <unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <time> and <unit> within curly braces ({}).

- **-endvalue <value>**  
(optional) The ending value of the counter. This option applies to Range counter patterns only. All other counter patterns start from 0 and go to the max value for that particular signal (for example, for a 3-bit signal, the start value will be 000 and end value will be 111).  
    <value> — Any positive integer.
- **-initialvalue <value>**  
(optional) The initial value for the waveform. Value must be appropriate for the type of waveform you are creating. Not applicable to counter patterns.  
    <value> — Any positive integer.
- **-language {vhdl | verilog}**  
(optional) Controls which language is used for modifying the wave.  
    vhdl — (default) Specifies the VHDL language.  
    verilog — Specifies the Verilog language.
- **-period <value>**  
(required) The period of the signal.
- **-portmode {input | output | inout | internal}**  
(optional) The port type for the waveform.  
    in — Ports of type IN. You can also specify “input” as an alias for in.  
    out — Ports of type OUT. You can also specify “output” as an alias for out.  
    inout — Ports of type INOUT.  
    internal — (default) Ports of type INTERNAL.
- **-random\_type exponential | normal | poisson | uniform**  
(required) Specifies a random pattern to generate.  
    exponential — Exponential distribution of waveform events.  
    normal — Normal or Gaussian distribution of waveform events.  
    poisson — Poisson distribution of waveform events.  
    uniform — (default) Uniform distribution of waveform events.
- **-range <msb lsb>**  
(optional) Identifies bit significance in a counter pattern.  
    msb lsb — Most significant bit and least significant bit. Both must be specified.
- **-repeat forever | <n> | never**  
(required) Controls duration of pattern repetition.  
    forever — Repeat the pattern for as long as the simulation runs.  
    <n> — Repeat the pattern <n> number of times where <n> is any positive integer.

never — Never repeat the pattern during simulation.

- **-seed <value>**

(optional) Specifies a seed value for a randomly generated waveform.

<value> — Any non-negative integer where the default is 5.

- **-sequence {val1 val2 val3 ...}**

(required) The set of values that you want repeated.

<val1> — Value must be appropriate for the type of waveform you are creating.

Multiple values are entered as a space separated list and enclosed in curly braces ({}).

- **-starttime {<time><unit>}**

(optional) The simulation time that the waveform should start. If omitted, the waveform starts at 0 simulation time units.

<time> — Specified as an integer or decimal number.

<unit> — (optional) A suffix specifying a unit of time where the default is to specify the current simulation resolution by omitting <unit>. Valid time units are: fs, ps, ns, us, ms, sec, min, and hr. If <unit> is specified, you must enclose <time> and <unit> within curly braces ({}).

- **-startvalue <value>**

(required when specifying -pattern counter) The initial value of the counter. This option applies to patterns specifying -type Range only. All other counter patterns start from 0 and go to the maximum value for that particular signal (e.g., for a 3-bit signal, the start value will be 000 and the end value will be 111).

<value> — Value must be appropriate for the type of waveform you are creating.

- **-step <value>**

(required) The step by which the counter is incremented/decremented.

<value> — Value must be appropriate for the type of waveform you are creating.

- **-type {binary | gray | johnson | onehot | range | zerohot}**

(optional) Specifies a counter format.

binary — Specifies a binary counter.

gray — Specifies a binary counter where two successive values differ in only one bit.

Also known as a reflected binary counter.

johnson — Specifies a twisted ring or Johnson counter.

onehot — Specifies a shift counter where only one bit at a time is set to “on” (1).

range — (default) Specifies a binary counter where the values range between -startvalue and -endvalue

zerohot — Specifies a shift counter where only one bit at a time is set to “off” (0).

- **-value <value>**  
(optional, recommended when specifying -pattern constant) Specifies a value for the constant pattern.  
  
    <value> — Value must be appropriate for the type of waveform you are creating.
- **<wave\_name>**  
(required) The name of an existing waveform created with the [wave create](#) command.

## Related Topics

[wave create](#)

[Generating Stimulus with Waveform Editor](#)

[Accessing the Create Pattern Wizard](#)

## wave sort

This command sorts signals in the Wave window by name or full path name.

### Syntax

**wave sort {ascending | descending | fa | fd}**

### Arguments

- **ascending | descending | fa | fd**

(required) Sort signals in one of the following orders.

ascending — Sort in ascending order by signal name.

descending — Sort in descending order by signal name.

fa — Sort in ascending order by the full path name.

fd — Sort in descending order by full path name.

### Examples

**wave sort ascending**

## when

This command instructs ModelSim to perform actions when specified conditions are met.

### Syntax

```
when [[-fast] [-id <id#>] [-label <label>] [-repeat] {<when_condition_expression>} {<command>}]
```

### Description

Use this command to control ModelSim activity for one or more specified conditions.

For example, you can use the command to break on a signal value or at a specific simulator time. Use the [nowhen](#) command to deactivate when commands.

The when command uses a when\_condition\_expression to determine whether or not to perform the action. Conditions can include VHDL signals and Verilog nets and registers. The when\_condition\_expression uses a simple restricted language (that is not related to Tcl), which permits only four operators and operands that may be either HDL object names, signame'event, or constants. ModelSim evaluates the condition every time any object in the condition changes, hence the restrictions.

Here are some additional points to keep in mind about the when command:

- The when command creates the equivalent of a VHDL process or a Verilog always block. It does not work like a looping construct you might find in other languages such as C.
- Virtual signals, functions, regions, types, and so forth, cannot be used in the when command. Neither can simulator state variables other than \$now.
- With no arguments, when will list the currently active when statements and their labels (explicit or implicit).

### Embedded Commands Allowed with the -fast Argument

You can use any Tcl command as a <command>, along with any of the following vsim commands:

- bp, bd
- change
- disablebp, enablebp
- echo
- examine
- force, noforce

- log, nolog
- stop
- when, nowhen

### Embedded Commands Not Allowed with the -fast Argument

- Any do commands
- Any Tk commands or widgets
- References to U/I state variables or tcl variables
- Virtual signals, functions, or types

### Using Global Tcl Variables with the -fast Argument

Embedded commands that use global Tcl variables for passing a state between the when command and the user interface need to declare the state using the Tcl uivar command. For example, the variable *i* below is visible in the GUI. From the command prompt, you can display it (by entering echo \$i) or modify it (for example, by entering set i 25).

```
set i 10
when -fast {clk == '0'} {
    uivar i
    set i [expr {$i - 1}]
    if {$i <= 0} {
        force reset 1 0, 0 250
    }
}
when -fast {reset == '0'} {
    uivar i
    set i 10
}
```

### Additional Restrictions on the -fast Argument

Accessing channels (such as files, pipes, sockets) that were opened outside of the embedded command will not work. For example:

```
set fp [open mylog.txt w]
when -fast {bus} {
    puts $fp "bus change: [examine bus]"
}
```

The channel that \$fp refers to is not available in the simulator, only in the user interface. Even using the uivar command does not work here because the value of \$fp has no meaning in the context of the -fast argument.

The following method of rewriting this example opens the channel, writes to it, then closes it within the when command:

```
when -fast {bus} {
    set fp [open mylog.txt a]
    puts $fp "bus change: [examine bus]"
    close $fp
}
```

The following example is a little more sophisticated method of doing the same thing:

```
when -fast {$now == 0ns} {
    set fp [open mylog.txt w]
}
when -fast {bus} {
    puts $fp "bus change: [examine bus]"
}
when -fast {$now == 1000ns} {
    close $fp
}
```

The general principle is that any embedded command done using the -fast argument is global to all other commands used with the -fast argument. Here, {\$now == 0ns} is a way to define Tcl processes that the -fast commands can use. These processes have the same restrictions that when bodies have, but the advantage is again speed as a proc will tend to execute faster than code in the when body itself.

It is recommended not to use virtual signals and expressions.

## Arguments

- **-fast**  
(optional) Causes the embedded <command> to execute within the simulation kernel, which provides faster execution and reduces impact on simulation runtime performance.  
Limitations on using the -fast argument are described above (in “[Embedded Commands Not Allowed with the -fast Argument](#)”). Disallowed commands still work, but they slow down the simulation.
- **-label <label>**  
(optional) Used to identify individual when commands.  
  
<label> — Associates a name or label with the specified when command. Adds a level of identification to the when command. The label may contain special characters.  
Quotation marks (" ") or braces ({ }) are required only if <label> contains spaces or special characters.
- **-id <id#>**  
(optional) Attempts to assign this id number to the when command.

<id#> — Any positive integer that is not already assigned. If the id number you specify is already used, ModelSim will return an error.

---

**Note**

 Id numbers for when commands are assigned from the same pool as those used for the bp command. So even if you have not specified a given id number for a when command, that number may still be used for a breakpoint.

---

- -repeat

(Limited to “when” breakpoint expressions involving “\$now”). Instructs the command to reestablish the breakpoint when triggered so that it will fire again for the next time period. Without this argument, expressions using \$now only trigger once.

- {<when\_condition\_expression>}

(required if a command is specified) Specifies the conditions to be met for the specified <command> to be executed. The condition is evaluated in the simulator kernel and can be an object name, in which case the curly braces can be omitted. The command will be executed when the object changes value. The condition can be an expression with these operators:

Name	Operator
equals	==, =
not equal	!=, /=
greater than	>
less than	<
greater than or equal	>=
less than or equal	<=
AND	&&, AND
OR	, OR

The operands may be object names, signame'event, or constants. Subexpressions in parentheses are permitted. The command will be executed when the expression is evaluated as TRUE or 1.

The formal BNF syntax is:

```
condition ::= Name | { expression }

expression ::= expression AND relation
            | expression OR relation
            | relation
```

```
relation ::= Name = Literal
           | Name /= Literal
           | Name ' EVENT
           | ( expression )

Literal ::= '<char>' | "<bitstring>" | <bitstring>
```

The " $=$ " operator can occur only between a Name and a Literal. This means that you cannot compare the value of two signals, i.e.,  $Name = Name$  is not possible.

Tcl variables can be used in the condition expression but you must replace the curly braces ( $\{\}$ ) with double quotes (""). This works like a macro substitution where the Tcl variables are evaluated once and the result is then evaluated as the when condition. Condition expressions are evaluated in the vsim kernel, which knows nothing about Tcl variables. That's why the condition expression must be evaluated in the GUI before it is sent to the vsim kernel. See below for an example of using a Tcl variable.

The " $>$ ", " $<$ ", " $\geq$ ", and " $\leq$ " operators are the standard ones for vector types, not the overloaded operators in the std\_logic\_1164 package. This may cause unexpected results when comparing objects that contain values other than 1 and 0. ModelSim does a lexical comparison (position number) for values other than 1 and 0. For example:

```
0000 < 1111 ## This evaluates to true
H000 < 1111 ## This evaluates to false
001X >= 0010 ## This also evaluates to false
```

- {<command>}

(required if a when expression is specified) The command(s) for this argument are evaluated by the Tcl interpreter within the ModelSim GUI. Any ModelSim or Tcl command or series of commands are valid with one exception—the **run** command cannot be used with the when command. The command sequence usually contains a **stop** command that sets a flag to break the simulation run after the command sequence is completed. Multiple-line commands can be used.

