OptiSPICE

Simulator Command Reference

Opto-Electronic Circuit Design Software

Version 5.2

OptiSPICE

Simulator Command Reference

Opto-Electronic Circuit Design Software

Disclaimer

Optiwave makes no representation or warranty with respect to the adequacy of this documentation or the programs which it describes for any particular purpose or with respect to its adequacy to produce any particular result. In no event shall Optiwave, its employees, its contractors or the authors of this documentation, be liable for special, direct, indirect, or consequential damages, losses, costs, charges, claims, demands, or claim for lost profits, fees, or expenses of any nature or kind.

Table of Contents

Simulator Commands 1

.TRAN 3
.OP 7
.DC 9
.AC 13
.NOISE 17
.OPTION 19
JOBCONTROL 23
.MONITOR 25
.PRINT 29
.IC 31
.PARAM 33
.INCLUDE 35
.SUBCKT 37

Simulator Commands

This section contains information on the following OptiSPICE simulator commands.

Analysis:

- .TRAN
- .OP
- .DC
- .AC
- .NOISE

Options:

- .OPTION
- JOBCONTROL

Output:

- .MONITOR
- .PRINT

Other commands:

- .IC
- .PARAM
- .INCLUDE
- .SUBCKT

.TRAN

The command ".TRAN" performs transient analysis which simulates a circuit from the time point zero, *t* = 0, to a user-specified time.

Syntax

Style	Form

SPICE	`.TRAN tstep tstop <SWEEP <Data=sweepModelName>`
	`+ <<SWvar=>sweepParam <SWstart=>val <SWstop=>val`
	`+ <SWstep=>val>> <solver=BACKE/TRAP/FLIP> <abstol=val>`
	`+ <reltol=val> <habstol=val> <hreltol=val> <c_abstol=val>`
	`+ <c_reltol=val> <trtol=val> <NoiseSim=0/1>`
	`+ <MaxBandwidth = val>`

Parameters

Name and description	Symbol	Default value	Units	Value range
Step size Initial step time	*tstep*	-	s	[0, +INF[
Stop time Stop time for transient analysis	*tstop*	-	s	[0, +INF[
Parameter sweep flag Enable parameter sweep if this keyword is given	SWEEP	-	-	-
Parameter sweep data model Name of the model which contains the parameters to be swept and the file name that contains sweep values	Data	0	-	-
Sweep parameter name Name of the parameter to be swept (if parameter Data is not provided)	SWvar	-	-	-
Parameter sweep start value Start value for the parameter to be swept (if parameter Data is not provided)	SWstart	0	-	]-INF, +INF[
Parameter sweep stop value Stop value for the parameter to be swept (if parameter Data is not provided)	SWstop	0	-	]-INF, +INF[

Name and description	Symbol	Default value	Units	Value range
Parameter sweep step value Increment value for the parameter to be swept (if parameter Data is not provided)	SWstep	0	-	[0, +INF[
Numerical integration method Numerical integration method used for the transient solver 1) TRAP - trapezoidal method, 2) BACKE - backward Euler method, 3) FLIP - starts with TRAP, but will switch to BACKE if necessary (in case of artificial oscillations)	SOLVER	FLIP	-	TRAP, BACKE, FLIP
Maximum step size Maximum limit time step the simulator allowed to take.	MAXST	-	s	[0, +INF[
Minimum step size Minimum time step the simulator allowed to take.	MINST	-	s	[0, +INF[
Absolute error tolerance Absolute error tolerance for the branch currents. If not given use abstol value defined by .Option statement.	abstol	1e-8	-	[0, +INF[
Relative error tolerance Relative error tolerance for the node voltages. If not given use reltol value defined by .Option statement.	reltol	1e-6	-	[0, +INF[
Transient absolute error tolerance If given, sets the absolute error tolerance for the branch currents during transient part of the simulation, otherwise abstol is used.	habstol	1e-8	-	[0, +INF[
Transient relative error tolerance If given, sets the relative error tolerance for the node voltages during transient part of the simulation, otherwise reltol is used	hreltol	1e-6	-	[0, +INF[
Absolute error tolerance for non-linear element currents Sets the absolute error tolerance required for the convergence check for currents flowing through non-linear elements	c_abstol	1e-15	-	[0, +INF[

