Allegro Current Sensor LTSpice Models

Interactive LTspice Model of Allegro Current Sensors

Kasey Hampton – Applications Systems Engineer
Manchester, NH

Table of Contents

Revision History

Revision	Date	Comment	Responsible
1.0	**February 2020**	**Initial Release**	**K. Hampton**

Table 1: Revision History

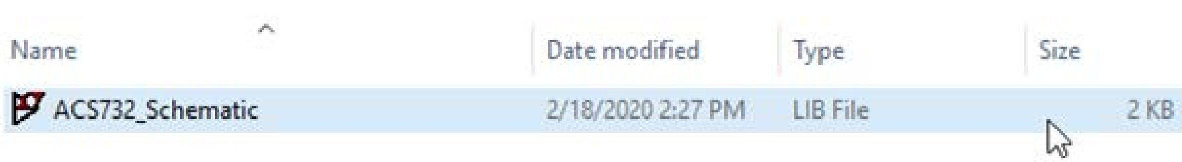
Abstract

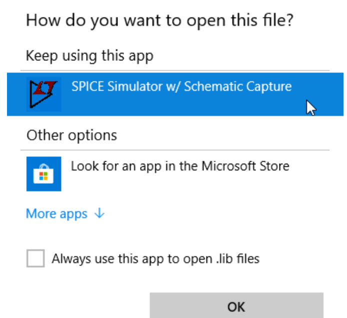
Behavioral models for Allegro current sensors were developed to be simulated in LTSpice. The document explains how to generate a model symbol from the provided netlist, how to change the model's sensitivity (among other device specific parameters), and how to create a simple test bench to assess the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

Procedure

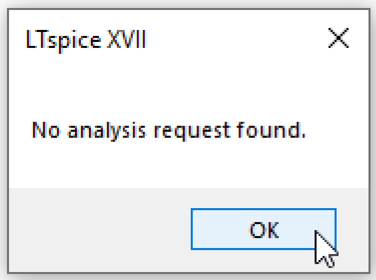
The LTSpice netlist for each Allegro current sensor is located on the device specific webpage. If the user does not already have LTSpice, the SPICE simulation software is free for download. Take the following steps to download, import, and simulate the provided netlist for each Allegro current sensor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





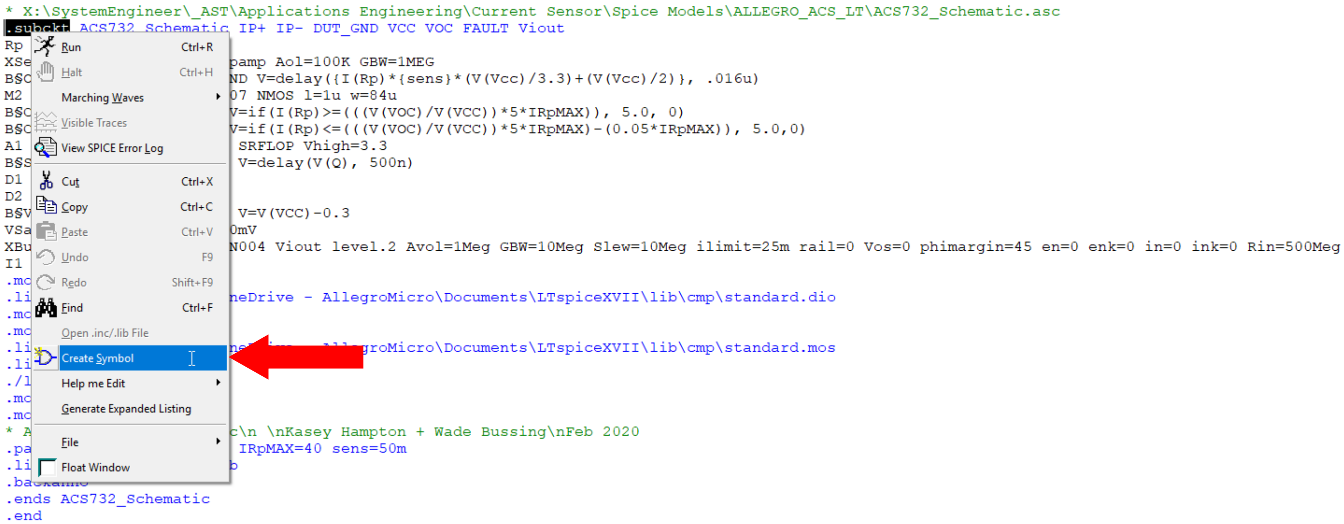
2. The user may see a warning message about a non-existing transient analysis after opening the netlist. Click "OK".



