

# Allegro Current Sensor LTSpice Models

Interactive LTspice Model of Allegro Current Sensors

#### Kasey Hampton – Applications Systems Engineer Manchester, NH

## Table of Contents

Table of Contents	.1
Revision History	.1
Abstract	.1

#### Revision History

Revision	Date	Comment	Responsible
1.0	February 2020	Initial Release	K. Hampton
Table 1: Pavision History			

Table 1: Revision History

#### Abstract

Behavioral models for Allegro current sensors were developed to be simulated in LTSpice. The document explains how to generate a model symbol from the provided netlist, how to change the model's sensitivity (among other device specific parameters), and how to create a simple test bench to assess the functionality of the model.

#### Procedure

The LTSpice netlist for each Allegro current sensor is located on the device specific webpage. If the user does not already have <u>LTSpice</u>, the SPICE simulation software is free for download. Take the following steps to download, import, and simulate the provided netlist for each Allegro current sensor model.

1. Begin by opening the provided netlist in LTSpice. This file is a \*.lib file. The file name will begin with ACS and include the three- or five-digit part number.

Name	Date modified	Туре	Size
ACS732_Schematic	2/18/2020 2:27 PM	LIB File	2 KB

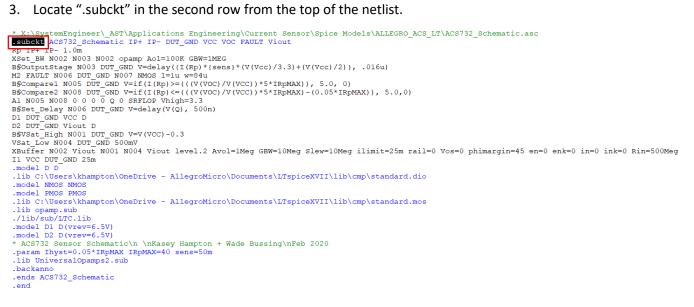


How do you want to open this file?			
Keep us	ing this app		
₽ s	PICE Simulator w/ Schematic Capture		
Other o	ptions		
Look for an app in the Microsoft Store			
More apps $\psi$			
Always use this app to open .lib files			
	ОК		

2. The user may see a warning message about a non-existing transient analysis after opening the netlist. Click "OK".

LTspice XVII	×
No analysis request found.	
ОК	L,

3. Locate ".subckt" in the second row from the top of the netlist.



4. Right click ".subckt". Select "Create Symbol".



			\Applications Engineering\Current Sensor\Spice Models\ALLEGRO_ACS_LT\ACS732_Schematic.asc
		ematic	IP+ IP- DUT_GND VCC VOC FAULT Viout
Rp 🄀 <u>R</u> un		Ctrl+R	
XSe Halt			pamp Aol=100K GBW=1MEG
D30 -			ND V=delay({I(Rp)*{sens}*(V(Vcc)/3.3)+(V(Vcc)/2)}, .016u) 07 NMOS 1=1u w=84u
M2 Marc BSC	ching <u>W</u> aves		<pre>// MNOS 1=1u ==54u P=if(I(kp))=(((V(VC)/V(VCC))*5*IRpMAX)), 5.0, 0)</pre>
BSC Visib	ole Traces		<pre>1(1(R)/-((V(VGC)/V(VCC))*3*IRPMAX), 5.0, 0)</pre>
	v SPICE Error Log		SRLOP Vhidh=3.3
B§S	Since entring		V=delay(V(Q), 500n)
D1 👗 Cut		Ctrl+X	
D2 BSV	у	Ctrl+C	V=V (VcC) -0.3
VSa 📄 Paste	e	Ctrl+V	DmV
XBU	o	F9	N004 Viout level.2 Avol=1Meg GBW=10Meg Slew=10Meg ilimit=25m rail=0 Vos=0 phimargin=45 en=0 enk=0 in=0 ink=0 Rin=500Meg
.mc ( Redo	o S	hift+F9	
			neDrive - AllegroMicro\Documents\LTspiceXVII\lib\cmp\standard.dio
.li .mc		cann	
.mc <u>O</u> per	n .inc/.lib File		new weeks and the second s
li Drea	ate <u>S</u> ymbol	I	in the second seco
	o me Edit	•	
.mc .mc <u>G</u> ene	erate Expanded Listi	ing	
* A			chn hnKasev Hampton + Wade BussinghnFeb 2020
.pa Eile		•	IRpMAX=40 sens=50m
	t Window		
.backanne			
.ends ACS .end	S732_Schema	atic	
.end			

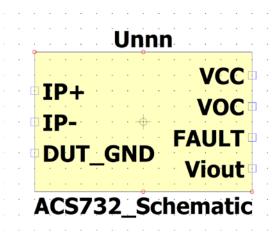
5. LTSpice will ask to automatically generate a symbol file. Select "Yes".