Name and description	Symbol	Default value	Units	Value range
Relative error tolerance for non-linear element currents Sets the relative error tolerance required for the convergence check for currents flowing through non-linear elements	c_reltol	1e-6	-	[0, +INF[
Tolerance factor for truncation error estimation Factor used to estimate the local truncation error when the timestep algorithm truncates the Taylor series expansion. As trtol increases, the time step increases.	trtol	7.0	-	[0, +INF[
Enable noise simulation Enable noise simulation for transient analysis	NoiseSim	0	-	0, 1
Maximum noise bandwidth Maximum bandwidth for noise simulation	MaxBandwidth	1e9	Hz	[0, +INF[

Examples

```
.TRAN 0.01ns 20ns
```

The above statement performs transient analysis with an initial time step of 0.01 ns and stop time of 20 ns.

```
.TRAN 0.01ns 20ns SWEEP Data = SweepModel
```

The above transient analysis is performed by enabling parameter sweep, and the parameters to be swept and their sweep values are given by a DATA model with the name SweepModel. See Notes section for more details about DATA model statement.

```
.TRAN 0.01ns 20ns SWEEP rval 50 100 10
```

The above transient analysis includes a single parameter sweep where a parameter rval is swept linearly from 50 to 100 with an increment value of 10.

```
.TRAN 0.01ns 20ns solver = TRAP Maxst = 1ns Minst = 0.01 ps
```

In the above example, the numerical integration algorithm is set to trapezoidal method and the internal step size which is calculated by OptiSPICE is restricted such that minimum value is 0.01 ps and maximum value is 1 ns.

```
.TRAN 0.01ns 20ns NoiseSim = 1 MaxBandwidth = 10e9
```

In the above example, noise simulation is enabled globally for transient simulation and the bandwidth of the noise is set to 10 GHz. Noise bandwidth forces the maximum possible internal step size to be 1/MaxBandwidth (0.01 ns for this example).

Notes:

DATA model statement is used to provide a sweep data given by a text file for a simulation analysis. It supports one or more sweeping parameters. Following is an example of sweeping of two parameters, rval and cval, given in a file sweep.dat.

```
Osp MODEL Name = SweepModel type=DATA params = [rval cval]
+ file = sweep.dat
```

Content of the sweep.dat is given as follows

```
50   100p
50   200p
100  100p
100  200p
150  100p
150  200p
```

The first column corresponds to the rval while the second one corresponds to the cval (in the order of the parameters listed). Separate simulation is performed for each row. i.e. first simulation is performed with rval = 50 and cval = 100p, second simulation is performed with rval = 50 and cval = 200p, and so on.

.OP

This command performs DC operating point analysis. The DC operating point is the starting solution for any transient simulation.

Syntax

Style	Form
SPICE	`.OP <SWEEP <Data=`*`sweepModelName`*`> <<SWvar=>`*`sweepParam`* `+ <SWstart=>`*`val`*` <SWstop=>`*`val`*` <SWstep=>`*`val`*`>>`

Parameters

Name and description	Symbol	Default value	Units	Value range
Parameter sweep flag Enable parameter sweep if this keyword is given	SWEEP	-	-	-
Parameter sweep data model Name of the model which contains the parameters to be swept and the file name that contains sweep values	Data	0	-	-
Sweep parameter name Name of the parameter to be swept (if parameter Data is not provided)	SWVar	-	-	-
Parameter sweep start value Start value for the parameter to be swept (if parameter Data is not provided)	SWstart	0	-	]-INF, +INF[
Parameter sweep stop value Stop value for the parameter to be swept (if parameter Data is not provided)	SWstop	0	-	]-INF, +INF[
Parameter sweep step value Increment value for the parameter to be swept (if parameter Data is not provided)	SWstep	0	-	[0, +INF[

Examples

```
.OP
```

The above statement performs DC operating point analysis.

```
.OP SWEEP rval 50 100 10
```

In the above example, a DC operating point analysis is performed for each parametric value of the rval, ranging from 50 to 100 with an increment value of 10 .

.DC

This command performs DC sweep analysis where a sweep of the DC value of an independent source is performed for a specified range.

Syntax

Style	Form

SPICE	`.DC srcName startVal stopVal incr <SWEEP <Data=sweepModelName>`
	`+ <<SWvar=>sweepParam <SWstart=>val <SWstop=>val`
	`+ <SWstep=>val>> <abstol=val> <reltol=val> <c_abstol=val>`
	`+ <c_reltol=val>`

