

DIGITAL INDUSTRIES SOFTWARE

Siemens EDA's full-flow portfolio helps engineers achieve optimum IC design verification efficiency



Contents

Introduction	3
S-Edit	3
Op Amp design	4
Simulation Setup	4
General options	6
Hierarchy priority	7
Parameters	7
DC OP and sweep analysis	8
Running a simulation	9
Simulation netlists and log files	10
DC sweep simulation results in EZwave	11
Results back annotation	13
Seamless EZwave calculator integration	14
Monte Carlo simulation setup and results	16
Stability analysis	18
AC analysis	19
PVT simulation	20
Transient analysis	21
Conclusion	23

Introduction

Designers demand an integrated set of tools that run seamlessly from front-end to back-end design. The Siemens EDA full-flow portfolio provides the tools to perform all essential tasks required in an analog/mixed-signal integrated circuit (IC) design. A quick overview of the front-end flow using the S-Edit schematic capture environment will be covered in this white paper, followed by a more detailed description and steps for using the Analog FastSPICE[™] (AFS) platform simulator to go through the verification of a basic amplifier design.

S-Edit

S-Edit seamlessly integrates with all Siemens simulator offerings through an intuitive and highly customizable GUI. S-Edit is very easy to use out of the box, eliminating the steep learning curve often required when switching tools. The comprehensive and customizable toolbar at the top of S-Edit enables easy access to many design entry shortcuts and accelerates productivity as there is no need to look under complicated sub-menu and drop-down options. The libraries and cells associated with the design, the command window, and properties associated with an instance or test bench are conveniently located in S-Edit and can be detached and easily reconfigured to suit any work style or display area.



Figure 1: S-Edit schematic capture environment.

Op Amp design

A simple schematic is drawn in S-Edit for an eight transistor, two-stage operational amplifier (op amp) design implementation. The dummy devices are drawn at the bottom to construct a complete rectangular, common centroid layout for the NMOS current sinks.

A simple DC test bench for an output compensated amplifier in unity-gain feedback configuration is shown in figure 3 for the op amp design.



Figure 2: Schematic of a two-stage operational amplifier.



Figure 3: DC test bench.

Simulation Setup

Simulation Setup

The Simulation Setup allows the user to set up options and simulation test benches for various types of analysis and can be launched using the relevant toolbar button as shown in figure 4.



Figure 4: Menu icon to launch Simulation Setup menu.

From the Simulation Setup menu, test benches can be created using the Sim test bench pulldown. Once a simulator is selected, all relevant simulation options and analysis on the GUI adapt accordingly. S-Edit supports three simulators: T-Spice, Eldo™, and AFS. For this design several DC simulation test benches have been created. A schematic test bench can be associated, with several different simulation test benches, all created, renamed, copied and saved under this menu.

🕡 Setup AFS Simu	lation of cell 'OpAmp_D	C_TB'					
Sim testbench:	DC_SWEEP_AFS DC_OFFSET_AFS		_	•	Simulator: Analog Fast	SPICE (AFS)	•
General Simulation Options	DC_UP_AFS DC_OP_PVT_AFS DC_SWEEP_AFS				l I Directory Names		
Netlisting Options	DC_SWEEP_PVT_AFS				Name		RESUL
Additional Comman	<rename> <copy></copy></rename>				n Results Folder Simulation Results		RESUL
Spectre Options	<delete></delete>			File Sear	tre Models and Includes ch Path	/	PDK/Ge
Sumphony Optio	ins			Include F	Files		
DC Operating Po	oint Analysis			Library F	i les Library Files	'Ge	eneric_
Transient Analys	Transient Analysis				Spectre Include Files		
 Monte Callo Ana DC Sweep Anal 	uysis Vsis		8	Simulat	or Setup		

Figure 5: Simulation test bench setup for DC Sweep analysis.

