ModelSim®

Xilinx Edition II

Command Reference

Version 5.7g

Published: 18/Jun/03

The world's most popular HDL simulator

ModelSim support

Support for ModelSim is available from your FPGA vendor. See the About ModelSim dialog box (accessed via the Help menu) for contact information.

Table of Contents

Syntax and conventions (CR-5)

Commands (CR-23)

Syntax and conventions

Chapter contents

Documentation conventions

This manual uses the following conventions to define ModelSim 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[a]
{ }	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 (#) or by two hyphens (--); useful for adding comments to DO files (macros)

a. 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: Neither the prompt at the beginning of a line nor the <Enter> key that ends a line is shown in the command examples.

Command return values

All simulator commands are invoked using Tcl. For most commands that write information to the Main window, that information is also available as a Tcl result. By using command substitution the results can be made available to another command or assigned to a Tcl variable. For example:

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

sets variable “aluinputs” to the result of the **find** command (CR-79).

Command shortcuts

- You may abbreviate command syntax, but there’s a catch — the minimum number of characters required to execute a command are those that make it unique. Remember, as we add new commands some of the old shortcuts may not work.
- Multiple commands may be entered 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
  ```

  You probably expect the **simstats** results to display in the Transcript window, but they will 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

The simulator command history may be reviewed, or commands may be reused, with these shortcuts at the ModelSim/VSIM prompt:

Shortcut	Description
up and down arrows	scrolls through the command history with the keyboard arrows
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
`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. Two styles can be used for VHDL numbers, one for Verilog.

VHDL numbering conventions

The first of two VHDL number styles is:

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

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

Note: A ‘-’ can also be used to designate a "don’t care" element when you search for a signal in the List or Wave window. 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 -0110--. If you don’t include the double quotes, ModelSim will read the ‘-’ as a negative sign.

Examples

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

The second VHDL number style is:

```
base "value"
```

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

Examples

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

Verilog numbering conventions

Verilog numbers are expressed in the style:

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

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

Note: A ‘-’ can also be used to designate a "don’t care" element when you search for a signal 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.

Examples

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

File and directory pathnames

Several ModelSim commands have arguments that point to files or directories. For example, the **-y** argument to vlog 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 quotes. For example:

```
vlog top.v -y C:/Documents\ and\ Settings/mcarnes/simprims
```

or

```
vlog top.v -y "C:/Documents and Settings/mcarnes/simprims"
```

HDL item names

VHDL and Verilog items are organized hierarchically. Each of the following HDL items 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

Item name syntax

The syntax for specifying item names in ModelSim is as follows:

```
[<datasetName><datasetSeparator>][<pathSeparator>][<hierarchicalPath>]<item
Name>[<elementSelection>]
```

where

`datasetName`
is the logical name of the WLF file in which the item 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. See *Chapter 7 - WLF files (datasets) and virtuals* for more information.

`datasetSeparator`
is the character used to terminate the dataset name. The default is ’:’, though a different character (other than ’\’) may be specified as the dataset separator via the DatasetSeparator (UM-353) variable in the *modelsim.ini* file. The default is ':'. This character must be different than the pathSeparator character.

`pathSeparator`
is the character used to separate hierarchical item names. Normally, '/' is used for VHDL and '.' is used for Verilog, although other characters (except '\') may be specified via the PathSeparator (UM-355) variable in the *modelsim.ini* file. This character must be different than the datasetSeparator.

`hierarchicalPath`
is a set of instance names each separated by a path separator.

`itemName`
is the name of an object in a design.

`elementSelection`
indicates some combination of the following:

Array indexing - Single array elements are specified using either parentheses "()" or square brackets "[]" around a single number.

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 ", ":".

Record field selection - A record field is specified using a period "." followed by the name of the field.

Specifying names

We distinguish between four "types" of item names: simple, relative, fully-rooted, and absolute.

A simple name does not contain any hierarchy. It is simply the name of an item (e.g., *clk* or *data[3:0]*) in the current context.

A relative name does not start with a path separator and may or may not include a dataset name or a hierarchical path (e.g., *u1/data* or *view:clk*). A relative name is relative to the current context in the current or specified dataset.

A fully-rooted name starts with a path separator and includes a hierarchical path to an item (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 (e.g., */u1/clk*). In this case, the first top-level instance in the design is assumed.

An absolute name is an exactly specified hierarchical name containing a dataset name and a fully rooted name (e.g., *sim:/top/u1/clk*).

The current dataset is used when accessing items 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** command (CR-74).

The current context in the current or specified dataset is used when accessing items 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** command (CR-74) to set the current context to a different instance.

Here are some examples of item names and what they specify:

Syntax	Description
clk	specifies the item *clk* in the current context
/top/clk	specifies the item *clk* in the top-level design unit.
/top/block1/u2/clk	specifies the item *clk*, two levels down from the top-level design unit
block1/u2/clk	specifies the item *clk*, two levels down from the current context
array_sig[4]	specifies an index of an array item
{array_sig(1 to 10)}	specifies a slice of an array item in VHDL syntax
{mysignal[31:0]}	specifies a slice of an array item in Verilog syntax
record_sig.field	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. See "Environment variables" (UM-345) for more information.

Note: Environment variable expansion *does not* occur in files that are referenced via the **-f** argument to **vcom**, **vlog**, or **vsim**.

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. 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 trailing space.

\\ext\ ident\!\\    # All non-alpha characters escaped
```

Wildcard characters

Wildcard characters can be used in HDL item names in some simulator commands. Conventions for wildcards are as follows:

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 *only* with the **find** command (CR-79)

The **WildcardFilter** Tcl preference variable filters matching items for the add wave, add log, add list, and find commands.

Note: A wildcard character will never match a path separator. For example, */dut/** will match */dut/siga* and */dut/clk*. However, */dut** won't match either of those.

ModelSim variables

Several variables are available to control simulation, provide simulator state feedback, or modify the appearance of the ModelSim GUI. To take effect, some variables, such as environment variables, must be set prior to simulation.

ModelSim variables can be referenced in simulator commands by preceding the name of the variable with the dollar sign ($) character. ModelSim uses global Tcl variables for simulator state variables, simulator control variables, simulator preference variables, and user-defined variables (see "Preference variables located in Tcl files" (UM-360) for more information).

See *Appendix A - ModelSim variables* in the User's Manual for more information on variables.

Variable settings report

The **report** command (CR-109) 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. See UserTimeUnit (UM-356).

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.

Comments in argument files

Argument files may be loaded with the **-f <filename>** argument of the **vcom**, **vlog**, and **vsim** commands. The **-f <filename>** argument specifies a file that contains more command line arguments.

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.

Also, program arguments can be placed on separate lines in the argument file, with the newline characters treated as space characters. There is no need to put '\' at the end of each line.

DOS pathnames require a backslash (\), but ModelSim will accept either a backslash or the forward slash (/).

Note: VHDL93 uses backslashes to denote extended identifiers. By default ModelSim PE/PLUS uses backslashes as pathname separators. Therefore it cannot recognize extended identifiers.

You can change this behavior so that backslashes on comment lines are used for extended identifiers, but then you can only use forward slashes when you need pathname delimiters. To do this, "uncomment" the following line in the *modelsim.ini* file and set its value to zero.

```
BackslashesArePathnameDelimiters = 0
```

This will allow command lines that can reference signals, variables, and design unit names that use extended identifiers; for example:

```
examine \clock 2x\
```

GUI_expression_format

The GUI_expression_format is an option of several simulator commands that operate within the ModelSim GUI environment. The commands that use the expression format are:

configure (CR-51), **examine** (CR-75), **searchlog** (CR-116), **virtual function** (CR-165), and **virtual signal** (CR-177)

Expression typing

GUI expressions are typed. The supported types consist of six scalar types and two 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' 'H' 'L' 'W' 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 array types supported 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)
```

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/vhdlsig
vlogsig[3]
vhdlsig(9)
vlogsig[5:2]
vhdlsig(5 downto 2)
```

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 we do not support leaving full arrays as elements in the result. (Please let us know if you need that option.)

If the elements being concatenated are of incompatible base type, a VHDL-style record will be created. The record object can be expanded in the Signals 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

The 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, the concatenation will be created with descending index range from (*n*-1) downto 0, where *n* is the number of elements in the array. The **concat_range** directive completely specifies the index range. The **concat_ascending** directive specifies that the index start at zero and increment upwards. The **concat_flatten** directive flattens the signal structure hierarchy. The **concat_sort_wild_ascending** directive gathers signals by name in ascending order (the default is descending).

```
(concat_range 31:0)<concatenationExpr> # Verilog syntax
(concat_range (31:0))<concatenationExpr> # Also Verilog syntax
(concat_range (31 downto 0))<concatenationExpr> # VHDL syntax
(concat_ascending) <concatenationExpr>
(concat_flatten) <concatenationExpr> # no hierarchy
(concat_sort_wild_ascending) <concatenationExpr>
```

Examples

```
&{ "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_ascending)&{ "mybusbasename*" }
```

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

```
(concat_ascending)((concat_sort_wild_ascending)&{"mybusbasename*" })
```

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

VHDL record field support

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:

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

Grouping and precedence

Operator precedence generally follows that of the C language, but we recommend liberal use of parentheses.

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 curly 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

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 X z Z U H L W 'U' 'X' '0' '1' 'Z' 'H' 'L' 'W' '-' 1'b0 1'b1

Array constants, expressed in any of the following formats

Type	Values
VHDL # notation	<int>#<alphanum>[#] Example: 16#abc123#
VHDL bitstring	"(U|X|0|1|Z|L|H|W|-)*" Example: "11010X11"
VLOG 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..., OB... ModelSim automatically zero fills unspecified upper bits.

Variables

Variable	Type
Name of a signal	The name may be a simple name, a VHDL or VLOG style extended identifier, or a VHDL or VLOG 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 -- VLOG net, VLOG register, VLOG integer, or VLOG 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

Variable	Type
Name of a signal	-- VHDL signals of type bit_vector or std_logic_vector -- VLOG register -- VLOG net array A subrange or index may be specified in either VHDL or VLOG 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" (CR-21) below for further details on ’delayed and ’hasX.

Operators

Operator	Description
&&	boolean and
||	boolean or
!	boolean not
==	equal
!=	not equal
===	exact equal
!==	exact not equal
<	less than
<=	less than or equal
>	greater than
>=	greater than or equal
not/NOT or ~	unary bitwise inversion
and/AND/&	bitwise and
nand/NAND	bitwise nand
or/OR/|	bitwise or
nor/NOR	bitwise nor
xor/XOR	bitwise xor
xnor/XNOR	bitwise xnor

Operator	Description
sll/SLL	shift left logical
sla/SLA	shift left arithmetic
srl/SRL	shift right logical
sra/SRA	shift right arithmetic
ror/ROR	rotate right
rol/ROL	rotate left
+	arithmetic add
-	arithmetic subtract
*	arithmetic multiply
/	arithmetic divide
mod/MOD	arithmetic modulus
rem/REM	arithmetic remainder
|<vector_expr>	OR reduction
^<vector_expr>	XOR reduction

Note: Arithmetic operators use the std_logic_arith package.

Casting

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

```
/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/adder* 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 the undelayed */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.

Commands

Chapter contents

The commands here are entered either in macro files or on the command line of the Main window. Some commands are automatically entered on the command line when you use the ModelSim graphical user interface.

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

Note: ModelSim commands are case sensitive. Type them as they are shown in this reference.

Command reference table

The following table provides a brief description of each ModelSim command. Command details, arguments and examples can be found at the page numbers given in the Command name column.

Command name	Action
abort (CR-30)	halts the execution of a macro file interrupted by a breakpoint or error
add dataflow (CR-31)	adds the specified item to the Dataflow window
add list (CR-32)	lists VHDL signals and variables, and Verilog nets and registers, and their values in the List window
add log	also known as the **log** command; see **log** (CR-87)
add wave (CR-35)	adds VHDL signals and variables, and Verilog nets and registers to the Wave window
alias (CR-39)	creates a new Tcl procedure that evaluates the specified commands
batch_mode (CR-40)	returns a 1 if ModelSim is operating in batch mode, otherwise returns a 0
bd (CR-41)	deletes a breakpoint
bookmark add wave (CR-42)	adds a bookmark to the specified Wave window
bookmark delete wave (CR-43)	deletes bookmarks from the specified Wave window
bookmark goto wave (CR-44)	zooms and scrolls a Wave window using the specified bookmark
bookmark list wave (CR-45)	displays a list of available bookmarks
bp (CR-46)	sets a breakpoint
cd (CR-49)	changes the ModelSim local directory to the specified directory
change (CR-50)	modifies the value of a VHDL variable or Verilog register variable
configure (CR-51)	invokes the List or Wave widget configure command for the current default List or Wave window
dataset alias (CR-55)	assigns an additional name to a dataset
dataset clear (CR-56)	clears the current simulation WLF file
dataset close (CR-57)	closes a dataset
dataset info (CR-58)	reports information about the specified dataset
dataset list (CR-59)	lists the open dataset(s)
dataset open (CR-60)	opens a dataset and references it by a logical name
dataset rename (CR-61)	changes the logical name of an opened dataset
dataset save (CR-62)	saves data from the current WLF file to a specified file

Command name	Action
dataset snapshot (CR-63)	saves data from the current WLF file at a specified interval
delete (CR-65)	removes HDL items from either the List or Wave window
describe (CR-66)	displays information about the specified HDL item
disablebp (CR-67)	turns off breakpoints and when commands
do (CR-68)	executes commands contained in a macro file
drivers (CR-69)	displays in the Main window the current value and scheduled future values for all the drivers of a specified VHDL signal or Verilog net
dumplog64 (CR-70)	dumps the contents of the *vsim.wlf* file in a readable format
echo (CR-71)	displays a specified message in the Main window
edit (CR-72)	invokes the editor specified by the EDITOR environment variable
enablebp (CR-73)	turns on breakpoints and when commands turned off by the **disablebp** command (CR-67)
environment (CR-74)	displays or changes the current dataset and region environment
examine (CR-75)	examines one or more HDL items, and displays current values (or the values at a specified previous time) in the Main window
exit (CR-78)	exits the simulator and the ModelSim application
find (CR-79)	displays the full pathnames of all HDL items in the design whose names match the name specification you provide
force (CR-82)	allows you to apply stimulus to VHDL signals and Verilog nets and registers, interactively
help (CR-85)	displays in the Main window a brief description and syntax for the specified command
history (CR-86)	lists the commands executed during the current session
log (CR-87)	creates a wave log format (WLF) file containing simulation data for all HDL items whose names match the provided specifications
lshift (CR-89)	takes a Tcl list as argument and shifts it in-place one place to the left, eliminating the 0th element
lsublist (CR-90)	returns a sublist of the specified Tcl list that matches the specified Tcl glob pattern
modelsim (CR-91)	starts the ModelSim GUI without prompting you to load a design; valid only for Windows platforms
noforce (CR-92)	removes the effect of any active **force** (CR-82) commands on the selected HDL items
nolog (CR-93)	suspends writing of data to the WLF file for the specified signals

Command name	Action
notepad (CR-95)	opens a simple text editor
noview (CR-96)	closes a window in the ModelSim GUI
nowhen (CR-97)	deactivates selected **when** (CR-208) commands
onbreak (CR-98)	specifies command(s) to be executed when running a macro that encounters a breakpoint in the source code
onElabError (CR-99)	specifies one or more commands to be executed when an error is encountered during elaboration
onerror (CR-100)	specifies one or more commands to be executed when a running macro encounters an error
pause (CR-101)	interrupts the execution of a macro
precision (CR-102)	determines how real numbers display in the GUI
printenv (CR-103)	echoes to the Main window the current names and values of all environment variables
project (CR-104)	performs common operations on new projects
pwd (CR-105)	displays the current directory path in the Main window
quietly (CR-106)	turns off transcript echoing for the specified command
quit (CR-107)	exits the simulator
radix (CR-108)	specifies the default radix to be used
report (CR-109)	displays the value of all simulator control variables, or the value of any simulator state variables relevant to the current simulation
restart (CR-111)	reloads the design elements and resets the simulation time to zero
resume (CR-113)	resumes execution of a macro file after a **pause** command (CR-101), or a breakpoint
run (CR-114)	advances the simulation by the specified number of timesteps
searchlog (CR-116)	searches one or more of the currently open logfiles for a specified condition
shift (CR-118)	shifts macro parameter values down one place
show (CR-119)	lists HDL items and subregions visible from the current environment
simstats (CR-120)	reports performance-related statistics about active simulations
status (CR-121)	lists all currently interrupted macros
step (CR-122)	steps to the next HDL statement
stop (CR-123)	stops simulation in batch files; used with the **when** command (CR-208)
tb (CR-124)	displays a stack trace for the current process in the Main window

Command name	Action
transcript (CR-125)	controls echoing of commands executed in a macro file; also works at top level in batch mode
transcript file (CR-126)	sets or queries the pathname for the transcript file
tssi2mti (CR-127)	converts a vector file in Fluence Technology (formerly TSSI) Standard Events Format into a sequence of **force** (CR-82) and **run** (CR-114) commands
vcd add (CR-128)	adds the specified items to the VCD file
vcd checkpoint (CR-129)	dumps the current values of all VCD variables to the VCD file
vcd comment (CR-130)	inserts the specified comment in the VCD file
vcd dumpports (CR-131)	creates a VCD file that captures port driver data
vcd dumpportsall (CR-133)	creates a checkpoint in the VCD file that shows the current value of all selected ports
vcd dumpportsflush (CR-134)	flushes the VCD buffer to the VCD file
vcd dumpportslimit (CR-135)	specifies the maximum size of the VCD file
vcd dumpportsoff (CR-136)	turns off VCD dumping and records all dumped port values as x
vcd dumpportson (CR-137)	turns on VCD dumping and records the current value of all selected ports
vcd file (CR-138)	specifies the filename and state mapping for the VCD file created by a **vcd add** command (CR-128)
vcd files (CR-140)	specifies the filename and state mapping for the VCD file created by a **vcd add** command (CR-128); supports multiple VCD files
vcd flush (CR-142)	flushes the contents of the VCD file buffer to the VCD file
vcd limit (CR-143)	specifies the maximum size of the VCD file
vcd off (CR-144)	turns off VCD dumping and records all VCD variable values as x
vcd on (CR-145)	turns on VCD dumping and records the current values of all VCD variables
vcd2wlf (CR-146)	translates VCD files into WLF files
vcom (CR-147)	compiles VHDL design units
vdel (CR-153)	deletes a design unit from a specified library
vdir (CR-154)	lists the contents of a design library
verror (CR-155)	prints a detailed description of a message number
vgencomp (CR-156)	writes a Verilog module's equivalent VHDL component declaration to standard output
view (CR-158)	opens a ModelSim window and brings it to the front of the display

Command name	Action
virtual count (CR-160)	counts the number of currently defined virtuals that were not read in using a macro file
virtual define (CR-161)	prints the definition of the virtual signal or function in the form of a command that can be used to re-create the object
virtual delete (CR-162)	removes the matching virtuals
virtual describe (CR-163)	prints a complete description of the data type of one or more virtual signals
virtual expand (CR-164)	produces a list of all the non-virtual objects contained in the virtual signal(s)
virtual function (CR-165)	creates a new signal that consists of logical operations on existing signals and simulation time
virtual hide (CR-168)	sets a flag in the specified real or virtual signals so that the signals do not appear in the Signals window
virtual log (CR-169)	causes the sim-mode dependent signals of the specified virtual signals to be logged by the simulator
virtual nohide (CR-171)	resets the flag set by a virtual hide command
virtual nolog (CR-172)	stops the logging of the specified virtual signals
virtual region (CR-174)	creates a new user-defined design hierarchy region
virtual save (CR-175)	saves the definitions of virtuals to a file
virtual show (CR-176)	lists the full path names of all the virtuals explicitly defined
virtual signal (CR-177)	creates a new signal that consists of concatenations of signals and subelements
virtual type (CR-180)	creates a new enumerated type
vlib (CR-182)	creates a design library
vlog (CR-183)	compiles Verilog design units
vmake (CR-189)	creates a makefile that can be used to reconstruct the specified library
vmap (CR-191)	defines a mapping between a logical library name and a directory by modifying the *modelsim.ini* file
vsim (CR-192)	loads a new design into the simulator
vsim<info> (CR-206)	returns information about the current vsim executable
vsource (CR-207)	specifies an alternative file to use for the current source file
when (CR-208)	instructs ModelSim to perform actions when the specified conditions are met
where (CR-213)	displays information about the system environment
wlf2log (CR-214)	translates a ModelSim WLF file(*vsim.wlf*) to a QuickSim II logfile

Command name	Action
wlfman (CR-216)	outputs information about or new WLF file from existing WLF file
wlfrecover (CR-218)	attempts to repair incomplete WLF files
write format (CR-219)	records the names and display options in a file of the HDL items currently being displayed in the List or Wave window
write list (CR-221)	records the contents of the List window in a list output file
write preferences (CR-222)	saves the current GUI preference settings to a Tcl preference file
write report (CR-223)	prints a summary of the design being simulated
write transcript (CR-224)	writes the contents of the Main window transcript to the specified file
write tssi (CR-225)	records the contents of the List window in a “TSSI format” file
write wave (CR-227)	records the contents of theWave window in PostScript format

abort

The **abort** command halts the execution of a macro file interrupted by a breakpoint or error. When macros are nested, you may choose to abort the last macro only, abort a specified number of nesting levels, or abort all macros. The **abort** command may be used within a macro to return early.

Syntax

```
abort
  [<n> | all]
```

Arguments

`<n> | all`
An integer giving the number of nested macro levels to abort; **all** aborts all levels. Optional. Default is 1.

See also

onbreak (CR-98), **onElabError** (CR-99), **onerror** (CR-100)

add dataflow

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

Syntax

```
add dataflow
  <item> [-window <wname>]
```

`<item>`
Specifies a process, signal, net, or register that you want to add to the Dataflow window. Required. Multiple items separated by spaces may be specified. Wildcards are allowed. (Note that the **WildcardFilter** Tcl preference variable identifies types to ignore when matching items with wildcard patterns.)

`-window <wname>`
Adds the items to the specified Dataflow window <wname> (e.g., dataflow2). Optional. Used to specify a particular window when multiple instances of that window type exist. Selects an existing window; does not create a new window. Use the **view** command (CR-158) with the **-new** option to create a new window.

See also

Dataflow window (UM-158)

add list

The **add list** command lists VHDL signals and variables and Verilog nets and registers in the List window, along with their associated values. User-defined buses may also be added for either language.

If no port mode is specified, **add list** will display all items in the selected region with names matching the item name specification.

Limitations: VHDL variables and Verilog memories can be listed using the variable's full name only (no wildcards).

Syntax

```
add list
  [-allowconstants] [-in] [-inout] [-internal]
  [[<item_name> | {<item_name> {sig1 sig2 sig3 ...}}] ...] ...
  [-label <name>] [-nodelta] [-notrigger | -trigger] [-out] [-ports]
  [-<radix>] [-recursive] [-width <n>]
```

Arguments

`-allowconstants`
For use with wildcard searches. Specifies that constants matching the wildcard search should be added to the List window. Optional. By default, constants are ignored because they do not change.

`-in`
For use with wildcard searches. Specifies that the scope of the search is to include ports of mode IN if they match the **item_name** specification. Optional.

`-inout`
For use with wildcard searches. Specifies that the scope of the search is to include ports of mode INOUT if they match the **item_name** specification. Optional.

`-internal`
For use with wildcard searches. Specifies that the scope of the search is to include internal items (non-port items) if they match the **item_name** specification. VHDL variables are not selected. Optional.

`<item_name>`
Specifies the name of the item to be listed. Optional. Wildcard characters are allowed. (Note that the **WildcardFilter** Tcl preference variable identifies types to ignore when matching items with wildcard patterns.) Variables may be added if preceded by the process name. For example,

```
add list myproc/int1
```

`{<item_name> {sig1 sig2 sig3 ...}}`
Creates a user-defined bus in place of **item_name**; 'sig*i*' are signals to be concatenated within the user-defined bus. Optional. Specified items may be either scalars or various sized arrays as long as they have the same element enumeration type.

`-label <name>`
Specifies an alternative signal name to be displayed as a column heading in the listing. Optional. This alternative name is not valid in a **force** (CR-82) or **examine** (CR-75) command; however, it can be used in a **searchlog** command (CR-116) with the **list** option.

`-nodelta`
Specifies that the delta column not be displayed when adding signals to the List window. Optional. Identical to **configure list -delta none**.

`-notrigger`
Specifies that items are to be listed, but does not cause the List window to be updated when the item changes. Optional.

`-out`
For use with wildcard searches. Specifies that the scope of the search is to include ports of mode OUT if they match the **item_name** specification. Optional.

`-ports`
For use with wildcard searches. Specifies that the scope of the search is to include all ports. Optional. Has the same effect as specifying **-in**, **-out**, and **-inout** together.

`-<radix>`
Specifies the radix for the items that follow in the command. Optional. Valid entries (or any unique abbreviations) are: binary, ascii character, unsigned decimal, octal, hex, symbolic, and default. If no radix is specified for an enumerated type, the default representation is used. You can change the default radix for the current simulation using the **radix** command (CR-108). You can change the default radix permanently by editing the DefaultRadix (UM-353) 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.

`-recursive`
For use with wildcard searches. Specifies that the scope of the search is to descend recursively into subregions. Optional; if omitted, the search is limited to the selected region.

`-trigger`
Specifies that items are to be listed and causes the List window to be updated when the items change. Optional. Default.

`-width <n>`
Specifies the column width in characters. Optional.

Examples

```
add list -r /*
```
Lists all items in the design.