Parameters

Name and description	Symbol	Default value	Units	Value range
Source name Name of the DC sweep source	*SrcName*	-	-	-
Start value Start value for the DC sweep	*startVal*	-	V, A, or Deg C	]-INF, +INF[
Stop value Stop value for the DC sweep	*stopVal*	-	V, A, or Deg C	]-INF, +INF[
Increment value Increment value for the DC sweep	*incr*	-	V, A, or Deg C	[0, +INF[
Parameter sweep flag Enable parameter sweep if this keyword is given	SWEEP	-	-	-
Parameter sweep data model Name of the model which contains the parameters to be swept and the file name that contains sweep values	Data	0	s	-
Sweep parameter name Name of the parameter to be swept (if parameter Data is not provided)	SWvar	-	-	-
Parameter sweep start value Start value for the parameter to be swept (if parameter Data is not provided)	SWstart	0	-	]-INF, +INF[

Name and description	Symbol	Default value	Units	Value range
Parameter sweep stop value Stop value for the parameter to be swept (if parameter Data is not provided)	SWstop	0	-	]-INF, +INF[
Parameter sweep step value Increment value for the parameter to be swept (if parameter Data is not provided)	SWstep	0	-	[0, +INF[
Absolute error tolerance Absolute error tolerance for the branch currents. If not given use abstol value defined by .Option statement.	abstol	1e-8	-	[0, +INF[
Relative error tolerance Relative error tolerance for the the node voltages. If not given use reltol value defined by .Option statement.	reltol	1e-6	-	[0, +INF[
Absolute error tolerance for non-linear element currents Sets the absolute error tolerance required for the convergence check for currents flowing through non-linear elements	c_abstol	1e-15	Q	[0, +INF[
Relative error tolerance for non-linear element currents Sets the relative error tolerance required for the convergence check for currents flowing through non-linear elements	c_reltol	1e-6	Q	[0, +INF[

Examples

```
.DC Vin 0 5 0.1
```

The above example performs DC sweep for the voltage source, Vin, from 0 to 5V with the increment of 0.1 V.

```
.DC I1 10m 100m 5m
```

The above example performs DC sweep for the current source, I1, from 10 mA to 100 mA, with the increment of 5 mA.

```
.DC ThermSrc1 20 200 10
```

The above example performs DC sweep for the thermal source, ThermSrc1, from 20 °C to 200 °C, with the increment of 10 °C.

```
.DC Vin 0 5 0.1 SWEEP Data = SweepModel
```

In the above example, DC analysis is performed by enabling parameter sweep, and the parameters to be swept and their sweep values are given by a DATA model with the name SweepModel.

```
.DC Vin 0 5 0.1 SWEEP rval 50 100 10
```

In the above example, DC analysis includes a single parameter sweep where a parameter rval is swept linearly from 50 to 100 with an increment value of 10.

.DC

.AC

This command performs AC analysis which simulates the circuit for a given frequency range.

Syntax

Style	Form

SPICE	`.AC <LIN/DEC> npoints fstart fstop <SWEEP <Data=sweepModelName>`
	`+ <<SWvar=>sweepParam <SWstart=>val <SWstop=>val`
	`+ <SWstep=>val>> <abstol=val> <reltol=val> <c_abstol=val>`
	`+ <c_reltol=val>`

Parameters

Name and description	Symbol	Default value	Units	Value range
AC sweep type Variation type of frequency points for AC sweep; LIN - linear variation, DEC - decade variation	-	LIN	-	LIN, DEC
Number of frequency points Number of frequency points for the AC analysis	*npoints*	0	-	[0, +INF[
Start frequency Starting frequency for the AC Analysis	*fstart*	0	Hz	[0, +INF[
Stop frequency Stopping frequency for the AC analysis	*fstop*	0	Hz	[0, +INF[
Parameter sweep data model Name of the model which contains the parameters to be swept and the file name that contains sweep values	Data	0	-	-
Sweep parameter name Name of the parameter to be swept (if parameter Data is not provided)	SWvar	-	-	-
Parameter sweep start value Start value for the parameter to be swept (if parameter Data is not provided)	SWstart	0	-	]-INF, +INF[

Name and description	Symbol	Default value	Units	Value range
Parameter sweep stop value Stop value for the parameter to be swept (if parameter Data is not provided)	SWstop	0	-	]-INF, +INF[
Parameter sweep step value Increment value for the parameter to be swept (if parameter Data is not provided)	SWstep	0	-	[0, +INF[
Absolute error tolerance Absolute error tolerance for the branch currents to perform operating point analysis in the presence of nonlinear elements. If not given use abstol value defined by .Option statement.	abstol	1e-8	-	[0, +INF[
Relative error tolerance Relative error tolerance for the the node voltages to perform operating point analysis in the presence of nonlinear elements. If not given use reltol value defined by .Option statement.	reltol	1e-6	-	[0, +INF[
Absolute error tolerance for non-linear element currents Sets the absolute error tolerance required for the convergence check for currents flowing through non-linear elements (for operating point analysis in the presence of nonlinear elements)	c_abstol	1e-15	-	[0, +INF[
Relative error tolerance for non-linear element currents Sets the relative error tolerance required for the convergence check for currents flowing through non-linear elements (for operating point analysis in the presence of nonlinear elements.	c_reltol	1e-6	-	[0, +INF[