It's worth noting that there are no hidden options or popup menus, and everything is shown on each corresponding sub window in the Simulation Setup. For example, the Simulation Options sub menu for AFS is shown in figure 6. Selecting an option shows a brief description from the user manual at the bottom. But let's look at each page analytically.

	-				
eneral	Simulation Options	<u>^</u>			
imulation Options	Accuracy				
letlisting Options		C Default			
lierarchy Priority	Performance/Accuracy Options	C Performance			
idditional Commands	And the Destination of Annual Annua	CArshoole			
arameters	Arsmode Performance/Accuracy Tuning	4 - moderate			
pectre Uptions	Enable RC Reduction	L			
Sumphany Onting	RC Reduction Max Frequency	1e+12			
DC Operating Date Analysis	Multithreading & Multi-Core				
Transient Analysis	Number of Threads per Process	1			
Monte Carlo Analysis	Number of Parallel Processes	1			
E DC Sweep Analysis	Temperature Simulation Options				
AC Application	Simulation Temperature (deg. C)				
Noise Applyin	Reference Temperature (deg. C)				
Stabilitu Analusis	Tolerance Simulation Options				
Transfer Function Analysis	iabstol	1e-12			
Temperature Sween	reitol	0.001			
Parameter Sween	vabstol	1e-06			
Corners	Convergence Simulation Options				
] DC Mismatch Analysis] S-Parameter Analysis] Harmonic Balance Analysis] HB AC Analysis] HB Noise Analysis	Homotopy	All C grin stepping Source stepping Damped pseudotransient C Pseudotransient C None			
] PSS Analysis	amin	1e-12			
] PSS AC Analysis	aminda				
] PSS Noise Analysis	Nodeset Series Resistance	1			
PSS Stability Analysis	Component Simulation Options				
PSS Transfer Function Analysis	Device Scale	1			
	Technology Scale Eactor				
	Device Noise Setup				
	Turn Noise Off for Linear D < Noise Date				
	Tanna instance All Instances (Natist and E	OK' Allow industrial device poice us impicu			
	Eachie Mains Constantias for Constitution	DK, Allow Individual device hose via briosy			
	Disable Moise Generation for Specific Instance	.05			
	Disable Noise Generation for Specific Instant	les .			
	Simulation Options				
	Maximum Warnings	5			
	Remove Zero Valued Voltage Sources	2 - Matrix Optimization			
	Enable Checklimit Statements	R			
	Assert Violations File	<u>×</u>			

Figure 6: Simulation Setup menu with help.

General options

The general options are where simulation netlist results and related files are generated and stored. Industry-standard models are supported and the definitions to the libraries are included on this page. This is a simple DC simulation test bench using typical Spice models of internal Siemens EDA generic process design kits (PDK). The ability to save all simulation results, time-stamped accordingly, or the ability to keep overriding existing ones is set on this menu.

	General File and Directory Names	
	Spice File Name	/RESULTS/\${Design}/\${Cell}/\${CurrentTestbench}/\${Cell}_\${CurrentTestbench}.scs
	Keep All Simulation Results	Tell}/\${CurrentTestbench}/
	File Search Path	/PDK/Generic_250nm/Models
	Library Files	'Generic_250nm.lib' TT
	spectre cibrary mes	
	Spectre Include Files	
Ξ	Simulator Setup	
	Simulator Command	
	Additional Command-Line Options	
	Post-Process Command	
Ξ	Results Viewing	
	Show Waveforms	After
	EZwave Tcl File	
	Enable Waveform Probing	
	Subcircuit Probe Level	

Figure 7: General setup for simulation.

Hierarchy priority

The hierarchy priority page will supersede the hierarchy setup in the schematic (selected by right clicking on an instance to select the view). The actual view type can be different on multiple instances of the same cell throughout the schematic hierarchy and can be easily selected by right-clicking on the schematic directly. The default hierarchy priority is Schematic, SPICE, Verilog-A, Verilog-AMS, Verilog, VHDL-AMS and VHDL.