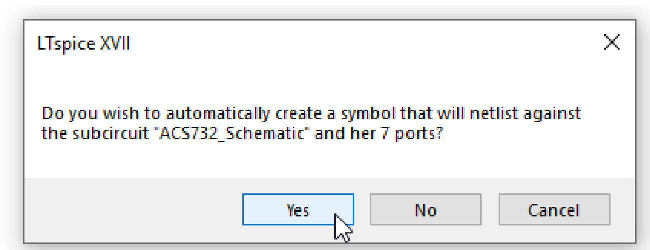
3. Locate ".subckt" in the second row from the top of the netlist.

```
* X:\SystemEngineer\_AST\Applications Engineering\Current Sensor\Spice Models\ALLEGRO_ACS_LT\ACS732_Schematic.asc
.subckt ACS732_Schematic IP+ IP- DUT_GND VCC VOC FAULT Viout
Rp IP+ IP- 1.0m
XSet_BW N002 N003 N002 opamp Aol=100K GBW=1MEG
B§OutputStage N003 DUT_GND V=delay({I(Rp)*{sens}*(V(Vcc)/3.3)+(V(Vcc)/2)}, .016u)
M2 FAULT N006 DUT_GND N007 NMOS l=1u w=84u
B§Compare1 N005 DUT_GND V=if(I(Rp)>=(((V(VOC)/V(VCC))*5*IRpMAX)), 5.0, 0)
B§Compare2 N008 DUT_GND V=if(I(Rp)<=(((V(VOC)/V(VCC))*5*IRpMAX)-(0.05*IRpMAX)), 5.0,0)
A1 N005 N008 0 0 0 0 Q 0 SRFLOP Vhigh=3.3
B§Set_Delay N006 DUT_GND V=delay(V(Q), 500n)
D1 DUT_GND VCC D
D2 DUT_GND Viout D
B§VSat_High N001 DUT_GND V=V(VCC)-0.3
VSat_Low N004 DUT_GND 500mV
XBuffer N002 Viout N001 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
I1 VCC DUT_GND 25m
.model D D
.lib C:\Users\khampton\OneDrive - AllegroMicro\Documents\LTspiceXVII\lib\cmp\standard.dio
.model NMOS NMOS
.model PMOS PMOS
.lib C:\Users\khampton\OneDrive - AllegroMicro\Documents\LTspiceXVII\lib\cmp\standard.mos
.lib opamp.sub
./lib/sub/LTC.lib
.model D1 D(vrev=6.5V)
.model D2 D(vrev=6.5V)
* ACS732 Sensor Schematic\n \nKasey Hampton + Wade Bussing\nFeb 2020
.param Ihyst=0.05*IRpMAX IRpMAX=40 sens=50m
.lib UniversalOpamps2.sub
.backanno
.ends ACS732_Schematic
.end
```

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



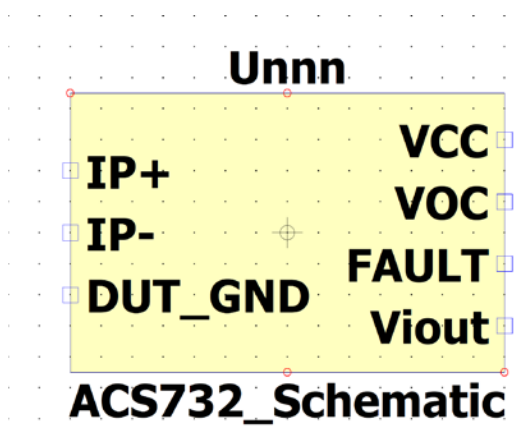
5. LTSpice will ask to automatically generate a symbol file. Select “Yes”.