```
add list *
```
Lists all items in the region.

```
add list -in *
```
Lists all input ports in the region.

```
add list a -label sig /top/lower/sig {array_sig(9 to 23)}
```
Displays a List window containing three columns headed *a*, *sig*, and *array_sig(9 to 23)*.

```
add list clk -notrigger a b c d
```
Lists *clk*, *a*, *b*, *c*, and *d* only when *clk* changes.

```
config list -strobeperiod {100 ns} -strobestart {0 ns} -usestrobe 1
add list -notrigger clk a b c d
```
Lists *clk*, *a*, *b*, *c*, and *d* every 100 ns.

```
add list -hex {mybus {msb {opcode(8 downto 1)} data}}
```
Creates a user-defined bus named "mybus" consisting of three signals; the bus is displayed in hex.

```
add list vec1 -hex vec2 -dec vec3 vec4
```
Lists the item *vec1* using symbolic values, lists *vec2* in hexadecimal, and lists *vec3* and *vec4* in decimal.

See also

add wave (CR-35), **log** (CR-87), "Extended identifiers" (CR-12)

add wave

The **add wave** command adds VHDL signals and variables and Verilog nets and registers to the Wave window. It also allows specification of user-defined buses.

If no port mode is specified, **add wave** will display all items in the selected region with names matching the item name specification.

Limitations: VHDL variables and Verilog memories can be listed using the variable's full name only (no wildcards).

Syntax

```
add wave
  [-allowconstants] [-color <standard_color_name>] [-expand <signal_name>]
  [-<format>] [-height <pixels>] [-in] [-inout] [-internal]
  [[-divider <divider_name>...] | [<item_name> | {<item_name> {sig1 sig2 sig3
  ...}}] ...] [-label <name>] [-noupdate] [-offset <offset>] [-out] [-ports]
  [-<radix>] [-recursive] [-scale <scale>]
```

Arguments

`-allowconstants`
For use with wildcard searches. Specifies that constants matching the wildcard search should be added to the Wave window. Optional. By default, constants are ignored because they do not change.

`-color <standard_color_name>`
Specifies the color used to display a waveform. Optional. These are the standard X Window color names, or rgb value (e.g., #357f77); enclose 2-word names ("light blue") in quotes.

`-divider <divider_name>`
Adds a divider with the specified name. Optional. You can specify one or more names. All names listed after **-divider** are taken to be names.

`-expand <signal_name>`
Causes a compound signal to be expanded immediately, but only one level down. Optional. The <signal_name> is required, and may include wildcards.

`-<format>`
Specifies the display format of the items:

```
literal
logic
analog-step
analog-interpolated
analog-backstep
```

Optional. Literal waveforms are displayed as a box containing the item value. Logic signals may be U, X, 0, 1, Z, W, L, H, or '-'.

Analog signals are sized by **-scale** and by **-offset**. Analog-step changes to the new time before plotting the new Y. Analog-interpolated draws a diagonal line. Analog-backstep plots the new Y before moving to the new time. See "Editing and formatting HDL items in the Wave window" (UM-228).

`-height <pixels>`
Specifies the height (in pixels) of the waveform. Optional.

`-in`
For use with wildcard searches. Specifies that the scope of the search is to include ports of mode IN if they match the item_name specification. Optional.

`-inout`
For use with wildcard searches. Specifies that the scope of the search is to include ports of mode INOUT if they match the item_name specification. Optional.

`-internal`
For use with wildcard searches. Specifies that the scope of the search is to include internal items (non-port items) if they match the item_name specification. Optional.

`<item_name>`
Specifies the names of HDL items to be included in the Wave window display. Optional. Wildcard characters are allowed. Note that the **WildcardFilter** Tcl preference variable identifies types to ignore when matching items with wildcard patterns. Variables may be added if preceded by the process name. For example,

```
add wave myproc/int1
```

`{<item_name> {sig1 sig2 sig3 ...}}`
Creates a user-defined bus with the name <item_name>; 'sig*i*' are signals to be concatenated within the user-defined bus. Optional.

Note: You can also select **Tools > Combine Signals** (Wave window) to create a user-defined bus.

`-label <name>`
Specifies an alternative name for the signal being added to the Wave window. Optional. For example,

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

adds the *clock* signal, labeled as "c", to the Wave window.

This alternative name is not valid in a **force** (CR-82) or **examine** (CR-75) command; however, it can be used in a **searchlog** command (CR-116) with the **wave** option.

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

`-offset <offset>`
Modifies an analog waveform's position on the display. Optional. The offset value is part of the wave positioning equation (see **-scale** below).

`-out`
For use with wildcard searches. Specifies that the scope of the search is to include ports of mode OUT if they match the item_name specification. Optional.

`-ports`
For use with wildcard searches. Specifies that the scope of the listing is to include ports of modes IN, OUT, or INOUT. Optional.

`-<radix>`
Specifies the radix for the items that follow in the command. Optional.

Valid entries (or any unique abbreviations) are: binary, ascii character, unsigned decimal, octal, hex, symbolic, and default. If no radix is specified for an enumerated type, the default representation is used. You can change the default radix for the current simulation using the **radix** command (CR-108). You can change the default radix permanently by editing the DefaultRadix (UM-353) 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.

`-recursive`

For use with wildcard searches. Specifies that the scope of the search is to descend recursively into subregions. Optional; if omitted, the search is limited to the selected region.

`-scale <scale>`
Scales analog waveforms. Optional. The scale value is part of the wave positioning equation shown below.

The position and size of the waveform is given by:

`(signal_value + <offset>) * <scale>`

If *signal_value* + <*offset*> = *0*, the waveform will be aligned with its name. The <*scale*> value determines the height of the waveform, 0 being a flat line.

Examples

`add wave -logic -color gold out2`
Displays an item named *out2*. The item is specified as being a logic item presented in gold.

`add wave -hex {address {a_7 a_6 a_5 a_4 a_3 a_2 a_1 a_0}}`
Displays a user-defined, hex formatted bus named *address*.

`add wave -r /*`
Waves all items in the design.

`add wave *`
Waves all items in the region.

`add wave -in *`
Waves all input ports in the region.

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

Creates 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.

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

Adds the item vec1 to the Wave window using symbolic values, adds vec2 in hexadecimal, and adds vec3 and vec4 in decimal.

See also

add list (CR-32), **log** (CR-87), "Extended identifiers" (CR-12), "Concatenation directives" (CR-16)

alias

The **alias** 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 ModelSim commands (e.g., run, env, etc.) cannot be aliased.

Syntax

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

Arguments

`<name>`
Specifies the new procedure name to be used when invoking the commands.

`"<cmds>"`
Specifies the command or commands to be evaluated when the alias is invoked.

Examples

`alias`
Lists all aliases currently defined.

`alias <name>`
Lists the alias definition for the specified name if one exists.

`alias <name>`
Lists the alias definition for the specified name if one exists.

```
alias myquit "write list ./mylist.save; quit -f"
```
Creates a Tcl procedure, "myquit", that when executed, writes the contents of the List window to the file *mylist.save* by invoking **write list** (CR-221), and quits ModelSim by invoking **quit** (CR-107).

batch_mode

The **batch_mode** command returns a 1 if ModelSim is operating in batch mode, otherwise it returns a 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 /*
}
```

See also

"Running command-line and batch-mode simulations" (UM-388)

bd

The **bd** command deletes a breakpoint. You must specify a filename and line number or a specific breakpoint id#. You may specify multiple filename/line number pairs and id#s.

Syntax

```
bd
  <filename> <line_number> | <id#>
```

Arguments

`<filename>`
Specifies the name of the source file in which the breakpoint is to be deleted. Required if an id# is not specified. The filename must match the one used previously to set the breakpoint, including whether a full pathname or a relative name was used.

`<line_number>`
Specifies the line number of the breakpoint to be deleted. Required if an id# is not specified.

`<id#>`
Specifies the id number of the breakpoint to be deleted. Required if a filename and line number are not specified. If you are deleting a C breakpoint, the id# will have a "c" prefix.

Examples

`bd alu.vhd 127`
Deletes the breakpoint at line 127 in the source file named *alu.vhd*.

`bd 5`
Deletes the breakpoint with id# 5.

`bd 6 alu.vhd 234`
Deletes the breakpoint with id# 6 and the breakpoint at line 234 in the source file named *alu.vhd*.

See also

bp (CR-46), **onbreak** (CR-98), *Chapter 13 - C Debug*

bookmark add wave

The **bookmark add wave** 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 (see **write format** command (CR-219)).

Syntax

```
bookmark add wave
  <label> <zoomrange> <topindex>
```

Arguments

`<label>`
Specifies the name for the bookmark. Required.

`<zoomrange>`
Specifies a list of two times with optional units. Required. These two times must be enclosed in braces ({}) or quotation marks ("").

`<topindex>`
Specifies the vertical scroll position of the window. Required. The number identifies which item the window should be scrolled to. For example, specifying 20 means the Wave window will be scrolled down to show the 20th item.

Examples

```
bookmark add wave foo {{10 ns} {1000 ns}} 20
```
Adds 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 item in the window.

See also

bookmark delete wave (CR-43), **bookmark goto wave** (CR-44), **bookmark list wave** (CR-45), **write format** (CR-219)

bookmark delete wave

The **bookmark delete wave** command deletes bookmarks from the specified Wave window.

Syntax

```
bookmark delete wave
  <label> [-all]
```

Arguments

`<label>`
Specifies the name of the bookmark to delete. Required unless the -all switch is used.

`-all`
Specifies that all bookmarks in the window be deleted. Optional.

Examples

`bookmark delete wave foo`
Deletes the bookmark named "foo" from the current default Wave window.

`bookmark delete wave -all -window wave1`
Deletes all bookmarks from the Wave window named "wave1".

See also

bookmark add wave (CR-42), **bookmark goto wave** (CR-44), **bookmark list wave** (CR-45), **write format** (CR-219)

bookmark goto wave

The **bookmark goto wave** command zooms and scrolls a Wave window using the specified bookmark.

Syntax

```
bookmark goto wave
  <label>
```

Arguments

`<label>`
Specifies the bookmark to go to. Required.

See also

bookmark add wave (CR-42), **bookmark delete wave** (CR-43), **bookmark list wave** (CR-45), **write format** (CR-219)

bookmark list wave

The **bookmark list wave** command displays a list of available bookmarks in the Main window transcript.

Syntax

```
bookmark list wave
```

Arguments

See also

bookmark add wave (CR-42), **bookmark delete wave** (CR-43), **bookmark goto wave** (CR-44), **write format** (CR-219)

bp

The **bp** or breakpoint command either sets a file-line breakpoint or returns a list of currently set breakpoints. A set breakpoint affects every instance in the design unless the -inst <region> argument is used.

Syntax

```
bp
  <filename> <line_number>
  [-id <id#>] [-inst <region>] [-disable] [-cond {<condition_expression>}]
  [{<command>...}] | [-query <filename> [<line_number> [line_number]]]
```

Arguments

`<filename>`
Specifies the name of the source file in which to set the breakpoint. Required if you are setting HDL breakpoints.

`<line_number>`
Specifies the line number at which the breakpoint is to be set. Required if you are setting HDL breakpoints.

`-id <id#>`
Attempts to assign this id number to the breakpoint. Optional. If the id number you specify is already used, ModelSim will return an error.

Note: Ids for breakpoints are assigned from the same pool as those used for the **when** command (CR-208). So, even if you haven't used an id number for a breakpoint, it's possible it is used for a **when** command.

`-inst <region>`
Sets the breakpoint so it applies only to the specified region. Optional.

`-disable`
Sets the breakpoint in a disabled state. Optional. You can enable the breakpoint later using the **enablebp** command (CR-73). By default, breakpoints are enabled when they are set.

`-cond {<condition_expression>}`
Specifies condition(s) that determine whether the breakpoint is hit. Optional. If the condition is true, the simulation stops at the breakpoint. If false, the simulation bypasses the breakpoint.

The condition can be an expression with these operators:

Name	Operator
equals	==, =
not equal	!=, /=
AND	&&, AND

Name	Operator
OR	||, OR

The operands may be item 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.

`{<command>...}`

Specifies one or more commands that are to be executed at the breakpoint. Optional. Multiple commands must be separated by semicolons (;) or placed on multiple lines. The entire command must be placed in curly braces.

Any commands that follow a **run** (CR-114) or **step** (CR-122) command will be ignored. A **run** or **step** command terminates the breakpoint sequence. This applies if macros are used within the **bp** command string as well. A **resume** (CR-113) command should not be used.

If many commands are needed after the breakpoint, they can be placed in a macro file.

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

Returns information about the breakpoints set in the specified file. The information returned varies depending on which arguments you specify. See the examples below for details.

Examples

`bp`
Lists all existing breakpoints in the design, including the source file names, line numbers, breakpoint id#s, and any commands that have been assigned to breakpoints.

`bp alu.vhd 147`
Sets a breakpoint in the source file *alu.vhd* at line 147.

`bp alu.vhd 147 {do macro.do}`
Executes the *macro.do* macro file after the breakpoint.

`bp -disable test.vhd 22 {echo [exa var1]; echo [exa var2]}`
Sets a breakpoint at line 22 of the file *test.vhd* and examines the values of the two variables *var1* and *var2*. This breakpoint is initially disabled. It can be enabled with the **enablebp** command (CR-73).

`bp test.vhd 14 {if {$now /= 100} then {cont}}`
Sets a breakpoint in every instantiation of the file *test.vhd* at line 14. When that breakpoint is executed, the command is run. This command causes the simulator to continue if the current simulation time is not 100.

`bp -query testadd.vhd`
Lists the line number and enabled/disabled status (1 = enabled, 0 = disabled) of all breakpoints in *testadd.vhd*.

`bp -query testadd.vhd 48`
Lists details about the breakpoint on line 48. The output comprises six pieces of information: the first item (0 or 1) designates whether a breakpoint exists on the line (1 = exists, 0 = doesn't exist); the second item is always 1; the third item is the file name in the compiled source; the fourth item is the breakpoint line number; the fifth item is the breakpoint id; and the sixth item (0 or 1) designates whether the breakpoint is enabled (1) or disabled (0).

`bp -query testadd.vhd 2 59`
Lists all executable lines in *testadd.vhd* between lines 2 and 59.

Note: Any breakpoints set in VHDL code and called by either resolution functions or functions that appear in a port map are ignored.

See also

add dataflow (CR-31), **bd** (CR-41), **disablebp** (CR-67), **enablebp** (CR-73), **onbreak** (CR-98), **when** (CR-208)

cd

The **cd** 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>`
The directory to which to change. Optional. If no directory is specified, ModelSim changes to your home directory.

change

The **change** command modifies the value of a VHDL variable or Verilog register variable.

Syntax

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

Arguments

`<variable>`
Specifies the name of a variable. Required. HDL variable names must specify a scalar type or a one-dimensional array of character enumeration. You may also specify a record subelement, an indexed array, a sliced array, or a bit or slice of a register, as long as the type is one of the above.

`<value>`
Defines a value for the variable. Required. The specified value must be appropriate for the type of the variable.

Examples

`change count 16#FFFF`
Changes the value of the variable *count* to the hexadecimal value FFFF.

`change rega[16] 0`
Changes the value of *rega* that is specified by the index (i.e., 16).

`change foo[20:22] 011`
Changes the value of foo that is specified by the slice (i.e., 20:22).

See also

force (CR-82)

configure

The **configure** (**config**) command invokes the List or Wave widget configure command for the current default List or Wave window. To change the default window, use the **view** command (CR-158).

Syntax

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

  [-delta [all | collapse | none]] [-gateduration [<duration_open>]]
  [-gateexpr [<expression>]] [-usegating [<value>]]
  [-strobeperiod [<period>]] [-strobestart [<start_time>]]
  [-usesignaltriggers [<value>]] [-usestrobe [<value>]]

  [-childrowmargin [<pixels>]] [-cursorlockcolor [<color>]]
  [-gridcolor [<color>]] [-griddelta [<pixels>]] [-gridoffset [<time>]]
  [-gridperiod [<time>]] [-namecolwidth [<width>]] [-rowmargin [<pixels>]]
  [-signalnamewidth [<value>]] [-timecolor [<color>]]
  [-timeline [<value>]] [-valuecolwidth [<width>]] [-vectorcolor [<color>]]
  [-waveselectcolor [<color>] [-waveselectenable [<value>]]
```

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`
Specifies either the List or Wave widget to configure. Required.

`<option> <value>`

`-bg <color>`
Specifies the window background color. Optional.

`-fg <color>`
Specifies the window foreground color. Optional.

`-selectbackground <color>`
Specifies the window background color when selected. Optional.

`-selectforeground <color>`
Specifies the window foreground color when selected. Optional.

`-font <font>`
Specifies the font used in the widget. Optional.

`-height <pixels>`
Specifies the height in pixels of each row. Optional.

Arguments, List window only

`-delta [all | collapse | none]`
The **all** option displays a new line for each time step on which items change; **collapse** displays the final value for each time step; and **none** turns off the display of the delta column. To use **-delta**, **-usesignaltriggers** must be set to 1 (on). Optional.

`-gateduration [<duration_open>]`
The duration for gating to remain open beyond when **-gateexpr** (below) becomes false, expressed in x number of timescale units. Extends gating beyond the back edge (the last list row in which the expression evaluates to true). Optional. 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).

`-gateexpr [<expression>]`
Specifies the expression for trigger gating. Optional. (Use the **-usegating** argument to enable trigger gating.) The expression is evaluated when the List window would normally have displayed a row of data.

`-usegating [<value>]`
Enables triggers to be gated on (a value of 1) or off (a value of 0) by an overriding expression. Default is off. Optional. (Use the **-gatexpr** argument to specify the expression.) See "Setting List window display properties" (UM-184) for additional information on using gating with triggers.

`-strobeperiod [<period>]`
Specifies the period of the list strobe. (When using a time unit, the time value and unit must be placed in curly braces.) Optional.

`-strobestart [<start_time>]`
Specifies the start time of the list strobe. Optional.

`-usesignaltriggers [<value>]`
If 1, uses signals as triggers; if 0, not. Optional.

`-usestrobe [<value>]`
If 1, uses the strobe to trigger; if 0, not. Optional.

Arguments, Wave window only

`-childrowmargin [<pixels>]`
Specifies the distance in pixels between child signals. Optional.

`-cursorlockcolor [<color>]`
Specifies the color of a locked cursor. Default is red.

`-gridcolor [<color>]`
Specifies the background grid color; the default is grey50. Optional.

`-griddelta [<pixels>]`
Specifies the closest (in pixels) two grid lines can be drawn before intermediate lines will be removed. Optional. Default is 40.

`-gridoffset [<time>]`
Specifies the time (in user time units) of the first grid line. Optional. Default is 0.

`-gridperiod [<time>]`
Specifies the time (in user time units) between subsequent grid lines. Optional. Default is 1.

`-namecolwidth [<width>]`
Specifies in pixels the width of the name column. Optional.

`-rowmargin [<pixels>]`
Specifies the distance in pixels between top-level signals.

`-signalnamewidth [<value>]`
Controls the number of hierarchical regions displayed as part of a signal name shown in the pathname pane. Optional. Default of 0 displays the full path. 1 displays only the leaf path element, 2 displays the last two path elements, and so on.

`-timecolor [<color>]`
Specifies the time axis color; the default is green. Optional.

`-timeline [<value>]`
Specifies whether the horizontal axis displays simulation time (default) or grid period count. Default is zero. When set to 1, the grid period count is displayed.

`-valuecolwidth [<width>]`
Specifies in pixels the width of the value column.

`-vectorcolor [<color>]`
Specifies the vector waveform color; the default is #b3ffb3. Optional.

`-waveselectcolor [<color>]`
Specifies the background highlight color of a selected waveform. Default is gray30.

`-waveselectenable [<value>]`
Specifies whether the waveform background highlights when an item is selected. The default of 1 enables highlighting; 0 disables highlighting.

Examples

`config list -strobeperiod`
Displays the current value of the strobeperiod attribute.

`config list -strobeperiod {50 ns} -strobestart 0 -usestrobe 1`
Sets the strobe waveform and turns it on.

`config wave -vectorcolor blue`
Sets the wave vector color to blue.

`config wave -signalnamewidth 1`
Sets the display in the current Wave window to show only the leaf path of each signal.

See also

view (CR-158), "Preference variables located in Tcl files" (UM-360)

dataset alias

The **dataset alias** command assigns an additional name (alias) to a dataset. The dataset can then be referenced by that alias. A dataset can have any number of aliases, but all dataset names and aliases must be unique.

Syntax

```
dataset alias
  <dataset_name> [<alias_name>]
```

Arguments

`<dataset_name>`
Specifies the name of the dataset to which to assign the alias. Required.

`<alias_name>`
Specifies the alias name to assign to the dataset. Optional. If you don't specify an alias_name, ModelSim lists current aliases for the specified dataset_name.

See also

dataset list (CR-59), **dataset open** (CR-60), **dataset save** (CR-62)

dataset clear

The **dataset clear** command removes all event data from the current simulation WLF file while keeping all currently logged signals logged. Subsequent run commands will continue to accumulate data in the WLF file.

Syntax

```
dataset clear
```

Example

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

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

See also

"WLF files (datasets)" (UM-126), **log** (CR-87)

dataset close

The **dataset close** command closes an active dataset. To open a dataset, use the **dataset open** command.

Syntax

```
dataset close
  <logicalname> | [-all]
```

Arguments

`<logicalname>`
Specifies the logical name of the dataset or alias you wish to close. Required if -all isn't used.

`-all`
Closes all open datasets including the simulation. Optional.

See also

dataset open (CR-60)

dataset info

The **dataset info** command reports a variety of information about a dataset.

Syntax

```
dataset info
  <option> <dataset_name>
```

Arguments

`<option>`
Identifies what information you want reported. Required. Only one option per command is allowed. The current options include:

`name` - Returns the actual name of the dataset. Useful for identifying the real dataset name of an alias.

`file` - Returns the name of the WLF file associated with the dataset.

`exists` - Returns "1" if the dataset exists; "0" if it doesn't.

`<dataset_name>`
Specifies the name of the dataset or alias for which you want information. Required.

See also

dataset alias (CR-55), **dataset list** (CR-59), **dataset open** (CR-60)

dataset list

The **dataset list** command lists all active datasets.

Syntax

```
dataset list
  -long
```

Arguments

`-long`
Lists the filename corresponding to each dataset's or alias' logical name. Optional.

See also

dataset alias (CR-55), **dataset save** (CR-62)

dataset open

The **dataset open** command opens a WLF file (representing a prior simulation) and assigns it the logical name that you specify. To close a dataset, use **dataset close**.

Syntax

```
dataset open
  <filename> [<logicalname>]
```

Arguments

`<filename>`
Specifies the WLF file to open as a view-mode dataset. Required.

`<logicalname>`
Specifies the logical name for the dataset. Optional. This is a prefix that will identify the dataset in the current session. By default the dataset prefix will be the name of the specified WLF file.

Examples

`dataset open last.wlf test`
Opens the dataset file *last.wlf* and assigns it the logical name *test*.

See also

dataset alias (CR-55), **dataset list** (CR-59), **dataset save** (CR-62), **vsim** (CR-192) -view option

dataset rename

The **dataset rename** command changes the logical name of a dataset to the new name you specify.

Syntax

```
dataset rename
  <logicalname> <newlogicalname>
```

Arguments

`<logicalname>`
Specifies the existing logical name of the dataset. Required.

`<newlogicalname>`
Specifies the new logical name for the dataset. Required.

Examples

```
dataset rename test test2
```
Renames the dataset file "test" to "test2".

See also

dataset alias (CR-55), **dataset list** (CR-59), **dataset open** (CR-60)

dataset save

The **dataset save** 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
  <logicalname> <newlogicalname>
```

Arguments

`<datasetname>`
Specifies the name of the dataset you want to save. Required.