---

**Note**

 If you want to stop the simulation using a when command, you must use a **stop** command within your when statement. DO NOT use an **exit** command or a quit command. The stop command acts like a breakpoint at the time it is evaluated.

---

## Examples

- The when command below instructs the simulator to display the value of object *c* in binary format when there is a clock event, the clock is 1, and the value of *b* is 01100111. Finally, the command tells ModelSim to stop.

```
when -label when1 {clk'event and clk='1' and b = "01100111"} {
    echo "Signal c is [exa -bin c]"
    stop
}
```

- The when command below echoes the simulator time when slice [3:1] of wire [15:0] count matches the hexadecimal value 7, and simulation time is between 70 and 111 nanoseconds.

```
when {$now > 70ns and count(3:1) == 3'h7 && $now < 111ns} {  
    echo "count(3:1) matched 3'h7 at time " $now  
}
```

- The commands below show an example of using a Tcl variable within a when command. Note that the curly braces ({} ) have been replaced with double quotes ("").

```
set clk_b_path /tb/ps/dprb_0/udprb/ucar_reg/uint_ram/clkb;  
when -label when1 "$clk_b_path'event and $clk_b_path ='1" {  
    echo "Detected Clk edge at path $clk_b_path"  
}
```

- The when command below is labeled *a* and will cause ModelSim to echo the message “*b* changed” whenever the value of the object *b* changes.

```
when -label a b {echo "b changed"}
```

- The multi-line when command below does not use a label and has two conditions. When the conditions are met, ModelSim runs an echo command and a stop command.

```
when {b = 1  
    and c /= 0 } {  
    echo "b is 1 and c is not 0"  
    stop  
}
```

- In the example below, for the declaration "wire [15:0] a;", the when command will activate when the selected bits match a 7:

```
when {a(3:1) = 3'h7} {echo "matched at time " $now}
```

- In the example below, we want to sample the values of the address and data bus on the first falling edge of clk after sstrb has gone high.

```
# ::strobe is our state variable
set ::strobe Zero
# This signal breakpoint only fires when sstrb changes to a '1'
when -label checkStrobe {/top/sstrb == '1'} {
    # Our state Zero condition has been met, move to state One
    set ::strobe One
}
# This signal breakpoint fires each time clk goes to '0'
when {/top/clk == '0'} {
    if {$::strobe eq "One"} {
        # Our state One condition has been met
        # Sample the busses
        echo Sample paddr=[examine -hex /top/paddr] :: sdata=[examine
-hex
        /top/sdata]
        # reset our state variable until next rising edge of sstrb
        (back to
            state Zero)
        set ::strobe Zero
    }
}
```

### Ending the simulation with the stop command

Batch mode simulations are often structured as "run until condition X is true," rather than "run for X time" simulations. The multi-line when command (shown below) sets a done condition, and ModelSim runs an echo command and a stop command when the condition is reached.

The simulation will not stop (even if a quit -f command is used) unless you enter a stop command. To exit the simulation and quit ModelSim, use an approach like the following:

```
onbreak {resume}
when {/done_condition == '1'} {
    echo "End condition reached"
    if [batch_mode] {
        set DoneConditionReached 1
        stop
    }
}
run 1000 us
if {$DoneConditionReached == 1} {
    quit -f
}
```

This example stops 100ns after a signal transition:

```
when {a = 1} {
    # If the 100ns delay is already set then let it go.
    if {[when -label a_100] == ""} {
        when -label a_100 { $now = 100 } {
            # delete this breakpoint then stop
            nowhen a_100
            stop
        }
    }
}
```

### Time-based breakpoints

You can build time-based breakpoints into a when statement with the following syntax.

For absolute time (indicated by @) use:

```
when {$now = @1750 ns} {stop}
```

You can also use:

```
when {errorFlag = '1' OR $now = 2 ms} {stop}
```

This example adds 2 ms to the simulation time at which the when statement is first evaluated, then stops. The white space between the value and time unit is required for the time unit to be understood by the simulator.

You can also use variables, as shown in the following example:

```
set time 1000
when "\$now = $time" {stop}
```

The quotes instruct Tcl to expand the variables before calling the command. So, the when command sees:

```
when "$now = 1000" stop
```

Note that "\$now" has the '\$' escaped. This prevents Tcl from expanding the variable, because if it did, you would get:

```
when "0 = 1000" stop
```

## where

This command displays information about the system environment. It is useful for debugging problems where ModelSim cannot find the required libraries or support files.

### Syntax

where

### Description

The command displays two results on consecutive lines:

- current directory

This is the current directory that ModelSim was invoked from, or that was specified on the ModelSim command line.

- current project file

This is the *.mpf* file ModelSim is using. All library mappings are taken from here when a project is open. If the design is not loaded through a project, this line displays the *modelsim.ini* file in the current directory.

### Arguments

None.

### Examples

- Design is loaded through a project:

**VSIM> where**

Returns:

```
# Current directory is: D:\Client
# Project is: D:/Client/monproj.mpf
```

- Design is loaded with no project (indicates the *modelsim.ini* file is under the *mydesign* directory):

**VSIM> where**

Returns:

```
# Current directory is: C:\Client\testcase\mydesign
# Project is: modelsim.ini
```

## wlf2log

This command translates a ModelSim WLF file (*vsim.wlf*) to a QuickSim II logfile. It reads the *vsim.wlf* WLF file generated by the add list, add wave, or log commands in the simulator and converts it to the QuickSim II logfile format.

---

**Note**

 This command should be invoked only after you have stopped the simulation using `quit -sim` or `dataset close sim`.

---

### Syntax

```
wlf2log <wlffile> [-bits] [-fullname] [-help] [-inout] [-input] [-internal] [-l <instance_path>]  
[-lower] [-o <outfile>] [-output] [-quiet]
```

### Arguments

- <wlffile>  
(required) Specifies the ModelSim WLF file that you are converting.
- -bits  
(optional) Forces vector nets to be split into 1-bit wide nets in the log file.
- -fullname  
(optional) Shows the full hierarchical pathname when displaying signal names.
- -help  
(optional) Displays a list of command options with a brief description for each.
- -inout  
(optional) Lists only the inout ports. This may be combined with the -input, -output, or -internal switches.
- -input  
(optional) Lists only the input ports. This may be combined with the -output, -inout, or -internal switches.
- -internal  
(optional) Lists only the internal signals. This may be combined with the -input, -output, or -inout switches.
- -l <instance\_path>  
(optional) Lists the signals at or below an HDL instance path within the design hierarchy.  
    <instance\_path> — Specifies an HDL instance path.

- **-lower**  
(optional) Shows all logged signals in the hierarchy. When invoked without the -lower switch, only the top-level signals are displayed.
- **-o <outfile>**  
(optional) Directs the output to be written to a file where the default destination for the logfile is standard out.  
    <outfile> — A user specified filename.
- **-output**  
(optional) Lists only the output ports. This may be combined with the -input, -inout, or -internal switches.
- **-quiet**  
(optional) Disables error message reporting.

## wlf2vcf

This command translates a ModelSim WLF file to a standard VCD file. Complex data types that are unsupported in the VCD standard (records, memories, etc.) are not converted.

---

### Note

---

 This command should be invoked only after you have stopped the simulation using [quit -sim](#) or [dataset close sim](#).

---

### Syntax

```
wlf2vcf <wlffile> [-help] [-o <outfile>] [-quiet]
```

### Arguments

- <wlffile>  
(required) Specifies the ModelSim WLF file that you are converting.
- -help  
(optional) Displays a list of command options with a brief description for each.
- -o <outfile>  
(optional) Specifies a filename for the output where the default destination for the VCD output is stdout.  
    <outfile> — A user specified filename.
- -quiet  
(optional) Disables warning messages that are produced when an unsupported type (for example, records) is encountered in the WLF file.

## wlfman

This is a collection of related commands you can use to get information about saved WLF files and perform various actions on them.

### Syntax

```
wlfman info <source_wlffile> [-v]
wlfman items <source_wlffile> [-n] [-v]
wlfman filter -o <out_wlffile> <source_wlffile> [-begin <time>] [-end <time>]
  [-collapsedelta | -collapsetime | -nocollapse] [-compress | -nocompress]
  [-f <object_list_file>] [-index | -noindex] [-r <object>] [-nowarn <number>] [-opt | -noopt]
  [-s <symbol>] [-t <resolution>]
wlfman profile <source_wlffile> [-rank] [-top <number>]
wlfman merge -o <out_wlffile> [<wlffile1> <wlffile2> ...] [-compress | -nocompress]
  [-index | -noindex] [-opt | -noopt]
wlfman monitor [-f | -i <intervalTime> | -p <endTime>] [-q | -v] <source_wlffile>
wlfman optimize -o <out_wlffile> <source_wlffile> [-compress | -nocompress]
  [-index | -noindex] [-opt | -noopt]
```

### Description

The following wlfman commands perform different functions on saved WLF files:

- wlfman info returns file information, resolution, versions, and so forth about the source WLF file.
- wlfman items generates a list of HDL objects (i.e., signals) and/or transaction streams from the source WLF file and outputs it to stdout. When redirected to a file, the output is called an object\_list\_file, and it can be read in by wlfman filter. Comments start with a '#' and continue to the end of the line. Wildcards are legal in the leaf portion of the name. Here is an example:

```
/top/foo # signal foo
/top/u1/* # all signals under u1
/top/u1 # same as line above
-r /top/u2 # recursively, all signals under u2
/top/stream1 # transaction stream stream1
```

object\_list\_files — The object\_list\_file is a list of objects and/or transaction streams, one per line. Transaction object lists can include a stream name, stream array name, or sub stream, all with a full path. Transaction objects recorded in the object\_list\_file:

Full path of a stream array — Logs and records all of the individual streams in the specified stream array, the sub-streams of each stream, and the phase sub-streams recursively for each

sub-stream including the attributes of the transactions present in the sub-streams and phase sub-streams. For example:

```
/top/<stream_array> # stream array full path (ex: /top/stream_array)
```

Full path of a stream object — Logs and records all of the sub-streams and recursively records all phase sub-streams for each specified sub-stream including the attributes of the transactions present in the sub-streams and phase sub-streams. For example:

```
/top/<stream_object> # individual stream full path (ex: /top/stream)
```

Sub-streams — Logs and records all of the attributes of the specified sub-stream. Other sub-streams of the main stream are not logged. Phase sub-streams cannot be individually logged. For example:

```
/top/<stream_object> # individual stream full path (ex: /top/stream)
```

```
/top/<stream_object>.<sub-stream> # sub-stream full path
```

(ex: /top/stream.s0)

---

**Note**

 You can produce these files from scratch but be careful with syntax. It is recommended that you use wlfman items as it always creates a legal object\_list\_file.

---

- wlfman filter reads in a WLF file and, optionally, an object\_list\_file, and writes a new WLF file containing filtered information from those sources. You determine the filtered information with the arguments you specify.
- wlfman monitor returns the current state of a WLF file to the transcript. Each time the state is monitored, a line of information is output. The state of the WLF file can be monitored at regular intervals, indicating the changes over time. For example:

```
wlfman monitor visim.wlf
File      Sim
State     Time
closed   14000
```

- wlfman profile generates a report of the estimated percentage of file space that each signal is taking in the specified WLF file. This command can identify signals that account for a large percentage of the WLF file size (such as a logged memory that uses a zero-delay integer loop to initialize the memory). You may be able to drastically reduce WLF file size by not logging those signals.