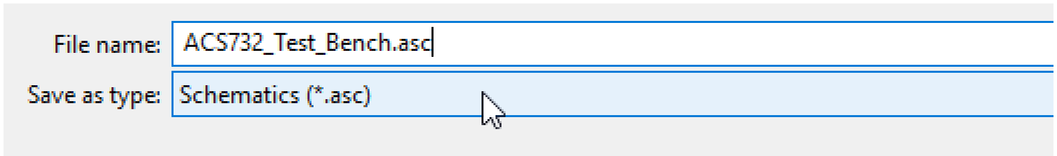
6. The generated symbol file will be *.ASY file type. The name of the symbol file will have the same name as the netlist: ACS followed by the three- or five-digit part number. The file should automatically be saved in the following default file location:

   C:\Users\User\Documents\LTspiceXVII\lib\sym\AutoGenerated.

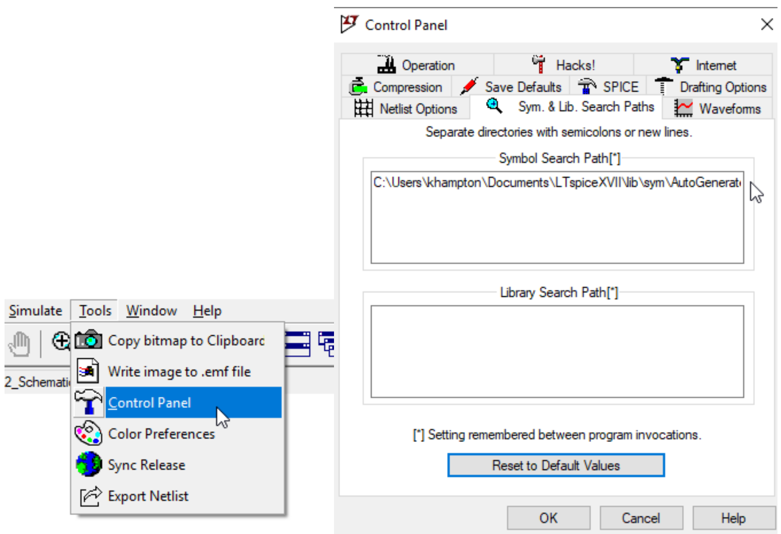
   (Note the user can also find the symbol library folders here: C:\Program Files\LTC\LTspiceXVII\lib\sym. If the symbol does not default save to the documents folder above, save here.)



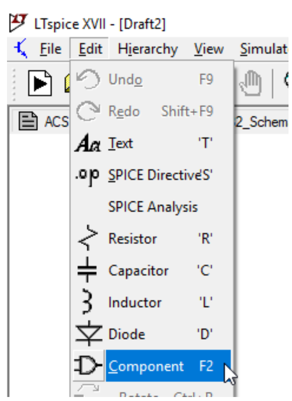
7. To place and use the generated current sensor model, open a new schematic by selecting the "New Schematic" icon (or "File" → "New Schematic" or Ctrl+N).
8. Save the new schematic file to a known local location with a file extension of .ASC.



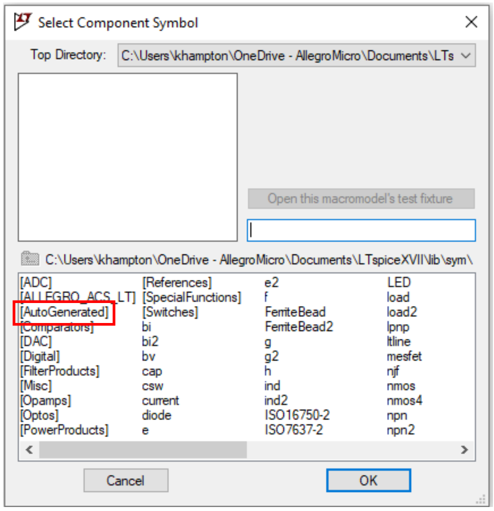
9. Before placing the current sensor block in the schematic, ensure that the "AutoGenerated" file location (or the chosen save location of the symbol) is a Symbol Search Path. Locate the Control Panel in "Tools" → "Control Panel". Navigate to the "Sym. & Lib. Search Paths" tab. Add "C:\Users\User\Documents\LTspiceXVII\lib\sym\AutoGenerated"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