`<filename>`
Specifies the name of the file to save. Required.

Examples

`dataset save sim gold.wlf`
Saves all current log data in the sim dataset to the file "gold.wlf".

See also

dataset snapshot (CR-63)

dataset snapshot

The **dataset snapshot** command saves data from the current WLF file (*vsim.wlf* by default) at a specified interval. This lets you take sequential or cumulative "snapshots" of your simulation data.

Syntax

```
dataset snapshot
  [-dir <directory>] [-disable] [-enable] [-file <filename>] [-filemode
  overwrite | increment] [-mode cumulative | sequential] [-report] [-reset]
  -size <file size> | -time <simulation time>
```

Arguments

`-dir <directory>`
Specifies a directory into which the files should be saved. Optional. Default is to save into the directory where ModelSim is writing the current WLF file.

`-disable`
Turns snapshotting off. Optional. All other options are ignored if you specify **-disable**.

`-enable`
Turns snapshotting on. Optional. Default.

`-file <filename>`
Specifies the name of the file to save. Optional. Default is "vsim_snapshot". ".wlf" will be appended to the file and possibly an incrementing suffix if **-filemode** is set to "increment".

`-filemode overwrite | increment`
Specifies whether to overwrite the snapshot file each time a snapshot occurs. Optional. Default is "overwrite". If you specify "increment", a new file is created for each snapshot. An incrementing suffix (0 to n) is added to each new file (e.g., *vsim_snapshot_0.wlf*).

`-mode cumulative | sequential`
Specifies whether to keep all data from the time signals are first logged. Optional. Default is "cumulative". If you specify "sequential", the current WLF file is cleared every time a snapshot is taken. See the examples for further details.

`-report`
Lists current snapshot settings in the Main window transcript. Optional. All other options are ignored if you specify **-report**.

`-reset`
Resets values back to defaults. Optional. The behavior is to reset to default, then apply remainder of arguments on command line. See examples below. If specified by itself without any other arguments, -reset disables dataset snapshot.

`-size <file size>`
Specifies that a snapshot occurs based on WLF file size. You must specify either **-size** or **-time**. See examples below.

`-time <simulation time>`
Specifies that a snapshot occurs based on simulation time. You must specify either **-time** or **-size**. See examples below.

Examples

`dataset snapshot -size 10`
Creates the file *vsim_snapshot.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 -mode sequential`
Similar to the previous example but in this case the current WLF file is cleared every time it reaches 10 MB.

`dataset snapshot -time 1000000 -file gold.wlf -mode sequential -filemode increment`
Assuming simulator time units are ps, this command saves a file called "gold_n.wlf" every 1000000 ps. If you ran for 3000000 ps, you'd have three files: *gold_0.wlf* with data from 0 to 1000000 ps, *gold_1.wlf* with data from 1000001 to 2000000, and *gold_2.wlf* with data from 2000001 to 3000000.

Note: Because this example uses "sequential" mode, if you ran the simulation for 3500000 ps, the resulting *vsim.wlf* (the default log file) file will contain data only from 3000001 to 3500000 ps.

`dataset snapshot -reset -time 10000`
Enables snapshotting with time=10000 and default mode (cumulative) and default filemode (overwrite).

See also

dataset save (CR-62)

delete

The **delete** command removes HDL items from either the List or Wave window.

Syntax

```
delete
  list|wave [-window <wname>] <item_name>
```

Arguments

`list|wave`
Specifies the target window for the **delete** command. Required.

`-window <wname>`
Specifies the name of the List or Wave window to target for the **delete** command (the **view** command (CR-158) allows you to create more than one List or Wave window). Optional. If no window is specified the default window is used; the default window is determined by the most recent invocation of the **view** command (CR-158).

`<item_name>`
Specifies the name of an item. Required. Must match an item name used in an **add list** (CR-32) or **add wave** (CR-35) command. Multiple item names may be specified. Wildcard characters are allowed.

Examples

```
delete list -window list2 vec2
```
Removes the item *vec2* from the list2 window.

See also

add list (CR-32), **add wave** (CR-35), and "Wildcard characters" (CR-13)

describe

The **describe** command displays information about the specified HDL item. The description is displayed in the Main window (UM-145). The following kinds of items can be described:

- **VHDL**
  signals, variables, and constants
- **Verilog**
  nets and registers

VHDL signals and Verilog nets and registers may be specified as hierarchical names. VHDL variables and constants can be described only when visible from the current process that is either selected in the Process window or is the currently executing process (at a breakpoint for example).

Syntax

```
describe
  <name>
```

Arguments

`<name>`
The name of an HDL item for which you want a description.

disablebp

The **disablebp** command turns off breakpoints and **when** commands. To turn the breakpoints or when statements back on again, use the **enablebp** command.

Syntax

```
disablebp
  [<id#>]
```

Arguments

`<id#>`
Specifies a breakpoint or **when** command id to disable. Optional. If you don't specify an id#, all breakpoints are disabled.

See also

bd (CR-41), **bp** (CR-46), **enablebp** command (CR-73), **onbreak** (CR-98), **resume** (CR-113), **when** (CR-208)

do

The **do** command executes commands contained in a macro file. A macro file can have any name and extension. An error encountered during the execution of a macro file causes its execution to be interrupted, unless an **onerror** command (CR-100), **onbreak** command (CR-98), or the OnErrorDefaultAction Tcl variable has specified the **resume** command (CR-113).

Syntax

```
do
  <filename> [<parameter_value>]
```

Arguments

`<filename>`
Specifies the name of the macro file to be executed. Required. The name can be a pathname or a relative file name.

Pathnames are relative to the current working directory if the **do** command is executed from the command line. If the **do** command is executed from another macro file, pathnames are relative to the directory of the calling macro file. This allows groups of macro files to be moved to another directory and still work.

`<parameter_value>`
Specifies values that are to be passed to the corresponding parameters $1 through $9 in the macro file. Optional. Multiple parameter values must be separated by spaces.

If you want to make the parameters optional (i.e., specify fewer parameter values than the number of parameters actually used in the macro), you must use the argc (UM-362) simulator state variable in the macro. See "Making macro parameters optional" (UM-340).

Note that there is no limit on the number of parameters that can be passed to macros, but only nine values are visible at one time. You can use the **shift** command (CR-118) to see the other parameters.

Examples

`do macros/stimulus 100`
This command executes the file *macros/stimulus*, passing the parameter value 100 to $1 in the macro file.

`do testfile design.vhd 127`
If the macro file *testfile* contains the line **bp** $1 $2, this command would place a breakpoint in the source file named *design.vhd* at line 127.

See also

Chapter 12 - Tcl and macros (DO files) (UM-323), "Running command-line and batch-mode simulations" (UM-388), "Using a startup file" (UM-358), DOPATH (UM-345)

drivers

The **drivers** command displays in the Main window the current value and scheduled future values for all the drivers of a specified VHDL signal or Verilog net. The driver list is expressed relative to the top-most design signal/net connected to the specified signal/net. If the signal/net is a record or array, each subelement is displayed individually. This command reveals the operation of transport and inertial delays and assists in debugging models.

Syntax

```
drivers
  <item_name>
```

Arguments

`<item_name>`
Specifies the name of the signal or net whose values are to be shown. Required. All signal or net types are valid. Multiple names and wildcards are accepted.

dumplog64

The **dumplog64** 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.

The **dumplog64** command cannot be used in a DO file.

Syntax

```
dumplog64
  <filename>
```

Arguments

`<filename>`
The name of the WLF file to be read. Required.

echo

The **echo** command displays a specified message in the Main window.

Syntax

```
echo
  [<text_string>]
```

Arguments

`<text_string>`
Specifies the message text to be displayed. Optional. If the text string is surrounded by quotes, blank spaces are displayed as entered. If quotes are omitted, two or more adjacent blank spaces are compressed into one space.

Examples

```
echo “The time is    $now ns.”
```
If the current time is 1000 ns, this command produces the message:

```
The time is     1000 ns.
```
If the quotes are omitted, all blank spaces of two or more are compressed into one space.

```
echo The time is    $now ns.
```
If the current time is 1000ns, this command produces the message:

```
The time is 1000 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 produces:

```
The hex value of counter is 15.
```

edit

The **edit** command invokes the editor specified by the EDITOR environment variable.

Syntax

```
edit
  [<filename>]
```

Arguments

`<filename>`
Specifies the name of the file to edit. Optional. If the <filename> is omitted, the editor opens the current source file. If you specify a non-existent filename, it will open a new file.

See also

notepad (CR-95), and the EDITOR (UM-345) environment variable

enablebp

The **enablebp** command turns on breakpoints and **when** commands that were previously disabled.

Syntax

```
enablebp
  [<id#>]
```

Arguments

`<id#>`
Specifies a breakpoint or when statement id to enable. Optional. If you don't specify an id#, all breakpoints are enabled.

See also

bd (CR-41), **bp** (CR-46), **disablebp** command (CR-67), **onbreak** (CR-98), **resume** (CR-113), **when** (CR-208)

environment

The **environment**, or **env** command, allows you to display or change the current dataset and region/signal environment.

Syntax

```
environment
  [-dataset] [-nodataset] [[<dataset_prefix>] [<pathname>]]
```

Arguments

`-dataset`
Displays the specified environment pathname *with* a dataset prefix. Optional. Dataset prefixes are displayed by default if more than one dataset is open during a simulation session.

`-nodataset`
Displays the specified environment pathname *without* a dataset prefix. Optional.

`<dataset_prefix>`
Changes all unlocked windows to the specified dataset context. Optional. The prefix is the logical name of the dataset followed by a colon (e.g., "sim:"). If the `<pathname>` argument is specified as well, it will change the environment to that specified context. If `<pathname>` is omitted, the environment reflects the previously set context. If you don't specify a dataset prefix, then the current dataset is used.

`<pathname>`
Specifies the pathname to which the current region/signal environment is to be changed. Optional. If omitted the command causes the pathname of the current region/signal environment to be displayed.

Multiple levels of a pathname must be separated by the character specified in the PathSeparator (UM-355). A single path separator character can be entered to indicate the top level. Two dots (..) can be entered to move up one level.

Examples

`env`
Displays the pathname of the current region/signal environment.

`env -dataset test`
Changes all unlocked windows to the context of the "test" dataset.

`env test:/top/foo`
Changes all unlocked windows to the context "test: /top/foo".

`env blk1/u2`
Moves down two levels in the design hierarchy.

`env /`
Moves to the top level of the design hierarchy.

examine

The **examine**, or **exa** command, examines one or more HDL items, and displays current values (or the values at a specified previous time) in the Main window (UM-145).

The following items can be examined:

- **VHDL**
  signals, shared variables, process variables, constants, and generics
- **Verilog**
  nets, registers, and variables

To display a previous value, specify the desired time using the **-time** option.

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 item:

- 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 item 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 item name.
- If no items of the specified name can be found in the specified context, then an upward search is done to look for a matching item 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 items 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** won't match either of those.

See "HDL item names" (CR-10) for more information on specifying names.

Syntax

```
examine
  [-delta <delta>] [-env <path>] [-in] [-out] [-inout] [-internal] [-ports]
  [-name] [-<radix>] [-time <time>] [-value] <name>...
```

Arguments

`-delta <delta>`
Specifies a simulation cycle at the specified time from which to fetch the value. Optional. The default is to use the last delta of the time step. The items 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 delta. This option can be used only with items that have been logged via the add list, add wave, or log command.

`-env <path>`
Specifies a path to look for a signal name. Optional.

`-in`
Specifies that `<name>` include ports of mode IN. Optional.

`-out`
Specifies that `<name>` include ports of mode OUT. Optional.

`-inout`
Specifies that `<name>` include ports of mode INOUT. Optional.

`-internal`
Specifies that `<name>` include internal (non-port) signals. Optional.

`-ports`
Specifies that `<name>` include all ports. Optional. Has the same effect as specifying -in, -inout, and -out together.

`-name`
Displays signal name(s) along with the value(s). Optional. Default is **-value** behavior (see below).

`-<radix>`
Specifies the radix for the items that follow in the command. Valid entries (or any unique abbreviations) are: binary, ascii character, unsigned decimal, octal, hex, symbolic, and default. If no radix is specified for an enumerated type, the default representation is used. You can change the default radix for the current simulation using the **radix** command (CR-108). You can change the default radix permanently by editing the DefaultRadix (UM-353) variable in the *modelsim.ini* file.

`-time <time>`
Specifies the time value between 0 and $now for which to examine the items. Optional. The items 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. This option can be used only with items that have been logged via the add list, add wave, or log command.

If the <time> field uses a unit, the value and unit must be placed in curly braces. For example, the following are equivalent for ps resolution:

```
exa -time {3.6 ns} signal_a
exa -time 3600 signal_a
```

`-value`
Returns value(s) as a curly-braces separated Tcl list. Default. Use to toggle off a previous use of **-name**.

`<name>...`
Specifies the name of any HDL item. Required. All item types are allowed, except those of the type file. Multiple names and wildcards are accepted. Spaces, square brackets, and extended identifiers require curly 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
```

Examples

`examine {rega[16]}`
Returns the value of *rega* that is specified by the index (i.e., 16). Note that you must use curly braces when examining subelements.

`examine {foo[20:22]}`
Returns the value of foo specified by the slice (i.e., 20:22). Note the curly braces.

`examine {/top/\My extended id\ }`
Note that when specifying an item that contains an extended identifier as the last part of the name, there must be a space after the closing '\' and before the closing '}'.

See also

"HDL item names" (CR-10), "Wildcard characters" (CR-13),

exit

The **exit** command exits the simulator and the ModelSim application.

Syntax

```
exit
  [-force]
```

Argument

`-force`
Quits without asking for confirmation. Optional; if this argument is omitted, ModelSim asks you for confirmation before exiting.

Note: If you want to stop the simulation using a **when** command (CR-208), you must use a **stop** command (CR-123) within your when statement. DO NOT use an **exit** command or a **quit** command (CR-107). The **stop** command acts like a breakpoint at the time it is evaluated.

find

The **find** command locates items in the design whose names match the name specification you provide. You must specify the type of item you want to find. 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.

When searching for nets and signals, the order in which arguments are specified is unimportant. When searching for virtuals, however, all optional arguments must be specified before any item names.

The following rules are used by the find command to locate an item:

- 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 item 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 item name.
- If no items of the specified name can be found in the specified context, then an upward search is done to look for a matching item 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 items 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** won't 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 "HDL item names" (CR-10) for more information on specifying names.

Syntax

```
find nets | signals
  [-in] [-inout] [-internal] <item_name> ... [-nofilter] [-out] [-ports]
  [-recursive]

find virtuals
  [-kind <kind>] [-unsaved] <item_name> ...

find classes
  [<class_name>]

find objects
  [-class <class_name>] [-isa <class_name>] [<object_name>]
```

Arguments for nets and signals

`-in`
Specifies that the scope of the search is to include ports of mode IN. Optional.

`-inout`
Specifies that the scope of the search is to include ports of mode INOUT. Optional.

`-internal`
Specifies that the scope of the search is to include internal items. Optional.

`<item_name> ...`
Specifies the net or signal for which you want to search. Required. Multiple nets and signals and wildcard characters are allowed. Wildcard characters are accepted for primary names only. Wildcards in index and record filed names are not supported. Spaces, square brackets, and extended identifiers require special syntax; see the examples below for more details.

`-nofilter`
Specifies that the **WildcardFilter** Tcl preference variable be ignored when finding signals or nets. Optional.

`-out`
Specifies that the scope of the search is to include ports of mode OUT. Optional.

`-ports`
Specifies that the scope of the search is to include all ports. Optional. Has the same effect as specifying **-in**, **-out**, and **-inout** together.

`-recursive`
Specifies that the scope of the search is to descend recursively into subregions. Optional. If omitted, the search is limited to the selected region.

Arguments for virtuals

`-kind <kind>`
Specifies the kind of virtual object for which you want to search. Optional. <**kind**> can be one of designs, explicits, functions, implicits, or signals.

`-unsaved`
Specifies that ModelSim find only virtuals that have not been saved to a format file.

`<item_name> ...`
Specifies the virtual object for which you want to search. Required. Multiple virtuals and wildcard characters are allowed.

Arguments for classes

`<class_name>`
Specifies the incrTcl class for which you want to search. Optional. 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 for more information.

Arguments for objects

`-class <class_name>`
Restricts the search to objects whose most-specific class is **class_name**. Optional.

`-isa <class_name>`
Restricts the search to those objects that have **class_name** anywhere in their heritage. Optional.

`<object_name>`
Specifies the incrTcl object for which you want to search. Optional. 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 for more information.

Examples

`find signals -r /*`
Finds all signals in the entire design.

`find nets -in /top/xy*`
Finds all input signals in region /top that begin with the letters "xy".

`find signals -r u1/u2/cl*`
Finds all signals in the design hierarchy at or below the region *<current_context>/u1/u2* whose names begin with "cl".

`find signals {s[1]}`
Finds a signal named *s1*. Note that you must enclose the item in curly braces because of the square bracket wildcard characters.

`find signals {s[123]}`
Finds signals *s1*, *s2*, or *s3*.

`find signals s(1)`
Finds 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 {/top/data(3 downto 0)}`
Finds a 4-bit array named *data*. Note that you must use curly braces due to the spaces in the array name.

`find signals {/top/\My extended id\ }`
Note that when specifying an item that contains an extended identifier as the last part of the name, there must be a space after the closing '\' and before the closing '}'.

See also

"HDL item names" (CR-10), "Wildcard characters" (CR-13)

force

The **force** command allows you to apply stimulus interactively to VHDL signals and Verilog nets. Since **force** commands (like all commands) can be included in a macro file, it is possible to create complex sequences of stimuli.

You can force Virtual signals (UM-133) if the number of bits corresponds to the signal value. You cannot force virtual functions. In VHDL and mixed models, you cannot force an input port that is mapped at a higher level or that has a conversion function on the input.

You cannot force Verilog registers or variables – reg, integer, time, real (or realtime). These must be changed. See the **change** command (CR-50).

Syntax

```
force
  [-freeze | -drive | -deposit] [-cancel <time>] [-repeat <time>] <item_name>
  <value> [<time>] [, <value> <time> ...]
```

Arguments

`-freeze`
Freezes the item at the specified value until it is forced again or until it is unforced with a **noforce** command (CR-92). Optional.

`-drive`
Attaches a driver to the item and drives the specified value until the item is forced again or until it is unforced with a **noforce** command (CR-92). Optional.

This option is illegal for unresolved signals.

`-deposit`
Sets the item to the specified value. The value remains until there is a subsequent driver transaction, or until the item is forced again, or until it is unforced with a **noforce** command (CR-92). Optional.

If one of the **-freeze**, **-drive**, or **-deposit** options is not used, then **-freeze** is the default for unresolved items and **-drive** is the default for resolved items.

If you prefer **-freeze** as the default for resolved and unresolved VHDL signals, change the default force kind in the DefaultForceKind (UM-353) preference variable.

`-cancel <time>`
Cancels the **force** command at the specified **<time>**. The time is relative to the current time unless an absolute time is specified by preceding the value with the character @. Cancellation occurs at the last simulation delta cycle of a time unit. A value of zero cancels the force at the end of the current time period. Optional.

`-repeat <time>`
Repeats the **force** command, where **<time>** is the time at which to start repeating the cycle. The time is relative to the current time. A repeating **force** command will force a value before other non-repeating **force** commands that occur in the same time step. Optional.

<item_name>
Specifies the name of the HDL item to be forced. Required. A wildcard is permitted only if it matches one item. See "HDL item names" (CR-10) for the full syntax of an item name. The item name must specify a scalar type or a one-dimensional array of character enumeration. You may also specify a record subelement, an indexed array, or a sliced array, as long as the type is one of the above. Required.

<value>
Specifies the value to which the item is to be forced. The specified value must be appropriate for the type. Required.

A VHDL 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):

Value	Description
1111	character literal sequence
2#1111	binary radix
10#15	decimal radix
16#F	hexadecimal radix

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.

<time>
Specifies the time to which the value is to be applied. The time is relative to the current time unless an absolute time is specified by preceding the value with the character @. If the time units are not specified, then the default is the resolution units selected at simulation start-up. Optional.

A zero-delay force command causes the change to occur in the current (rather than the next) simulation delta cycle.

Examples

```
force input1 0
```
Forces *input1* to 0 at the current simulator time.

```
force bus1 01XZ 100 ns
```
Forces *bus1* to 01XZ at 100 nanoseconds after the current simulator time.

```
force bus1 16#f @200
```
Forces *bus1* to 16#F at the absolute time 200 measured in the resolution units selected at simulation start-up.

```
force input1 1 10, 0 20 -r 100
```
Forces *input1* to 1 at 10 time units after the current simulation time and to 0 at 20 time units after the current simulation time. This cycle repeats starting at 100 time units after the current simulation time, so the next transition is to 1 at 100 time units after the current simulation time.

```
force input1 1 10 ns, 0 {20 ns} -r 100ns
```
Similar to the previous example, but also specifies the time units. Time unit expressions preceding the "-r" must be placed in curly braces.

```
force s 1 0, 0 100 -repeat 200 -cancel 1000
```
Forces signal *s* to alternate between values 1 and 0 every 100 time units until time 1000. Cancellation occurs at the last simulation delta cycle of a time unit. So,

```
force s 1 0 -cancel 0
```
will force signal *s* to 1 for the duration of the current time period.

```
when {/mydut/siga = 10#1} {
  force -deposit /mydut/siga 10#85
}
```
Forces *siga* to decimal value 85 whenever the value on the signal is 1.

See also

noforce (CR-92), **change** (CR-50)

Note: You can configure defaults for the force command by setting the **DefaultForceKind** variable in the *modelsim.ini* file. See "Force command defaults" (UM-359).

help

The **help** command displays in the Main window a brief description and syntax for the specified command.