Figure 8: Hierarchy priority menus

Parameters

The Parameters page is where all design variables and parameters are entered. You can easily enter a new parameter by clicking on the corresponding button and filling out at a minimum the Name and Value fields. Holding down the control key when pressing the same button automatically imports all variables found in a design.

White Paper - Siemens EDA's full-flow portfolio helps engineers achieve optimum IC design verification efficiency

	Simulator Analog FartSPICE (AFS) Run with no analysis chose
- Burnatur	× c
- Parameters	
vip_oc	
vip_nign	
biss.	0.7
doad	10
442-5	25
	Add Parameter Name: Name: Value: Cancel Description:

Figure 9: Menu to set up parameters for simulation.

DC OP and sweep analysis

All simulator related and supported analyses options can be selected through the checkboxes on the Simulation Setup GUI and stored in various simulation test benches. In figure 10, a simple DC OP simulation and DC sweep analysis are carried out. The positive input voltage of the amplifier is swept from 0.4 volts to 0.8 volts in steps of 100 μ V, ultimately checking to see if the N2 tail current sink is pushed into a linear region at low voltages or not.

The amplifier's tail current sink, N2, as shown in figure 2 must be in saturation to be a proper current sink. If N2 goes into linear region, it acts as a resistor (not a current sink) and the amplifier performance starts to degrade. N2 is in saturation when N2_Margin=Vds-Vdst > 0 (most designers allow enough margin, so at least 50mV instead of 0). By sweeping the positive input voltage of the amplifier (the amplifier is in unity gain feedback) from a very low voltage to something higher, we can quickly identify the minimum input voltage we can get away with for N2 to be saturated.

Sim testbench: DC_SWEEP_AFS	Simulator: Analog Fa	stSPICE (AFS)
Seneral	DC Sweep Analysis	
Simulation Options	DC Sweep	
Netlisting Options	DC Analysis Name	VIN SWEEP
lierarchy Priority	DC Sweep Variable	1000
dditional Commands	Sween Variable	Design Variable Sweep
arameters	Darion Variable	vin de
pectre Options	Composed Name	10_0t
esuits	Component Name	
] Symphony Options	Component Parameter	
DC Operating Point Analysis	Model Name	
] Transient Analysis	Model Parameter	
Monte Carlo Analysis	DC Sweep Range	
DC Sweep Analysis	Sweep Type	Linear-Step
] AC Analysis	Start Value	0.4
Noise Analysis	Stop Value	0.8
Stability Analysis	Step	0.1
] Transfer Function Analysis	Number of Points	0.1
] Temperature Sweep	List of Points	0.4
Parameter Sweep	- DC Sween Ontions	
Corners	Husterack Super	
DC Mismatch Analysis	DC Schools Sheep	
3 S-Farameter Analysis	Under Starte Strate Ones Salution	
1 UP AC Application	write Pirst Sweep Step Solution	
J HD AC Analysis 1 HD Noise Analysis	Initial Condition	
DEC Analysis	Set of Initial Conditions	None
1 PSS AC Analysis	Initial Condition File	
I PSS Noise Analysis	Additional Parameters	
1 PSS Stahiltu Analusis	Additional Analysis Parameters	
PSS Transfer Function Analysis	DC Estimate File	file naths. This revnests is evaluated

Figure 10: DC Sweep Analysis setup.

Running a simulation

When all simulation test bench customizations are finished, you can simply run a simulation or click OK to save the changes. The simulation netlist can also be generated and viewed before a simulation is run simply by clicking on the "T" button in the toolbar. The simulation can also be launched from the S-Edit toolbar.



Figure 11: Start simulation icon.