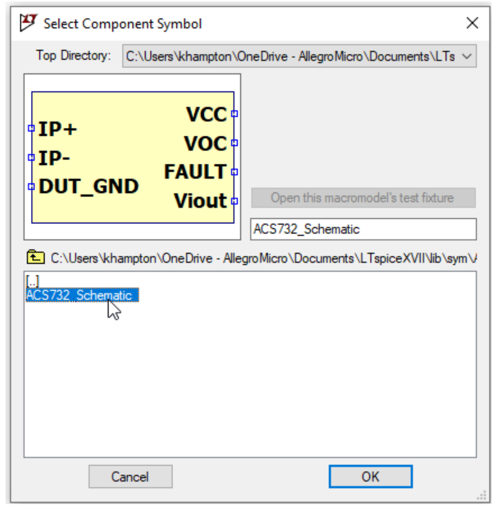


10. To place the generated symbol block, select F2 or "Edit" → "Component".

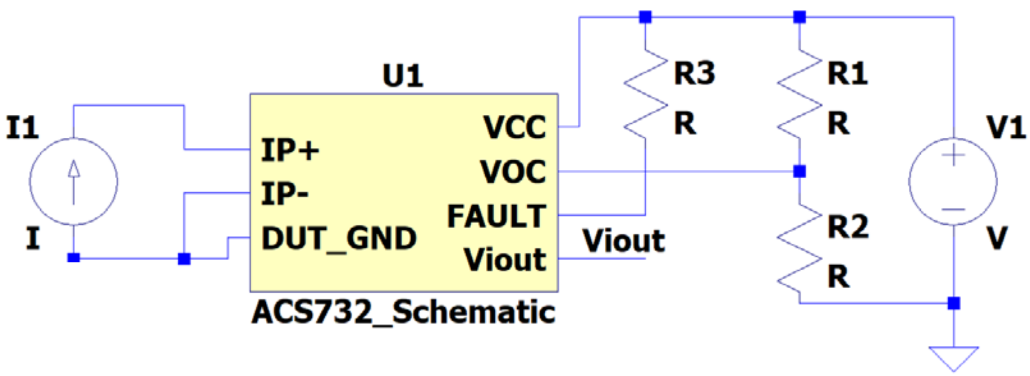


11. This will open the "Select Component Symbol" dialogue window. Open the "AutoGenerated" directory. Select "OK" to place the symbol.



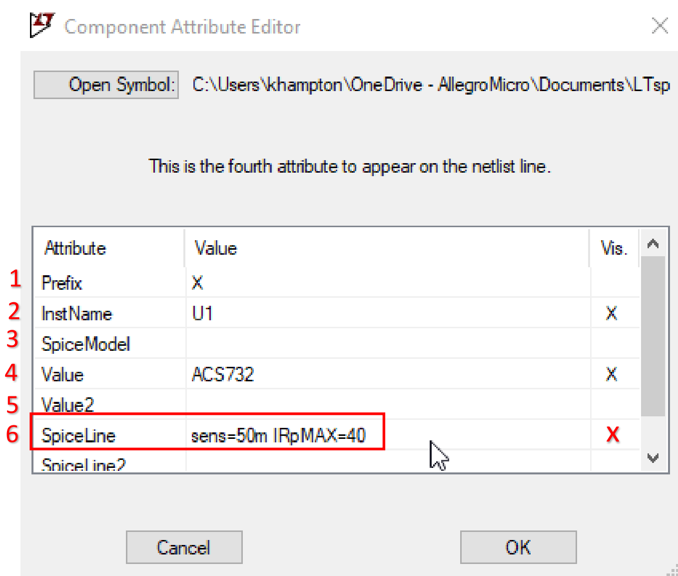


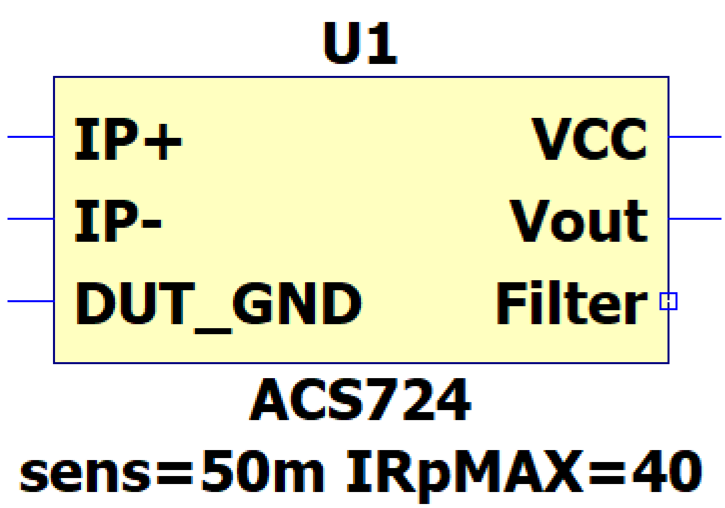
12. After placing the symbol block, the user can generate a system specific test bench in the schematic file. The test bench below, for example, represents the ACS732 typical application. Each component, including resistors and capacitors, current supplies, and voltage supplied, were placed by going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