Examples

```
.AC LIN 1000 1k 1MEG
```

The above statement performs frequency sweep for 1000 frequency points ranging from 1 kHz to 1 MHz, varying linearly.

```
.AC DEC 10 10MEG 5G
```

The above example performs frequency sweep by 10 points per decade from 10 MHz to 5 GHz.

```
.AC DEC 10 10MEG 5G SWEEP Data = SweepModel
```

In the above example, AC analysis is performed by enabling parameter sweep, and the parameters to be swept and their sweep values are given by a DATA model with the name SweepModel.

```
.AC DEC 10 10MEG 5G SWEEP cval 100p 500p 50p
```

In the above example, AC analysis includes a single parameter sweep where capacitance parameter cval is swept linearly from 100 pF to 500 pF with an increment value of 50 pF.

.AC

.NOISE

Performs AC noise analysis when used in conjunction with .AC statement.

Syntax

Style	Form
SPICE	`.NOISE V(`*`outnode`*`) `*`inputsource interval`*

Parameters

Name and description	Symbol	Default value	Units	Value range
outnode Name of the output node where all the noises be summed	-	-	-	-
inputsource Name of the independent voltage or current source to be used as the input reference noise	-	-	-	-
interval Number of frequency points to be selected each time from the AC sweep for printing noise analysis summary	-	-	-	[0, +INF[

Example

```
.NOISE V(5) Vin 10
```

In this statement, node 5 is specified as the output node where the total output noise is desired. Input noise is referred to the voltage source Vin. Ten frequency points are selected each time from the AC sweep for printing noise analysis summary.

.OPTION

This command sets a wide variety of simulation options.

Syntax

Style	Form
SPICE	`.OPT(ION(S))` <param1 = *val1*> <param2 = *val2*> ....

Parameters

Name and description	Symbol	Default value	Units	Value range
Minimum conductance Minimum conductance allowed	GMIN	1e-12	-	[0, +INF[
Minimum conductance for DC Minimum conductance for DC solution	GMINDC	1e-12	-	[0, +INF[
Cache all mode shapes Option to save (cache) mode shape of all optical signals to file	CacheAllModeShapes	0	-	0, 1
Mode shape array size Array size for spatial mode shapes	ModeShapeArySize	128	-	[0, +INF[
Absolute error tolerance Absolute error tolerance for the branch currents	ABSTOL	1e-8	-	[0, +INF[
Relative error tolerance Relative error tolerance for the node voltages	RELTOL	1e-6	-	[0, +INF[
Number of primary digits Number of digits to be printed to the right of the decimal point for typical X axis variables such as time, frequency, etc.	PrimaryNumDGT	16	-	-
Number of digits Number of digits to be printed to the right of the decimal point for the output results (voltage, current, optical power, etc)	NUMDGT	8	-	-
Cache all mode shapes Option to cache or save results of all mode shapes for post-processing	CacheAllModeShapes	1	-	[0, 1]

Name and description	Symbol	Default value	Units	Value range
Use Intel MKL When enabled and set to 1 (USEMKL=1), the Intel MKL sparse matrix (SM) solver will be used, otherwise the default UMFPACK SM solver will be used.	USEMKL	1	-	[0, 1]
MKLThreads Sets the number of threads/cores to be used by the Intel MKL SM solver. By default 50% of the available threads will be used.	MKLTHREADS	-	-	-
Parameter hierarchy To set the parameter scope rules, either local or global	PARHIER	GLOBAL	-	GLOBAL, LOCAL
Element parameter scaling factor Scaling factor for element parameter	SCALE	1	-	[0, +INF[
Model parameter scaling factor Scaling factor for model parameter	SCALM	1	-	[0, +INF[
Reference temperature Reference temperature for the simulation	TNOM	25	Deg C	]-INF, +INF[
MOSFET drain diffusion area Sets default MOSFET drain diffusion area	DEFAD	0	-	[0, +INF[
MOSFET source diffusion area Sets default MOSFET source diffusion area	DEFAS	0	-	[0, +INF[
MOSFET channel length Sets default MOSFET channel length	DEFL	1e-4	-	[0, +INF[
MOSFET channel width Sets default MOSFET channel width	DEFW	1e-4	-	[0, +INF[
MOSFET number of drain squares Sets default number of squares for drain resistance in a MOSFET	DEFNRD	0	-	[0, +INF[
MOSFET number of source squares Sets default number of squares for source resistance in a MOSFET	DEFNRS	0	-	[0, +INF[