Syntax

```
help
  [<command> | <topic>]
```

Arguments

`<command>`
Specifies the command for which you want help. The entry is case and space sensitive. Optional.

`<topic>`
Specifies a topic for which you want help. The entry is case and space sensitive. Optional. 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

The **history** command lists the commands you have executed during the current session. History is a Tcl command. For more information, consult the Tcl Man Pages.

Syntax

```
history
  [clear] [keep <value>]
```

Arguments

`clear`
Clears the history buffer. Optional.

`keep <value>`
Specifies the number of executed commands to keep in the history buffer. Optional. The default is 50.

log

The **log** command creates a wave log format (WLF) file containing simulation data for all HDL items whose names match the provided specifications. Items (VHDL signals and variables, and Verilog nets and registers) that are displayed using the **add list** (CR-32) and **add wave** (CR-35) commands are automatically recorded in the WLF file. The log is stored in a WLF file (formerly a WAV file) in the working directory. By default the file is named *vsim.wlf*. You can change the default name using the -wlf option of the **vsim** (CR-192) command.

If no port mode is specified, the WLF file contains data for all items in the selected region whose names match the item name specification.

The WLF file is the source of data for the List and Wave windows. An item that has been logged and is subsequently added to the List or Wave window will have its complete history back to the start of logging available for listing and waving.

Limitations: Verilog memories and VHDL variables can be logged using the variable's full name only (no wildcards).

Syntax

```
log
  [-flush] [-howmany] [-in] [-inout] [-internal] [-out] [-ports]
  [-recursive] <item_name> ...
```

Arguments

`-flush`
Adds region data to the WLF file after each individual log command. Optional. Default is to add region data to the log file only when a command that advances simulation time is executed (e.g., run, step, etc.) or when you quit the simulation.

`-howmany`
Returns an integer indicating the number of signals found. Optional.

`-in`
Specifies that the WLF file is to include data for ports of mode IN whose names match the specification. Optional.

`-inout`
Specifies that the WLF file is to include data for ports of mode INOUT whose names match the specification. Optional.

`-internal`
Specifies that the WLF file is to include data for internal items whose names match the specification. Optional.

`-out`
Specifies that the WLF file is to include data for ports of mode OUT whose names match the specification. Optional.

`-ports`
Specifies that the scope of the search is to include all ports. Optional.

`-recursive`
Specifies that the scope of the search is to descend recursively into subregions. Optional; if omitted, the search is limited to the selected region.

`<item_name>`
Specifies the item name which you want to log. Required. Multiple item names may be specified. Wildcard characters are allowed. (Note that the **WildcardFilter** Tcl preference variable identifies types to ignore when matching items with wildcard patterns.)

Examples

```
log -r /*
```
Logs all items in the design.

```
log -out *
```
Logs all output ports in the current design unit.

See also

add list (CR-32), **add wave** (CR-35), **nolog** (CR-93), and "Wildcard characters" (CR-13)

Note: The log command is also known as the "add log" command.

lshift

The **lshift** command takes a Tcl list as an argument and shifts it in-place, one place to the left, eliminating the 0th element. The number of shift places may also be specified. Returns nothing.

Syntax

```
lshift
  <list> [<amount>]
```

Arguments

`<list>`
Specifies the Tcl list to target with **lshift**. Required.

`<amount>`
Specifies the number of places to shift. Optional. Default is 1.

Examples

```
proc myfunc args {

        # throws away the first two arguments

        lshift args 2

        ...

    }
```

See also

See the Tcl man pages (**Help > Tcl Man Pages**) for details.

lsublist

The **lsublist** command returns a sublist of the specified Tcl list that matches the specified Tcl glob pattern.

Syntax

```
lsublist
  <list> <pattern>
```

Arguments

`<list>`
Specifies the Tcl list to target with **lsublist**. Required.

`<pattern>`
Specifies the pattern to match within the <list> using Tcl glob-style matching. Required.

Examples

In the example below, variable ‘t’ returns "structure signals source".

```
set window_names "structure signals variables process source wave list
dataflow"

set t [lsublist $window_names s*]
```

See also

The **set** command is a Tcl command. See the Tcl man pages (**Help > Tcl Man Pages**) for details.

modelsim

The **modelsim** command starts the ModelSim GUI without prompting you to load a design. This command 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 shortcut's properties. (Arguments work on the DOS command line too, of course.)

The simulator may be invoked from either the MODELSIM prompt after the GUI starts or from a DO file called by **modelsim**.

Syntax

```
modelsim
  [-do <macrofile>] [-project <project file>]
```

Arguments

`-do <macrofile>`
Specifies the DO file to execute when **modelsim** is invoked. Optional.

Note: In addition to the macro 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 (CR-192) is invoked.

`-project <project file>`
Specifies the *modelsim.ini* file to load for this session. Optional.

See also

vsim (CR-192), **do** (CR-68), and "Using a startup file" (UM-358)

noforce

The **noforce** command removes the effect of any active **force** (CR-82) commands on the selected HDL items. The **noforce** command also causes the item's value to be re-evaluated.

Syntax

```
noforce
  <item_name> ...
```

Arguments

<item_name>
Specifies the name of a item. Required. Must match an item name used in a previous **force** command (CR-82). Multiple item names may be specified. Wildcard characters are allowed.

See also

force (CR-82) and "Wildcard characters" (CR-13)

nolog

The **nolog** command suspends writing of data to the wave log format (WLF) file for the specified signals. 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 (CR-87) 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. If you are using dumplog64.c, you will need to get an updated version.

Syntax

```
nolog
  [-all] | [-reset] | [-recursive] [-in] [-out] [-inout] [-ports]
  [-internal] [-howmany] <item_name> ...
```

Arguments

`-all`
Turns off logging for all signals currently logged. Optional. Must be used alone without other arguments.

`-reset`
Turns logging back on for all signals unlogged. Optional. Must be used alone without other arguments.

`-recursive`
Specifies that the scope of the search is to descend recursively into subregions. Optional. If omitted, the search is limited to the selected region.

`-in`
Specifies that the WLF file is to turn off logging for ports of mode IN whose names match the specification. Optional.

`-out`
Specifies that the WLF file is to turn off logging for ports of mode OUT whose names match the specification. Optional.

`-inout`
Specifies that the WLF file is to turn off logging for ports of mode INOUT whose names match the specification. Optional.

`-ports`
Specifies that the scope of the search is to turn off logging for all ports. Optional.

`-internal`
Specifies that the WLF file is to turn off logging for internal items whose names match the specification. Optional.

`-howmany`
Returns an integer indicating the number of signals found. Optional.

`<item_name>`
Specifies the item name which you want to unlog. Required. Multiple item names may be specified. Wildcard characters are allowed.

Examples

```
nolog -r /*
```
Unlogs all items in the design.

```
nolog -out *
```
Unlogs all output ports in the current design unit.

See also

add list (CR-32), **add wave** (CR-35), **log** (CR-87)

notepad

The **notepad** command opens a simple text editor. It may be used to view and edit ASCII files or create new files. This mode can be changed from the Notepad Edit menu. See "Mouse and keyboard shortcuts" (UM-156) for a list of editing shortcuts.

Returns nothing.

Syntax

```
notepad
  [<filename>] [-r | -edit]
```

Arguments

`<filename>`
Name of the file to be displayed. Optional.

`-r | -edit`
Selects the notepad editing mode: -r for read-only, and -edit for edit mode. Optional. Edit mode is the default.

noview

The **noview** command closes a window in the ModelSim GUI. To open a window, use the **view** command.

Syntax

```
noview
  [*] <window_name>...
```

Arguments

*
  Wildcards can be used, for example: l* (List window), s* (Signal, Source, and Structure windows), even * alone (all windows). Optional.

<window_name>...
  Specifies the ModelSim window type to close. Multiple window types may be used; at least one type (or wildcard) is required. Available window types are:

  dataflow, list, process, signals, source, structure, variables, and wave

Examples

noview wave1
  Closes the Wave window named "wave1".

noview l*
  Closes all List windows.

noview s*
  Closes all Structure, Signals, and Source windows.

See also

view (CR-158)

nowhen

The **nowhen** command deactivates selected **when** (CR-208) commands.

Syntax

```
nowhen
  [<label>]
```

Arguments

`<label>`
Specifies an individual when command. Optional. Wildcards may be used to select more than one when command.

Examples

```
when -label 99 b {echo “b changed”}
…
nowhen 99
```
This **nowhen** command deactivates the **when** (CR-208) command labeled 99.

```
nowhen *
```
This **nowhen** command deactivates all **when** (CR-208) commands.

onbreak

The **onbreak** command is used within a macro. It specifies one or more commands to be executed when running a macro that encounters a breakpoint in the source code. Using the **onbreak** command without arguments will return the current **onbreak** command string. Use an empty string to change the **onbreak** command back to its default behavior (i.e., onbreak ""). In that case, the macro will be interrupted after a breakpoint occurs (after any associated **bp** command (CR-46) string is executed).

onbreak commands can contain macro calls.

Syntax

```
onbreak
  {[<command> [; <command>] ...]}
```

Arguments

`<command>`
Any command can be used as an argument to **onbreak**. 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 curly braces. It is an error to execute any commands within an **onbreak** command string following a **run** (CR-114), **run -continue**, or **step** (CR-122) command. This restriction applies to any macros or Tcl procedures used in the **onbreak** command string. Optional.

Examples

`onbreak {exa data ; cont}`
Examine the value of the HDL item data when a breakpoint is encountered. Then continue the **run** command (CR-114).

`onbreak {resume}`
Resume execution of the macro file on encountering a breakpoint.

See also

abort (CR-30), **bd** (CR-41), **bp** (CR-46), **do** (CR-68), **onerror** (CR-100), **resume** (CR-113), **status** (CR-121)

onElabError

The **onElabError** command specifies one or more commands to be executed when an error is encountered during elaboration. The command is used by placing it within the *modelsim.tcl* file or a macro. During initial design load **onElabError** may be invoked from within the *modelsim.tcl* file; during a simulation restart **onElabError** may be invoked from a macro.

Use the **onElabError** command without arguments to return to a prompt.

Syntax

```
onElabError
  {[<command> [; <command>] ...]}
```

Arguments

`<command>`
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 curly braces. Optional.

See also

do (CR-68)

onerror

The **onerror** command is used within a macro; it specifies one or more commands to be executed when a running macro encounters an error. Using the **onerror** command without arguments will return the current **onerror** command string. Use an empty string to change the **onerror** command back to its default behavior (i.e., onerror ""). Use **onerror** with a **resume** command (CR-113) to allow an error message to be printed without halting the execution of the macro file.

Syntax

```
onerror
  {[<command> [; <command>] ...]}
```

Arguments

`<command>`
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 curly braces. Optional.

Example

```
onerror {quit -f}
```
Forces the simulator to quit if an error is encountered while the macro is running.

See also

abort (CR-30), **do** (CR-68), **onbreak** (CR-98), **resume** (CR-113), **status** (CR-121)

Note: You can also set the global **OnErrorDefaultAction** Tcl variable in the *pref.tcl* file to dictate what action ModelSim takes when an error occurs. The onerror command is invoked only when an error occurs in the macro file that contains the onerror command. Conversely, **OnErrorDefaultAction** will run even if the macro does not contain a local onerror command. This can be useful when you run a series of macros from one script, and you want the same behavior across all macros.

pause

The **pause** command placed within a macro interrupts the execution of that macro.

Syntax

```
pause
```

Arguments

None.

Description

When you execute a macro and that macro gets interrupted, the prompt will change to:

```
VSIM (pause)7>
```

This “pause” prompt reminds you that a macro has been interrupted.

When a macro is paused, you may invoke another macro, and if that one gets interrupted, you may even invoke another — up to a nesting level of 50 macros.

If the status of nested macros gets confusing, use the **status** command (CR-121). It will show you which macros are interrupted, at what line number, and show you the interrupted command.

To resume the execution of the macro, use the **resume** command (CR-113). To abort the execution of a macro use the **abort** command (CR-30).

See also

abort (CR-30), **do** (CR-68), **resume** (CR-113), **run** (CR-114)

precision

The **precision** command determines how real numbers display in the graphic interface (e.g., Signals, Wave, Variables, and List windows). It does not affect the internal representation of a real number and therefore precision values over 17 are not allowed.

Syntax

```
precision
  [<digits>[#]]
```

Arguments

`<digits>[#]`
Specifies the number of digits to display. Optional. Trailing zeros are not displayed unless you append the '#' sign. See examples for more details.

Examples

`precision 4`
Results in 4 digits of precision. For example:

```
1.234 or 6543
```

`precision 8#`
Results in 8 digits of precision including trailing zeros. For example:

```
1.2345600 or 6543.2100
```

`precision 8`
Results in 8 digits of precision but doesn't print trailing zeros. For example:

```
1.23456 or 6543.21
```

printenv

The **printenv** command echoes to the Main window the current names and values of all environment variables. If variable names are given as arguments, prints only the names and values of the specified variables. Returns nothing. All results go to the Main window.

Syntax

```
printenv
  [<digits>[#]]
```

Arguments

`<var>...`
Specifies the name(s) of the environment variable(s) to print. Optional.

Examples

`printenv`
Prints all environment variable names and their current values. For example,

```
# CC = gcc
# DISPLAY = srl:0.0
...
```

`printenv USER HOME`
Prints the specified environment variables:

```
# USER = vince
# HOME = /scratch/srl/vince
```

project

The **project** commands are used to perform common operations on projects. Use this command outside of a simulation session.

Syntax

```
project
  [addfile <filename>] | [close] | [compileall] | [delete <project>] | [env]
  | [history] | [new <home_dir> <proj_name> [<defaultlibrary>]
  [<use_current>]] | [open <project>] | [removefile <filename>]
```

Arguments

`addfile <filename>`
Adds the specified file to the current open project. Optional.

`close`
Closes the current project. Optional.

`compileall`
Compiles all files in the current project. Optional.

`delete <project>`
Deletes a specified project file. Optional.

`env`
Returns the current project file. Optional.

`history`
Lists a history of manipulated projects. Optional.

`new <home_dir> <proj_name> [<defaultlibrary>] [<use_current>]`
Creates a new project under a specified home directory with a specified name and optionally a default library. Optional. If use_current is set to 1, then ModelSim uses the current *modelsim.ini* file when creating the project rather than the default.

`open <project>`
Opens a specified project file, making it the current project. Changes the current working directory to the project's directory. Optional.

`removefile <filename>`
Removes the specified file from the current project. Optional.

Examples

`vsim> project open /user/george/design/test3/test3.mpf`
Makes */user/george/design/test3* the current project and changes the current working directory to */user/george/design/test3*.

`vsim> project compile all`
Executes current project library build scripts.

pwd

The Tcl **pwd** command displays the current directory path in the Main window.

Syntax

```
pwd
```

Arguments

None.

quietly

The **quietly** command turns off transcript echoing for the specified command.

Syntax

```
quietly
  <command>
```

Arguments

`<command>`
Specifies the command for which to disable transcript echoing. Required. Any results normally echoed by the specified command will not be written to the Main window transcript. To disable echoing for all commands use the **transcript** command (CR-125) with the **-quietly** option.

See also

transcript (CR-125)

quit

The **quit** command exits the simulator.

Syntax

```
quit
```

Arguments

`-f or -force`
Quits without asking for confirmation. Optional; if this argument is omitted, ModelSim asks you for confirmation before exiting. (The -f and -force arguments are equivalent.)

`-sim`
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*).

Note: If you want to stop the simulation using a **when** command (CR-208), you must use a **stop** command (CR-123) within your when statement. DO NOT use an **exit** command (CR-78) or a **quit** command. The **stop** command acts like a breakpoint at the time it is evaluated.

radix

The **radix** command specifies the default radix to be used for the current simulation. The command can be used at any time. The specified radix is used for all commands (**force** (CR-82), **examine** (CR-75), **change** (CR-50), etc.) as well as for displayed values in the Signals, Variables, Dataflow, List, and Wave windows. You can change the default radix permanently by editing the DefaultRadix (UM-353) variable in the *modelsim.ini* file.

Syntax

```
radix
  [-symbolic | -binary | -octal | -decimal | -hexadecimal |
  -unsigned | -ascii]
```

Arguments

Entries may be truncated to any length. For example, **-symbolic** could be expressed as **-s** or **-sy**, etc. Optional.

Also, **-signed** may be used as an alias for **-decimal**. The **-unsigned** radix will display as unsigned decimal. The **-ascii** radix will display a Verilog item as a string equivalent using 8 bit character encoding.

If no arguments are used, the command returns the current default radix.

report

The **report** command displays the value of all simulator control variables, or the value of any simulator state variables relevant to the current simulation.

Syntax

```
report
  simulator control | simulator state
```

Arguments

`simulator control`
Displays the current values for all simulator control variables.

`simulator state`
Displays the simulator state variables relevant to the current simulation.

Examples

`report simulator control`
Displays all simulator control variables.

```
# UserTimeUnit = ns
# RunLength = 100
# IterationLimit = 5000
# BreakOnAssertion = 3
# DefaultForceKind = default
# IgnoreNote = 0
# IgnoreWarning = 0
# IgnoreError = 0
# IgnoreFailure = 0
# CheckpointCompressMode = 1
# NumericStdNoWarnings = 0
# StdArithNoWarnings = 0
# PathSeparator = /
# DefaultRadix = symbolic
# DelayFileOpen = 0
```

`report simulator state`
Displays all simulator state variables. Only the variables that relate to the design being simulated are displayed:

```
# now = 0.0
# delta = 0
# library = work
# entity = type_clocks
# architecture = full
# resolution = 1ns
```

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 the **Tools > Edit Preferences**.

See also

"Preference variables located in INI files" (UM-349), and "Preference variables located in Tcl files" (UM-360)

restart

The **restart** 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.) 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 the **-keeploaded** option to the **vsim** command (CR-192) 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.
- Shared libraries that implement VHDL foreign subprograms remain loaded (they are not reloaded) even if they also contain code for a foreign architecture.

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. See *Chapter 6 - Verilog PLI / VPI* for more information on the Verilog PLI.

Syntax

```
restart
  [-force] [-nobreakpoint] [-nolist] [-nolog] [-nowave]
```

Arguments

`-force`
Specifies that the simulation will be restarted without requiring confirmation in a popup window. Optional.

`-nobreakpoint`
Specifies that all breakpoints will be removed when the simulation is restarted. Optional. The default is for all breakpoints to be reinstalled after the simulation is restarted.

`-nolist`
Specifies that the current List window environment will **not** be maintained after the simulation is restarted. Optional. The default is for all currently listed HDL items and their formats to be maintained.

`-nolog`
Specifies that the current logging environment will **not** be maintained after the simulation is restarted. Optional. The default is for all currently logged items to continue to be logged.

`-nowave`
Specifies that the current Wave window environment will **not** be maintained after the simulation is restarted. Optional. The default is for all items displayed in the Wave window to remain in the window with the same format.

Note: You can configure defaults for the restart command by setting the **DefaultRestartOptions** variable in the *modelsim.ini* file. See "Restart command defaults" (UM-359).

See also

vsim (CR-192)

resume

The **resume** command is used to resume execution of a macro file after a **pause** command (CR-101), or a breakpoint. It may be input manually or placed in an **onbreak** (CR-98) command string. (Placing a **resume** command in a **bp** (CR-46) command string does not have this effect.) The **resume** command can also be used in an **onerror** (CR-100) command string to allow an error message to be printed without halting the execution of the macro file.

Syntax

```
resume
```

Arguments

None.

See also

abort (CR-30), **do** (CR-68), **onbreak** (CR-98), **onerror** (CR-100), **pause** (CR-101)

run

The **run** command advances the simulation by the specified number of timesteps.

Syntax

```
run
  [<timesteps>[<time_units>]] | -all | -continue | -next | -step | -over
```

Arguments

`<timesteps>[<time_units>]`
Specifies the number of timesteps for the simulation to run. The number may be fractional, or may be specified absolute by preceding the value with the character @. Optional. In addition, optional <time_units> may be specified as:

fs, ps, ns, us, ms, or **sec**

The default <timesteps> and <time_units> specifications can be changed during a ModelSim session by selecting **Simulate > Simulation Options** (Main window). See "Setting default simulation options" (UM-263). Time steps and time units may also be set with the RunLength (UM-355) and UserTimeUnit (UM-356) variables in the *modelsim.ini* file.

`-all`
Causes the simulator to run the current simulation forever, or until it hits a breakpoint or specified break event. Optional.

`-continue`
Continues the last simulation run after a **step** (CR-122) command, **step -over** command or a breakpoint. A **run -continue** command may be input manually or used as the last command in a **bp** (CR-46) command string. Optional.

`-next`
Causes the simulator to run to the next event time. Optional.

`-step`
Steps the simulator to the next HDL statement. Optional.

`-over`
Specifies that VHDL procedures, functions and Verilog tasks are to be executed but treated as simple statements instead of entered and traced line by line. Optional.

Examples

`run 1000`
Advances the simulator 1000 timesteps.

`run 10.4 ms`
Advances the simulator the appropriate number of timesteps corresponding to 10.4 milliseconds.

`run @8000`
Advances the simulator to timestep 8000.

See also

step (CR-122)

searchlog

The **searchlog** command searches one or more of the currently open logfiles for a specified condition. 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.

Syntax

```
searchlog
  [-count <n>] [-deltas] [-env <path>] [-expr {<expr>}] [-reverse]
  [-rising | -falling | -anyedge] [-startDelta <num>] <startTime>
  [-value <string>] <pattern>
```

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

`-count <n>`
Specifies to search for the <n>-th occurrence of the match condition, where <n> is a positive integer. Optional.

`-deltas`
Indicates to test for match on simulation delta cycles. Otherwise, matches are only tested for at the end of each simulation time step. Optional.

`-env <path>`
Provides a design region in which to look for the signal names. Optional.

`-expr {<expr>}`
Specifies a general expression of signal values and simulation time. Optional. **searchlog** will search until the expression evaluates to true. The expression must have a boolean result type. See "GUI_expression_format" (CR-15) for the format of the expression.

`-reverse`
Specifies to search backwards in time from <startTime>. Optional.

`-rising | -falling | -anyedge`
Specifies an edge to look for on a scalar signal. Optional. This option is ignored for compound signals. If no options are specified, the default is -anyedge.

`-startDelta <num>`
Indicates a simulation delta cycle on which to start. Optional.

`<startTime>`
Specifies the simulation time at which to start the search. Required. The time may be specified as an integer number of simulation units, or as {<num> <timeUnit>}, where <num> can be integer or with a decimal point, and <timeUnit> is one of the standard VHDL time units (fs, ps, ns, us, ms, sec).

`-value <string>`
Specifies to search until a single scalar or compound signal takes on this value. Optional.

`<pattern>`
Specifies one or more signal names or wildcard patterns of signal names to search on. Required unless the **-expr** argument is used.

See also

virtual signal (CR-177), **virtual log** (CR-169), **virtual nolog** (CR-172)

shift

The **shift** 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, etc. The previous value of $1 is discarded.

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 (UM-362) variable.

Syntax

```
shift
```

Arguments

None.

Description

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.

See also

do (CR-68)

show

The **show** command lists HDL items and subregions visible from the current environment. The items listed include:

- **VHDL**
  signals and instances
- **Verilog**
  nets, registers, tasks, functions, instances and memories

The **show** command returns formatted results to stdout. To eliminate formatting (to use the output in a Tcl script), use the **Show** command instead.

Syntax

```
show
  [-all] [<pathname>]
```

Arguments

`-all`
Display all names at and below the specified path recursively. Optional.

`<pathname>`
Specifies the pathname of the environment for which you want the items and subregions to be listed. Optional; if omitted, the current environment is assumed.

Examples

`show`
Lists the names of all the items and subregion environments visible in the current environment.

`show /uut`
Lists the names of all the items and subregions visible in the environment named /uut.