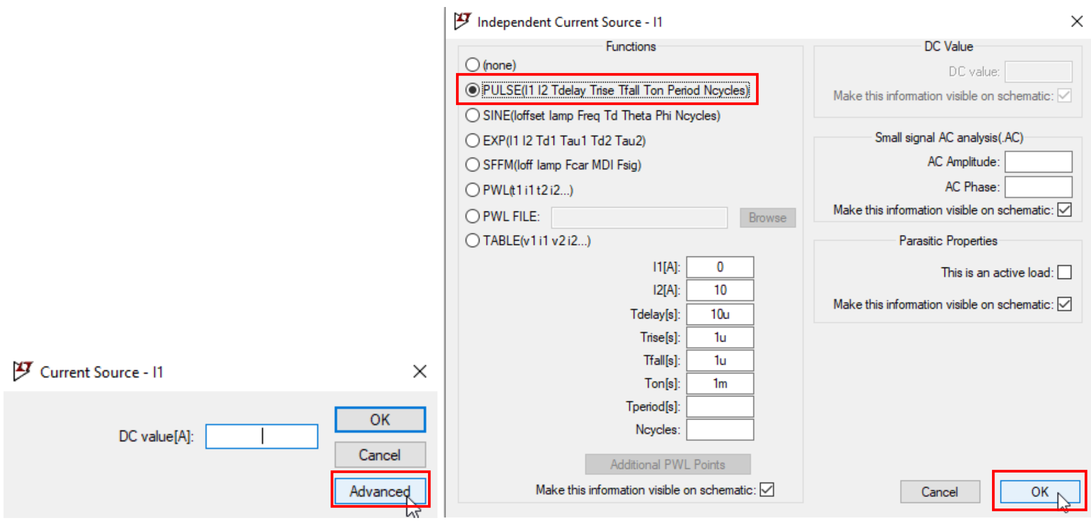




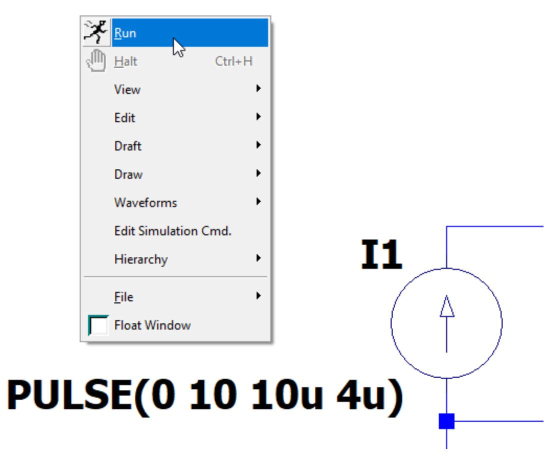
DC Analysis

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.

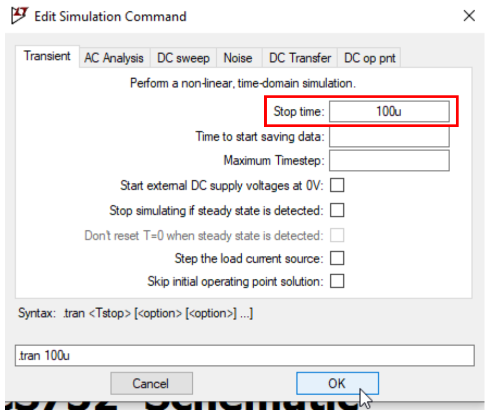
1. Add a current supply to the schematic. Connect the current supply to the IP pins of the sensor by adding wires (F3). Right click on the current supply component. Select "Advanced". Using the advanced settings of the current supply, the user can customize various characteristics of the input current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