Simulation netlists and log files

Regardless of the format choice of the simulation models, S-Edit exports a netlist and then launches the AFS simulation in the background. While the simulation is running, the simulation status and AFS log file are printed in the command window. When the simulation is finished, links to the netlist, log file and results directory are shown in the command window alongside the simulation statistics. Clicking on the netlist opens the simulation netlist in the S-Edit text viewer. While clicking on the log file link, the log file opens in a separate text viewer tab.



Figure 12: Simulation netlists and log files.

DC sweep simulation results in EZwave

EZwave[™] is the default waveform viewer and results postprocessor for AFS and is immediately launched after an AFS simulation finishes. Expanding the DC sweep simulation database shows all available results and measurements taken from the simulation. Plotting measurement N2_margin across the input voltage sweep shows that the tail current sink N2 is saturated for any input voltage higher than approximately 521µV. As the input voltage gets lower, so does the tail voltage of the amplifier (i.e., the drain of N2) which reduces the Vds of N2 and hence N2_Margin. Visual Distress Signal = Drain-Source voltage, Vdsat = Saturation voltage, gram = Transconductance. It's nice to show what happens to the gm of N3 (this is the amplifier input device) when the input voltage (vip) is too low, and as a result N2 is in the linear region (i.e., not a good current sink, if at all a sink). N2 is in the linear region when N2_Margin < 0V or so.



Figure 13: DC sweep simulation results in EZwave.

EZwave comes with an advanced calculator for the postprocessing of results. The calculator can be accessed through the menu or relevant toolbar button. It features loads of built-in functions, such as Measurements, Signal Processing, and Statistical, to mention a few. A waveform can either be dragged and dropped into the calculator from the database navigator or directly from a graph.



Figure 14: EZwave Waveform Calculator.

Expressions can easily be constructed, in this case the output voltage minus the input voltage to get the systematic offset of the amplifier across the DC Sweep, and evaluated. The output voltage and the offset voltage graphs are appended as shown in figure 15, where name and other visual effects can easily be changed to the desired ones.



Figure 15: Output voltage and systematic offset waveforms.

Results back annotation

Once a simulation is run, the DC OP voltage or current can be back annotated on the schematic. This can be done using the toolbar dropdown menu and selecting voltage or current. The node voltages are back annotated, and the currents show that this op amp consumes a static current of 13μ A.



Figure 16: Results back annotation.

In addition, small-signal parameters can be back annotated using this toolbar icon. All relevant parameters and device region information are shown for the op amp. All available data set results associated with the simulation test bench can be accessed and switched to using this toolbar icon.





Figure 17: Small-signal parameters back annotation.

Seamless EZwave calculator integration

In the Simulation Setup GUI launched directly from the schematic test bench, there is a new Results pane which integrates the EZwave calculator seamlessly for any advanced expression and output measurement construction during design. Net voltages and node currents to be saved or plotted are easily selected using the toolbar icons and stored under the corresponding signals section in the Results pane.



Figure 18: Seamless EZwave calculator integration.

After a simulation is run, EZwave pops up and plots the expressions from the Results pane that evaluate a waveform. Scalar results, like the phase margin or the DC loop gain, are also evaluated and shown in the same pane.



Figure 19: EZwave plots from calculator expressions.

An expression can be sent to and from the calculator for further editing or be manually modified in the Results pane. Right-clicking on an expression and selecting recalculate evaluates a new measurement. All scalar expressions and waveforms can be re-evaluated and plotted again at any point, providing that the results are present.



Figure 20: Menu to send expressions to EZwave for recalculation.