LTspice XVII	×
Do you wish to automatically create a symbol that will netlist against the subcircuit "ACS732_Schematic" and her 7 ports?	
Yes No Cancel	

6. The generated symbol file will be \*.ASY file type. The name of the symbol file will have the same name as the netlist: ACS followed by the three- or five-digit part number. The file should automatically be saved in the following default file location:

<u>C:\Users\User\Documents\LTspiceXVII\lib\sym\AutoGenerated</u>.

(Note the user can also find the symbol library folders here: <u>C:\Program Files\LTC\LTspiceXVII\lib\sym</u>. If the symbol does not default save to the documents folder above, save here.)





- 7. To place and use the generated current sensor model, open a new schematic by selecting the "New Schematic" icon  $\blacktriangleright$  (or "File"  $\rightarrow$  "New Schematic" or Ctrl+N).
- 8. Save the new schematic file to a known local location with a file extension of .ASC.

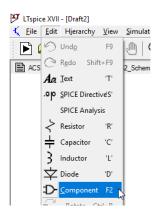
File name:	ACS732_Test_Bench.asc	
Save as type:	Schematics (*.asc)	N
		~

Before placing the current sensor block in the schematic, ensure that the "AutoGenerated" file location (or the chosen save location of the symbol) is a Symbol Search Path. Locate the Control Panel in "Tools"
 → "Control Panel". Navigate to the "Sym. & Lib. Search Paths" tab. Add

"C:\Users\User\Documents\LTspiceXVII\lib\sym\AutoGenerated" to the Symbol Search Path.

	🗗 Control Panel	×
		Y Internet
	💼 Compression 🥖 Save Defaults 👕 SPIG	CE Trafting Options
	H Netlist Options Sym. & Lib. Search	Paths 🔛 Waveforms
	Separate directories with semicolons	or new lines.
	Symbol Search Path[*]	
	C:\Users\khampton\Documents\LTspiceXVII	lib∖sym∖AutoGenerati
Simulate Tools Window Help	Library Search Path[*]	
🕛 🗎 🄁 Copy bitmap to Clipboard 🛛 🧮 🕻	e l	
Schemati	-	
Control Panel		
Color Preferences	[*] Setting remembered between program	m invocations.
🅎 Sync Release	Reset to Default Values	
Export Netlist		
	OK	Cancel Help

10. To place the generated symbol block, select F2 or "Edit"  $\rightarrow$  "Component".

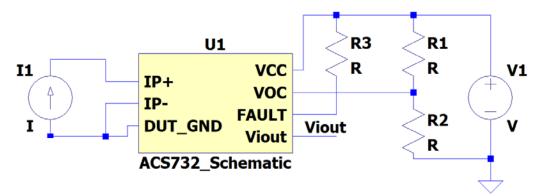


11. This will open the "Select Component Symbol" dialogue window. Open the "AutoGenerated" directory. Select "OK" to place the symbol.



🍠 Select Component Symbol	×	Select Component Symbol	×
C.\Users\khampton\OneDrive - Allegro [ADC] [References]	Drive - Allegro Micro \Documents \LTs ~ Open this macromodel's test fixture Micro \Documents \LTspice XVI/Nib \sym\ e2 LED	Top Directory: C:\Users\khampton\OneDrive - Allegro Micro \Docum	ents/LTs ~
[Alt JGGRO_ACS_LT]     [SpecialFunctions]       [AutoGenerated]     [Switches]       [DAC]     bi       [DAC]     bi2       [Data]     bv       [FiterProducts]     cap       [Optos]     current       [Optos]     diode       [PowerProducts]     e       <	f load FerriteBead load2 FerriteBead2 lonp g tline g2 merfet h nff ind nmos isO 16750-2 npn ISO 7637-2 npn2 SO 7637-2 NPA2	CS732 Schemate	

12. After placing the symbol block, the user can generate a system specific test bench in the schematic file. The test bench below, for example, represents the ACS732 typical application. Each component, including resistors and capacitors, current supplies, and voltage supplied, were placed by going to the Select Component Symbol (F2), and placing the default LTSpice component.