When the WLF file contains transaction streams and/or stream arrays, wlfman profile generates an additional report of estimated file space used for each transaction as a percentage of total file size. You may be able to drastically reduce WLF file size by not logging some transactions or streams.

The stated size of a transaction is equal to the size of the transaction without any user attributes plus the sum of the sizes of every attribute in that transaction. Also, an assumption is made that every transaction will have its attributes recorded if one of the transactions in a sub-stream has that attribute. If the object is a stream array, the sum of the sizes of all the streams is presented in which case the sizes of the individual elements are not presented.

- `wlfman merge` combines two WLF files with different signals or transaction objects into one WLF file. It *does not* combine wlf files containing the same signals at different runtime ranges (for example, `mixedhdl_0ns_100ns.wlf` & `mixedhdl_100ns_200ns.wlf`). When merging two WLF files containing the same transaction streams, the first stream's data is recorded, the second stream is ignored, and a warning is issued that a horizontal merge is not supported.
- `wlfman optimize` copies the data from the WLF file to the output WLF file, adding or replacing the indexing and optimization information.

The different command are intended to be used together. For example, you might run `wlfman profile` and identify a signal or transaction stream that accounts for 50% of the WLF file size. If you do not actually need that object, you can then run `wlfman filter` to remove it from the WLF file.

## Arguments

- `-o <out_wlffile>`  
(required) Specifies the name of the output WLF file. The output WLF file will contain all objects specified by the preceding arguments. Output WLF files are always written in the latest WLF version regardless of the source WLF file version.
- `<source_wlffile>`  
(required) Specifies the WLF file from which you want information.
- `<wlffile1> <wlffile2> ...`  
(required) Specifies the WLF files whose objects you want to copy into one WLF file. Specified as a space separated list.
- `-begin <time>`  
(optional) Specifies the simulation time at which to start reading information from the source WLF file where the default is to include the entire length of time recorded in `<source_wlffile>`. If a transaction on a particular stream started prior to the time specified, then that transaction is ignored.  
  
`<time>` —
- `-collapsedelta | -collapsetime | -nocollapse`  
(optional) Controls preservation of events in the resulting WLF file. The data preserved depends on how events were recorded in the input WLF file. Specifying a finer granularity of preservation than the input WLF file will have no additional affect.

- collapsedelta — (default) Preserves only the values at the end of a delta.
- collapsetime — Preserves only the values at the end of a time step.
- nocollapse — Preserves all events.
- -compress | -nocompress
  - (optional) Controls compression of the output WLF file.
    - compress — Enables compression. (default)
    - nocompress — Disables compression.
- -end <time>
  - (optional) Specifies the simulation time at which filtering of <source\_wlffile> is stopped and no further data is logged.
- -f
  - (optional) Repeat status update every 10 seconds of real time unless an alternate time interval is specified with -i <intervalTime>.
- -f <object\_list\_file>
  - (optional) Specifies an object\_list\_file created by wlfman items or by the user to include in <out\_wlffile>.

For user created object list files, the object list can include stream name, stream array name, or sub stream with a full path. All
- -i <intervalTime>
  - (optional) Specifies the time delay before the next status update where the default is 10 seconds of real time if not specified.
    - <intervalTime> — Any positive integer.
- -index | -noindex
  - (optional) Controls indexing when writing the output WLF file. Indexing makes viewing wave data faster, however performance during optimization will be slower because indexing and optimization require significant memory and CPU resources. Disabling indexing makes viewing wave data slower unless the display is near the start of the WLF file. Disabling indexing also disables optimization of the WLF file but may provide a significant performance boost when archiving WLF files. Indexing and optimization information can be added back to the file using the wlfman optimize command.
    - index — Enables indexing. (default)
    - noindex — Disables indexing and optimization.
- -n
  - (optional) Lists regions only (no signals).
- -nowarn <number>
  - (optional) Selectively disables a category of warning messages.

1 — Disables "Skipping unsupported object" warning message.

- **-opt | -noopt**  
(optional) Controls optimization of the output WLF file.
  - opt — Enables WLF file optimization. (default)
  - noopt — Disables WLF file optimization.
- **-p <endTime>**  
(optional) Specifies the simulation time at which wlfman will stop monitoring the WLF file.  
<endTime> — Any positive integer.
- **-q**  
(optional) Suppress normal status messages while monitoring.
- **-r <object>**  
(optional) Specifies an object (region) to recursively include in the output. If <object> is a signal, the output would be the same as using -s.
- **-rank**  
(optional) Sorts the wlfman profile report by percentage of the total file space used by each signal.
- **-s <symbol>**  
(optional) Specifies an object to include in the output. By default all objects are included.
- **-t <resolution>**  
(optional) Specifies the time resolution of the new WLF file. By default the resolution is the same as the source WLF file.
- **-top <number>**  
(optional) Filters the wlfman profile report so that only the top <number> signals in terms of file space percentage are displayed.
- **-v**  
(optional) Produces verbose output that lists the object type next to each object.

## Examples

- Specifying the command:

```
wlfman profile -rank top_vh.wlf
```

returns:

```
#Repeated ID #'s mean those signals share the same
#space in the wlf file.
#
# ID      Transitions   File %  Name
#-----  -----
#      1          2192      33 %  /top_vh/pdata
#      1          1224      18 %  /top_vh/ptrans
#      1          1216      18 %  /top_vh/sdata
#      3          675       10 %  /top_vh/strans
#      5          423       6  %  /top_vh/cache/gen_3/s/data_out
#      6          135       3  %  /top_vh/paddr.
#
#      .
#      .
#      .
```

- Specifying the command:

**wlfman profile -top 3 trans.wlf**

returns:

```
#The following table lists the number of transitions and approximate
#wlf file space consumed (prior to compression) for each signal
#logged in the wlf file.
#Repeated ID #'s mean those signals share the same
#space in the wlf file.
#
# ID      Transitions   File %  Name
#-----  -----
#      1          1001      11 %  /top/t3
#      2          1001      11 %  /top/t1
#      3          1         0 %   /top/s2
#The following table lists the number of transactions and
#approximate wlf file space consumed (prior to compression) for each
#stream or stream array logged in the wlf file.
#
# ID      Transactions   File %  Name
#-----  -----
#      1          3000      61 %  /top/stream2
#      1          1000      17 %  /top/stream1
```

- Specifying the command:

**wlfman monitor -f -p 1000000000 vsim.wlf**

Returns:

```
Setting end time to 100000000, measuring progress %
File      File      Percent
State     Time      Complete
open      7239185  7.2%
open      7691785  7.7%
open      8144385  8.1%
open      8596625  8.6%
```

## Related Topics

[Recording Simulation Results With Datasets](#)

[WLF File Parameter Overview](#)

## wlrecover

This command attempts to "repair" WLF files that are incomplete due to a crash or if the file was copied prior to completion of the simulation. Use this command if you receive a "bad magic number" error message when opening a WLF file. You can run the command from the VSIM> or ModelSim> prompt or from a shell.

### Syntax

```
wlrecover <filename> [-force] [-q]
```

### Arguments

- <filename>  
(required) Specifies the WLF file to repair.
- -force  
(optional) Disregards file locking and attempts to repair the file.
- -q  
(optional) Hides all messages unless there is an error while repairing the file.

### Related Topics

[Saving a Simulation to a WLF File](#)

## write format

This command records the names and display options of the HDL objects currently being displayed in the Analysis, List, Memory, Message Viewer, Test Browser, and Wave windows.

### Syntax

```
write format {<window_type>} <filename>  
write format restart [<option option1 ...>] <filename>
```

### Description

The file created is primarily a list of [add list](#) or [add wave](#) commands, though a few other commands are included (refer to "Output" below).

This file may be invoked with the [do](#) command to recreate the window format on a subsequent simulation run (refer to [restart](#) below).

Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

### Arguments

- <window\_type>  
(required unless specifying restart) Specifies that the contents of the designated window are recorded in the file specified by <filename>.
  - breakpoints — Records file line and signal breakpoints.
  - list — Records objects of the List window.
  - memory — Records objects of the Memory window.
  - msgviewer — Records objects of the Message Viewer window.
  - watch — Records objects of the Watch window.
  - wave — Records objects of the Wave window.
  - restart — Records objects of all windows and breakpoints in the *.do* file.
- restart  
(required) Creates a *.do* file that recreates all debug windows, all file/line breakpoints, and all signal breakpoints created using the [when](#) command. If the [ShutdownFile](#) *modelsim.ini* variable is set to this *.do* filename, it will call the write format restart command upon exit.

When you load a format file, ModelSim verifies the existence of the datasets required by that file. ModelSim displays an error message if the requisite datasets do not all exist. To force the execution of the format file even if all datasets are not present, use the *-force* switch with your *do* command. For example:

```
VSIM> do format.do -force
```

---

**Note**

---

 Note that using the -force switch when datasets are not present will result in error messages for signals referencing the nonexistent datasets. Also, -force is recognized by the format file not the do command.

---

- <option option1 ...>  
(optional) Excludes a specific type of information from write format restart *.do* file.
  - nobreak — Do not record breakpoints.
  - nolastnow — Do not report last now value.
  - nolist — Do not record the List window format.
  - nomemory — Do not record Memory window views.
  - nosource — Do not record source files.
  - novsim — Do not record the vsim command.
  - nowave — Do not record the Wave window format.
- <filename>  
(required) Specifies the name of the output file where the data is to be written. You must specify the *.do* extension.

## Examples

- Save the current data in the List window in a file named *alu\_list.do*.

```
write format list alu_list.do
```
- Save the current data in the Wave window in a file named *alu\_wave.do*.

```
write format wave alu_wave.do
```

- An example of a saved Wave window format file:

```
onerror {resume}
quietly WaveActivateNextPane {} 0
add wave -noupdate -format Logic /cntr_struct/ld
add wave -noupdate -format Logic /cntr_struct/rst
add wave -noupdate -format Logic /cntr_struct/clk
add wave -noupdate -format Literal /cntr_struct/d
add wave -noupdate -format Literal /cntr_struct/q
TreeUpdate [SetDefaultTree]
quietly WaveActivateNextPane
add wave -noupdate -format Logic /cntr_struct/p1
add wave -noupdate -format Logic /cntr_struct/p2
add wave -noupdate -format Logic /cntr_struct/p3
TreeUpdate [SetDefaultTree]
WaveRestoreCursors {0 ns}
WaveRestoreZoom {0 ns} {1 us}
configure wave -namecolwidth 150
configure wave -valuecolwidth 100
configure wave -signalnamewidth 0
configure wave -justifyvalue left
```

In the example above, five signals are added with the *-noupdate* argument to the default window. The TreeUpdate command then refreshes all five waveforms. The second WaveActivateNextPane command creates a second pane which contains three signals. The WaveRestoreCursors command restores any cursors you set during the original simulation, and the WaveRestoreZoom command restores the Zoom range you set. These four commands are used only in saved Wave format files; therefore, they are not documented elsewhere.