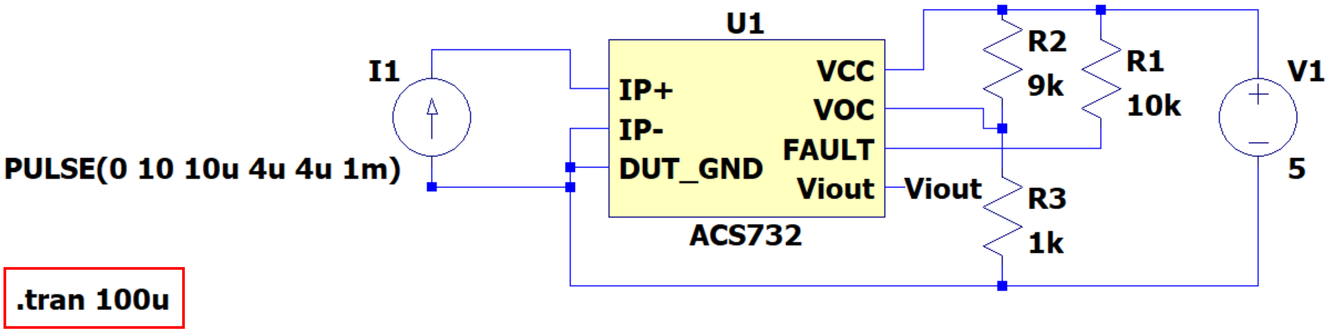
2. Ensure all other components in the test bench are given values. Right click on a blank portion of the schematic and select "Run".



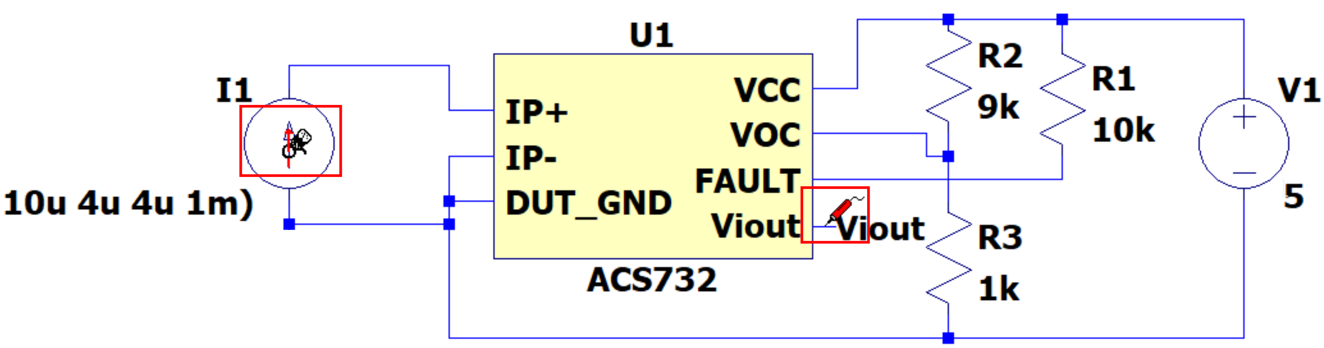
3. The "Edit Simulation Command" dialogue window will appear. Select the "Transient" tab. Enter a "Stop time". Click "OK".

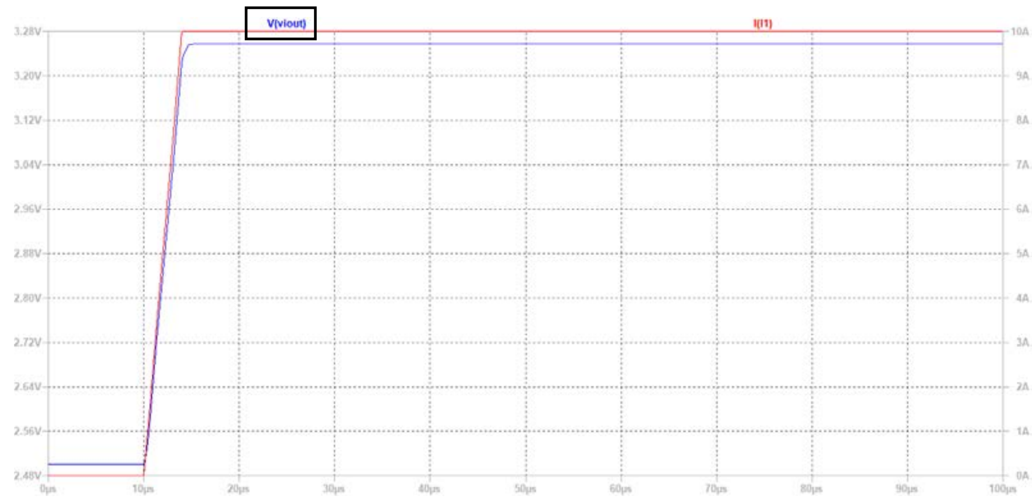


4. The transient simulation command will appear in the bottom left of the window.



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.