`show sub_region`
Lists the names of all the items and subregions visible in the environment named sub_region which is directly visible in the current environment.

See also

find (CR-79)

simstats

The **simstats** command returns performance-related statistics about the simulation.

If executed without arguments, the command returns a list of pairs like the following:

```
{memory 57376} {{working set} 56152} {time 0} {{cpu time} 0} {context 0} /
{{page faults} 0}
```

See the arguments below for descriptions of each pair.

Note: Some of the values may not be available on all platforms and other values may be approximates. Different operating systems report these numbers differently.

Syntax

```
simstats
  [memory | working | time | cpu | context | faults]
```

Arguments

`memory`
Returns the amount of virtual memory that the OS has allocated for vsim. Optional.

`working`
Returns the portion of allocated virtual memory that is currently being used by all vsim processes. Optional. If this number exceeds memory size, you will encounter performance degradation.

`time`
Returns the cumulative "wall clock time" of the run commands. Optional.

`cpu`
Returns the cumulative processor time of the run commands. Optional. Processor time differs from wall clock time in that processor time is only counted when the cpu is actually running vsim. If vsim is swapped out for another process, cpu time does not increase.

`context`
Returns the number of context swaps (vsim being swapped out for another process) that have occurred during the run commands. Optional.

`faults`
Returns the number of page faults that have occurred during the run commands. Optional.

status

The **status** command lists summary information about currently interrupted macros. 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** (CR-98) or **onerror** (CR-100) commands that have been defined for each interrupted macro.

Syntax

```
status
  [file | line]
```

Arguments

`file`
Reports the file pathname of the current macro.

`line`
Reports the line number of the current macro.

Examples

The transcript below contains examples of **resume** (CR-113), and **status** commands.

```
VSIM (pause) 4> 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 (pause) 5> resume
# Resuming execution of macro resume_test.do at line 4
```

See also

abort (CR-30), **do** (CR-68), **pause** (CR-101), **resume** (CR-113)

step

The **step** command steps to the next HDL statement. Current values of local HDL variables may be observed at this time using the Variables window. VHDL procedures and functions and Verilog tasks and functions can optionally be skipped over. When a wait statement or end of process is encountered, time advances to the next scheduled activity. The Process and Source windows will then be updated to reflect the next activity.

Syntax

```
step
  [-over] [<n>]
```

Arguments

`-over`
Specifies that VHDL procedures and functions and Verilog tasks and functions should be executed but treated as simple statements instead of entered and traced line by line. Optional.

`<n>`
Any integer. Optional. Will execute ‘n’ steps before returning.

See also

run (CR-114)

stop

The **stop** command is used with the **when** command (CR-208) 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
```

Arguments

None.

Use the **run** command (CR-114) with the **-continue** option to continue the simulation run, or the **resume** command (CR-113) 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 (CR-208), you must use a **stop** command within your when statement. DO NOT use an **exit** command (CR-78) or a **quit** command (CR-107). The **stop** command acts like a breakpoint at the time it is evaluated.

See also

bp (CR-46), **resume** (CR-113), **run** (CR-114), **when** (CR-208)

tb

The **tb** (traceback) command displays a stack trace for the current process in the Main window. This lists the sequence of HDL function calls that have been entered to arrive at the current state for the active process.

Syntax

```
tb
```

transcript

The **transcript** command controls echoing of commands executed in a macro file; it also works at top level in batch mode. If no option is specified, the current setting is reported.

Syntax

```
transcript
  [<filename> | off | -q | quietly]
```

Arguments

`on`
Specifies that commands in a macro file will be echoed to the Main window as they are executed. Optional.

`off`
Specifies that commands in a macro file will not be echoed to the Main window as they are executed. Optional.

`-q`
Returns "0" if transcripting is turned off or "1" if transcripting is turned on. Useful in a Tcl conditional expression. Optional.

`quietly`
Turns off the transcript echo for all commands. To turn off echoing for individual commands see the **quietly** command (CR-106). Optional.

Examples

`transcript on`
Commands within a macro file will be echoed to the Main window as they are executed.

`transcript`
If issued immediately after the previous example, the message:

```
Macro transcripting is turned on.
```

appears in the Main window.

See also

echo (CR-71)

transcript file

The **transcript file** command sets or queries the pathname for the transcript file. 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.

Syntax

```
transcript file
  [<filename>]
```

Arguments

`<filename>`
Specifies the full path and filename for the transcript file. Optional. 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 setting is returned.

Examples

```
transcript file ""
```
Closes the current transcript file and stops writing data to the file. This is a method for reducing the size of your transcript.

```
transcript file ""
run 1 ms
transcript file transcript
run 1 ms
```
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.

See also

"Transcript" (UM-147)

tssi2mti

The **tssi2mti** command is used to convert a vector file in Fluence Technology (formerly TSSI) Standard Events Format into a sequence of **force** (CR-82) and **run** (CR-114) commands. The stimulus is written to the standard output.

The source code for **tssi2mti** is provided in the file *tssi2mti.c* in the *examples* directory.

Syntax

```
tssi2mti
  <signal_definition_file> [<sef_vector_file>]
```

Arguments

`<signal_definition_file>`
Specifies the name of the Fluence Technology signal definition file describing the format and content of the vectors. Required.

`<sef_vector_file>`
Specifies the name of the file containing vectors to be converted. If none is specified, standard input is used. Optional.

Examples

```
tssi2mti trigger.def trigger.sef > trigger.do
```
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 exactly the same as the previous one, but uses the standard input instead.

See also

force (CR-82), **run** (CR-114), **write tssi** (CR-225)

vcd add

The **vcd add** command adds the specified items to a VCD file. The allowed items are Verilog nets and variables and VHDL signals of type bit, bit_vector, std_logic, and std_logic_vector (other types are silently ignored).

All **vcd add** commands must be executed at the same simulation time. The specified items 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

Syntax

```
vcd add
  [-r] [-in] [-out] [-inout] [-internal] [-ports] [-file <filename>]
  <item_name>
```

Arguments

`-r`
Specifies that signal and port selection occurs recursively into subregions. Optional. If omitted, included signals and ports are limited to the current region.

`-in`
Includes only port driver changes from ports of mode IN. Optional.

`-out`
Includes only port driver changes from ports of mode OUT. Optional.

`-inout`
Includes only port driver changes from ports of mode INOUT. Optional.

`-internal`
Includes only internal variable or signal changes. Excludes port driver changes. Optional.

`-ports`
Includes only port driver changes. Excludes internal variable or signal changes. Optional.

`-file <filename>`
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 (CR-140).

`<item_name>`
Specifies the Verilog or VHDL item to add to the VCD file. Required. Multiple items may be specified by separating names with spaces. Wildcards are accepted.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files. Verilog tasks are documented in the IEEE 1364 standard.

vcd checkpoint

The **vcd checkpoint** 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>`
Specifies the name of the VCD file. Optional. If omitted the command is executed on the file designated by the **vcd file** command (CR-138) or "dump.vcd" if **vcd file** was not invoked.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files.

vcd comment

The **vcd comment** command inserts the specified comment in the specified VCD file.

Syntax

```
vcd comment
  <comment string> [<filename>]
```

Arguments

`<comment string>`
Comment to be included in the VCD file. Required. Must be quoted by double quotation marks or curly braces.

`<filename>`
Specifies the name of the VCD file. Optional. If omitted the command is executed on the file designated by the **vcd file** command (CR-138) or "dump.vcd" if **vcd file** was not invoked.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files.

vcd dumpports

The **vcd dumpports** command creates a VCD file that includes port driver data.

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 task: $dumpports

Syntax

```
vcd dumpports
  [-direction] [-file <filename>] [-in] [-inout] [-out] [-unique]
  <item_name>
```

Arguments

`-direction`
Affects both VHDL and Verilog ports. Optional. Specifies that the port/variable type recorded in the VCD header for VHDL and Verilog ports shall be one of the following:

in, out, inout, internal, ports (includes in, out, and inout); the default is all ports

Note: The **-direction** argument is obsolete in ModelSim versions 5.5c and later. It is supported only for backwards compatibility with an NEC flow. See http://www.model.com/products/documentation/resim_vcd.pdf for information regarding its use in earlier versions.

`-file <filename>`
Specifies the path and name of a VCD file to create. Optional. Defaults to the current working directory and the filename *dumpports.vcd*. Multiple filenames can be opened during a single simulation.

`-in`
Includes ports of mode IN. Optional.

`-inout`
Includes ports of mode INOUT. Optional.

`-out`
Includes ports of mode OUT. Optional.

`-unique`
Generates unique vcd variable names for ports, even if those ports are connected to the same collapsed net. Optional.

`<item_name>`
Specifies the Verilog or VHDL item to add to the VCD file. Required. Multiple items may be specified by separating names with spaces. Wildcards are accepted.

Examples

```
vcd dumpports -in -file counter.vcd /test_counter/dut/*
```

Creates a VCD file named *counter.vcd* of all IN ports in the region /test_counter/dut/.

```
vcd dumpports -file addern.vcd /testbench/uut/*
vsim -vcdstim addern.vcd addern -gn=8 -do "add wave /*; run 1000"
```

These two commands resimulate a design from a VCD file. See "Resimulating a design from a VCD file" (UM-315) for further details.

vcd dumpportsall

The **vcd dumpportsall** 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

Syntax

```
vcd dumpportsall
  [<filename>]
```

Arguments

`<filename>`
Specifies the name of the VCD file. Optional. If omitted the command is executed on all open VCD files.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files.

vcd dumpportsflush

The **vcd dumpportsflush** command flushes the contents of the VCD file buffer to the specified VCD file.

Related Verilog task: $dumpportsflush

Syntax

```
vcd dumpportsflush
  [<filename>]
```

Arguments

`<filename>`
Specifies the name of the VCD file. Optional. If omitted the command is executed on all open VCD files.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files.

vcd dumpportslimit

The **vcd dumpportslimit** 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.

Related Verilog task: $dumpportslimit

Syntax

```
vcd dumpportslimit
  <dumplimit> [<filename>]
```

Arguments

`<dumplimit>`
Specifies the maximum VCD file size in bytes. Required.

`<filename>`
Specifies the name of the VCD file. Optional. If omitted the command is executed on all open VCD files.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files.

vcd dumpportsoff

The **vcd dumpportsoff** command turns off VCD dumping and records all dumped port values as x.

Related Verilog task: $dumpportsoff

Syntax

```
vcd dumpportsoff
  [<filename>]
```

Arguments

`<filename>`
Specifies the name of the VCD file. Optional. If omitted the command is executed on all open VCD files.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files.

vcd dumpportson

The **vcd dumpportson** 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>`
Specifies the name of the VCD file. Optional. If omitted the command is executed on all open VCD files.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files.

vcd file

The **vcd file** command specifies the filename and state mapping for the VCD file created by a **vcd add** command (CR-128). The **vcd file** command is optional. If used, it must be issued before any **vcd add** commands.

Related Verilog task: $dumpfile

Note: vcd file is included for backward compatibility. Use the **vcd files** command (CR-140) if you want to use multiple VCD files during a single simulation.

Syntax

```
vcd file
  [-direction] [-dumpports] [<filename>] [-map <mapping pairs>] [-nomap]
```

Arguments

`-direction`
Affects only VHDL ports. Optional. It specifies that the port/variable type recorded in the VCD header for VHDL ports shall be one of the following:

in, out, inout, internal, ports (includes in, out, and inout); the default is all ports

Note: The **-direction** argument is obsolete in ModelSim versions 5.5c and later. It is supported only for backwards compatibility with an NEC flow. See http://www.model.com/products/documentation/resim_vcd.pdf for information regarding its use in earlier versions.

`-dumpports`
Capture detailed port driver data for Verilog ports and VHDL std_logic ports. Optional. This option works only on ports, and subsequent **vcd add** command (CR-128) will accept only qualifying ports (silently ignoring all other specified items).

`<filename>`
Specifies the name of the VCD file that is created (the default is *dump.vcd*). Optional.

`-map <mapping pairs>`
Affects only VHDL signals of type std_logic. Optional. It allows you to override the default mappings. The mapping is 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. For example, to map L and H to z:

```
vcd file -map "L z H z"
```

Note that the quotes in the example above are a Tcl convention for command strings that include spaces.

`-nomap`
Affects only VHDL signals of type std_logic. Optional. 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.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files. Verilog tasks are documented in the IEEE 1364 standard.

vcd files

The **vcd files** command specifies a filename and state mapping for a VCD file created by a **vcd add** command (CR-128). The **vcd files** command is optional. If used, it must be issued before any **vcd add** commands.

Related Verilog task: $fdumpfile

Syntax

```
vcd files
  [-direction] <filename> [-map <mapping pairs>] [-nomap]
```

Arguments

`-direction`
Affects both VHDL and Verilog ports. Optional. It specifies that the port/variable type recorded in the VCD header for VHDL and Verilog ports shall be one of the following:

in, out, inout, internal, ports (includes in, out, and inout); the default is all ports

Note: The **-direction** argument is obsolete in ModelSim versions 5.5c and later. It is supported only for backwards compatibility with an NEC flow. See http://www.model.com/products/documentation/resim_vcd.pdf for information regarding its use in earlier versions.

`<filename>`
Specifies the name of a VCD file to create. Required. Multiple files can be opened during a single simulation.

`-map <mapping pairs>`
Affects only VHDL signals of type std_logic. Optional. It allows you to override the default mappings. The mapping is 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. For example, to map L and H to z:

```
vcd files -map "L z H z"
```

Note that the quotes in the example above are a Tcl convention for command strings that include spaces.

`-nomap`
Affects only VHDL signals of type std_logic. Optional. 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.

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 and the **vcd on** (CR-145) and **vcd off** (CR-144) 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
```

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files. Verilog tasks are documented in the IEEE 1364 standard.

vcd flush

The **vcd flush** 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>`
Specifies the name of the VCD file. Optional. If omitted the command is executed on the file designated by the **vcd file** command (CR-138) or *dump.vcd* if **vcd file** was not invoked.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files. Verilog tasks are documented in the IEEE 1364 standard.

vcd limit

The **vcd limit** command specifies the maximum size of a 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.

Related Verilog tasks: $dumplimit, $fdumplimit

Syntax

```
vcd limit
  <filesize> [<filename>]
```

Arguments

`<filesize>`
Specifies the maximum VCD file size in bytes. Required.

`<filename>`
Specifies the name of the VCD file. Optional. If omitted the command is executed on the file designated by the **vcd file** command (CR-138) or *dump.vcd* if **vcd file** was not invoked.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files. Verilog tasks are documented in the IEEE 1364 standard.

vcd off

The **vcd off** 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>`
Specifies the name of the VCD file. Optional. If omitted the command is executed on the file designated by the **vcd file** command (CR-138) or *dump.vcd* if **vcd file** was not invoked.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files. Verilog tasks are documented in the IEEE 1364 standard.

vcd on

The **vcd on** command turns on VCD dumping to the specified file and records the current values of all VCD variables. By default, **vcd on** is automatically performed at the end of the simulation time that the **vcd add** (CR-128) commands are performed.

Related Verilog tasks: $dumpon, $fdumpon

Syntax

```
vcd on
  [<filename>]
```

Arguments

`<filename>`
Specifies the name of the VCD file. Optional. If omitted the command is executed on the file designated by the **vcd file** command (CR-138) or *dump.vcd* if **vcd file** was not invoked.

See also

See *Chapter 11 - Value Change Dump (VCD) Files* for more information on VCD files. Verilog system tasks are documented in the IEEE 1364 standard.

vcd2wlf

vcd2wlf is a utility that translates a VCD (Value Change Dump) file into a WLF file that can be displayed in ModelSim using the **vsim -view** argument.

Syntax

```
vcd2wlf
  [-splitio] [-splitio_in_ext <extension>] [-splitio_out_ext <extension>]
  <vcd filename> <wlf filename>
```

Arguments

`-splitio`
Specifies extended VCD port values be split into their corresponding input and output components by creating 2 signals instead of just 1 in the resulting .wlf file. Optional. 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 a "__o" appended to it.

`-splitio_in_ext <extension>`
Specifies an extension to add to input-component signal names created by using `-splitio`. Optional.

`-splitio_out_ext <extension>`
Specifies an extension to add to output-component signal names created by using `-splitio`. Optional.

`<vcd filename>`
Specifies the name of the VCD file you want to translate into a WLF file. Required.

`<wlf filename>`
Specifies the name of the output WLF file. Required.

vcom

The **vcom** command is used to invoke VCOM, the Model Technology VHDL compiler. Use VCOM to compile VHDL source code into a specified working library (or to the **work** library by default).

This command may be invoked from within ModelSim or from the operating system command prompt. This command may also be invoked during simulation.

Compiled libraries are version dependent. For example you cannot use a library compiled with 5.5 in a simulation using 5.6 **vsim**.

Syntax

```
vcom
  [-87] [-93] [+acc[=<spec>][+<entity>[(architecture)]]] [-check_synthesis]
  [-debugVA] [-defercheck] [-explicit] [-f <filename>]
  [-force_refresh] [-help] [-ignoredefaultbinding] [-ignorevitalerrors]
  [-just abcep] [-skip abcep] [-line <number>] [-no1164]
  [-noaccel <package_name>] [-nocasestaticerror] [-nocheck]
  [-noindexcheck] [-nologo] [-nonstddriverinit]
  [-noothersstaticerror] [-norangecheck] [-novital] [-novitalcheck]
  [-nowarn <number>] [-O0] [-pedanticerrors]
  [-performdefaultbinding] [-quiet] [-rangecheck] [-refresh] [-s] [-source]
  [-time] [-version]
  [-work <library_name>] <filename>
```

Arguments

`-87`
Disables support for VHDL 1076-1993. This is the VCOM default. Optional. See additional discussion in the examples. Note that the default can be changed with the *modelsim.ini* file; see "Preference variables located in INI files" (UM-349).

`-93`
Specifies that the simulator is to support VHDL 1076-1993. Optional. Default is -87. See additional discussion in the examples.

`+acc[=<spec>][+<entity>[(architecture)]]`
Enables access to design objects that would otherwise become unavailable due to optimizations. Optional. Note that using this option may reduce optimizations.

<spec> currently has only one choice:

v–Enable access to variables, constants, and aliases in processes that would otherwise be merged due to optimizations.

<entity> and **(<architecture>)** specify the design unit(s) in which to allow the access. If **(<architecture>)** is not specified, then all architectures of a given **<entity>** are enabled for access.

`-check_synthesis`
Turns on limited synthesis rule compliance checking. Specifically, it checks to see that signals read by a process are in the sensitivity list. Optional. The checks understand only combinational logic, not clocked logic. Edit the CheckSynthesis (UM-350) variable in the *modelsim.ini* file to set a permanent default.

`-debugVA`
Prints a confirmation if a VITAL cell was optimized, or an explanation of why it was not, during VITAL level-1 acceleration. Optional.

`-defercheck`
Defers until run-time all compile-time range checking on constant index and slice expressions . As a result, index and slice expressions with invalid constant ranges that are never evaluated will not cause compiler error messages to be issued. Optional.

`-explicit`
Directs the compiler to resolve ambiguous function overloading by favoring the explicit function definition over the implicit function definition. Optional. 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 <filename>`
Specifies a file with more command line arguments. Optional. Allows complex argument strings to be reused without retyping. Environment variable expansion (for example in a pathname) *does not* occur in **-f** files.

`-force_refresh`
Forces the refresh of a module. Optional. When the compiler refreshes a design unit, it checks each dependency to ensure its source has not been changed and recompiled. If a dependency has been changed and recompiled, the compiler will not refresh the dependent design unit (unless you use **-force_refresh**). To avoid potential errors or mismatches caused by the dependency recompilation, you should recompile the dependent design unit's source rather than use this switch.

`-help`
Displays the command's options and arguments. Optional.

`-ignoredefaultbinding`
Instructs the compiler not to generate a default binding during compilation. Optional. You must explicitly bind all components in the design to use this switch.

`-ignorevitalerrors`
Directs the compiler to ignore VITAL compliance errors. Optional. 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.

`-just abcep`
Directs the compiler to "just" include:
`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 optional switch.

`-skip abcep`
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 optional switch.

`-line <number>`
Starts the compiler on the specified line in the VHDL source file. Optional. By default, the compiler starts at the beginning of the file.

`-no1164`
Causes the source files to be compiled without taking advantage of the built-in version of the IEEE **std_logic_1164** package. Optional. This will typically result in longer simulation times for VHDL programs that use variables and signals of type **std_logic**.

`-noaccel <package_name>`
Turns off acceleration of the specified package in the source code using that package.

`-nocasestaticerror`
Suppresses case static warnings. Optional. VHDL standards require that case 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`
Disables index and range checks. Optional. You can disable these individually using the **-noindexcheck** and **-norangecheck** arguments, respectively.

`-noindexcheck`
Disables checking on indexing expressions to determine whether indices are within declared array bounds. Optional.

`-nologo`
Disables startup banner. Optional.

`-nonstddriverinit`
Forces ModelSim to match pre-5.7c behavior in initializing drivers in a particular case. Optional. 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 connect 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 only does this in the specific cases where older versions used the actual signal's initial value.

`-noothersstaticerror`
Disables warnings that result from array aggregates with multiple choices having "others" clauses that are not locally static. Optional. If the **-pedanticerrors** switch is specified, this switch is ignored.

`-norangecheck`
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**. See "Range and index checking" (UM-51) for additional information.

`-novital`
Causes **vcom** to use VHDL code for VITAL procedures rather than the accelerated and optimized timing and primitive packages built into the simulator kernel. Optional. Allows breakpoints to be set in the VITAL behavior process and permits single stepping through the VITAL procedures to debug your model. Also all of the VITAL data can be viewed in the variables or signals windows.

`-novitalcheck`
Disables VITAL 2000 compliance checking if you are using VITAL 2.2b. Optional.

`-nowarn <number>`
Selectively disables an individual warning message. Optional. Multiple **-nowarn** switches are allowed. Warnings may be disabled for all compiles via the Main window **Options > Compile Options** menu command or the *modelsim.ini* file (see the "[vcom] VHDL compiler control variables" (UM-350)).

The warning message numbers are:

```
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 = compliance checks
7 = optimization messages
9 = signal value used in expression evaluated at elaboration
```

`-O0`
Lower the optimization to a minimum with **-O0** (capital oh zero). Optional. 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`
Forces ModelSim to error (rather than warn) on two conditions: 1) when a choice in a case statement is not a locally static expression; 2) when an array aggregate with multiple choices doesn't have a locally static "others" choice. Optional. This argument overrides **-nocasestaticerror** and **-noothersstaticerror** (see above).

`-performdefaultbinding`
Enables default binding when it has been disabled via the **RequireConfigForAllDefaultBinding** option in the *modelsim.ini* file. Optional.

`-quiet`
Disable 'loading' messages. Optional.

`-rangecheck`
Enables run time range checking. Default. Range checking can be disabled using the **-norangecheck** argument. See "Range and index checking" (UM-51) for additional information.

`-refresh`
Regenerates a library image. Optional. By default, the work library is updated; use **-work <library>** to update a different library. See **vcom** "Examples" (CR-152) for more information.

`-s`
Instructs the compiler not to load the **standard** package. Optional. This argument should only be used if you are compiling the **standard** package itself.

`-source`
Displays the associated line of source code before each error message that is generated during compilation. Optional; by default, only the error message is displayed.

`-time`
Reports the "wall clock time" **vcom** takes to compile the design. Optional. Note that if many processes are running on the same system, wall clock time may differ greatly from the actual "cpu time" spent on **vcom**.

`-version`
Returns the version of the compiler as used by the licensing tools, such as "Model Technology ModelSim SE vcom 5.5 Compiler 2000.01 Jan 29 2000".

`-work <library_name>`
Specifies a logical name or pathname of a library that is to be mapped to the logical library **work**. Optional; 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.