## write list

This command records the contents of the List window in a list output file.

### Syntax

**write list [-events] <filename>**

### Description

This file contains simulation data for all HDL objects displayed in the List window: VHDL signals and variables and Verilog nets and registers.

Arguments to this command are order-dependent. Please read through the argument descriptions for more information.

### Arguments

- **-events**  
(optional) Specifies to write print-on-change format where the default is tabular format.
- **<filename>**  
(required) Specifies the name of the output file where the data is to be written.

### Examples

- Save the current data in the List window in a file named *alu.lst*.

**write list alu.lst**

### Related Topics

[write tssi](#)

## write preferences

This command saves the current GUI preference settings to a Tcl preference file. Settings saved include Wave, Objects, and Locals window column widths; Wave, Objects, and Locals window value justification; and Wave window signal name width.

### Syntax

```
write preferences <preference file name>
```

### Arguments

- <preference file name>

(required) Specifies the name for the preference file. If the file is named *modelsim.tcl*, ModelSim will read the file each time vsim is invoked. To use a preference file other than *modelsim.tcl* you must specify the alternative file name with the [MODELSIM\\_TCL](#) environment variable.

You can modify variables by editing the preference file with the ModelSim [notepad](#):

```
notepad <preference file name>
```

# write report

This command prints a summary of the design being simulated including a list of all design units (VHDL configurations, entities, and packages, and Verilog modules) with the names of their source files. The summary includes a list of all source files used to compile the given design.

## Syntax

```
write report [-capacity [-l | -s] [-line] [-qdas | -vmem]] |  
[-l | -s] | [-tcl] | [<filename>]
```

## Description

The Simulation Report contains the following information:

- Design Simulated — directory path of the design's top-level module
- Number of signals/nets in the design
- Number of processes in the design
- Simulator Parameters, including:
  - Current directory
  - Project file directory
  - Simulation time resolution
- List of design units used, including:
  - Module name
  - Architecture, if applicable
  - Library directory
  - Source file
  - Timescale
  - Occurrences

Some arguments to this command are order-dependent. Please read through the argument descriptions for more information.

## Arguments

- **-capacity**

(optional) Reports data on memory usage of various types of SystemVerilog constructs in the design. Collects memory usage data for dynamic objects.

ModelSim collects memory usage data for dynamic objects.

Must be specified first when specifying -qdas.

To display memory data for all object types, specify -capacity -l.

- <filename>  
(optional) Specifies the name of the output file where the data is to be written. If <filename> is omitted, the report is written to the Transcript window.
- -l  
(optional) Generates more detailed information about the design, including a list of sparse memories or the memory capacity for all object types. You must precede this argument with -capacity when specifying a capacity report.
- -line  
(optional) Generates point of allocation (line) based report. If -line is not used, the report will be generated based on declaration. Vsim must be run with -capacity=line to print a point of allocation (line) based report.
- -qdas  
(optional) Reports memory usage data for queues, dynamic arrays, associative arrays, and strings (each is provided in its own section in the report). You must precede this argument with -capacity when specifying a capacity report.
- -s  
(optional) Generates a short list of design information. You must precede this argument with -capacity when specifying a capacity report.
- -tcl  
(optional) Generates a Tcl list of design unit information. This argument cannot be used with a filename.
- -vmem  
(optional) When specified with capacity, -vmem reports usage data for Verilog memories.

## Examples

- Save information about the current design in a file named *alu\_rpt.txt*.

**write report alu\_rpt.txt**

- Create a Simulation Report for the current simulation

**write report -l**

returns:

```
##  
## SIMULATION REPORT           Generated on Mon Aug 10 12:56:15 2009  
##  
##  
## Design simulated: <directory>\work.top(fast)  
## Number of signals/nets in design: 89  
## Number of processes in design: 74  
##  
## Simulator Parameters:  
##  
##      Current directory: <directory>\  
##      Project file: <directory>\win32/../modelsim.ini  
##      Simulation time resolution: 1ns  
##  
## List of Design units used:  
##  
##      Module: top  
##      Architecture: fast  
##      Library: <directory>\work  
##      Source File: top.v  
##      Timescale: 1ns / 1ns  
##      Occurrences: 1  
##  
##      Module: proc  
##      Architecture: fast  
##      Library: <directory>\work  
##      Source File: proc.v  
##      Timescale: 1ns / 1ns  
##      Occurrences: 1  
  
...
```

## write timing

This command displays path delays and timing check limits, unadjusted for delay net delays, for the specified instance.

### Syntax

```
write timing [-recursive] [-file <filename>] [<instance_name1>...<instance_nameN>]  
[-simvalues]
```

### Description

When the write timing command reports interconnect delays on a Verilog module instance you will see either MIPDs (Module Input Port Delays) or MITDs (Module Transport Port Delays) reported. If you specify either the `+multisource_int_delays` or the `+transport_int_delays` argument with the `vsim` command, INTERCONNECT delays will be reported as MITDs. Otherwise they will be reported as MIPDs. An MIPD report may look like the following:

```
# /top/u1: [mymod:src/5/test.v(18)]  
# MIPD(s):  
#   Port clk_in: (6, 6, 6)
```

An MITD report may look like the following:

```
# /top/u1: [mymod:src/5/test.v(18)]  
# MITDs to port clk_in:  
# From port /top/p/y = (6)
```

When the `+multisource_int_delays` argument is specified without `+sdf_verbose` on the `vsim` command line, "write timing" does not report the individual bits of vector source ports of SDF INTERCONNECT delays.

For example, assume the SDF file contains the following two INTERCONNECT statements:

```
(INTERCONNECT p/y[0] n/bus_in[0] (3))  
(INTERCONNECT p/y[1] n/bus_in[1] (4))
```

The corresponding "write timing" output looks like this:

```
# MITDs to port bus_in[0]:  
#   From port /tb12/p/y = (3)  
# MITDs to port bus_in[1]:  
#   From port /tb12/p/y = (4)
```

Notice that the specific bits of the source port are not reported.

When "+sdf\_verbose" is added to the vsim command line the "write timing" output becomes:

```
# MITDs to port bus_in[0]:  
#   From port /tb12/p/y[0] = (3)  
# MITDs to port bus_in[1]:  
#   From port /tb12/p/y[1] = (4)
```

Notice that the specific bits of the source port are now reported.

## Arguments

- **-file <filename>**  
(optional) Specifies the name of the output file where the data is to be written. If the -file argument is omitted, timing information is written to the Transcript window.  
    <filename> — Any valid filename. May include special characters and numbers.
- **<instance\_name1>...<instance\_nameN>**  
(required) The name(s) of the instance(s) for which timing information will be written. If <instance\_name> is omitted, the command returns nothing.
- **-recursive**  
(optional) Generates timing information for the specified instance and all instances underneath it in the design hierarchy.
- **-simvalues**  
(optional) Displays optimization-adjusted values for delay net delays.

## Examples

- Write timing about /top/u1 and all instances underneath it in the hierarchy to the file timing.txt.

```
write timing -r -f timing.txt /top/u1
```

- Write timing information about the designated instances to the Transcript window.

```
write timing /top/u1 /top/u2 /top/u3 /top/u8
```

## write transcript

This command writes the contents of the Transcript window to the specified file. The resulting file can then be modified to replay the transcribed commands as a DO file (macro).

---

### Note

---

 The command cannot be used in batch mode. In batch mode use the standard Transcript file or redirect stdout.

---

### Syntax

write transcript [<filename>]

### Arguments

- <filename>  
(optional) Specifies the name of the output file where the data is to be written. If the <filename> is omitted, the transcript is written to a file named *transcript*.

### Related Topics

[Saving a Transcript File as a DO file](#)

# write tssi

This command records the contents of the List window in a "TSSI format" file.

## Syntax

```
write tssi <filename>
```

## Description

The file contains simulation data for all HDL objects displayed in the List window that can be converted to TSSI format (VHDL signals and Verilog nets). A signal definition file is also generated.

The List window needs to be using symbolic radix in order for write tssi to produce useful output.

If the <filename> has a file extension (e.g., *listfile.lst*), then the definition file is given the same file name with the extension *.def* (e.g., *listfile.def*). The values in the listfile are produced in the same order that they appear in the List window. The directionality is determined from the port type if the object is a port, otherwise it is assumed to be bidirectional (mode INOUT).

Objects that can be converted to SEF are VHDL enumerations with 255 or fewer elements and Verilog nets. The enumeration values U, X, 0, 1, Z, W, L, H and - (the enumeration values defined in the IEEE Standard 1164 std\_ulogic enumeration) are converted to SEF values according to the table below. Other values are converted to a question mark (?) and cause an error message. Though the write tssi command was developed for use with std\_ulogic, any signal which uses only the values defined for std\_ulogic (including the VHDL standard type bit) will be converted.

std_ulogic State Characters	SEF State Characters		
	Input	Output	Bidirectional
U	N	X	?
X	N	X	?
0	D	L	0
1	U	H	1
Z	Z	T	F
W	N	X	?
L	D	L	0
H	U	H	1
-	N	X	?

Bidirectional logic values are not converted because only the resolved value is available. The TSSI TDS ASCII In Converter and ASCII Out Converter can be used to resolve the directionality of the signal and to determine the proper forcing or expected value on the port. Lowercase values x, z, w, l, and h are converted to the same values as the corresponding capitalized values. Any other values will cause an error message to be generated the first time an invalid value is detected on a signal, and the value will be converted to a question mark (?).

---

**Note**

 The TDS ASCII In Converter and ASCII Out Converter are part of the TDS software. ModelSim outputs a vector file, and TSSI tools determine whether the bidirectional signals are driving or not.

---

## Arguments

- <filename>  
(required) Specifies the name of the output file where the data is to be written.

# write wave

This command records the contents of the Wave window in PostScript format. The output file can then be printed on a PostScript printer.

## Syntax

```
write wave <filename> [-end <time>] [-landscape] [-height <real_num>]  
[-margin <real_num>] [-perpage <time>] [-portrait][-start <time>] [-width <real_num>]
```

## Arguments

- <filename>  
(required) Specifies the name of the PostScript (.ps) output file.
- -end <time>  
(optional) The simulation time at which the record will end.  
<time> — Specified as a positive integer or decimal number where the units are the current simulation time resolution.
- -height <real\_num>  
(optional) Specifies the paper height in inches.  
<real\_num> — Specified as a positive integer or decimal number where the default is 11.0.
- -landscape  
(optional) Use landscape (horizontal) orientation. (default)
- -margin <real\_num>  
(optional) Specifies the margin in inches.  
<real\_num> — Specified as a positive integer or decimal number where the default is 0.5.
- -perpage <time>  
(optional) Specifies the time width per page of output.  
<time> — Specified as a positive integer or decimal number where the units are the current simulation time resolution.
- -portrait  
(optional) Use portrait (vertical) orientation where the default is landscape (horizontal).
- -start <time>  
(optional) Specifies the start time to be written.  
<time> — Specified as a positive integer or decimal number where the units are the current simulation time resolution.

- **-width <real\_num>**

(optional) Specifies the paper width in inches.

<real\_num> — Specified as a positive integer or decimal number where the default is 8.5.

## Examples

- Save the current data in the Wave window in a file named *alu.ps*.