Name and description	Symbol	Default value	Units	Value range
MOSFET drain perimeter Sets the default MOSFET drain diode perimeter	DEFPD	0	-	[0, +INF[
MOSFET source perimeter Sets the default MOSFET source diode perimeter	DEFPS	0	-	[0, +INF[
Random number generator seed Sets the seed value for random number generator used for Monte Carlo noise analysis	SEED	0	-	[0, +INF[

Example

```
.OPTION ABSTOL = 1e-10 RELTOL = 1e-8 TNOM = 27 CacheAllModeShapes = 1
```

This statement sets the absolute tolerance, relative tolerance, and nominal temperature, as well as enables the option to save all optical signal mode shape to be saved to files.

JOBCONTROL

JOBCONTROL provides special control options for simulation process, especially for debugging purposes.

Syntax

Style	Form

OptiSPICE	`Osp JOBCONTROL <Verbosity = SILENT/LOG/DEBUG>`
	`+ <TempControl=TEMPCIR/NOTEMPCIR> <AddDCtoPWL = 0/1>`
	`+ <LogCircuit = 0/1>`

Parameters

Name and description	Symbol	Default value	Units	Value range
Verbosity level Sets the level of displaying details during simulation process. In SILENT mode, only simulation progress is displayed. In LOG mode, a log file containing Netlist data and progress is generated. In DEBUG mode, more simulation related details such as internal time steps and convergence failures are displayed and stored in a log file.	Verbosity	LOG	-	SILENT, LOG, DEBUG
Temperature control Sets the temperature dependency of the circuit. If the circuit to be affected by temperature variation set to TEMPCIR, otherwise set to NOTEMPCIR.	TempControl	TEMPCIR	-	NOTEMPCIR, TEMPCIR
Add DC offset to PWL transient source function Enable DC offset to be added to a piece wise linear (PWL) source	AddDCtoPWL	0	-	0, 1
Log circuit element details Option to enable printing element details to log file	LogCircuit	0	-	0, 1

Name and description	Symbol	Default value	Units	Value range
Job control for all monitors When set to 1 or 2, monitors shall be generated for all electrical and optical nodes. Option 2 will also generate monitors for all nodes contained within sub-circuits. NOTE 1: For large circuits the number of monitors may be very high. NOTE 2: This control function is independent of manually inserted probes (monitors).	JobControl AllMon	0	-	0, 1, 2

Example

```
Osp JOBCONTROL Verbosity = SILENT TempControl = NOTEMPCIR

+ AddDCtoPWL = 1
```

This statement makes the verbosity level to silent so that only simulation progress is displayed, the circuit to be independent of temperature, and DC offset to be added to PWL source.

```
Osp JOBCONTROL AllMon=1
```

This statement sets the Job control for all monitors to Level 1. Monitors shall be generated for all electrical and optical nodes of top level elements (monitors within sub-circuits shall not be generated)

.MONITOR

Allows users to probe nodal voltages, current through devices, and optical field. Results are stored in specific files according to the analysis type. Many type of the results for transient analysis produced by Monitor statement can be visualized using OptiSystem.

Syntax

Style	Form
OptiSPICE	.MONITOR *V/ACNoiseV node* <Format=*CMPLX/MAG/MAGPHI*> .MONITOR *I elemName port* <Format=*CMPLX/MAG/MAGPHI*> .MONITOR *OptXXX/ACNoiseOptFields elemName port* + <Dir=*IN/OUT/BOTH*> <Pol=X/Y/BOTH> <Format=*CMPLX/MAG*> + <ChannelLambda=val> <Sum=*YES/NO*> <P0dB=0/1>

Parameters

Name and description	Symbol	Default value	Units	Value range
Output type Specifies the type of output; I - current, V - voltage, OptFields - optical field, OptPower - optical power, OptPhase - optical phase, OptChirp - optical chirp, OptLambda - optical carrier wavelength, ACNoiseV - noise voltage when AC noise analysis is performed, ACNoiseOptFields - optical field noise when AC noise analysis is performed	*type*	-	-	I, V, OptFields, OptPower, OptPhase, OptChirp, OptLambda, ACNoiseV, ACNoiseOptFields
Node Node at which voltage is monitored	*node*	-	-	-
Port number Port number of the device where the output is monitored. Applicable to current and optical signal, not for voltage.	*port*	-	-	1, 2, 3, ...
Direction Direction of optical wave propagation to be monitored in a device: FWD - forward direction only, REV - reverse direction only, BOTH - both forward and reverse direction	Dir	OUT	-	IN, OUT, BOTH