`<filename>`
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 or wildcards may be used (e.g., **.vhd*).

If no filenames are given, a dialog box pops up allowing you to graphically select the options and enter a filename.

Examples

```
vcom example.vhd
```
Compiles the VHDL source code contained in the file *example.vhd*.

```
vcom -87 o_units1.vhd o_units2.vhd
vcom -93 n_unit91.vhd n_unit92.vhd
```
ModelSim supports designs that use elements conforming to both the 1993 and the 1987 standards. Compile the design units separately using the appropriate switches.

Note that in the example above, the **-87** switch on the first line is redundant since the VCOM default is to compile to the 1987 standard.

```
vcom -noaccel numeric_std example.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 -explicit 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.

```
ARITHMETIC."="(left, right)
```

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.

```
vcom -work mylib -refresh
```
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).

vdel

The **vdel** command deletes a design unit from a specified library.

Syntax

```
vdel
  [-help] [-verbose] [-lib <library_name>] [-all | <design_unit>
  [<arch_name>]]
```

Arguments

`-help`
Displays the command's options and arguments. Optional.

`-verbose`
Displays progress messages. Optional.

`-lib <library_name>`
Specifies the logical name or pathname of the library that holds the design unit to be deleted. Optional; by default, the design unit is deleted from the **work** library.

`-all`
Deletes an entire library. Optional. BE CAREFUL! Libraries cannot be recovered once deleted, and you are not prompted for confirmation.

`<design_unit>`
Specifies the entity, package, configuration, or module to be deleted. Required unless **-all** is used.

`<arch_name>`
Specifies the name of an architecture to be deleted. Optional; if omitted, all of the architectures for the specified entity are deleted. Invalid for a configuration or a package.

Examples

`vdel -all`
Deletes the **work** library.

`vdel -lib synopsys -all`
Deletes the **synopsys** library.

`vdel xor`
Deletes the entity named **xor** and all its architectures from the **work** library.

`vdel xor behavior`
Deletes the architecture named **behavior** of the entity **xor** from the **work** library.

`vdel base`
Deletes the package named **base** from the **work** library.

vdir

The **vdir** command selectively lists the contents of a design library.

This command can also be used to check compatibility of a vendor library. If vdir cannot read a vendor-supplied library, the library may not be ModelSim compatible.

Syntax

```
vdir
  [-help] [-l] [-r] [-lib <library_name>] [<design_unit>]
```

Arguments

`-help`
Displays the command's options and arguments. Optional.

`-l`
Prints the version of **vcom** or **vlog** that each design unit was compiled under. Also prints the object-code version number that indicates which versions of **vcom/vlog** and ModelSim are compatible. This example was printed by **vdir -l** for the counter module in the **work** library:

```
# MODULE counter
# Verilog Version: OzO;ZAVlR1jO;>KYTg2kY2
# Source directory: ..\examples\projects\mixed
# Source modified time: 944001078
# Source file: ../examples/projects/verilog/counter.v
# Opcode format: 5.4 Beta 4; VLOG EE Object version 17
# Version number: e:VQh7zF_VJYN9MbEXUG_3
# Optimized Verilog design root: 1
# Language standard: 1
```

`-r`
Prints architecture information for each entity in the output.

`-lib <library_name>`
Specifies the logical name or the pathname of the library to be listed. Optional. By default, the contents of the **work** library are listed.

`<design_unit>`
Indicates the design unit to search for within the specified library. If the design unit is a VHDL entity, its architectures are listed. Optional. By default, all entities, configurations, modules, and packages in the specified library are listed.

Example

```
vdir -lib design my_asic
```
Lists the architectures associated with the entity named **my_asic** that reside in the HDL design library called **design**.

verror

The **verror** command prints a detailed description about a message number. It may also point to additional documentation related to the error.

Syntax

```
verror
  <msgNum>...
```

Arguments

<msgNum>
Specifies the message number of a ModelSim message. Required. This number can be obtained from messages that have the format:

** <Level>: ([<Tool>-[<Group>-]]<MsgNum>) <FormattedMsg>

Example

Say you see the following message in the transcript:

```
** Error (vsim-3601) foo.v(22): Too many Verilog port connections.
```

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 the **vgencomp** command to write its equivalent VHDL component declaration to standard output. Optional switches allow you to generate bit or vl_logic port types; std_logic port types are generated by default.

Syntax

```
vgencomp
  [-help] [-lib <library_name>] [-b] [-s] [-v] <module_name>
```

Arguments

`-help`
Displays the command's options and arguments. Optional.

`-lib <library_name>`
Specifies the pathname of the working library. If not specified, the default library **work** is used. Optional.

`-b`
Causes **vgencomp** to generate bit port types. Optional.

`-s`
Used for the explicit declaration of default std_logic port types. Optional.

`-v`
Causes **vgencomp** to generate vl_logic port types. Optional.

`<module_name>`
Specifies the name of the Verilog module to be accessed. Required.

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;
```

view

The **view** command will open a ModelSim window and bring that window to the front of the display.

To remove a window, use the **noview** command (CR-96).

Syntax

```
view
  [*] [-height <n>] [-icon] [-title {New Window Title} <window_type>] [-width
  <n>] [-x <n>] [-y <n>] <window_type>...
```

Arguments

`*`
Specifies that all windows be opened. Optional.

`-height <n>`
Specifies the window height in pixels. Optional.

`-icon`
Toggles the view between window and icon. Optional.

`-title {New Window Title} <window_type>`
Specifies the window title of the designated window. Curly braces are only needed for titles that include spaces. Double quotes can be used in place of braces, for example "New Window Title". If the new window title does not include spaces, no braces or quotes are needed. For example: *-title new_wave wave* assigns the title *new_wave* to the Wave window.

`-width <n>`
Specifies the window width in pixels. Optional.

`<window_type>...`
Specifies the ModelSim window type to view. Required. You do no need to type the full type (see examples below); implicit wildcards are accepted; multiple window types may be used. Available window types are:

```
dataflow, list, process, signals, source, structure, variables, wave
```

`-x <n>`
Specifies the window upper-left-hand x-coordinate in pixels. Optional.

`-y <n>`
Specifies the window upper-left-hand y-coordinate in pixels. Optional.

Examples

`view d`
Opens the Dataflow window.

`view si pr`
Opens the Signals and Process windows.

`view s`
Opens the Signals, Source, and Structure windows.

`view -title {My Wave Window} wave`
Opens a new wave window with My Wave Window as its title.

See also

noview (CR-96)

virtual count

The **virtual count** command counts the number of currently defined virtuals that were not read in using a macro file.

Syntax

```
virtual count
  [-kind <kind>]
```

Arguments

`-kind <kind>`
Specifies a subset of virtuals to look at. Optional. <*kind*> can be any of the following: signals, functions, designs, implicits, and explicits. Unique abbreviations are accepted.

`-unsaved`
Specifies that the count include only those virtuals that have not been saved. Optional.

See also

virtual define (CR-161), **virtual save** (CR-175), **virtual show** (CR-176), "Virtual Objects (User-defined buses, and more)" (UM-133)

virtual define

The **virtual define** command prints to the Main window the definition of the virtual signal or function in the form of a command that can be used to re-create the object.

Syntax

```
virtual define
  [-kind <kind>] <pathname>
```

Arguments

`-kind <kind>`
Specifies a subset of virtuals to look at. Optional. <*kind*> can be any of the following: signals, functions, designs, implicits, and explicits. Unique abbreviations are accepted.

`<pathname>`
Specifies the path to the virtual(s) for which you want definitions. Required. Wildcards can be used.

Examples

`virtual define -kind explicits *`
Shows the definitions of all the virtuals you have explicitly created.

See also

virtual describe (CR-163), **virtual show** (CR-176), "Virtual Objects (User-defined buses, and more)" (UM-133)

virtual delete

The **virtual delete** command removes the matching virtuals.

Syntax

```
virtual delete
  [-kind <kind>] <pathname>
```

Arguments

`-kind <kind>`
Specifies a subset of virtuals to look at. Optional. <*kind*> can be any of the following: signals, functions, designs, implicits, and explicits. Unique abbreviations are accepted.

`<pathname>`
Specifies the path to the virtual(s) you want to delete. Required. Wildcards can be used.

Examples

```
virtual delete -kind explicits *
```
Deletes all of the virtuals you have explicitly created.

See also

virtual signal (CR-177), **virtual function** (CR-165), "Virtual Objects (User-defined buses, and more)" (UM-133)

virtual describe

The **virtual describe** command prints to the Main window a complete description of the data type of one or more virtual signals. Similar to the existing **describe** command.

Syntax

```
virtual describe
  [-kind <kind>] <pathname>
```

Arguments

`-kind <kind>`
Specifies a subset of virtuals to look at. Optional. <*kind*> can be any of the following: signals, functions, designs, implicits, and explicits. Unique abbreviations are accepted.

`<pathname>`
Specifies the path to the virtual(s) for which you want descriptions. Required. Wildcards can be used.

Examples

`virtual describe -kind explicits *`
Describes the data type of all virtuals you have explicitly created.

See also

virtual define (CR-161), **virtual show** (CR-176), "Virtual Objects (User-defined buses, and more)" (UM-133)

virtual expand

The **virtual expand** command produces a list of all the non-virtual objects contained in the specified virtual signal(s). This can be used to create a list of arguments for a command that does not accept or understand virtual signals.

Syntax

```
virtual expand
  [-base] <pathname>
```

Arguments

`-base`
Causes the root signal parent to be output in place of a subelement. Optional. For example:

```
vcd add [virtual expand -base myVirtualSignal]
```

the resulting command after substitution would be:

```
vcd add signala signalb signalc
```

`<pathname>`
Specifies the path to the signals and virtual signals to expand. Required. Wildcards can be used. Any number of paths can be specified.

Examples

`vcd add [virtual expand myVirtualSignal]`
Adds 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. So if myVirtualSignal is a concatenation of signala, signalb.rec1 and signalc(5 downto 3), the resulting command after substitution would be:

```
vcd add signala signalb.rec1 {signalc(5 downto 3)}
```

The slice of signalc is quoted in curly braces, because it contains spaces.

See also

virtual signal (CR-177), "Virtual Objects (User-defined buses, and more)" (UM-133)

virtual function

The **virtual function** 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>**. It cannot handle bit selects and slices of Verilog registers. Please see "Syntax and conventions" (CR-5) 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 Signals 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.

Syntax

```
virtual function
  [-env <path>] [-install <path>] [-implicit] [-delay <time>]
  {<expressionString>} <name>
```

Arguments

Arguments for **virtual function** are the same as those for **virtual signal**, except for the contents of the expression string.

`-env <path>`
Specifies a hierarchical context for the signal names in **<expressionString>** so they don't all have to be full paths. Optional.

`-install <path>`
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. Optional.

`-implicit`
Used internally to create virtuals that are automatically saved with the List or Wave format. Optional.

`-delay <time>`
Specifies a value by which the virtual function will be delayed. Optional. You can use negative values to look forward in time. If units are specified, the <time> option must be enclosed in curly braces. See the examples below for more details.

`{<expressionString>}`
A text string expression in the MTI GUI expression format. Required. See "GUI_expression_format" (CR-15) for more information.

`<name>`
The name you define for the virtual signal. Required. 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

`virtual function { not /chip/section1/clk } clk_n`
Creates a signal */chip/section1/clk_n* that is the inverse of */chip/section1/clk*.

`virtual function -install /chip { (std_logic_vector) chip.vlog.rega } rega_slv`
Creates a std_logic_vector equivalent of a verilog register *rega* and installs it as */chip/rega_slv*.

`virtual function { /chip/addr[11:0] == 0xfab } addr_eq_fab`
Creates 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 { gate:/chip/siga XOR rtl:/chip/siga) } siga_diff`
Creates 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 -delay {10 ns} {/top/signalA AND /top/signalB} myDelayAandB`
Creates a virtual signal consisting of the logical "AND" function of */top/signalA* with */top/signalB*, and delays it by 10 ns.

`virtual function { | (gate:/chip/outbus XOR rtl:/chip/outbus) } outbus_diff`
Creates 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.

Commands fully compatible with virtual functions

add dataflow (CR-31)	add log /log (CR-87)	add wave (CR-35)
delete (CR-65)	describe (CR-66) ("virtual describe" is a little faster)	examine (CR-75)
find (CR-79)	restart (CR-111)	searchlog (CR-116)
show (CR-119)		

Commands not currently compatible with virtual functions

drivers (CR-69)	force (CR-82)	noforce (CR-92)
vcd add (CR-128)	when (CR-208)	

See also

virtual count (CR-160)	virtual define (CR-161)	virtual delete (CR-162)
virtual describe (CR-163)	virtual expand (CR-164)	virtual hide (CR-168)
virtual log (CR-169)	virtual nohide (CR-171)	virtual nolog (CR-172)
virtual region (CR-174)	virtual save (CR-175)	virtual show (CR-176)
virtual signal (CR-177)	virtual type (CR-180)	Virtual Objects (User-defined buses, and more) (UM-133)

virtual hide

The **virtual hide** command sets a flag in the specified real or virtual signals, so those signals do not appear in the Signals 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 <kind>]|[-region <path>] <pattern>
```

Arguments

`-kind <kind>`
Specifies a subset of virtuals to look at. Optional. <kind> can be any of the following: signals, functions, designs, implicits, and explicits. Unique abbreviations are accepted.

`-region <path>`
Used in place of -kind to specify a region of design space in which to look for the signal names. Optional.

`<pattern>`
Indicates which signal names or wildcard patterns should be used in finding the signals to hide. Required. Any number of names or wildcard patterns may be used.

See also

virtual nohide (CR-171), "Virtual Objects (User-defined buses, and more)" (UM-133)

virtual log

The **virtual log** 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 <kind>]|[-region <path>] [-recursive] [-only] [-in] [-out] [-inout]
  [-internal] [-ports] <pattern>
```

Arguments

`-kind <kind>`
Specifies a subset of virtuals to look at. Optional. <kind> can be any of the following: signals, functions, designs, implicits, and explicits. Unique abbreviations are accepted.

`-region <path>`
Used in place of -kind to specify a region of design space in which to look for signals to log. Optional.

`-recursive`
Specifies that the scope of the search is to descend recursively into subregions. Optional. If omitted, the search is limited to the selected region.

`-only`
Can be used with a wildcard to specify that only virtual signals (as opposed to all signals) found by the wildcard should be logged. Optional.

`-in`
Specifies that the kernal log data for ports of mode IN whose names match the specification. Optional.

`-out`
Specifies that the kernel log data for ports of mode OUT whose names match the specification. Optional.

`-inout`
Specifies that the kernel log data for ports of mode INOUT whose names match the specification. Optional.

`-internal`
Specifies that the kernel log data for internal items whose names match the specification. Optional.

`-ports`
Specifies that the kernel log data for all ports. Optional.

`<pattern>`
Indicates which signal names or wildcard patterns should be used in finding the signals to log. Required. Any number of names or wildcard patterns may be used.

See also

virtual nolog (CR-172), "Virtual Objects (User-defined buses, and more)" (UM-133)

virtual nohide

The **virtual nohide** command reverses the effect of a **virtual hide** command. It resets the flag in the specified real or virtual signals, so those signals reappear in the Signals window.

Syntax

```
virtual nohide
  [-kind <kind>]|[-region <path>] <pattern>
```

Arguments

`-kind <kind>`
Specifies a subset of virtuals to look at. Optional. <kind> can be any of the following: signals, functions, designs, implicits, and explicits. Unique abbreviations are accepted.

`-region <path>`
Used in place of -kind to specify a region of design space in which to look for the signal names. Optional.

`<pattern>`
Indicates which signal names or wildcard patterns should be used in finding the signals to expose. Required. Any number of names or wildcard patterns may be used.

See also

virtual hide (CR-168), "Virtual Objects (User-defined buses, and more)" (UM-133)

virtual nolog

The **virtual nolog** 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 <kind>]|[-region <path>] [-recursive] [-only] [-in] [-out] [-inout]
  [-internal] [-ports] <pattern>
```

Arguments

`-kind <kind>`
Specifies a subset of virtuals to look at. Optional. <kind> can be any of the following: signals, functions, designs, implicits, and explicits. Unique abbreviations are accepted.

`-region <path>`
Used in place of -kind to specify a region of design space in which to look for signals to unlog. Optional.

`-recursive`
Specifies that the scope of the search is to descend recursively into subregions. Optional. If omitted, the search is limited to the selected region.

`-only`
Can be used with a wildcard to specify that only virtual signals (as opposed to all signals) found by the wildcard should be unlogged. Optional.

`-in`
Specifies that the kernel exclude data for ports of mode IN whose names match the specification. Optional.

`-out`
Specifies that the kernel exclude data for ports of mode OUT whose names match the specification. Optional.

`-inout`
Specifies that the kernel exclude data for ports of mode INOUT whose names match the specification. Optional.

`-internal`
Specifies that the kernel exclude data for internal items whose names match the specification. Optional.

`-ports`
Specifies that the kernel exclude data for all ports. Optional.

`<pattern>`
Indicates which signal names or wildcard pattern should be used in finding the signals to unlog. Required. Any number of names or wildcard patterns may be used.

See also

virtual log (CR-169), "Virtual Objects (User-defined buses, and more)" (UM-133)

virtual region

The **virtual region** command creates a new user-defined design hierarchy region.

Syntax

```
virtual region
  <parentPath> <regionName>
```

Arguments

`<parentPath>`
The full path to the region that will become the parent of the new region. Required.

`<regionName>`
The name you want for the new region. Required.

See also

virtual function (CR-165), **virtual signal** (CR-177), "Virtual Objects (User-defined buses, and more)" (UM-133)

Note: Virtual regions cannot be used in the **when** (CR-208) command.

virtual save

The **virtual save** command saves the definitions of virtuals to a file.

Syntax

```
virtual save
  [-kind <kind>] [-append] [<filename>]
```

Arguments

`-kind <kind>`
Specifies a subset of virtuals to look at. Optional. <kind> can be any of the following: signals, functions, designs, implicits, and explicits. Unique abbreviations are accepted.

`-append`
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. Optional.

`<filename>`
Used for writing the virtual definitions. Optional. 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.

See also

virtual count (CR-160), "Virtual Objects (User-defined buses, and more)" (UM-133)

virtual show

The **virtual show** command lists the full path names of all explicitly defined virtuals.

Syntax

```
virtual show
  [-kind <kind>]
```

Arguments

`-kind <kind>`
Specifies a subset of virtuals to look at. Optional. <kind> can be any of the following: signals, functions, designs, implicits, and explicits. Unique abbreviations are accepted.

See also

virtual define (CR-161), **virtual describe** (CR-163), "Virtual Objects (User-defined buses, and more)" (UM-133)

virtual signal

The **virtual signal** 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>**. It cannot handle bit selects and slices of Verilog registers. Please see "Syntax and conventions" (CR-5) for more details on syntax.

Syntax

```
virtual signal
  [-env <path>] [-install <path>] [-implicit] [-delay <time>]
  {<expressionString>} <name>
```

Arguments

`-env <path>`
Specifies a hierarchical context for the signal names in **<expressionString>**, so they don't all have to be full paths. Optional.

`-install <path>`
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. Optional.

`-implicit`
Used internally to create virtuals that are automatically saved with the List or Wave format. Optional.

`-delay <time>`
Specifies a value by which the virtual signal will be delayed. Optional. You can use negative values to look forward in time. If units are specified, the <time> option must be enclosed in curly braces. See the examples below for more details.

`{<expressionString>}`
A text string expression in the MTI GUI expression format that defines the signal and subelement concatenation. Can also be a literal constant or computed subexpression. Required. For details on syntax, please see "Syntax and conventions" (CR-5).

`<name>`
The name you define for the virtual signal. Required. 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

```
virtual signal -env sim:/chip/alu { (concat_range (4 downto 0))(a_04 & a_03
& a_02 & a_01 & a_00) } a
```
Reconstructs a bus *sim:/chip/alu/a(4 downto 0)*, using VHDL notation, assuming that *a_ii* are scalars all of the same type.

```
virtual signal -env sim:chip.alu { (concat_range [4:0])&{a_04, a_03, a_02,
a_01, a_00} } a
```
Reconstructs a bus *sim:chip.alu.a[4:0]*, using Verilog notation. Note that the concatenation notation starts with "&{" rather than "{".

```
virtual signal -install sim:/testbench { /chipa/alu/a(19 downto 13) &
/chipa/decode/inst & /chipa/mode } stuff
```
Creates 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 -delay {10 ps} {/top/signalA} myDelayedSignalA
```
Creates a virtual signal that is the same as */top/signalA* except it is delayed by 10 ps.

```
virtual signal { chip.instruction[23:21] } address_mode
```
Creates a three-bit signal, *chip.address_mode*, as an alias to the specified bits.

```
virtual signal {a & b & c & 3'b000} myextendedbus
```
Concatenates signals *a*, *b*, and *c* with the literal constant '000'.

```
virtual signal {num & "000"} fullbus
add wave -unsigned fullbus
```
Adds three missing bits to the bus *num*, creates a virtual signal *fullbus*, and then adds that signal to the wave window.

```
virtual signal { num31 & num30 & num29 & ... & num4 & num3 & "000" } fullbus
add wave -unsigned fullbus
```
Reconstructs 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 (i.e. num28, num27, etc.) represented by the ... in the syntax above.

```
virtual signal {(aold == anew) & (bold == bnew)} myequalityvector
```
Creates 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 *c*, the second bit is false (0). Each subexpression is evaluated independently.

Commands fully compatible with virtual signals

add list (CR-32)	add log / log (CR-87)	add wave (CR-35)
delete (CR-65)	describe (CR-66) ("virtual describe" is a little faster)	examine (CR-75)
find (CR-79)	force (CR-82)/noforce (CR-92)	restart (CR-111)
searchlog (CR-116)	show (CR-119)	

Commands compatible with virtual signals using [virtual expand <signal>]

drivers (CR-69)	vcd add (CR-128)	

Commands not currently compatible with virtual signals

when (CR-208)

See also

virtual count (CR-160)	virtual define (CR-161)	virtual delete (CR-162)
virtual describe (CR-163)	virtual expand (CR-164)	virtual function (CR-165)
virtual hide (CR-168)	virtual log (CR-169)	virtual nohide (CR-171)
virtual nolog (CR-172)	virtual region (CR-174)	virtual save (CR-175)
virtual show (CR-176)	virtual type (CR-180)	Virtual Objects (User-defined buses, and more) (UM-133)

virtual type

The **virtual type** 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>
```

Arguments

`-delete <name>`
Deletes a previously defined virtual type. **<name>** is the name you gave the virtual type when you originally defined it. Optional.

`{<list_of_strings>}`
A list of values and their associated character strings. Required. Values can be expressed in decimal or based notation. 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.

There is currently no restriction on the contents of each string, but if strings contain spaces they would need to be quoted, and if they contain characters treated specially by Tcl (square brackets, curly braces, backslashes...), they would need to be quoted with curly braces.

See the examples below for further syntax.

`<name>`
The user-defined name of the virtual type. Required. 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

```
virtual type {state0 state1 state2 state3} mystateType
virtual function {(mystateType)mysignal} myConvertedSignal
add wave myConvertedSignal
```

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 Signals window, the string "state0" will appear when *mysignal* == 0, "state1" when *mysignal* == 1, "state2" when *mysignal* == 2, etc.

```
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
```

Uses 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 -delete mystateType
```

Deletes the virtual type "mystateType".

See also

virtual function (CR-165), "Virtual Objects (User-defined buses, and more)" (UM-133)

Note: Virtual types cannot be used in the **when** (CR-208) command.

vlib

The **vlib** command creates a design library. You must use **vlib** rather than operating system commands to create a library directory or index file. If the specified library already exists as a valid ModelSim library, the **vlib** command will exit with a warning message without touching the library.