### **Changing Parameter Values**

The sensitivity of the current sensor, and thus the maximum current, is user programmable. To change the sensitivity of the sensor, right click on the symbol block. The "Component Attribute Editor" dialogue window will appear. Locate the "SpiceLine" attribute, the sixth attribute on the list. Add the following variables to the line: sens=VALUE IRpMAX=VALUE. The values of sensitivity and maximum current are per the device specific datasheet and the sensor used in the customer specific application. Double click the Vis. column to show the variable on the symbol.

l	🎔 Component A	ttribute Editor		×
	Open Symbol:	C:\Users\khampton\OneDri	ive - AllegroMicro\Documents\L	Tsp
	This	is the fourth attribute to appe	ar on the netlist line.	
	Attribute	Value	Vis.	^
1	Prefix	x		
2	InstName	U1	Х	
3	SpiceModel			
4	Value	ACS732	X	
5	Value2			
6	SpiceLine SpiceLine2	sens=50m IRpMAX=40	×	
		U1	OK	.:
	IP+		VCC	L
	IP-		Vout	
	TL		vout	
	DUT	_GND	Filter	þ

ACS724 sens=50m IRpMAX=40



#### Simulations

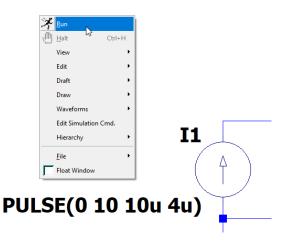
#### DC Analysis

Running a DC analysis allows the user to input a current step to the sensor and observe the proportional voltage output. To simulate and view the step response of the sensor, follow the steps below.

 Add a current supply to the schematic. Connect the current supply to the IP pins of the sensor by adding wires (F3). Right click on the current supply component. Select "Advanced". Using the advanced settings of the current supply, the user can customize various characteristics of the input current step including the rise time and on time of the current pulse. Click "OK".

	11 Independent Current Source - 11	×
	Functions	DC Value
	O (none)	DC value:
	PULSE(I1 I2 Tdelay Trise Tfall Ton Period Ncycles)	Make this information visible on schematic:
	◯ SINE(loffset lamp Freq Td Theta Phi Ncycles)	
	O EXP(I1 I2 Td1 Tau1 Td2 Tau2)	Small signal AC analysis(.AC)
	◯ SFFM(loff lamp Fcar MDI Fsig)	AC Amplitude:
	O PWL(t1i1t2i2)	AC Phase:
	O PWL FILE: Browse	Make this information visible on schematic: $\square$
	O TABLE(v1 i1 v2 i2)	Parasitic Properties
	I1[A]: 0	This is an active load:
	I2[A]: 10	_
	Tdelay[s]: 10u	Make this information visible on schematic: 🗹
	Trise[s]: 1u	
🕑 Current Source - I1 🛛 🕹 🗙	Tfall[s]: 1u	
	Ton[s]: 1m	
01	Tperiod[s]:	
DC value[A]:	Ncycles:	
Cancel	Additional PWL Points	
[ Advanced ]	Make this information visible on schematic:	
Advanced	Marke this information visible on schematic.	Cancel

2. Ensure all other components in the test bench are given values. Right click on a blank portion of the schematic and select "Run".

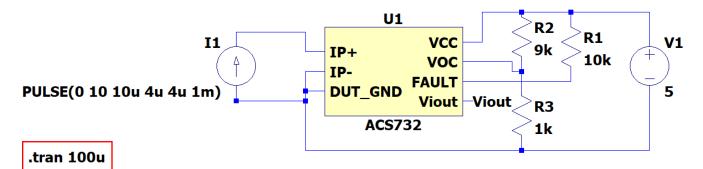


3. The "Edit Simulation Command" dialogue window will appear. Select the "Transient" tab. Enter a "Stop time". Click "OK".

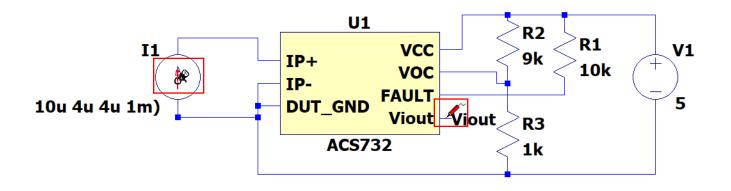


🄁 Edit Simulation Command					×		
Transient	AC Analysis	DC sweep	Noise	DC Transfer	DC op pnt		
	Perform a non-linear, time-domain simulation.						
	Stop time: 100u						
	Time to start saving data:						
	Maximum Timestep:						
	Start external DC supply voltages at 0V:						
	Stop simulating if steady state is detected:						
	Don't reset T=0 when steady state is detected:						
	Step the load current source:						
	Skip initial operating point solution:						
Syntax: tran <tstop> [<option> [<option>]]</option></option></tstop>							
.tran 100u							
	Car	ncel		0	к		

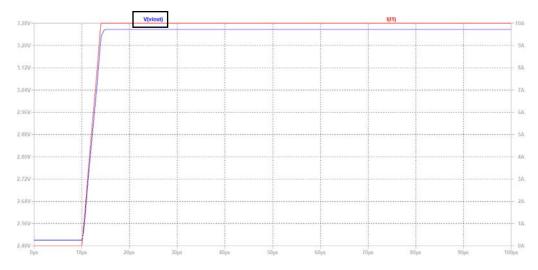
4. The transient simulation command will appear in the bottom left of the window.