Name and description	Symbol	Default value	Units	Value range
Polarization Polarization of the optical signal to be monitored; X - X polarization only, Y - prints Y polarized, BOTH - both X and Y polarizations	Pol	X	-	X, Y, BOTH
Output format Output format of complex values for optical signal and frequency response of voltage and current; CMPLX - prints real and imaginary values, MAG - prints magnitude only, MAGPHI - prints magnitude and phase of electrical signal in case of AC analysis	Format	MAGPHI	-	CMPLX, MAG, MAGPHI
Channel wavelength Specifies what channel to print. If given only the specific channel info will be printed.	ChannelLambda	0	nm	[0, +INF[
Mode summation option If set to YES, sums the power of all optical modes together and prints only summed power. Only applicable for orthogonal modes and if the monitor type is OptPower.	Sum	NO	-	YES, NO
Magnitude in dB If set the magnitude of the optical power will be printed in dB	P0dB	0	-	0, 1

Examples

Electrical

```
.MONITOR V OUT
```

The above statement prints voltage at node named as OUT. In case of AC analysis, this statement prints out real and imaginary values of the voltage.

```
.MONITOR I M1 1
```

The above statement prints current flowing into the pin 1 (drain) of a MOSFET with name M1. Current flowing into the given pin is considered as the positive current.

```
.MONITOR V OUT Format = MAG
```

The above statement prints magnitude of the voltage at the node OUT when AC analysis is performed.

```
.MONITOR V OUT Format = MAGPHI
```

In this above statement, phase is also printed out along with magnitude. The unit of the phase is in radians.

```
.MONITOR I Q1 3 Format = MAGPHI
```

Prints magnitude and phase of the current flowing into the pin 3 (emitter) of a bibolar junction transistor with the name Q1 when an AC analysis is performed.

```
.MONITOR ACNoiseV OUT Format = MAG
```

Prints magnitude of the noise voltage at the node, OUT, when an AC analysis is performed.

Optical

```
.MONITOR OptFields Laser1 3
```

The above statement prints electric field at the third pin of the Laser1. With *Dir*, *Pol*, and *Format* are not given, they take their default values such that it prints both directions, both polarizations, and real and imaginary value of the fields.

```
.MONITOR OptFields Laser1 3 Dir = OUT Pol = X Format = MAG
```

Prints only the magnitude of the X polarized field propagating out of the Laser1.

```
.MONITOR OptPower Fiber1 1 Dir = IN
```

Prints the optical power propagating into the pin 1 of the Fiber1.

```
.MONITOR OptPhase Fiber1 1 Dir = IN
```

Prints the phase of the field propagating into the pin 1 of the Fiber1.

```
.MONITOR OptChirp Laser1 3 Dir = OUT
```

Prints the chirp of the field propagating out of Laser1.

```
.MONITOR OptLambda Laser1 3 Dir = OUT
```

Prints the carrier wavelength of Laser1.

```
.MONITOR ACNoiseOptFields Laser1 3 Dir = OUT
```

Prints the noise in optical field of Laser1 when AC analysis is performed.

.PRINT

Provides basic compatibility with HSPICE for printing voltage, current, and AC output noise.

Syntax

Style	Form

SPICE	`.PRINT <TRAN/DC/AC> V(node1) V(node2) ...`
	`.PRINT <TRAN/DC/AC> I(elem1) I(elem2) ...`
	`.PRINT NOISE ONOISE`

Parameters

Name and description	Symbol	Default value	Units	Value range
Analysis type Analysis type for the print	-	-	-	TRAN, DC, AC, NOISE
Print type Voltage (V) or current (I) to be printed	-	-	-	V, I, ONOISE
Node name Node name for printing voltage at the specific node	-	-	-	-
Element name Element name for printing current through the element	-	-	-	-

Examples

```
.PRINT TRAN V(out)
```

The above statement prints the voltage at the node named as ‘out’ for the transient analysis performed

```
.PRINT DC I(M1)
```

Prints the current through MOSFET M1 (through three terminals: drain, gate, and source) for the DC analysis performed.

```
.PRINT TRAN V(1) V(2) V(3)
```

Prints the voltages at node 1, 2, and 3 for the transient analysis.

```
.PRINT TRAN V(X1.1)
```

Prints the voltage at node 1 of the subcircuit X1.

```
.PRINT AC V(OUT)
```

Prints AC magnitude at the node OUT.

```
.PRINT NOISE ONOISE
```

Prints AC noise at a node where output is desired. The output node is specified by .NOISE statement.