Syntax

```
vlib
  [-archive [-compact <percent>]] [-help] [-dos | -short | -unix | -long]
  <name>
```

Arguments

`-archive [-compact <percent>]`
Causes design units that are compiled into the created library to be stored in archives rather than in subdirectories. Optional. See "Archives" (UM-39) for more details.

You may optionally specify a decimal number between 0 and 1 that denotes the allowed percentage of wasted space before archives are compacted. By default archives are compacted when 50% (.5) of their space is wasted. See an example below.

`-help`
Displays the command's options and arguments. Optional.

`-dos`
Specifies that subdirectories in a library have names that are compatible with DOS. Not recommended if you use the **vmake** (CR-189) utility. Optional.

`-short`
Interchangeable with the **-dos** argument. Optional.

`-unix`
Specifies that subdirectories in a library may have long file names that are NOT compatible with DOS. Optional. Default for ModelSim SE.

`-long`
Interchangeable with the **-unix** argument. Optional.

`<name>`
Specifies the pathname or archive name of the library to be created. Required.

Examples

`vlib design`
Creates the design library *design*. You can define a logical name for the library using the **vmap** command (CR-191) or by adding a line to the library section of the *modelsim.ini* file that is located in the same directory.

`vlib -archive -compact .3 uut`
Creates the design library *uut* and specifies that any design units compiled in to the library are created as archives. Also specifies that each archive be compacted when 30% of the its space is wasted.

vlog

The **vlog** command is used to invoke VLOG, the Model Technology Verilog compiler. Use **vlog** to compile Verilog source code into a specified working library (or to the **work** library by default).

vlog may be invoked from within ModelSim or from the operating system command prompt. It may also be invoked during simulation.

Compiled libraries are version dependent. For example you cannot use a library compiled with 5.5 in a simulation using 5.6 **vsim**.

Syntax

```
vlog
  [-93] [-help] [-compat] [-compile_uselibs[=<directory_name>]]
  [+define+<macro_name>[=<macro_text>]] [+delay_mode_distributed]
  [+delay_mode_path] [+delay_mode_unit] [+delay_mode_zero] [-f <filename>]
  [-hazards] [+incdir+<directory>] [-incr] [+libext+<suffix>] [+librescan]
  [-line <number>] [-lint] [+maxdelays] [+mindelays] [-noincr] [+nolibcell]
  [-nologo] [+nospecify] [+notimingchecks] [+nowarn<CODE>] [-O0] [-quiet] [-
  R [<simargs>]] [-refresh] [-source] [-time] [+typdelays] [-u] [-v
  <library_file>] [-version] [-vlog95compat] [-work <library_name>]
  [-y <library_directory>] <filename>
```

Arguments

`-93`
Specifies that the VHDL interface to Verilog modules use VHDL 1076-1993 extended identifiers to preserve case in Verilog identifiers that contain uppercase letters.

`-help`
Displays the command's options and arguments. Optional.

`-compat`
Disables optimizations that result in different event ordering than Verilog-XL. Optional.

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. See "Event ordering in Verilog designs" (UM-80) for additional information.

`-compile_uselibs[=<directory_name>]`
Locates source files specified in a **'uselib** directive (see "Verilog-XL `uselib compiler directive" (UM-75)), compiles those files into automatically created libraries, and updates the *modelsim.ini* file with the logical mappings to the new libraries. Optional. 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.

`+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>
```

Optional. Multiple **+define** options are allowed on the command line. A command line macro overrides a macro of the same name defined with the **‘define** compiler directive.

`+delay_mode_distributed`
Disables path delays in favor of distributed delays. Optional. See "Delay modes" (UM-88) for details.

`+delay_mode_path`
Sets distributed delays to zero in favor of using path delays. Optional. See "Delay modes" (UM-88) for details.

`+delay_mode_unit`
Sets path delays to zero and non-zero distributed delays to one time unit. Optional. See "Delay modes" (UM-88) for details.

`+delay_mode_zero`
Sets path delays and distributed delays to zero. Optional. See "Delay modes" (UM-88) for details.

`-f <filename>`
Specifies a file with more command line arguments. Optional. Allows complex arguments to be reused without retyping. Nesting of **-f** options is allowed. Environment variable expansion (for example in a pathname) *does not* occur in **-f** files.

`-hazards`
Detects event order hazards involving simultaneous reading and writing of the same register in concurrently executing processes. Optional. You must also specify this argument when you simulate the design with **vsim** (CR-192). See "Hazard detection" (UM-83) for more details.

Important: Enabling **-hazards** implicitly enables the **-compat** argument. As a result, using this argument may affect your simulation results.

`+incdir+<directory>`
Specifies directories to search for files included with **‘include** compiler directives. Optional. By default, the current directory is searched first and then the directories specified by the **+incdir** options in the order they appear on the command line. You may specify multiple **+incdir** options as well as multiple directories separated by "+" in a single **+incdir** option.

`-incr`
Performs an incremental compile. Optional. 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 if you use **-incr** or not. May be used with **-fast**.

`+libext+<suffix>`
Works in conjunction with the **-y** option. Specifies file extensions for the files in a source library directory. Optional. By default the compiler searches for files without extensions. If you specify the **+libext** option, 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 extensions are tried in the order they appear in the **+libext** option.

`+librescan`
Scans libraries in command-line order for all unresolved modules. Optional.

`-line <number>`
Starts the compiler on the specified line in the Verilog source file. Optional. By default, the compiler starts at the beginning of the file.

`-lint`
Instructs ModelSim to perform three lint-style checks: 1) warn when Module ports are NULL; 2) warn when assigning to an input port; 3) warn when referencing undeclared variables/nets in an instantiation. The warnings are reported as WARNING[8]. Can also be enabled using the Show_Lint variable in the *modelsim.ini* file.

`+maxdelays`
Selects maximum delays from the "min:typ:max" expressions. Optional. If preferred, you can defer delay selection until simulation time by specifying the same option to the simulator.

`+mindelays`
Selects minimum delays from the "min:typ:max" expressions. Optional. If preferred, you can defer delay selection until simulation time by specifying the same option to the simulator.

`-noincr`
Disables incremental compile previously turned on with **-incr**. Optional.

`+nolibcell`
By default all modules compiled from a source library are treated as though they contain a **‘celldefine** compiler directive. This option disables this default. The **‘celldefine** directive only affects the PLI access routines **acc_next_cell** and **acc_next_cell_load**. Optional.

`-nologo`
Disables the startup banner. Optional.

`+nospecify`
Disables specify path delays and timing checks. Optional.

`+notimingchecks`
Removes all timing check entries from the design as it is parsed. Optional.

`+nowarn<CODE>`
Disables warning messages in the category specified by <CODE>. Optional. 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>.
```

This warning message can be disabled with the **+nowarnTFMPC** option.

`-O0`
Lower the optimization to a minimum with **-O0** (capital oh zero). Optional. 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.

`-quiet`
Disables ’loading’ messages. Optional.

`-R [<simargs>]`
Instructs the compiler to invoke the simulator (**vsim** (CR-192)) after compiling the design. The compiler automatically determines which top-level modules are to be simulated. The command line arguments following **-R** are passed to the simulator, not the compiler. Place the **-R** option at the end of the command line or terminate the simulator command line arguments with a single "-" character to differentiate them from compiler command line arguments.

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`
Regenerates a library image. Optional. By default, the work library is updated; use **-work <library_name>** to update a different library. See **vlog** examples for more information.

`-source`
Displays the associated line of source code before each error message that is generated during compilation. Optional; by default, only the error message is displayed.

`-time`
Reports the "wall clock time" **vlog** takes to compile the design. Optional. Note that if many processes are running on the same system, wall clock time may differ greatly from the actual "cpu time" spent on **vlog**.

`+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`
Converts regular Verilog identifiers to uppercase. Allows case insensitivity for module names. Optional.

`-v <library_file>`
Specifies a source library file containing module and UDP definitions. Optional. See "Verilog-XL compatible compiler arguments" (UM-74) 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`
Returns the version of the compiler as used by the licensing tools, such as "Model Technology ModelSim SE vlog 5.5 Compiler 2000.01 Jan 28 2000".

`-vlog95compat`
Some requirements in Verilog 2001 conflict with requirements in the 1995 LRM. Use of this argument ensures that code that was valid according to the 1995 LRM can still be compiled. Optional. Edit the vlog95compat (UM-351) variable in the *modelsim.ini* file to set a permanent default.

`-work <library_name>`
Specifies a logical name or pathname of a library that is to be mapped to the logical library **work**. Optional; 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.

`-y <library_directory>`
Specifies a source library directory containing module and UDP definitions. Optional. See "Verilog-XL compatible compiler arguments" (UM-74) 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 if your filenames differ from your module names. See additional discussion in the examples.

Important: 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

`vlog example.vlg`
Compiles the Verilog source code contained in the file *example.vlg*.

`vlog -L work -L libA -L libB top.v`
This command demonstrates how to compile hierarchical modules organized into separate libraries that have sub-module names that overlap among the libraries. Assume you have a top-level module *top* that instantiates module *modA* from library *libA* and module *modB* from library *libB*. Furthermore, *modA* and *modB* both instantiate modules named *cellA*, but the definition of *cellA* compiled into *libA* is different from that compiled into *libB*. In this case, you can't just specify **-L libA - L libB** because instantiations of *cellA* from *modB* resolve to the *libA* version of *cellA*. See "Library usage" (UM-72) for further information.

`vlog top.v -v und1`
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 +libext+.v+.u -y vlog_lib`
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 -work mylib -refresh
```

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).

If your library contains VHDL design units be sure to regenerate the library with the **vcom** command (CR-147) using the **-refresh** option as well. See "Regenerating your design libraries" (UM-47) for more information.

```
vlog module1.v -u -O0 -incr
```

The **-incr** option determines whether or not the module source or compile options have changed as module1 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.

vmake

The **vmake** utility allows you to use a Windows MAKE program to maintain libraries. You run **vmake** on a compiled design library, and the utility outputs a makefile. You can then run the makefile with a version of MAKE (not supplied with ModelSim) to reconstruct the library. 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 must be invoked from the Windows/DOS prompt.

Syntax

```
vmake
  [-fullsrcpath] [-help] [<library_name>] [><makefile>]
```

Arguments

`-fullsrcpath`
Produces complete source file paths within generated makefiles. Optional. By default source file paths are relative to the directory in which compiles originally occurred. This argument makes it possible to copy and evaluate generated makefiles within directories that are different from where compiles originally occurred.

`-help`
Displays the command's options and arguments. Optional.

`<library_name>`
Specifies the library name; if none is specified, then **work** is assumed. Optional.

`><makefile>`
Specifies the makefile name. Optional.

Examples

Here is an example of how to use **vmake** and MAKE on your **work** library:

```
C:\MIXEDHDL> vmake >makefile
```

Edit an HDL source file within the work library then enter:

```
C:\MIXEDHDL> make
```

Your design gets recompiled for you. You can change the design again and re-run MAKE to recompile additional changes.

You can also run **vmake** on libraries other than **work**. For example,

```
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. With no arguments, **vmap** reads the appropriate *modelsim.ini* file(s) and prints the current logical library to physical directory mappings. Returns nothing.

Syntax

```
vmap
  [-help] [-c] [-del] [<logical_name>] [<path>]
```

Arguments

`-help`
Displays the command's options and arguments. Optional.

`-c`
Copies the default *modelsim.ini* file from the ModelSim installation directory to the current directory. Optional.

Note: 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`
Deletes the mapping specified by <logical_name> from the current project file. Optional.

`<logical_name>`
Specifies the logical name of the library to be mapped. Optional.

`<path>`
Specifies the pathname of the directory to which the library is to be mapped. Optional. If omitted, the command displays the mapping of the specified logical name.

vsim

The **vsim** command is used to invoke the VSIM simulator, or to view the results of a previous simulation run (when invoked with the **-view** switch). You can specify a configuration, an entity/architecture pair, or a module for simulation. If a configuration is specified, it is invalid to specify an architecture. With no options, **vsim** brings up the Load Design dialog box, allowing you to specify the design and options; the Load Design dialog box will not be presented if you specify any options. During elaboration **vsim** determines if the source has been modified since the last compile.

This command may be used in batch mode from the Windows command prompt. See "Tips and techniques" (UM-387) for more information on the VSIM batch mode.

To **manually interrupt design elaboration** use the Break key.

The **vsim** command may also be invoked from the command line within ModelSim with most of the options shown below (all except the **vsim -c** and **-restore** options).

Syntax

```
vsim
  [-assertfile <filename>] [-c] [-do “<command_string>” | <macro_file_name>]
  [+dumpports+direction] [+dumpports+unique] [-f <filename>] [-
  g<Name>=<Value> ...] [-G<Name>=<Value> ...] [-gui]
  [-help] [-i] [-keeploaded] [-keeploadedrestart]
  [-keepstdout] [-l <filename>] [-multisource_delay min | max |
  latest][+multisource_int_delays]
  [+no_notifier][+no_tchk_msg] [+notimingchecks] [-quiet]

  [-sdfmin | -sdftyp | -sdfmax[@<delayScale>] [<instance>=]<sdf_filename>]
  [-sdfmaxerrors <n>] [-sdfnoerror] [-sdfnowarn] [+sdf_verbose]
  [-t [<multiplier>]<time_unit>]
  [-tag <string>] [-title <title>][-trace_foreign <int>]
  [-vcdstim <filename>] [-version] [-view [<dataset_name>=]<WLF_filename>]
  [-wlf <filename>] [-wlfcompress] [-wlfnocompress] [-wlfslim <size>]
  [-wlftlim <duration>]

  [-absentisempty]  [-nocollapse] [-nofileshare]
  [-noglitch] [+no_glitch_msg] [-std_input <filename>]
  [-std_output <filename>] [-strictvital] [-vital2.2b]

  [+alt_path_delays] [-extend_tcheck_data_limit <percent>]
  [-extend_tcheck_ref_limit <percent>]
  [-hazards] [+int_delays] [-L <library_name> ...] [-Lf <library_name> ...]
  [+maxdelays] [+mindelays] [+no_cancelled_e_msg] [+no_neg_tchk]
  [+no_notifier] [+no_path_edge] [+no_pulse_msg] [+no_show_cancelled_e]
  [+no_tchk_msg] [+nosdferror] [+nosdfwarn] [+nospecify] [+nowarn<CODE>]
  [+ntc_warn] [-pli "<object list>"][+<plusarg>]
  [+pulse_e/<percent>] [+pulse_e_style_ondetect] [+pulse_e_style_onevent]
  [+pulse_int_e/<percent>] [+pulse_int_r/<percent>] [+pulse_r/<percent>]
  [+sdf_nocheck_celltype] [+show_cancelled_e] [+transport_int_delays]
  [+transport_path_delays] [+typdelays]
  [-v2k_int_delays]

  [<library_name>.<design_unit>]
```

VSIM arguments are grouped alphabetically by language:

Arguments, VHDL and Verilog

`-assertfile <filename>`
Designates an alternative file for recording assertion messages. Optional. By default assertion messages are output to the file specified by the TranscriptFile variable in the *modelsim.ini* file (see "Creating a transcript file" (UM-357)).

`-c`
Specifies that the simulator is to be run in command line mode. Optional. Also see for more information.

`-do "<command_string>" | <macro_file_name>`
Instructs VSIM to use the command(s) specified by **<command_string>** or the macro file named by **<macro_file_name>** rather than the startup file specified in the *.ini* file, if any. Optional. Multiple commands should be separated by semi-colons (;).

`+dumpports+direction`
Modifies the format of extended VCD files to contain direction information. Optional.

`+dumpports+unique`
Generates unique vcd variable names for ports in a VCD file, even if those ports are connected to the same collapsed net. Optional.

`-f <filename>`
Specifies a file with more command line arguments. Optional. Allows complex argument strings to be reused without retyping. Environment variable expansion (for example in a pathname) *does not* occur in **-f** files.

`-g<Name>=<Value> ...`
Assigns a value to all specified VHDL generics and Verilog parameters that have not received explicit values in generic maps, instantiations, or via defparams (such as top-level generics/parameters and generics/parameters that would otherwise receive their default values). Optional. Note there is no space between **-g** and **<Name>=<Value>**.

Name is the name of the generic/parameter, exactly as it appears in the VHDL source (case is ignored). **Value** is 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. VHDL type mismatches will cause the specification to be ignored (including no error messages).

No spaces are allowed anywhere in the specification, except within quotes when specifying a string value. Multiple **-g** options are allowed, one for each generic/parameter.

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*.

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, put a forward tick 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"}
```

Note: When you compile Verilog code with **-fast** (see vlog (CR-183)), all parameter values are set at compile time. Therefore, the **-g** option has no effect on these parameters.

`-G<Name>=<Value> ...`
Same as **-g** (see above) except that it will also override generics/parameters that received explicit values in generic maps, instantiations, or via defparams. Optional. Note there is no space between **-G** and **<Name>=<Value>**.

`-gui`
Starts the ModelSim GUI without loading a design. Optional.

`-help`
Displays the command's options and arguments. Optional.

`-i`
Specifies that the simulator is to be run in interactive mode. Optional.

`-keeploaded`
Prevents the simulator from unloading/reloading any FLI/PLI/VPI shared libraries when it restarts or loads a new design. Optional. 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`
Prevents the simulator from unloading/reloading any FLI/PLI/VPI shared libraries during a restart. Optional. 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`
For use with foreign programs. Instructs the simulator to not redirect the stdout stream to the Main window. Optional.

`-l <filename>`
Saves the contents of the "Main window" (UM-145) transcript to <filename>. Optional. Default is *transcript*. Can also be specified using the *.ini* (see "Creating a transcript file" (UM-357)) file or the*.tcl* preference file.

`-multisource_delay min | max | latest`
Controls the handling of multiple PORT or INTERCONNECT constructs that terminate at the same port. Optional. 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`
Enables multisource interconnect delay with pulse handling and transport delay behavior. Optional. 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** switches (described below).

`+no_notifier`
Disables the toggling of the notifier register argument of all timing check system tasks. Optional. 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.

`+no_tchk_msg`
Disables error messages issued by timing check system tasks when timing check violations occur. Optional. Notifier registers are still toggled and may result in the propagation of Xs for timing check violations..

`+notimingchecks`
Disables Verilog and VITAL timing checks for faster simulation. Optional. 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.

`-quiet`
Disable 'loading' messages during batch-mode simulation. Optional.

`-sdfmin | -sdftyp | -sdfmax[@<delayScale>] [<instance>=]<sdf_filename>`
Annotates VITAL or Verilog cells in the specified SDF file (a Standard Delay Format file) with minimum, typical, or maximum timing. Optional.

The optional argument @<delayScale> scales all values by the specified value. For example, if you specify -sdfmax@1.5..., all maximum values in the SDF file will be scaled to 150% of their original value.

The use of [<instance>=] with <sdf_filename> is also optional; it is used when the backannotation is not being done at the top level. See "Specifying SDF files for simulation" (UM-298).

`-sdfmaxerrors <n>`
Controls the number of Verilog SDF missing instance messages that will be emitted before terminating vsim. Optional. <n> is the maximum number of missing instance error messages to be emitted. The default number is 5.

`-sdfnoerror`
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. Optional.

`-sdfnowarn`
Disables warnings from the SDF reader. Optional. See *Chapter 4 - VHDL simulation* for an additional discussion of SDF.

`+sdf_verbose`
Turns on the verbose mode during SDF annotation. The Main window provides detailed warnings and summaries of the current annotation. Optional.

`-t [<multiplier>]<time_unit>`
Specifies the simulator time resolution. Optional. <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 (i.e., 10fs, not 10 fs).

If you omit the **-t** argument, the default time resolution depends on design type: in a Verilog design with **'timescale** directives, the minimum time precision is used (see "Simulator resolution limit" (UM-78) for further details); in Verilog designs without *any* timescale directives, or in a VHDL or mixed design, the value specified for the Resolution (UM-355) variable in the *modelsim.ini* file is used.

Once you've begun simulation, you can determine the current simulator resolution by invoking the **report** command (CR-109) with the **simulator state** option.

`-tag <string>`
Specifies a string tag to append to foreign trace filenames. Optional. Used with the **-trace_foreign <int>** option. Used when running multiple traces in the same directory.

`-title <title>`
Specifies the title to appear for the ModelSim Main window. Optional. If omitted the current ModelSim version is the window title. Useful when running multiple simultaneous simulations. Text strings with spaces must be in quotes (e.g., "my title").

`-trace_foreign <int>`
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 thePLI/VPI code.

`-vcdstim <filename>`
Resimulates a design from a VCD file. Optional. The VCD file must have been created in a previous ModelSim simulation using the **vcd dumpports** command (CR-131). See "Resimulating a design from a VCD file" (UM-315) for more information.

`-version`
Returns the version of the simulator as used by the licensing tools, such as "Model Technology ModelSim SE vsim 5.5 Simulator 2000.01 Jan 28 2000".

`-view [<dataset_name>=]<WLF_filename>`
Specifies a wave log format (WLF) file for **vsim** to read. Allows you to use VSIM to view the results from an earlier simulation. The Structure, Signals, 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). See additional discussion in "Examples" (CR-205).

`-wlf <filename>`
Specifies the name of the wave log format (WLF) file to create. The default is *vsim.wlf*. Optional.

`-wlfcompress`
Creates compressed WLF files. Default. Use **-wlfnocompress** to turn off compression.

`-wlfnocompress`
Causes VSIM to create uncompressed WLF files. Optional. Beginning with version 5.5, 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 (UM-356) variable in the *modelsim.ini* file.

`-wlfslim <size>`
Specifies a size restriction in megabytes for the event portion of the WLF file. Optional. The default is infinite size (0). The **<size>** must be an integer.

Note that a WLF file contains event, header, and symbol portions. The size restriction is placed on the event portion only. When ModelSim exits, the entire header and symbol portion of the WLF file is written. Consequently, the resulting file will be larger than the size specified with **-wlfslim**.

If used in conjunction with **-wlftlim**, the more restrictive of the limits will take effect.

This option may also be specified with the WLFSizeLimit (UM-356) variable in the *modelsim.ini* file.

`-wlftlim <duration>`
Specifies the duration of simulation time for WLF file recording. Optional. 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 nanoseconds regardless of the current simulator resolution.

The time range begins at current simulation time and moves back in simulation time for the specified duration. For example,

```
vsim -wlftlim 5000
```

writes at least 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 (UM-356) variable in the *modelsim.ini* file.

Note: The **-wlfslim** and **-wlftlim** switches were designed to help users limit WLF file sizes for long or heavily logged simulations. When small values are used for these switches, the values may be overridden by the internal granularity limits of the WLF file format.

Arguments, VHDL

`-absentisempty`
Causes VHDL files opened for read that target non-existent files to be treated as empty, rather than ModelSim issuing fatal error messages. Optional.

`-nocollapse`
Disables the optimization of internal port map connections. Optional.

`-nofileshare`
By default ModelSim shares a file descriptor for all VHDL files opened for write or append that have identical names. The -nofileshare switch turns off file descriptor sharing. Optional.

`-noglitch`
Disables VITAL glitch generation. Optional.

See *Chapter 4 - VHDL simulation* for additional discussion of VITAL.

`+no_glitch_msg`
Disable VITAL glitch error messages. Optional.

`-std_input <filename>`
Specifies the file to use for the VHDL TextIO STD_INPUT file. Optional.

`-std_output <filename>`
Specifies the file to use for the VHDL TextIO STD_OUTPUT file. Optional.

`-strictvital`
Exactly match the VITAL package ordering for messages and delta cycles. Optional. Useful for eliminating delta cycle differences caused by optimizations not addressed in the VITAL LRM. Using this argument negatively impacts simulator performance.

`-vital2.2b`
Selects SDF mapping for VITAL 2.2b (default is VITAL 2000). Optional.

Arguments, Verilog

`+alt_path_delays`
Configures path delays to operate in inertial mode by default. Optional. In inertial mode, a pending output transition is cancelled 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 cancelled 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 cancelled pending value of the net. This option has no effect in transport mode (see **+pulse_e/<percent>** and **+pulse_r/<percent>**).

`-extend_tcheck_data_limit <percent>`

`-extend_tcheck_ref_limit <percent>`
Causes a one-time extension of qualifying data or reference limits in an attempt to provide a delay net delay solution prior to any limit zeroing. A limit qualifies if it bounds a violation region which does not overlap a related violation region.

`<percent>` is the maximum percent of limit relaxation. See "Extending check limits without zeroing" (UM-85) for an example of how to calculate the percentage.

`-hazards`
Enables event order hazard checking in Verilog modules. Optional. You must also specify this argument when you compile your design with **vlog** (CR-183). See "Hazard detection" (UM-83) for more details.

Important: Enabling **-hazards** implicitly enables the **-compat** argument. As a result, using this argument may affect your simulation results.

`+int_delays`
Optimizes annotation of interconnect delays for designs that have been compiled using **-fast** (see **vlog** command (CR-183)). Optional. This argument causes **vsim** to insert "placeholder" delay elements at optimized cell inputs, resulting in faster backannotation of interconnect delay from an SDF file.

`-L <library_name> ...`
Specifies the library to search for design units instantiated from Verilog. See "Library usage" (UM-72) 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.

`-Lf <library_name> ...`
Same as **-L** but libraries are searched before ‘uselib directives. See "Library usage" (UM-72) for more information. Optional.

`+maxdelays`
Selects the maximum value in min:typ:max expressions. Optional. The default is the typical value. Has no effect if you specified the min:typ:max selection at compile time.

`+mindelays`
Selects the minimum value in min:typ:max expressions. Optional. The default is the typical value. Has no effect if you specified the min:typ:max selection at compile time.

`+no_cancelled_e_msg`
Disables negative pulse warning messages. Optional. 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`
Disables negative timing check limits by setting them to zero. Optional. 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`
Disables the toggling of the notifier register argument of all timing check system tasks. Optional. 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`
Causes ModelSim to ignore the input edge specified in a path delay. Optional. 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`
Disables the warning message for specify path pulse errors. Optional. 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_show_cancelled_e`
Filters negative pulses on specify path delays so they don't show on the output. Default. Use **+show_cancelled_e** to drive a pulse error state.

`+no_tchk_msg`
Disables error messages issued by timing check system tasks when timing check violations occur. Optional. Notifier registers are still toggled and may result in the propagation of Xs for timing check violations.

`+nosdferror`
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. Optional.

`+nosdfwarn`
Disables warnings from the SDF annotator. Optional.

`+nospecify`
Disables specify path delays and timing checks. Optional.

`+nowarn<CODE>`
Disables warning messages in the category specified by <CODE>. Optional. 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>.
```

This warning message can be disabled with **+nowarnTFMPC**.