Monte Carlo simulation setup and results

A Monte Carlo simulation using the initial DC schematic test bench is set up using the corresponding simulation test bench selected under the same Simulation Setup menu. A mismatch on a Monte Carlo simulation of a 100 runs is set up and ready to be run. Note the input voltage offset measurement and histogram command in the Calculator menu from the Results page.

ieneral				<u> </u>	Simulator: Ar	alog FastSPIC	e (AFS) 💌	
- Inform On Construct				Ionte Carlo Analysis				
mulation Uptions			M	lonte Carlo Analysis Name	mc			
etlisting Options			N	umber Of MC Runs	100			
ierarchy Priority			Fi	irst Run				
dditional Commands			V.	ariation	All			
arameters			D	o Nominal Run	2			
pectre Options			S	eed				
esults			S	ampling				
] Symphony Options			S	ampling Method	Standard Monte Carlo			
DC Uperating Point A	Analysis		N	umber Of LHS Bins				
J Transient Analysis				C Application				
] AC Analysis] Noise Analysis	Enable	Plot	Name	Expression		Plot Options	🙄 🗙 🛧 🖌 Value	
Stability Analysis	R		Vofs_wf	wf(" <sim op="">vout")-wf("<sim op="">vip")</sim></sim>	wf(" <sim op="">vout")-wf("<sim op="">vip")</sim></sim>		Wave	
			Vofs_data	[i[1] for i in wftodata(Vofs_wf)]			[-28.49	
		M	hist_vofs	histogram(Vofs_wf, normalized = 0)	÷		Wave	

Figure 21: Monte Carlo simulation setup and calculator expressions.

In this simulation the random offset of the amplifier is being characterized. That is the input minus the output voltage, which in theory (for a very high gain op amp) should be zero. The random offset mainly comes from the amplifier's input devices (i.e., N3 and N4) in figure 2. On the schematic level (without any layout or real-world effects) for a fixed power budget, this is mostly determined by the input device's operating region (must be in saturation and ideally moderate or weak inversion) and size.

When the simulation finishes, EZwave appends the Monte Carlo results under the Currently Opened Databases menu and the histogram plot of the random offset voltage of the amplifier is displayed. A Gaussian approximation curve and all relevant statistical details and measurements are also shown in the graph. These details can be customized under the EZwave display preferences setup.



The offset voltage can also be plotted across the Monte Carlo runs by simply doubleclicking on the measurement as shown in figure 23.

- - - A

VOPOR

Interaction of the second second

Figure 23: Offset voltage plot.

Workspacel ×

stol gau

EZwave 2021.2 update2

下生 単成 時間 でる

70.0

🔨 Wave:2

Figure 22: Histogram plot of the random offset voltage of the amplifier.

Stability analysis

Stability analysis is an important simulation for an amplifier. It shows how stable your design is across any perturbations. If the design is less stable, any signal perturbation affecting the amplifier produces ringing effects that can be seen in the transient response. The more stable a design is, the slower it gets; generally, therefore we need a nice balance. We use metrics to measure the stability performance of an amplifier, and these are the phase margin and gain margin. A good amplifier has 60° of phase margin and anything over 10dB of gain margin.

A test bench for a loop-stability analysis of the op amp where the loop is broken using a standard DC voltage source of zero volts is shown in figure 25. The source probe name determines where the loop is broken, and the rest of the stability page is the same as that of an AC analysis. A stability simulation test bench across a specific PVT corner set has been defined for the process, output capacitance, voltage, and temperature variables.

After running the PVT simulations, the stability results are appended at the bottom of EZwave as before. AFS automatically exports all required stability summary measurements under the measures section and the phase margin measurement is plotted across the corner run number.



Figure 24: Stability analysis setup.



Figure 25: Phase margin measurement.



Figure 26: Stability analysis magnitude and phase waveforms.

Also, the classical loop-gain magnitude and phase waveforms can be accessed and plotted for all 135 PVT corners.