.IC

This command sets initial voltage values (at time point zero, $t = 0$) for the specified nodes for transient and operating point analysis. These nodal voltages remain fixed with the specified values until the operating point analysis is completed and only released after transient simulation starts ($t > 0$).

Syntax

Style	Form
SPICE	`.IC V(`*node1*`)=`*value1* `V(`*node2*`)=`*value2* `V(`*node3*`)=`*value3* `...`

Parameters

Name and description	Symbol	Default value	Units	Value range
Node names Name of the nodes initial voltages to be set	*node1, node2, ...*	-	-	-

Examples

```
.IC V(1)=0.5 V(3)=1.2
```

Sets initial voltage for node 1 at 0.5 V and node 3 at 1.2 V.

.PARAM

This command defines a parameter with a value or a mathematical expression.

Syntax

Style	Form
SPICE	`.PARAM paramName1=val1 paramName2=val2 paramName3=val3 ...` `.PARAM paramName1='equation1' paramName2='equation2' ...`

Parameters

Name and description	Symbol	Default value	Units	Value range
Parameter name Name of the parameter to be defined	*paramName*	-	-	-

Examples

```
.PARAM Rval = 50
```

Defines the parameter Rval with a value of 50.

```
.PARAM Cval = '1e-11 + 1-12 * exp(-TIME)'
```

Defines parameter Cval with a mathematical expression where TIME is the time value in transient analysis.

.PARAM

.INCLUDE

This command includes an external Netlist which is specified by a file path. File path can be absolute or relative.

Syntax

Style	Form
SPICE	`.INCLUDE "filepath"`

Parameters

Name and description	Symbol	Default value	Units	Value range
File path File path of the netlist to be included	*filepath*	-	-	-

Examples

```
.INCLUDE "C:\Circuits\MOSCircuit.sp"
```

Includes the netlist file MOSCircuit.sp with the absolute path.

```
.INCLUDE "LNA.sp"
```

Includes the netlist LNA.sp form the current working directory.

.INCLUDE

.SUBCKT

This command defines a sub-circuit. A sub-circuit definition ends with .ENDS command.

Syntax

Style	Form
SPICE	`.SUBCKT` *`subcktname n1 n2 n3 ... <param1=val1> <param2=val2> ...`*

Parameters

Name and description	Symbol	Default value	Units	Value range
Subcircuit name Name of the subcircuit to be defined	*subcktname*	-	-	-
Node names Node names for external reference	*n1, n2, n3, ...*	-	-	-
Parameter names Parameters to be used inside the subcircuit. These parameters are given with the default values.	*param1, param2, ...*	-	-	-

Examples

```
.SUBCKT OpAmp Vinp Vinm VEEp VEEm Vout Gain = 1e6
```

Defines the subcircuit OpAmp with external reference node names Vinp, Vinm, VEEp, VEEm and Vout. Parameter Gain is defined with the default value of 1e6. Any instance of this subcircuit can provide its own Gain value. However, if the value is not defined, then this default value is used.

\

When you create a new design, you must define the simulation parameters. These parameters are critical to the simulation. These parameters can be entered through the **Setup** dialog box.

The simulation parameters are organized by following categories in the **Setup** dialog box.

- Main
- Transient
- AC
- DC
- Options
- Control
- Parameters
- Libraries.

The parameters for each category are listed below. See OptiSPICE Simulator Command Reference fore more details.

Main

Main tab contain parameters that specify the type of the simulation.

Name and description	Symbol name	Default value	Units	Value range
Simulation type Used to set the desired simulation type	-	Transient	-	[Transient, AC, DC, Operating point, None]
Enable sweep Check box to enable parameter sweep	-	Disabled (Unchecked)	-	[Enabled, Disabled]
Spice Command Used to setup a customized entry of SPICE commands (this script is automatically placed at the beginning of the generated Netlist file)	-	-	-	-
Python File Name The name of the Python executable file to be used for post processing of simulation results	-	-	-	-
Script Engine Execution path Directory location of the Python executable file	-	-	-	-

Transient

The **Transient** tab contains parameters to perform transient analysis.