**write wave alu.ps**

- Write two separate pages to *top.ps*. The first page contains data from 600ns to 700ns, and the second page contains data from 701ns to 800ns.

**write wave -start 600ns -end 800ns -perpage 100ns top.ps**

To make the job of creating a PostScript waveform output file easier, use the **File > Print Postscript** menu selection in the Wave window.

# Index

---

## — Symbols —

'delayed, 35  
'hasX, 35  
'hasX, hasX, 35  
+define+, 393  
+delay\_mode\_distributed, 393  
+delay\_mode\_path, 393  
+delay\_mode\_unit, 394  
+delay\_mode\_zero, 394  
+inmdir+, 397  
+maxdelays, 399  
+mindelays, 399  
+nowarn, 402  
+typdelays, 410

## — A —

abort command, 63  
absolute time, using @, 26  
add dataflow command, 64  
add list command, 66  
add log command, 199  
add memory command, 71  
add message command, 73  
add watch command, 75  
add wave command, 77  
add\_cmdhelp command, 84  
addTime command, 288  
alias command, 86  
analog  
    signal formatting, 79  
annotating interconnect delays,  
    v2k\_int\_delays, 447  
archive load command, 87  
archive write command, 88  
array  
    slices, 16  
arrays  
    indexes, 15  
arrays, VHDL, searching for, 30

assertions

    testing for with onbreak command, 229  
attributes, of signals, using in expressions, 33  
automatic saving of coverage, 419

## — B —

batch\_mode command, 89  
batch-mode simulations  
    halting, 485  
bd (breakpoint delete) command, 90  
binary radix, mapping to std\_logic values, 41  
bookmark add wave command, 92  
bookmark delete wave command, 94  
bookmark goto wave command, 95  
bookmark list wave command, 96  
bp (breakpoint) command, 97  
break  
    on signal value, 478  
breakpoints  
    conditional, 478  
    continuing simulation after, 260  
    deleting, 90  
    listing, 97  
    setting, 97  
    signal breakpoints (when statements), 478  
    time-based  
        in when statements, 485  
busses  
    user-defined, 82

## — C —

call command, 103  
case choice, must be locally static, 336  
case sensitivity  
    VHDL vs. Verilog, 20  
cd (change directory) command, 108  
change command, 109  
-check\_synthesis argument, 330  
class instance garbage collector, 192, 194  
class member selection, syntax, 16

---

class objects, viewing, 114, 116, 118, 121, 124, 126, 128, 130  
classinfo command, 114, 116, 118, 121, 124, 126, 128, 130  
commands  
    virtual define, 365  
Color  
    radix, 246  
    example, 247  
combining signals, busses, 82  
commands  
    abort, 63  
    add dataflow, 64  
    add list, 66  
    add memory, 71  
    add message, 73  
    add wave, 77  
    add\_cmdhelp, 84  
    alias, 86  
    archive load, 87  
    archive write, 88  
    batch\_mode, 89  
    bd (breakpoint delete), 90  
    bookmark add wave, 92  
    bookmark delete wave, 94  
    bookmark goto wave, 95  
    bookmark list wave, 96  
    bp (breakpoint), 97  
    call, 103  
    cd (change directory), 108  
    change, 109  
    classinfo, 114, 116, 118, 121, 124, 126, 128, 130  
    configure, 132  
    dataset close, 141  
    dataset current, 144  
    dataset info, 145  
    dataset list, 146  
    dataset open, 147  
    dataset rename, 149  
    dataset save, 151  
    delete, 155  
    describe, 156  
    disablebp, 157  
    do, 158  
    drivers, 160  
    dumplog64, 162  
    echo, 163  
    edit, 164  
    enablebp, 165  
    encoding, 166  
    environment, 167  
    examine, 168  
    exit, 175  
    find, 177  
    find connections, 181  
    find infiles, 182  
    find insource, 183  
    force, 185  
    gc configure, 192  
    gc run, 194  
    help, 195  
    history, 196  
    layout, 197  
    log, 199  
    lshift, 202  
    lsublist, 203  
    mem compare, 204  
    mem display, 205  
    mem list, 208  
    mem load, 209  
    mem save, 213  
    mem search, 216  
    modelsim, 219  
    nolog, 221  
    notepad, 223  
    noview, 224  
    nowhen, 225  
    onbreak, 226  
    onElabError, 229  
    onerror, 230  
    onfinish, 232  
    pause, 233  
    printenv, 234, 235  
    process report, 236  
    pwd, 240  
    quietly, 241  
    quit, 242  
    radix, 243  
    radix define, 245

---

radix list, 250  
radix name, 251  
readers, 253  
report, 254  
restart, 256  
resume, 258  
run, 259  
runStatus, 262  
searchlog, 264  
see, 267  
setenv, 268  
shift, 269  
show, 270  
simstats, 271  
simstatslist, 273  
stack down, 275  
stack frame, 276  
stack level, 277  
stack up, 279  
status, 280  
stop, 283  
suppress, 284  
Time, 288  
transcript, 291  
transcript file, 292  
transcript path, 294  
transcript sizelimit, 295  
TreeUpdate, 501  
tssi2mti, 299  
unsetenv, 302  
variables referenced in, 26  
vcf add, 303  
vcf checkpoint, 305  
vcf comment, 306  
vcf dumpports, 307  
vcf dumpportsall, 310  
vcf dumpportsflush, 311  
vcf dumpportslimit, 312  
vcf dumpportsoff, 314  
vcf dumpportson, 315  
vcf file, 316  
vcf files, 318  
vcf flush, 321  
vcf limit, 322  
vcf off, 324  
vcf on, 325  
vcom, 329  
vdel, 345  
vencrypt, 350  
verror, 354  
vgencomp, 356  
vhencrypt, 358  
virtual count, 364  
virtual delete, 366  
virtual describe, 367  
virtual expand, 368  
virtual function, 369  
virtual hide, 372  
virtual log, 373  
virtual nohide, 375  
virtual nolog, 376  
virtual region, 378  
virtual save, 379  
virtual show, 380  
virtual signal, 381  
vlib, 387  
vlog, 391  
vmake, 413  
vmap, 415  
vsimVersion, 451  
vsources, 453  
wave, 455  
wave create, 459  
wave edit, 465  
wave export, 469  
wave import, 471  
wave modify, 472  
wave sort, 477  
WaveActivateNextPane, 501  
WaveRestoreCursors, 501  
WaveRestoreZoom, 501  
when, 478  
where, 486  
wlf2log, 487  
wlf2vcf, 489  
wlfman, 490  
wlfrecover, 497  
write format, 498  
write list, 501  
write preferences, 502

---

write report, 503  
write timing, 506  
write transcript, 508  
write tssi, 509  
write wave, 511  
commands formatTime, 191  
comment characters in VSIM commands, 14  
compiling  
    range checking in VHDL, 340  
    Verilog, 391  
    VHDL, 329  
        selected design units (-just eapbc), 334  
        standard package (-s), 340  
    VHDL-2008  
        REAL\_VECTOR, 333  
compressing files  
    VCD files, 307, 318  
concatenation  
    directives, 40  
    of signals, 39  
conditional breakpoints, 478  
configurations, simulating, 419  
configure command, 132  
constants  
    in case statements, 336  
    values of, displaying, 156, 168  
conversion  
    radix, 243  
coverage, automatic save, 419  
coverstore, auto-saved coverage, 419

## — D —

dataset close command, 141  
dataset current command, 144  
dataset info command, 145  
dataset list command, 146  
dataset open command, 147  
dataset rename command, 149  
dataset save command, 151  
datasets  
    environment command, specifying with, 167  
design loading, interrupting, 419  
declarations, hiding implicit with explicit, 343  
delay  
    interconnect, 427

delete command, 155  
deltas  
    collapsing in WLF files, 437  
dependency errors, 395  
describe command, 156  
design units  
    report of units simulated, 503  
    Verilog  
        adding to a library, 391  
directories  
    mapping libraries, 415  
disablebp command, 157  
dividers  
    adding from command line, 78  
divTime command, 288  
do command, 158  
DO file  
    executing, 158  
DO files, 158  
    breakpoints, executing at, 98  
    forcing signals, nets, or registers, 185  
    parameters  
        passing, 158  
    relative directories, 158  
    shifting parameter values, 269  
-dpiheader, vlog, 394, 442  
drivers command, 160  
dump files, viewing in the simulator, 326  
dumplog64 command, 162

## — E —

echo command, 163  
edit command, 164  
enablebp command, 165  
encoding command, 166  
environment command, 167  
environment variables  
    reading into Verilog code, 393  
    specifying UNIX editor, 164  
    state of, 235  
    using in pathnames, 20  
environment, displaying or changing  
    pathname, 167  
eqTime command, 288  
errors  
    getting details about messages, 354

---

onerror command, 230  
SDF, disabling, 431  
event order  
  changing in Verilog, 392  
examine command, 168  
exit command, 175  
extended identifiers, 20

## — F —

file compression  
  VCD files, 307, 318  
find command, 177  
find connections command, 181  
find infiles command, 182  
find insource command, 183  
fixed point radix, 245  
floating point radix, 245  
force  
  remove wire model, 429  
force command, 185  
format file  
  List window, 498  
  Wave window, 498  
formatTime command, 191, 288

## — G —

gc configure command, 192  
gc run command, 194  
generics  
  assigning or overriding values with -g and -G, 422  
  examining generic values, 168  
  limitation on assigning composite types, 423  
glitches  
  disabling generation  
    from command line, 440  
global visibility  
  PLI/FLI shared objects, 424  
gotolink modelsim\_user  
  DPI File Loading, 447  
gteTime command, 288  
gtTime command, 288  
GUI\_expression\_format, 32  
  syntax, 33

## — H —

hazards  
  -hazards argument to vlog, 397  
  -hazards argument to vsim, 442  
help command, 195  
history  
  of commands  
    shortcuts for reuse, 29  
history command, 196

## — I —

implicit operator, hiding with vcom -explicit, 343  
interconnect de lays, 427  
interconnect delays  
  annotating per Verilog 2001, 447  
internal signals, adding to a VCD file, 304  
interruptin g design loading, 419  
intToTime command, 288

## — K —

keywords  
  enabling SystemVerilog keywords, 407

## — L —

layout command, 197  
LD\_LIBRARY\_PATH, disabling default  
  internal setting of, 427  
libraries  
  design libraries, creating, 387  
  refreshing library images, 405  
  Verilog, 425  
lint-style checks, 399  
List window  
  adding items to, 66  
loading designs, interrupti ng, 419  
log command, 199  
log file  
  log command, 199  
  nolog command, 221  
  QuickSim II format, 487  
  redirecting wi th -l, 425  
  redirecting with -l, 426  
  virtual log command, 373  
  virtual nolog command, 376  
ls hift command, 202

---

lsublist command, 203  
lteTime command, 288  
ltTime command, 288

— M —

mc\_scan\_plusargs, PLI routine, 446  
mem compare command, 204  
mem display command, 205  
mem list command, 208  
mem load command, 209  
mem save command, 213  
mem search command, 216  
memory window  
    add memory command, 71  
    adding items to, 71  
memory, comparing contents, 204  
memory, displaying contents, 205  
memory, listing, 208  
memory, loading contents, 209  
memory, saving contents, 213  
memory, searching for patterns, 216  
messages  
    echoing, 163  
    getting more information, 354  
    loading, disabling with -quiet, 404  
    loading, disabling with -quiet, 340  
-mfcu, 399  
modelsim command, 219  
mulTime command, 288  
multi-source interconnect de lays, 427