5. Right click on a blank portion of the schematic and again select "Run". A simulation window will appear. To view the input step, hover over the current supply and click the supply when the cursor resembles an ammeter. To view the voltage output, hover over the Viout pin and click the supply when the cursor resembles a probe. Note: it is helpful to add labels to nodes that will be probed, i.e Viout, VCC, and the applied current step. Click F4 or "Edit" → "Label Net" to place labels. When the node is probed, the user input label name will appear as the name of the simulation curve.







#### AC Analysis

Running an AC analysis allows the user to simulate the frequency response of the current sensor. To simulate and view the frequency response of the sensor, follow the steps below.

1. Add a voltage supply to the schematic. Right click on the voltage supply component. Select "Advanced". Using the advanced settings of the voltage supply, the user can customize various characteristics of the power supply. Add a value to the "AC Amplitude" characteristic. Click "Ok".

	🤔 Independent Voltage Source - V1	×
	Functions  (none)  PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)  SINE(Voffset Vamp Freq Td Theta Phi Ncycles)	DC Value DC value: 5 Make this information visible on schematic:
	Sintervolace value in reg to finde in hebrids in hebrids)     EXP(V1 V2 Td1 Tau1 Td2 Tau2)     SFFM(Voff Vamp Fcar MDI Fsig)     PWL(t1 v1 t2 v2)	Small signal AC analysis(AC) AC Amplitude: 1 AC Phase:
	O PWL FILE: Browse	Make this information visible on schematic: ☑ Parasitic Properties Series Resistance[Ω]: Parallel Capacitance[F]: Make this information visible on schematic: ☑
😕 Voltage Source - V1 🛛 🗙		
DC value[V]: 5 Cancel Series Resistance[Ω]: Advanced	Additional PWL Points Make this information visible on schematic: 🗹	Cancel

- 2. Ensure all other components pins of the symbol are given values. If an analysis has already been run, take the steps below. If no simulation has been run, right click on a blank portion of the schematic and select "Run". This will open the "Edit Simulation Command" dialogue window where the user can chose and customize the desired simulation.
  - a. Right click the previous analysis command. The "Edit Simulation Command" dialogue window will appear.
  - b. Select the "AC Analysis" tab.
  - c. Input values for "Type of sweep", "Number of points per decade", "Start frequency", and "Stop frequency". Note clicking "OK" will NOT implement the analysis.

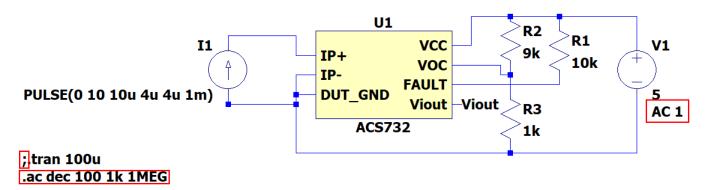


😕 Edit Simulation Command						;	×
Transient AC	Analysis	DC sweep	Noise	DC Transfer	DC op pnt		
Compute the small signal AC behavior of the circuit linearized about its DC operating point.							
	Type of sweep: Decade 🗸						
	Number of points per decade:			100			
	Start frequency:			1k			
	Stop frequency:			10MEG			
Syntax: .ac <oct, dec,="" lin=""> <npoints> <startfreq> <endfreq></endfreq></startfreq></npoints></oct,>							
.ac dec 100 1k 10MEG							
Cancel							

- d. Copy the .ac command and click "Cancel".
- e. The "Edit Text on the Schematic" dialogue window will appear. Here, the user should add a new line to the text window by typing Ctrl+M.
- f. Paste the copied AC analysis on the second line.
- g. Before clicking "OK", ensure the previous analysis is deleted or commented out. To comment out an analysis, place a semicolon (;) in front of the unused analysis.

	🗗 Edit Text on the Schematic:					
	O Comment Left	ication Font Size           V         1.5(default)           al Text	OK Cancel			
;—	tran 100u .ac dec 100 1k 1MEG		< 、			
	Type Ctrl-M to start a new line.					

3. The simulation commands will update and appear in the bottom left of the window.



4. Right click on a blank portion of the schematic and again select "Run". The simulation window will appear. To view the frequency response of the sensor output, hover over the Viout pin and click the



supply when the cursor resembles a probe. The frequency response will appear in the simulation window. Here, the user can measure the bandwidth of the sensor.