Name and description	Symbol name	Default value	Units	Value range
Step time Initial step time	TStep	1p	s	[0, +INF]
Stop time Stop time for the transient simulation.	TStop	10n	s	[0, +INF]
Maximum step size Maximum time interval the simulator is allowed to take.	MAXST	1e-3	s	[0, +INF]
Minimum step size Minimum time interval the simulator is allowed to take.	MINST	1e-9	s	[0, +INF]

Name and description	Symbol name	Default value	Units	Value range
Numerical integration method The numerical integration algorithm that is used to solve the system of equations numerically. The following methods are available: • BACKE - Backward Euler method • TRAP - Trapezoidal method • FLIP - Automatic flipping between trapezoidal and backward Euler methods by the simulator in order to perform accurate simulation.	SOLVER	FLIP	-	[BACKE, TRAP, FLIP]
Enable noise simulation Enable noise simulation for transient analysis. By default this parameter is disabled (Enable box is unchecked)	NoiseSim	1	-	0, 1
Maximum noise bandwidth Maximum bandwidth for noise simulation	MaxBandwidth	1e9	Hz	[0, +INF[

AC

AC tab contains parameters to perform AC analysis.

Name and description	Symbol name	Default value	Units	Value range
AC sweep type Frequency variation type for the AC sweep.	TYPE	LIN	-	[LIN, OCT, DEC]
Number of frequency points Total number of number of frequency points for the AC analysis	NPOINTS	0	-	[0, +INF]
Start frequency Starting frequency for the AC Analysis	FSTART	0	Hz	[0, +INF]
Stop frequency Stopping frequency for the AC analysis	FSTOP	0	Hz	[0, +INF]

DC

DC tab contains parameters to perform DC analysis.

Name and description	Symbol name	Default value	Units	Value range
Source name Name of the voltage/current source for DC sweep	SOURCE	-	-	-
Start value Starting DC value of the voltage/current source	START	0.0	V, A or °C	[0, +INF]
Stop value Stop value for the DC sweep	STOP	0.0	V, A or °C	[0, +INF]
Increment value DC increment value for the voltage/current source	STEP	0.0	V, A or °C	[0, +INF]

Options

Options tab contains parameters for setting simulation options.

Name and description	Symbol name	Default value	Units	Value range
Reference temperature Reference temperature for the simulation	TNOM	25.0	°C	[-INF, +INF]
Minimum conductance Minimum conductance allowed	GMIN	1e-12	-	[0, +INF[
Absolute tolerance Absolute error tolerance during DC and transient analysis	ABSTOL	1e-8	-	[0, +INF]
Relative tolerance Relative error tolerance for the solving variables from iteration to iteration	RELTOL	1e-3	-	[0, +INF]
Number of significant digits Number of significant digits to be printed in the output results	NUMDGT	0.0	V or A	[0, +INF]
Cache all mode shapes Option to cache or save results of all mode shapes for post-processing	CacheAllModeShapes	1	-	[0, 1]

Name and description	Symbol name	Default value	Units	Value range
Use Intel MKL When enabled and set to 1, the Intel MKL sparse matrix (SM) solver will be used, otherwise the default UMFPACK SM solver will be used.	USEMKL	1	-	[0, 1]

Control

Control tab contains parameters that controls the simulator operations.

Name and description	Symbol name	Default value	Units	Value range
Verbosity The level of details printed to the output	Verbosity	LOG	-	[SILENT, LOG, DEBUG]
Temperature control Simulation option for temperature dependency of the circuit.	TempControl	TEMPCIR	-	[NOTEMPCIR, TEMPCIR]
Add DC offset to PWL transient source function Enable (1) DC offset to be added to a transient piece wise linear (PWL) source	AddDCtoPWL	0	-	0, 1
Job control for all monitors When set to 1 or 2, monitors will be generated for all electrical and optical nodes. Option 2 will also generate monitors for all nodes contained within sub-circuits. NOTE 1: For large circuits the number of monitors may be very high. NOTE 2: This control function is independent of manually inserted probes (monitors).	JobControl AllMon	0	-	0, 1, 2

Parameters

Parameters tab contains global parameters set by users for any specific design.

Libraries

Libraries tab is used for including device model files and external Netlist files to the current design.

Name and description	Symbol name	Default value	Units	Value range
Libraries Model libraries path list	-	Electrical.libx; Optical.libx; Optoelectronic.libx; Subcircuits.libx	-	-
Include files External Netlist files to be included	-	-	-	-

Optiwave
7 Capella Court
Ottawa, Ontario, K2E 7X1, Canada

Tel.: 1.613.224.4700
Fax: 1.613.224.4706

E-mail: support@optiwave.com
URL: www.optiwave.com