— N —

name case sensitivity, VHDL vs. Verilog, 20  
negative pulses  
    driving an error state, 447  
neqTime command, 288  
nets  
    drivers of, displaying, 160  
    readers of, displaying, 253  
    stimulus, 185  
    values of  
        examining, 168  
-no\_risefall\_delaynets, 443  
nolog command, 221  
notepad command, 223  
noview command, 224

nowhen command, 225

— O —

object\_list\_file, WLF files, 490  
onbreak command, 226  
onElabError command, 229  
onerror command, 230  
onfinish command, 232  
optimizations

    disabling for VHDL designs, 339  
    optimizing wlf files, 492

order of events

    changing in Verilog, 392

— P —

parameters  
    using with DO files, 158  
pathnames  
    in VSIM commands, 15  
    spaces in, 14  
pause command, 233  
PLI  
    loading shared objects with global symbol  
        visibility, 424  
preference variables  
    WildcardFilter, 22  
printenv command, 234, 235  
process report command, 236  
projects  
    override mapping for work directory with  
        vcom, 343  
    override mapping for work directory with  
        vlog, 411  
propagation, preventing X propagation, 428  
pulse error state, 447  
pwd command, 240

— Q —

QuickSim II logfile format, 487  
quietly command, 241  
quit command, 242

— R —

Radix  
    color, 246  
        example, 247  
radix

---

display values in debug windows, 243  
of signals being examined, 68, 81, 171  
user defined, 245  
radix command, 243  
Radix define command  
    setting rad ix color, 247  
    setting radix color, 246  
radix define command, 245  
    fixed point radix, 245  
    floating point radix, 245  
radix list command, 250  
radix name command, 251  
range checking  
    disabling, 338  
    enabling, 340  
readers command, 253  
RealToTime command, 288  
record field selection, syntax, 16  
refresh, dependency check errors, 395  
refreshing library images, 405  
report command, 254  
reporting  
    processes in the Process Window, 236  
    variable settings, 26  
resolution  
    specifying with -t argument, 433  
restart command, 256  
resume command, 258  
run command, 259  
runStatus command, 262

— S —

scaleTime command, 288  
scope resolution operator, 17  
scope, setting region environment, 167  
SDF  
    annotation verbose mode, 431  
    controlling missing instance messages, 431  
    errors on loading, disabling, 431  
    warning messages, disabling, 431  
search libraries, 425  
searching  
    binary signal values in the GUI, 41  
    List window  
        signal values, transitions, and names, 32  
            VHDL arrays, 30  
            searchlog command, 264  
            see command, 267  
            setenv command, 268  
            shared objects  
                loading with global symbol visibility, 424  
            shift command, 269  
            shortcuts  
                command history, 29  
                command line caveat, 28  
            show command, 270  
            signals  
                alternative names in the Wave window (-label), 80  
                attributes of, using in expressions, 33  
                breakpoints, 478  
                combining into a user-defined bus, 82  
                drivers of, displaying, 160  
                environment of, displaying, 167  
                force time, specifying, 188  
                log file, creating, 199  
                pathnames in VSIM commands, 15  
                radix  
                    specifying for examine, 68, 81, 171  
                readers of, displaying, 253  
                stimulus, 185  
                values of  
                    examining, 168  
            simstats command, 271  
            simstatslist command, 273  
            simulating  
                delays, specifying time units for, 26  
                design unit, specifying, 419  
                saving simulations, 199, 437  
                stopping simulation in batch mode, 485  
            simulations  
                saving results, 151  
            Simulator commands, 63  
            simulator resolution  
                vsim -t argument, 433  
            simulator version, 436, 451  
            simultaneous events in Verilog  
                changing order, 392  
            spaces in pathnames, 14  
            sparse memories

---

listing with write report, 504  
specify path delays, 447  
stack down command, 275  
stack frame command, 276  
stack level command, 277  
stack up command, 279  
startup  
    alternate to startup.do (vsim -do), 421  
status command, 280  
Std\_logic  
    mapping to binary radix, 41  
stop command, 283  
subTime command, 288  
suppress command, 284  
synthesis  
    rule compliance checking, 330  
SystemC  
    class and structure member naming syntax, 16  
SystemVerilog  
    enabling with -sv argument, 407  
    multiple files in a compilation unit, 399  
    scope resolution, 17  
SystemVerilog classes  
    call command, 103

— T —

Tcl  
    history shortcuts, 29  
    variable  
        in when commands, 482  
TFMPC  
    disabling warning, 444  
time  
    absolute, using @, 26  
    simulation time units, 26  
time collapsing, 437  
Time commands, 288  
time resolution  
    setting  
        with vsim command, 433  
time, time units, simulation time, 26  
timescale directive warning  
    disabling, 444  
timing  
    disabling checks, 402

disabling checks for entire design, 428  
title, Main window, changing, 434  
transcript  
    redirecting with -l, 425, 426  
transcript command, 291  
transcript file command, 292  
transcript path command, 294  
transcript sizelimit command, 295  
TreeUpdate command, 501  
TSCALE, disabling warning, 444  
TSSI, 509  
tssi2mti command, 299

— U —

-u, 410  
undeclared nets, reporting an error, 399  
unsetenv command, 302  
user-defined bus, 82  
User-defined radix, 245

— V —

-v, 410  
v2k\_int\_delays, 447  
validTime command, 288  
values  
    describe HDL items, 156  
    examine HDL item values, 168  
variable settings report, 26  
variables  
    describing, 156  
    referencing in commands, 26  
    value of  
        changing from command line, 109  
        examining, 168  
vcd add command, 303  
vcd checkpoint command, 305  
vcd comment command, 306  
vcd dumpports command, 307  
vcd dumpportsall command, 310  
vcd dumpportsflush command, 311  
vcd dumpportslimit command, 312  
vcd dumpportsoff command, 314  
vcd dumpportson command, 315  
vcd file command, 316  
VCD files  
    adding items to the file, 303

---

capturing port driver data, 307  
converting to WLF files, 326  
creating, 303  
dumping variable values, 305  
flushing the buffer contents, 321  
generating from WLF files, 489  
inserting comments, 306  
internal signals, adding, 304  
specifying maximum file size, 322  
specifying name of, 318  
specifying the file name, 316  
state mapping, 316, 318  
turn off VCD dumping, 324  
turn on VCD dumping, 325  
viewing files from another tool, 326

vcd files command, 318  
vcd flush command, 321  
vcd limit command, 322  
vcd off command, 324  
vcd on command, 325  
vcd2wlf command, 326  
vcom command, 329  
vdel command, 345  
vector elements, initializing, 109  
vencrypt command, 350  
Verilog  
    capturing port driver data with -dumports, 316  
verror command, 354  
version  
    obtaining with vsim command, 436  
    obtaining with vsimcommands, 451  
vgencomp command, 356  
VHDL  
    arrays  
        searching for, 30  
    binding, ignore default, 333  
    field naming syntax, 16  
VHDL-2008  
    package STANDARD  
        REAL\_VECTOR, 333  
vhencrypt command, 358  
viewing  
    waveforms, 437  
virtual count commands, 364

virtual define command, 365  
virtual delete command, 366  
virtual describe command, 367  
virtual expand commands, 368  
virtual function command, 369  
virtual hide command, 372  
virtual log command, 373  
virtual nohide command, 375  
virtual nolog command, 376  
virtual region command, 378  
virtual save command, 379  
virtual show command, 380  
virtual signal command, 381  
vlib command, 387  
vlog  
    multiple file compilation, 399  
vlog command, 391  
vmake command, 413  
vmap command, 415  
vsim  
    disabling internal setting of  
        LD\_LIBRARY\_PATH, 427

— W —

warnings  
    SDF, disabling, 431  
    suppressing VCOM warning messages, 338, 402  
    suppressing VLOG warning messages, 402  
    suppressing VSIM warning messages, 444

watch window  
    add watch command, 75  
    watching signal values, 75  
wave commands, 455  
wave create command, 459  
wave cursor commands, 455  
wave edit command, 465  
wave export command, 469  
wave import command, 471  
wave log format (WLF) file, 437  
    of binary signal values, 199  
wave modify command, 472  
wave sort command, 477  
Wave window  
    adding items to, 77  
WaveActivateNextPane command, 501

---

waveform editor  
    creating waves, [459](#)  
    editing commands, [465](#)  
    importing vcd stimulus file, [471](#)  
    modifying existing waves, [472](#)  
    saving waves, [469](#)  
waveform logfile  
    log command, [199](#)  
waveforms  
    saving and viewing, [199](#)  
WaveRestoreCursors command, [501](#)  
WaveRestoreZoom command, [501](#)  
when command, [478](#)  
when statement  
    time-based breakpoints, [485](#)  
where command, [486](#)  
wildcard characters  
    for pattern matching in simulator  
        commands, [21](#)  
WildcardFilter Preference Variable, [22](#)  
windows  
    List window  
        output file, [501](#)  
        saving the format of, [498](#)  
    Wave window  
        path elements, changing, [135](#)  
WLF files  
    collapsing deltas, [437](#)  
    collapsing time steps, [437](#)  
    converting to VCD, [489](#)  
    creating from VCD, [326](#)  
    indexing, [492](#)  
    limiting size, [438](#)  
    log command, [199](#)  
    merging, [492](#)  
    optimizing, [492](#)  
    repairing, [497](#)  
    saving, [151](#)  
    specifying name, [437](#)  
    wlfman command, [490](#)  
wlf2log command, [487](#)  
wlf2vcd command, [489](#)  
wlfman command, [490](#)  
wlfrecover command, [497](#)  
write format command, [498](#)

write list command, [501](#)  
write preferences command, [502](#)  
write report command, [503](#)  
write timing command, [506](#)  
write transcript command, [508](#)  
write tssi command, [509](#)  
write wave command, [511](#)

— X —

X propagation  
    disabling for entire design, [428](#)

— Y —

-y, [411](#)

— Z —

zoom  
    wave window  
        returning current range, [456](#)

# End-User License Agreement

The latest version of the End-User License Agreement is available on-line at:  
[www.mentor.com/eula](http://www.mentor.com/eula)

## IMPORTANT INFORMATION

**USE OF ALL SOFTWARE IS SUBJECT TO LICENSE RESTRICTIONS. CAREFULLY READ THIS LICENSE AGREEMENT BEFORE USING THE PRODUCTS. USE OF SOFTWARE INDICATES CUSTOMER'S COMPLETE AND UNCONDITIONAL ACCEPTANCE OF THE TERMS AND CONDITIONS SET FORTH IN THIS AGREEMENT. ANY ADDITIONAL OR DIFFERENT PURCHASE ORDER TERMS AND CONDITIONS SHALL NOT APPLY.**

## END-USER LICENSE AGREEMENT ("Agreement")

This is a legal agreement concerning the use of Software (as defined in Section 2) and hardware (collectively "Products") between the company acquiring the Products ("Customer"), and the Mentor Graphics entity that issued the corresponding quotation or, if no quotation was issued, the applicable local Mentor Graphics entity ("Mentor Graphics"). Except for license agreements related to the subject matter of this license agreement which are physically signed by Customer and an authorized representative of Mentor Graphics, this Agreement and the applicable quotation contain the parties' entire understanding relating to the subject matter and supersede all prior or contemporaneous agreements. If Customer does not agree to these terms and conditions, promptly return or, in the case of Software received electronically, certify destruction of Software and all accompanying items within five days after receipt of Software and receive a full refund of any license fee paid.