AC analysis

There is a more advanced simulation that tests the circuit's ability to reject power supply noise. An ideal amplifier only has gain from its input to its output; any other perturbation should be rejected. If the power supply varies, the output of the amplifier should remain intact. In practice, it never does. The way to measure the amplifier's performance in that regard is by using a metric called Power Supply Rejection Ratio (PSRR). PSSR is defined as the small-signal gain from the power supply in

unch: [PSRR_PVT_AFS	- ac a	alucic			 Simulator: Analog FastSI 	PICE (AFS) 💌 🕅 Run i	with no analysis chosen		
		alucie Nama	AC.						
Options	Erecu	Erequency Sween							
stions	Sugar	Turne	Logwithmic)acada					
ommands	Start 8	requency	1	/00000					
ione	Ston F	requency	100M						
y Options	Numbe	v of Fragmancian	Firequescies 100		Additional SPICE Commands				
iting Point Analysis Analysis	Freque	ancy List	100		.WEASURE AC PSRR DC FIND '-vdb(vout)' AT=1				
arlo Analysis	E State	-File			.MEASURE AC PSRR_MIN	RIN '-vdb(vout)			
ip Analysis	DC Ed	imate File			.MEASURE AC PSRR_MIN_	FREQ WHEN par('-vdb(v	out)')=PSRR_MIN		
alycis	Like Pr	evious Operating Point	Π		-				
Function Analysis	🗆 Initia	Condition							
ture Sweep	Set of	Darameter Sweep							
a oweep	Initial	Parameter Sweep							
atch Analysis	Skip D	Parameter 1 Sweep	Definition (swep	ot for each va	CLOAD_SWEEP Parameter Para				
Balance Analysis	🗆 Addit	Sweep Name		CLOAD_					
valgtio Azolunis	Additi	What to Sweep		Paramete					
lycis	_	Sweep Variable		cload					
e Analysis		Instance Name							
olity Analysis safat Exection Analysis		Model Name							
		Sweep Type		List of	Corner Simulations				
		Start Value							
		Stop Value			Variable	Turne	DUT (4E)		
		Step			Variable	Type	FVI (45)		
		Number of Points			Generic_250nm.lib	Library	TT SS FF SF FS		
		List of Points		200p.d	vdd2v5_dc	Parameter	2.3 2.5 2.7		
		Data Tabla		2000	temp	Temperature	-40 27 125		
		Data Table				1			
	- 1			_					

Figure 27: Setup Simulation for PSSR.

question to the output, divided by the loop gain of the amplifier (i.e., input to output). For an amplifier with high loop gain (A β >>10), this ratio turns out to be the same as the small-signal output voltage divided by the small-signal power supply perturbation voltage. Setting the latter to an AC signal of a magnitude of 1 simplifies the calculation of PSRR to be equal to the output voltage of the amplifier.

The AC analysis test bench in figure 28 has a small signal applied on the power supply VDD2V5 of the op amp in closed-loop mode. The output voltage across a frequency sweep provides the PSSR of the op amp for a frequency up to a decade or so below its gain bandwidth. A similar PVT corner is defined as the previous PVT corner set, with the exception that the output capacitance is swept under the Parameters Sweep analysis; therefore the corner setup consists of only 45 combinations.

The Setup Simulation menu supports an Additional Commands page for those users who need to enter additional Spice/Spectre commands or have test benches already setup with probe and measurement commands. For this example, three. MEASURE commands have been added under the additional commands page for this test bench.

PVT simulation

After running the Process Voltage Temperature (PVT) simulation, the AC results are displayed in EZwave. The output voltage can either be plotted directly from the results or cross probed from the schematic as follows. In S-Edit, a voltage or current can be cross probed to EZwave by simply clicking on the corresponding probe icon on the toolbar and selecting the desired node on the schematic. EZwave then plots the probed net, in this case, the output voltage magnitude and phase.



Figure 28: Probe nets to plot waveforms.

In addition, the minimum PSRR measurement across the AC sweep can be plotted against the parameter sweep C-load for all PVT conditions. This same measurement can be plotted against a different x-axis by simply right-clicking on it in the results browser and selecting, for example, corner_group1 as shown in figure 29.



Figure 29: Power Supply Rejection Ratio waveform results.