`+ntc_warn`
Enables warning messages from the negative timing constraint algorithm. Optional. 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.

`-pli "<object list>"`
Loads a space-separated list of PLI shared objects. Optional. 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, see "Preference variables located in INI files" (UM-349). You can use environment variables as part of the path.

`+<plusarg>`
Arguments preceded with "+" are accessible by the Verilog PLI routine **mc_scan_plusargs()**. Optional.

`+pulse_e/<percent>`
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. Optional.

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`
Selects the "on detect" style of propagating pulse errors (see **+pulse_e**). Optional. 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`
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_int_e/<percent>`
Analogous to **+pulse_e**, except it applies to interconnect delays only. Optional. Used in conjunction with **+multisource_int_delays** (see above).

`+pulse_int_r/<percent>`
Analogous to **+pulse_r**, except it applies to interconnect delays only. Optional. Used in conjunction with **+multisource_int_delays** (see above).

`+pulse_r/<percent>`
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. Optional.

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`
Disables error check 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. Optional.

`+show_cancelled_e`
Drives a pulse error state ('X') for the duration of a negative pulse on a specify path delay. Optional. By default ModelSim filters negative pulses.

`+transport_int_delays`
Selects transport mode with pulse control for single-source nets (one interconnect path). Optional. 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 independently from **+multisource_int_delays**.

`+transport_path_delays`
Selects transport mode for path delays. Optional. 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.

`+typdelays`
Selects the typical value in min:typ:max expressions. Default. Has no effect if you specified the min:typ:max selection at compile time.

`-v2k_int_delays`
Causes interconnect delay to be visible at the load module port per the IEEE 1364-2001 spec. Optional. By default ModelSim annotates INTERCONNECT delays in a manner compatible with Verilog-XL. If you have $sdf_annotate() calls in your design that are not getting executed, add the Verilog task $sdf_done() after your last $sdf_annotate() to remove any zero-delay MIPDs that may have been created (see "ModelSim Verilog system tasks" (UM-95) for more information). May be used in tandem with **+multisource_int_delays** argument (see above).

Arguments, object

The object arguments may be a <library_name>.<design_unit>, *.mpf* file, *.wlf* file, or a text file. If no object specification is made, VSIM will open the Load a Design dialog box. Multiple design units may be specified for Verilog modules and mixed VHDL/Verilog configurations.

`<library_name>.<design_unit>`
Specifies a library and associated design unit; multiple library/design unit specifications can be made. Optional. If no library is specified, the **work** library is used. Environment variables can be used. <design_unit> may be one of the following:

`<configuration>`	Specifies the VHDL configuration to simulate.
`<module> ...`	Specifies the name of one or more top-level Verilog modules to be simulated. Optional.
`<entity> [<architecture>]`	Specifies the name of the top-level VHDL entity to be simulated. Optional. 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.

`<MPF_file_name>`
Opens the specified project. Optional.

`<WLF_file_name>`
Opens the specified dataset. Optional.

`<text_file_name>`
Opens the specified text file in a Source window. Optional.

Examples

```
vsim -gedge=’"low high"’ -gVCC=4.75 cpu
```

Invokes **vsim** on the entity **cpu** and assigns values to the generic parameters **edge** and **VCC**. If working within the ModelSim GUI, you would enter the command as follows:

```
vsim {-gedge="low high"} -gVCC=4.75 cpu
```

```
vsim -view test=sim2.wlf
```

Instructs 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** option to specify the name of the WLF file to create if you plan to create many files for later viewing. For example:

```
vsim -wlf my_design.i01 my_asic structure
vsim -wlf my_design.i02 my_asic structure
```

```
vsim -sdfmin /top/u1=myasic.sdf
```

Annotates instance */top/u1* using the minimum timing from the SDF file *myasic.sdf*.

Use multiple switches to annotate multiple instances:

```
vsim -sdfmin /top/u1=sdf1 -sdfmin /top/u2=sdf2 top
```

```
vsim ’mylib.top(only)’ gatelib.cache_set
```

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 -do "set PrefMain(forceQuit) 1; run -all" work.test_counter
```

Invokes **vsim** on *test_counter* and instructs the simulator to run until a break event and quit when it encounters a **$finish** task.

vsim<info>

The **vsim<info>** commands return information about the current vsim executable.

`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 5.4".

`vsimVersion`
Returns the version as used by the licensing tools, such as "1999.04".

`vsimVersionString`
Returns the full vsim version string.

This same information can be obtained using the **-version** argument of the **vsim** command (CR-192).

vsource

The **vsource** 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

```
vsource
  [<filename>]
```

Arguments

`<filename>`
Specifies a relative or full pathname. Optional. If filename is omitted the source file for the current design context is displayed.

Examples

```
vsource design.vhd
vsource /old/design.vhd
```

when

The **when** command instructs ModelSim to perform actions when the specified conditions are met. For example, you can use the **when** command to break on a signal value or at a specific simulator time (see "Time-based breakpoints" (CR-212)). Conditions can include the following HDL items: VHDL signals, and Verilog nets and registers. Use the **nowhen** command (CR-97) to deactivate **when** commands.

The **when** command uses a **when_condition_expression** to determine whether or not to perform the action. 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 item names, signame'event, or constants. ModelSim evaluates the condition every time any item in the condition changes, hence the restrictions.

With no arguments, **when** will list the currently active when statements and their labels (explicit or implicit).

Note: Virtual signals, functions, regions, types, etc. cannot be used in the **when** command. Neither can simulator state variables other than $now.

Syntax

```
when
  [[-label <label>] [-id <id#>] {<when_condition_expression>} {<command>}]
```

Arguments

`-label <label>`
Used to identify individual **when** commands. Optional.

`-id <id#>`
Attempts to assign this id number to the when command. Optional. If the id number you specify is already used, ModelSim will return an error.

Note: Ids for when commands are assigned from the same pool as those used for the **bp** command (CR-46). So, even if you haven't used an id number for a when command, it's possible it is used for a breakpoint.

`{<when_condition_expression>}`
Specifies the conditions to be met for the specified **<command>** to be executed. Required. The condition is evaluated in the simulator kernal and can be an item name, in which case the curly braces can be omitted. The command will be executed when the item changes value. The condition can be an expression with these operators:

Name	Operator
equals	==, =
not equal	!=, /=
greater than	>

Name	Operator
less than	<
greater than or equal	>=
less than or equal	<=
AND	&&, AND
OR	||, OR

The operands may be item 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 ">", "<", ">=", and "<=" 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 items 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>}`
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 (CR-114) cannot be used with the **when** command. Required. The command sequence usually contains a **stop** command (CR-123) 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 (CR-123) within your when statement. DO NOT use an **exit** command (CR-78) or a **quit** command. The **stop** command acts like a breakpoint at the time it is evaluated. See "Ending the simulation with the stop command" (CR-211) for examples.

Examples

The **when** command below instructs the simulator to display the value of item *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 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 clkb_path /tb/ps/dprb_0/udprb/ucar_reg/uint_ram/clkb;

when -label when1 "$clkb_path'event and  $clkb_path ='1'" {
     echo "Detected Clk edge at path $clkb_path"
}
```

This next example uses the Tcl **set** command (UM-329) to disable arithmetic package warnings at time 0. Note that the time unit (ns in this case) would vary depending on your simulation resolution.

```
when {$now = @1ns } {set NumericStdNoWarnings 1}
run -all
```

The **when** command below is labeled *a* and will cause ModelSim to echo the message "b changed" whenever the value of the item *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, an **echo** (CR-71) and a **stop** (CR-123) command will be executed.

```
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}
```

If you encounter a vectored net caused by compiling with **-fast,** use the 'event qualifier to prevent the command from falsely evaluating when unrelated bits of 'a' change:

```
when {a(3:1) = 3'h7 and a(3:1)'event} {echo "matched at time" $now}
```

The first when command below sets up a trigger for the falling edge of RESET. When this happens, a second when command is executed which sets up a trigger to occur 200us after the current time.

```
force SIGA 1
when {RESET'falling} {
    when {$now == 200us} {
        noforce SIGA
    }
}
run -all
```

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 below sets a done condition and executes an **echo** (CR-71) and a **stop** (CR-123) command when the condition is reached.

The simulation will not stop (even if a **quit -f** command is used) unless a **stop** command is executed. 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
}
```

Here's another example that 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 = @1750ns} {stop}
```

You can also use:

```
when {errorFlag = '1' OR $now = 2ms} {stop}
```

This example adds 2ms to the simulation time at which the **when** statement is first evaluated, then stops.

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
```

See also

bp (CR-46), **disablebp** (CR-67), **enablebp** (CR-73), **nowhen** (CR-97)

where

The **where** command displays information about the system environment. This command is useful for debugging problems where ModelSim cannot find the required libraries or support files.

Syntax

```
where
```

Arguments

None.

Description

The **where** command displays two system settings:

`current directory`
This is the current directory that ModelSim was invoked from, or was specified on the ModelSim command line.

`current project file`
This is the initialization file ModelSim is using. All library mappings are taken from here. Window positions, and other parameters are taken from the *modelsim.tcl* file.

wlf2log

The **wlf2log** command translates a ModelSim WLF file (*vsim.wlf*) to a QuickSim II logfile. The command 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.

Important: This command should be invoked only after you have stopped the simulation using **quit -sim** or **dataset close sim**.

Syntax

```
wlf2log
  [-bits] [-fullname] [-help] [-inout] [-input] [-internal]
  [-l <instance_path>] [-lower] [-o <outfile>] [-output] [-quiet] <wlffile>
```

Arguments

`-bits`
Forces vector nets to be split into 1-bit wide nets in the log file. Optional.

`-fullname`
Shows the full hierarchical pathname when displaying signal names. Optional.

`-help`
Displays a list of command options with a brief description for each. Optional.

`-inout`
Lists only the inout ports. Optional. This may be combined with the -input, -output, or -internal switches.

`-input`
Lists only the input ports. Optional. This may be combined with the -output, -inout, or -internal switches.

`-internal`
Lists only the internal signals. Optional. This may be combined with the -input, -output, or -inout switches.

`-l <instance_path>`
Lists the signals at or below the specified HDL instance path within the design hierarchy. Optional.

`-lower`
Shows all logged signals in the hierarchy. Optional. When invoked without the -lower switch, only the top-level signals are displayed.

`-o <outfile>`
Directs the output to be written to the file specified by <outfile>. Optional. The default destination for the logfile is standard out.

`-output`
Lists only the output ports. Optional. This may be combined with the -input, -inout, or -internal switches.

`-quiet`
Disables error message reporting. Optional.

`<wlffile>`
Specifies the ModelSim WLF file that you are converting. Required.

wlfman

The wlfman command allows you to get information about and manipulate WLF files. The command performs three functions depending on which mode you use.

- **wlfman info** generates file information, resolution, versions, etc.
- **wlfman items** generates a list of HDL items (i.e., signals) from the source WLF file and outputs it to stdout. When redirected to a file, the output is called an item-list-file, and it can be read in by **wlfman filter**. The item_list_file is a list of items, one per line. 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
  ```

  Note that you can produce these files from scratch but be careful with syntax. **wlfman items** always creates a legal item_list_file.
- **wlfman filter** reads in a WLF file and optionally an item_list_file and writes out another WLF file containing filtered information from those sources. You determine the filtered information with the arguments you specify.

Syntax

```
wlfman info
  <wlffile>

wlfman items
  [-n] [-v] <wlffile>

wlfman filter
  [-b <time>] [-e <time>] [-f <item-list-file>] [-r <item>] [-s <symbol>]
  [-t <resolution>] <sourcewlffile> <outwlffile>
```

Arguments for wlfman info

`<wlffile>`
Specifies the WLF file from which you want information. Required.

Arguments for wlfman items

`-n`
Lists regions only (no signals). Optional.

`-v`
Produces "verbose" output that lists item type next to each item. Optional.

`<wlffile>`
Specifies the WLF file from which you want item information. Required.

Arguments for wlfman filter

`-b <time>`
Specifies the simulation time at which you want to begin reading information from the source WLF file. Optional. By default the output includes the entire time that is recorded in the source WLF file.

`-e <time>`
Specifies the simulation time at which you want to end reading information from the source WLF file. Optional.

`-f <item-list-file>`
Specifies an item-list-file created by **wlfman items** to include in the output WLF file. Optional.

`-r <item>`
Specifies an item (region) to recursively include in the output. If **<item>** is a signal, the output would be the same as using **-s**. Optional.

`-s <symbol>`
Specifies an item to include in the output. Optional. By default all items are output.

`-t <resolution>`
Specifies the time resolution of the new WLF file. Optional. By default the resolution is the same as the source WLF file.

`<sourcewlffile>`
Specifies the source WLF file from which you want items. Required.

`<outwlffile>`
Specifies the name of the output WLF file. Required. The output WLF file will contain all items specified by **-f <item-list-file>**, **-r <item>**, and **-s <item>**. Output WLF files are always written in the latest WLF version regardless of the source WLF file version.

See also

Chapter 7 - WLF files (datasets) and virtuals

wlfrecover

The **wlfrecover** tool attempts to "repair" WLF files that are incomplete due to a crash or the file being copied prior to completion of the simulation. The tool works only on WLF files created by ModelSim versions 5.6 or later. You can run the tool from the VSIM prompt or from a shell.

Syntax

```
wlfrecover
  <filename> [-force] [-q]
```

Arguments

`<filename>`
Specifies the WLF file to repair. Required.

`-force`
Disregards file locking and attempts to repair the file. Required for PCs.

`-q`
Hides all messages unless there is an error while repairing the file. Optional.

write format

The **write format** command records the names and display options of the HDL items currently being displayed in the List or Wave window. The file created is primarily a list of **add list** (CR-32)or **add wave** (CR-35) commands, though a few other commands are included (see "Output" below). This file may be invoked with the **do** command (CR-68) to recreate the List or Wave window format on a subsequent simulation run.

When you load a wave or list format file, ModelSim verifies the existence of the datasets required by the format file. ModelSim displays an error message if the requisite datasets do not all exist. To force the execution of the wave or list format file even if all datasets are not present, use the **-force** switch with your **do** command. For example:

```
VSIM> do wave.do -force
```

Note that this will result in error messages for signals referencing nonexistent datasets. Also, -force is recognized by the format file not the **do** command.

Syntax

```
write format
  list | wave <filename>
```

Arguments

`list | wave`
Specifies that the contents of either the List or the Wave window are to be recorded. Required.

`<filename>`
Specifies the name of the output file where the data is to be written. Required.

Examples

`write format list alu_list.do`
Saves the current data in the List window in a file named *alu_list.do*.

`write format wave alu_wave.do`
Saves the current data in the Wave window in a file named *alu_wave.do*.

Output

Below is 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 pane. 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.

See also

add list (CR-32), **add wave** (CR-35)

write list

The **write list** command records the contents of the List window in a list output file. This file contains simulation data for all HDL items displayed in the List window: VHDL signals and variables and Verilog nets and registers.

Syntax

```
write list
  [-events] <filename>
```

Arguments

`-events`
Specifies to write print-on-change format. Optional. Default is tabular format.

`<filename>`
Specifies the name of the output file where the data is to be written. Required.

Examples

`write list alu.lst`
Saves the current data in the List window in a file named *alu.lst*.

See also

write tssi (CR-225)

write preferences

The **write preferences** command saves the current GUI preference settings to a Tcl preference file. Settings saved include current window locations and sizes; Wave, Signals and Variables window column widths; Wave, Signals and Variables window value justification; and Wave window signal name width.

Syntax

```
write preferences
  <preference file name>
```

Arguments

`<preference file name>`
Specifies the name for the preference file. Optional. 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** (UM-346) environment variable.

See also

You can modify variables by editing the preference file with the ModelSim **notepad** (CR-95):

```
notepad <preference file name>
```

write report

The **write report** 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.

Syntax

```
write report
  [[<filename>] [-l | -s]] | [-tcl]
```

Arguments

`<filename>`
Specifies the name of the output file where the data is to be written. Optional. If the <filename> is omitted, the report is written to the Main window transcript.

`-l`
Generates more detailed information about the design. Default.

`-s`
Generates a short list of design information. Optional.

`-tcl`
Generates a Tcl list of design unit information. Optional. This argument cannot be used with a filename.

Examples

`write report alu.rep`
Saves information about the current design in a file named *alu.rep*.

write transcript

The **write transcript** command writes the contents of the Main window Transcript to the specified file. The resulting file can be used to replay the transcribed commands as a DO file (macro).

The command cannot be used in batch mode. In batch mode use the standard "Transcript" (UM-147) file or redirect std out.

Syntax

```
write transcript
  [<filename>]
```

Arguments

`<filename>`
Specifies the name of the output file where the data is to be written. Optional. If the <*filename*> is omitted, the transcript is written to a file named *transcript*.

See also

do (CR-68)

write tssi

The **write tssi** command records the contents of the List window in a "TSSI format" file. The file contains simulation data for all HDL items 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.

Syntax

```
write tssi
  <filename>
```

Arguments

`<filename>`
Specifies the name of the output file where the data is to be written. Required.

Description

"TSSI format" is documented in the Fluence TDS Software System, Chapter 2 of Volume I, Getting Started, R11.1, dated November 15, 1999. In that document, TSSI format is called Standard Events Format (SEF).

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 item is a port, otherwise it is assumed to be bidirectional (mode INOUT).

Items 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

std_ulogic State Characters	SEF State Characters		
	Input	Output	Bidirectional
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 Fluence (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 from Fluence Technology. ModelSim outputs a vector file, and Fluence's tools determine whether the bidirectional signals are driving or not.

See also

tssi2mti (CR-127)

write wave

The **write wave** 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
  [-width <real_num>] [-height <real_num>]
  [-margin <real_num>] [-start <time>] [-end <time>] [-perpage <time>]
  [-landscape] [-portrait] <filename>
```

Arguments

`-width <real_num>`
Specifies the paper width in inches. Optional. Default is 8.5.

`-height <real_num>`
Specifies the paper height in inches. Optional. Default is 11.0.

`-margin <real_num>`
Specifies the margin in inches. Optional. Default is 0.5.

`-start <time>`
Specifies the start time (on the waveform time scale) to be written. Optional.

`-end <time>`
Specifies the end time (on the waveform time scale) to be written. Optional.

`-perpage <time>`
Specifies the time width per page of output. Optional.

`-landscape`
Use landscape (horizontal) orientation. Optional. This is the default orientation.

`-portrait`
Use portrait (vertical) orientation. Optional. The default is landscape (horizontal).

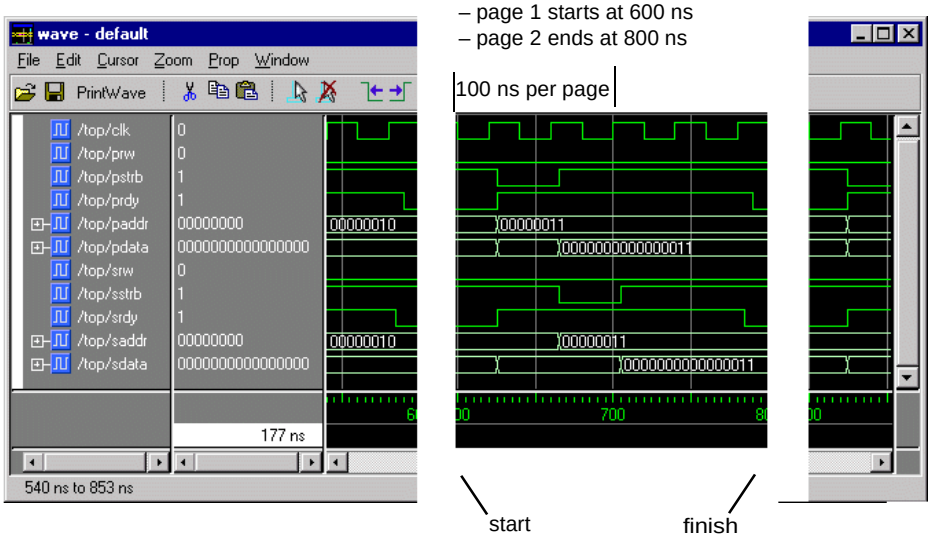
`<filename>`
Specifies the name of the PostScript output file. Required.

Examples

`write wave alu.ps`
Saves the current data in the Wave window in a file named *alu.ps.*

```
write wave -start 600ns -end 800ns -perpage 100ns top.ps
```

Writes two separate pages to *top.ps* as indicated in the illustration (the actual PostScript print out will show all items listed in the Wave window, not just the portion in view):



To make the job of creating a PostScript waveform output file easier, use the **File > Print Postscript** menu selection in the Wave window. See "Saving waveforms" (UM-242) for more information.

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

Index

CR = Command Reference, UM = User's Manual

Symbols

Numerics

A

B

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

C

D

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

E

F

G

M

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

N

O

P

Q

R

S

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

T

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

U

V

W

X

Y

Z