### 1. ORDERS, FEES AND PAYMENT.

- 1.1. To the extent Customer (or if agreed by Mentor Graphics, Customer's appointed third party buying agent) places and Mentor Graphics accepts purchase orders pursuant to this Agreement (each an "Order"), each Order will constitute a contract between Customer and Mentor Graphics, which shall be governed solely and exclusively by the terms and conditions of this Agreement, any applicable addenda and the applicable quotation, whether or not those documents are referenced on the Order. Any additional or conflicting terms and conditions appearing on an Order or presented in any electronic portal or automated order management system, whether or not required to be electronically accepted, will not be effective unless agreed in writing and physically signed by an authorized representative of Customer and Mentor Graphics.
- 1.2. Amounts invoiced will be paid, in the currency specified on the applicable invoice, within 30 days from the date of such invoice. Any past due invoices will be subject to the imposition of interest charges in the amount of one and one-half percent per month or the applicable legal rate currently in effect, whichever is lower. Prices do not include freight, insurance, customs duties, taxes or other similar charges, which Mentor Graphics will state separately in the applicable invoice. Unless timely provided with a valid certificate of exemption or other evidence that items are not taxable, Mentor Graphics will invoice Customer for all applicable taxes including, but not limited to, VAT, GST, sales tax, consumption tax and service tax. Customer will make all payments free and clear of, and without reduction for, any withholding or other taxes; any such taxes imposed on payments by Customer hereunder will be Customer's sole responsibility. If Customer appoints a third party to place purchase orders and/or make payments on Customer's behalf, Customer shall be liable for payment under Orders placed by such third party in the event of default.
- 1.3. All Products are delivered FCA factory (Incoterms 2010), freight prepaid and invoiced to Customer, except Software delivered electronically, which shall be deemed delivered when made available to Customer for download. Mentor Graphics retains a security interest in all Products delivered under this Agreement, to secure payment of the purchase price of such Products, and Customer agrees to sign any documents that Mentor Graphics determines to be necessary or convenient for use in filing or perfecting such security interest. Mentor Graphics' delivery of Software by electronic means is subject to Customer's provision of both a primary and an alternate e-mail address.

2. **GRANT OF LICENSE.** The software installed, downloaded, or otherwise acquired by Customer under this Agreement, including any updates, modifications, revisions, copies, documentation, setup files and design data ("Software") are copyrighted, trade secret and confidential information of Mentor Graphics or its licensors, who maintain exclusive title to all Software and retain all rights not expressly granted by this Agreement. Except for Software that is embeddable ("Embedded Software"), which is licensed pursuant to separate embedded software terms or an embedded software supplement, Mentor Graphics grants to Customer, subject to payment of applicable license fees, a nontransferable, nonexclusive license to use Software solely: (a) in machine-readable, object-code form (except as provided in Subsection 4.2); (b) for Customer's internal business purposes; (c) for the term of the license; and (d) on the computer hardware and at the site authorized by Mentor Graphics. A site is restricted to a one-half mile (800 meter) radius. Customer may have Software temporarily used by an employee for telecommuting purposes from locations other than a Customer office, such as the employee's residence, an airport or hotel, provided that such employee's primary place of employment is the site where the Software is authorized for use. Mentor Graphics' standard policies and programs, which vary depending on Software, license fees paid or services purchased, apply to the following: (a) relocation of Software; (b) use of Software, which may be limited, for example, to execution of a single session by a single user on the authorized hardware or for a restricted period of time (such limitations may be technically implemented through the use of authorization codes or similar devices); and (c) support services provided, including eligibility to receive telephone support, updates, modifications, and revisions. For the avoidance of doubt, if Customer provides any feedback or requests any change or enhancement to Products, whether in the course of receiving support or consulting services, evaluating Products, performing beta testing or otherwise, any inventions, product improvements, modifications or developments made by Mentor Graphics (at Mentor Graphics' sole discretion) will be the exclusive property of Mentor Graphics.

### 3. BETA CODE.

- 3.1. Portions or all of certain Software may contain code for experimental testing and evaluation (which may be either alpha or beta, collectively "Beta Code"), which may not be used without Mentor Graphics' explicit authorization. Upon Mentor Graphics' authorization, Mentor Graphics grants to Customer a temporary, nontransferable, nonexclusive license for experimental use to test and evaluate the Beta Code without charge for a limited period of time specified by Mentor Graphics. Mentor Graphics may choose, at its sole discretion, not to release Beta Code commercially in any form.
- 3.2. If Mentor Graphics authorizes Customer to use the Beta Code, Customer agrees to evaluate and test the Beta Code under normal conditions as directed by Mentor Graphics. Customer will contact Mentor Graphics periodically during Customer's use of the Beta Code to discuss any malfunctions or suggested improvements. Upon completion of Customer's evaluation and testing, Customer will send to Mentor Graphics a written evaluation of the Beta Code, including its strengths, weaknesses and recommended improvements.
- 3.3. Customer agrees to maintain Beta Code in confidence and shall restrict access to the Beta Code, including the methods and concepts utilized therein, solely to those employees and Customer location(s) authorized by Mentor Graphics to perform beta testing. Customer agrees that any written evaluations and all inventions, product improvements, modifications or developments that Mentor Graphics conceived or made during or subsequent to this Agreement, including those based partly or wholly on Customer's feedback, will be the exclusive property of Mentor Graphics. Mentor Graphics will have exclusive rights, title and interest in all such property. The provisions of this Subsection 3.3 shall survive termination of this Agreement.

### 4. RESTRICTIONS ON USE.

- 4.1. Customer may copy Software only as reasonably necessary to support the authorized use. Each copy must include all notices and legends embedded in Software and affixed to its medium and container as received from Mentor Graphics. All copies shall remain the property of Mentor Graphics or its licensors. Except for Embedded Software that has been embedded in executable code form in Customer's product(s), Customer shall maintain a record of the number and primary location of all copies of Software, including copies merged with other software, and shall make those records available to Mentor Graphics upon request. Customer shall not make Products available in any form to any person other than Customer's employees and on-site contractors, excluding Mentor Graphics competitors, whose job performance requires access and who are under obligations of confidentiality. Customer shall take appropriate action to protect the confidentiality of Products and ensure that any person permitted access does not disclose or use Products except as permitted by this Agreement. Customer shall give Mentor Graphics written notice of any unauthorized disclosure or use of the Products as soon as Customer becomes aware of such unauthorized disclosure or use. Customer acknowledges that Software provided hereunder may contain source code which is proprietary and its confidentiality is of the highest importance and value to Mentor Graphics. Customer acknowledges that Mentor Graphics may be seriously harmed if such source code is disclosed in violation of this Agreement. Except as otherwise permitted for purposes of interoperability as specified by applicable and mandatory local law, Customer shall not reverse-assemble, disassemble, reverse-compile, or reverse-engineer any Product, or in any way derive any source code from Software that is not provided to Customer in source code form. Log files, data files, rule files and script files generated by or for the Software (collectively "Files"), including without limitation files containing Standard Verification Rule Format ("SVRF") and Tcl Verification Format ("TVF") which are Mentor Graphics' trade secret and proprietary syntaxes for expressing process rules, constitute or include confidential information of Mentor Graphics. Customer may share Files with third parties, excluding Mentor Graphics competitors, provided that the confidentiality of such Files is protected by written agreement at least as well as Customer protects other information of a similar nature or importance, but in any case with at least reasonable care. Customer may use Files containing SVRF or TVF only with Mentor Graphics products. Under no circumstances shall Customer use Products or Files or allow their use for the purpose of developing, enhancing or marketing any product that is in any way competitive with Products, or disclose to any third party the results of, or information pertaining to, any benchmark.
- 4.2. If any Software or portions thereof are provided in source code form, Customer will use the source code only to correct software errors and enhance or modify the Software for the authorized use, or as permitted for Embedded Software under separate embedded software terms or an embedded software supplement. Customer shall not disclose or permit disclosure of source code, in whole or in part, including any of its methods or concepts, to anyone except Customer's employees or on-site contractors, excluding Mentor Graphics competitors, with a need to know. Customer shall not copy or compile source code in any manner except to support this authorized use.
- 4.3. Customer agrees that it will not subject any Product to any open source software ("OSS") license that conflicts with this Agreement or that does not otherwise apply to such Product.
- 4.4. Customer may not assign this Agreement or the rights and duties under it, or relocate, sublicense, or otherwise transfer the Products, whether by operation of law or otherwise ("Attempted Transfer"), without Mentor Graphics' prior written consent and payment of Mentor Graphics' then-current applicable relocation and/or transfer fees. Any Attempted Transfer without Mentor Graphics' prior written consent shall be a material breach of this Agreement and may, at Mentor Graphics' option, result in the immediate termination of the Agreement and/or the licenses granted under this Agreement. The terms of this Agreement, including without limitation the licensing and assignment provisions, shall be binding upon Customer's permitted successors in interest and assigns.
- 4.5. The provisions of this Section 4 shall survive the termination of this Agreement.

5. **SUPPORT SERVICES.** To the extent Customer purchases support services, Mentor Graphics will provide Customer with updates and technical support for the Products, at the Customer site(s) for which support is purchased, in accordance with Mentor Graphics' then current End-User Support Terms located at <http://supportnet.mentor.com/supportterms>.
6. **OPEN SOURCE SOFTWARE.** Products may contain OSS or code distributed under a proprietary third party license agreement, to which additional rights or obligations ("Third Party Terms") may apply. Please see the applicable Product documentation (including license files, header files, read-me files or source code) for details. In the event of conflict between the terms of this Agreement

(including any addenda) and the Third Party Terms, the Third Party Terms will control solely with respect to the OSS or third party code. The provisions of this Section 6 shall survive the termination of this Agreement.