Transient analysis

Finally, figure 30 shows a test bench for a transient analysis where an input voltage pulse is applied to the positive input of the op amp in the unity gain feedback mode. The Transient Analysis section of the simulation test bench shows the stop time, accuracy, and maximum time step to be used. A similar PVT corner set is used, including two different values of the output capacitance, resulting in a total of 90 corner simulations. Measurements are calculated during simulation to ultimately calculate the worst-case overshoot and undershoot values using the built-in calculator functions.

				-			_			
ieneral		_ Signals								
mulation Options	Transient Analysi	is						🔁 🗙 🗲		
etlisting Options	ons Transient Analysis Name TRAN			TRAN						
ierarchy Priority	Stop Time 50u		50u		motor Emoor					
dditional Commands		CLiberal			Parameter 1 Sweep Definition (swept for each value of Sweep 2) (Inner Most Sweep					
arameters	Accuracy Defaults (e	rrpreset) Moderate C Conservative		efaults (errpreset) Moderate		ccuracy Defaults (errpreset) O Moderate Sweep Name		Sweep Name vdd2v5_sweep		(
pectre Options				What	t to Sweep	Parameter	ameter			
esults	Maximum Time Step	ln			Swee	sp Variable	vdd2v5			
] Symphony Options 🗧 🗧					Insta	ince Name				
DC Operating Point Analysis	1				1900s	JI Name	Linear Over			
Transient Analysis					Start	type Value	2.3			
Monte Carlo Analysis					Stop	Value	2.7			
DC Sweep Analysis					Step		0.2			
] AC Analysis					Numb	ber of Points	0.2			
Noise Analysis		Calculator			List o	if Points	2.3			
Stability Analysis					Data	Table				
Transfer Function Analysis			_	_	_					
-		Fashia	Plot	Mamo		Evorecsion		Dist On Value		
] Temperature Sweep		Enable	100	Ivallie		Expression 1		Ploc Op Value		
Temperature Sweep Parameter Sweep		Enable		Overshoot	max(overshoot(wf(" <sim tran="">vout"), x_end</sim>	= 19.5u, option = 'WF')) 🚑	[1		

Figure 30: Simulation Setup for transient simulation.

The transient analysis results are displayed under Currently Open Databases in the EZwave results browser. The output voltage is plotted for all PVT combinations as shown in figure 31, and the overshoot and undershoot measurement results plotted as histograms in figure 32.



Figure 31: Output voltage is plotted for all PVT.



Figure 32: Overshoot and undershoot histograms.

Conclusion

Greater efficiency in analog design verification can now be achieved using our enhanced inter-tool communication in Siemens EDA's full-flow design environment. Integrating S-Edit, AFS, and EZwave helps engineers achieve an optimum IC design outcome. The tight integration with the EZwave calculator allows for easy construction of advanced expressions and pre-simulation or post-simulation measurement modifications that are all stored in states under a schematic test bench.

Siemens Digital Industries Software

Americas: 1 800 498 5351 EMEA: 00 800 70002222 Asia-Pacific: 001 800 03061910

For additional numbers, click here.

About Siemens Digital Industries Software

Siemens Digital Industries Software is driving transformation to enable a digital enterprise where engineering, manufacturing and electronics design meet tomorrow. Xcelerator, the comprehensive and integrated portfolio of software and services from Siemens Digital Industries Software, helps companies of all sizes create and leverage a comprehensive digital twin that provides organizations with new insights, opportunities and levels of automation to drive innovation. For more information on Siemens Digital Industries Software products and services, visit <u>siemens.com/software</u> or follow us on <u>LinkedIn</u>, <u>Twitter</u>, <u>Facebook</u> and <u>Instagram</u>. Siemens Digital Industries Software – Where today meets tomorrow.

siemens.com/software

© 2021 Siemens. A list of relevant Siemens trademarks can be found <u>here</u>. Other trademarks belong to their respective owners.

84151-D6 12/21 K