## 7. LIMITED WARRANTY.

- 7.1. Mentor Graphics warrants that during the warranty period its standard, generally supported Products, when properly installed, will substantially conform to the functional specifications set forth in the applicable user manual. Mentor Graphics does not warrant that Products will meet Customer's requirements or that operation of Products will be uninterrupted or error free. The warranty period is 90 days starting on the 15th day after delivery or upon installation, whichever first occurs. Customer must notify Mentor Graphics in writing of any nonconformity within the warranty period. For the avoidance of doubt, this warranty applies only to the initial shipment of Software under an Order and does not renew or reset, for example, with the delivery of (a) Software updates or (b) authorization codes or alternate Software under a transaction involving Software re-mix. This warranty shall not be valid if Products have been subject to misuse, unauthorized modification, improper installation or Customer is not in compliance with this Agreement. MENTOR GRAPHICS' ENTIRE LIABILITY AND CUSTOMER'S EXCLUSIVE REMEDY SHALL BE, AT MENTOR GRAPHICS' OPTION, EITHER (A) REFUND OF THE PRICE PAID UPON RETURN OF THE PRODUCTS TO MENTOR GRAPHICS OR (B) MODIFICATION OR REPLACEMENT OF THE PRODUCTS THAT DO NOT MEET THIS LIMITED WARRANTY. MENTOR GRAPHICS MAKES NO WARRANTIES WITH RESPECT TO: (A) SERVICES; (B) PRODUCTS PROVIDED AT NO CHARGE; OR (C) BETA CODE; ALL OF WHICH ARE PROVIDED "AS IS."
- 7.2. THE WARRANTIES SET FORTH IN THIS SECTION 7 ARE EXCLUSIVE. NEITHER MENTOR GRAPHICS NOR ITS LICENSORS MAKE ANY OTHER WARRANTIES EXPRESS, IMPLIED OR STATUTORY, WITH RESPECT TO PRODUCTS PROVIDED UNDER THIS AGREEMENT. MENTOR GRAPHICS AND ITS LICENSORS SPECIFICALLY DISCLAIM ALL IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NON-INFRINGEMENT OF INTELLECTUAL PROPERTY.

8. **LIMITATION OF LIABILITY.** TO THE EXTENT PERMITTED UNDER APPLICABLE LAW, IN NO EVENT SHALL MENTOR GRAPHICS OR ITS LICENSORS BE LIABLE FOR INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES (INCLUDING LOST PROFITS OR SAVINGS) WHETHER BASED ON CONTRACT, TORT OR ANY OTHER LEGAL THEORY, EVEN IF MENTOR GRAPHICS OR ITS LICENSORS HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN NO EVENT SHALL MENTOR GRAPHICS' OR ITS LICENSORS' LIABILITY UNDER THIS AGREEMENT EXCEED THE AMOUNT RECEIVED FROM CUSTOMER FOR THE HARDWARE, SOFTWARE LICENSE OR SERVICE GIVING RISE TO THE CLAIM. IN THE CASE WHERE NO AMOUNT WAS PAID, MENTOR GRAPHICS AND ITS LICENSORS SHALL HAVE NO LIABILITY FOR ANY DAMAGES WHATSOEVER. THE PROVISIONS OF THIS SECTION 8 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

## 9. THIRD PARTY CLAIMS.

- 9.1. Customer acknowledges that Mentor Graphics has no control over the testing of Customer's products, or the specific applications and use of Products. Mentor Graphics and its licensors shall not be liable for any claim or demand made against Customer by any third party, except to the extent such claim is covered under Section 10.
- 9.2. In the event that a third party makes a claim against Mentor Graphics arising out of the use of Customer's products, Mentor Graphics will give Customer prompt notice of such claim. At Customer's option and expense, Customer may take sole control of the defense and any settlement of such claim. Customer WILL reimburse and hold harmless Mentor Graphics for any LIABILITY, damages, settlement amounts, costs and expenses, including reasonable attorney's fees, incurred by or awarded against Mentor Graphics or its licensors in connection with such claims.
- 9.3. The provisions of this Section 9 shall survive any expiration or termination of this Agreement.

## 10. INFRINGEMENT.

- 10.1. Mentor Graphics will defend or settle, at its option and expense, any action brought against Customer in the United States, Canada, Japan, or member state of the European Union which alleges that any standard, generally supported Product acquired by Customer hereunder infringes a patent or copyright or misappropriates a trade secret in such jurisdiction. Mentor Graphics will pay costs and damages finally awarded against Customer that are attributable to such action. Customer understands and agrees that as conditions to Mentor Graphics' obligations under this section Customer must: (a) notify Mentor Graphics promptly in writing of the action; (b) provide Mentor Graphics all reasonable information and assistance to settle or defend the action; and (c) grant Mentor Graphics sole authority and control of the defense or settlement of the action.
- 10.2. If a claim is made under Subsection 10.1 Mentor Graphics may, at its option and expense: (a) replace or modify the Product so that it becomes noninfringing; (b) procure for Customer the right to continue using the Product; or (c) require the return of the Product and refund to Customer any purchase price or license fee paid, less a reasonable allowance for use.
- 10.3. Mentor Graphics has no liability to Customer if the action is based upon: (a) the combination of Software or hardware with any product not furnished by Mentor Graphics; (b) the modification of the Product other than by Mentor Graphics; (c) the use of other than a current unaltered release of Software; (d) the use of the Product as part of an infringing process; (e) a product that Customer makes, uses, or sells; (f) any Beta Code or Product provided at no charge; (g) any software provided by Mentor Graphics' licensors who do not provide such indemnification to Mentor Graphics' customers; (h) OSS, except to the extent that the infringement is directly caused by Mentor Graphics' modifications to such OSS; or (i) infringement by Customer that is deemed willful. In the case of (i), Customer shall reimburse Mentor Graphics for its reasonable attorney fees and other costs related to the action.
- 10.4. THIS SECTION 10 IS SUBJECT TO SECTION 8 ABOVE AND STATES THE ENTIRE LIABILITY OF MENTOR GRAPHICS AND ITS LICENSORS, AND CUSTOMER'S SOLE AND EXCLUSIVE REMEDY, FOR DEFENSE,

SETTLEMENT AND DAMAGES, WITH RESPECT TO ANY ALLEGED PATENT OR COPYRIGHT INFRINGEMENT OR TRADE SECRET MISAPPROPRIATION BY ANY PRODUCT PROVIDED UNDER THIS AGREEMENT.

**11. TERMINATION AND EFFECT OF TERMINATION.**

- 11.1. If a Software license was provided for limited term use, such license will automatically terminate at the end of the authorized term. Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement immediately upon written notice if Customer: (a) exceeds the scope of the license or otherwise fails to comply with the licensing or confidentiality provisions of this Agreement, or (b) becomes insolvent, files a bankruptcy petition, institutes proceedings for liquidation or winding up or enters into an agreement to assign its assets for the benefit of creditors. For any other material breach of any provision of this Agreement, Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement upon 30 days written notice if Customer fails to cure the breach within the 30 day notice period. Termination of this Agreement or any license granted hereunder will not affect Customer's obligation to pay for Products shipped or licenses granted prior to the termination, which amounts shall be payable immediately upon the date of termination.
- 11.2. Upon termination of this Agreement, the rights and obligations of the parties shall cease except as expressly set forth in this Agreement. Upon termination of this Agreement and/or any license granted under this Agreement, Customer shall ensure that all use of the affected Products ceases, and shall return hardware and either return to Mentor Graphics or destroy Software in Customer's possession, including all copies and documentation, and certify in writing to Mentor Graphics within ten business days of the termination date that Customer no longer possesses any of the affected Products or copies of Software in any form.
12. **EXPORT.** The Products provided hereunder are subject to regulation by local laws and European Union ("E.U.") and United States ("U.S.") government agencies, which prohibit export, re-export or diversion of certain products, information about the products, and direct or indirect products thereof, to certain countries and certain persons. Customer agrees that it will not export or re-export Products in any manner without first obtaining all necessary approval from appropriate local, E.U. and U.S. government agencies. If Customer wishes to disclose any information to Mentor Graphics that is subject to any E.U., U.S. or other applicable export restrictions, including without limitation the U.S. International Traffic in Arms Regulations (ITAR) or special controls under the Export Administration Regulations (EAR), Customer will notify Mentor Graphics personnel, in advance of each instance of disclosure, that such information is subject to such export restrictions.
13. **U.S. GOVERNMENT LICENSE RIGHTS.** Software was developed entirely at private expense. The parties agree that all Software is commercial computer software within the meaning of the applicable acquisition regulations. Accordingly, pursuant to U.S. FAR 48 CFR 12.212 and DFAR 48 CFR 227.7202, use, duplication and disclosure of the Software by or for the U.S. government or a U.S. government subcontractor is subject solely to the terms and conditions set forth in this Agreement, which shall supersede any conflicting terms or conditions in any government order document, except for provisions which are contrary to applicable mandatory federal laws.
14. **THIRD PARTY BENEFICIARY.** Mentor Graphics Corporation, Mentor Graphics (Ireland) Limited, Microsoft Corporation and other licensors may be third party beneficiaries of this Agreement with the right to enforce the obligations set forth herein.
15. **REVIEW OF LICENSE USAGE.** Customer will monitor the access to and use of Software. With prior written notice and during Customer's normal business hours, Mentor Graphics may engage an internationally recognized accounting firm to review Customer's software monitoring system and records deemed relevant by the internationally recognized accounting firm to confirm Customer's compliance with the terms of this Agreement or U.S. or other local export laws. Such review may include FlexNet (or successor product) report log files that Customer shall capture and provide at Mentor Graphics' request. Customer shall make records available in electronic format and shall fully cooperate with data gathering to support the license review. Mentor Graphics shall bear the expense of any such review unless a material non-compliance is revealed. Mentor Graphics shall treat as confidential information all information gained as a result of any request or review and shall only use or disclose such information as required by law or to enforce its rights under this Agreement. The provisions of this Section 15 shall survive the termination of this Agreement.
16. **CONTROLLING LAW, JURISDICTION AND DISPUTE RESOLUTION.** The owners of certain Mentor Graphics intellectual property licensed under this Agreement are located in Ireland and the U.S. To promote consistency around the world, disputes shall be resolved as follows: excluding conflict of laws rules, this Agreement shall be governed by and construed under the laws of the State of Oregon, U.S., if Customer is located in North or South America, and the laws of Ireland if Customer is located outside of North or South America or Japan, and the laws of Japan if Customer is located in Japan. All disputes arising out of or in relation to this Agreement shall be submitted to the exclusive jurisdiction of the courts of Portland, Oregon when the laws of Oregon apply, or Dublin, Ireland when the laws of Ireland apply, or the Tokyo District Court when the laws of Japan apply. Notwithstanding the foregoing, all disputes in Asia (excluding Japan) arising out of or in relation to this Agreement shall be resolved by arbitration in Singapore before a single arbitrator to be appointed by the chairman of the Singapore International Arbitration Centre ("SIAC") to be conducted in the English language, in accordance with the Arbitration Rules of the SIAC in effect at the time of the dispute, which rules are deemed to be incorporated by reference in this section. Nothing in this section shall restrict Mentor Graphics' right to bring an action (including for example a motion for injunctive relief) against Customer in the jurisdiction where Customer's place of business is located. The United Nations Convention on Contracts for the International Sale of Goods does not apply to this Agreement.
17. **SEVERABILITY.** If any provision of this Agreement is held by a court of competent jurisdiction to be void, invalid, unenforceable or illegal, such provision shall be severed from this Agreement and the remaining provisions will remain in full force and effect.
18. **MISCELLANEOUS.** This Agreement contains the parties' entire understanding relating to its subject matter and supersedes all prior or contemporaneous agreements. Any translation of this Agreement is provided to comply with local legal requirements only. In the event of a dispute between the English and any non-English versions, the English version of this Agreement shall govern to the extent not prohibited by local law in the applicable jurisdiction. This Agreement may only be modified in writing, signed by an authorized representative of each party. Waiver of terms or excuse of breach must be in writing and shall not constitute subsequent consent, waiver or excuse.