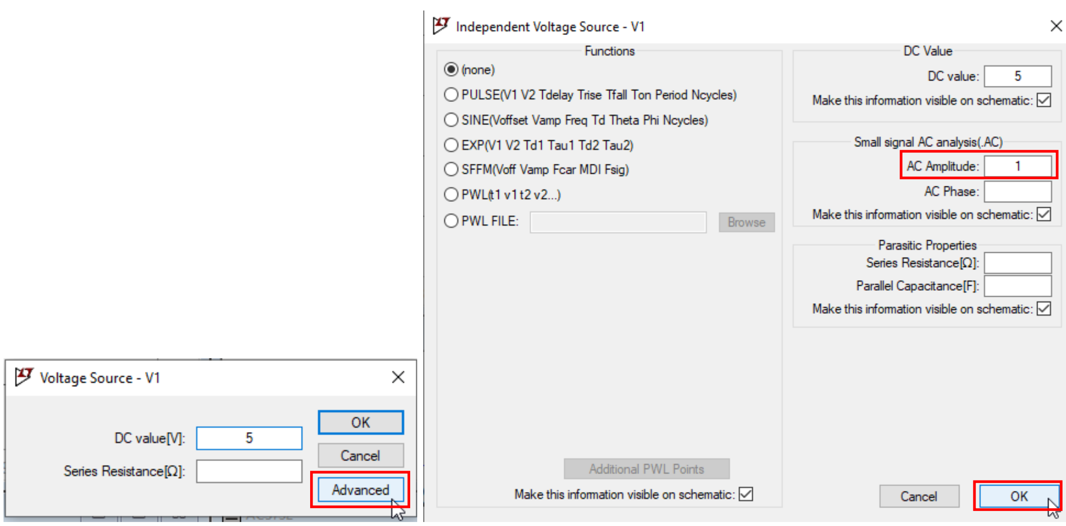




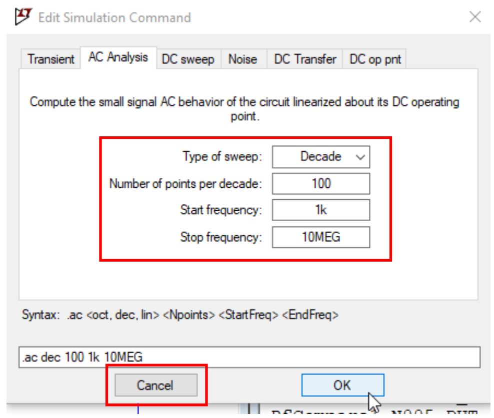
AC Analysis

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.

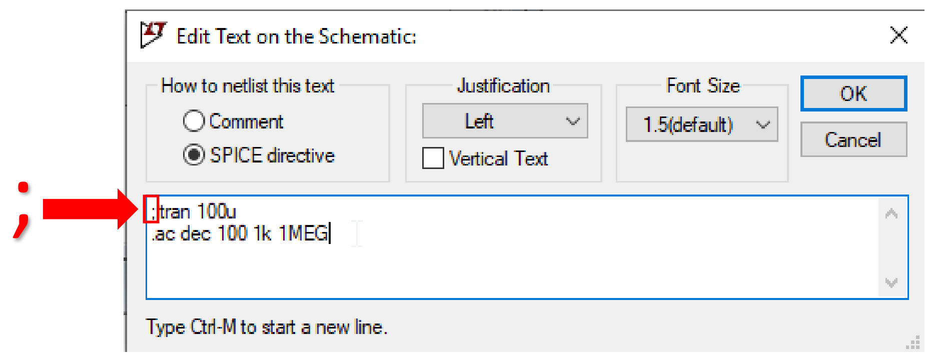
1. Add a voltage supply to the schematic. Right click on the voltage supply component. Select “Advanced”. Using the advanced settings of the voltage supply, the user can customize various characteristics of the power supply. Add a value to the “AC Amplitude” characteristic. Click “Ok”.



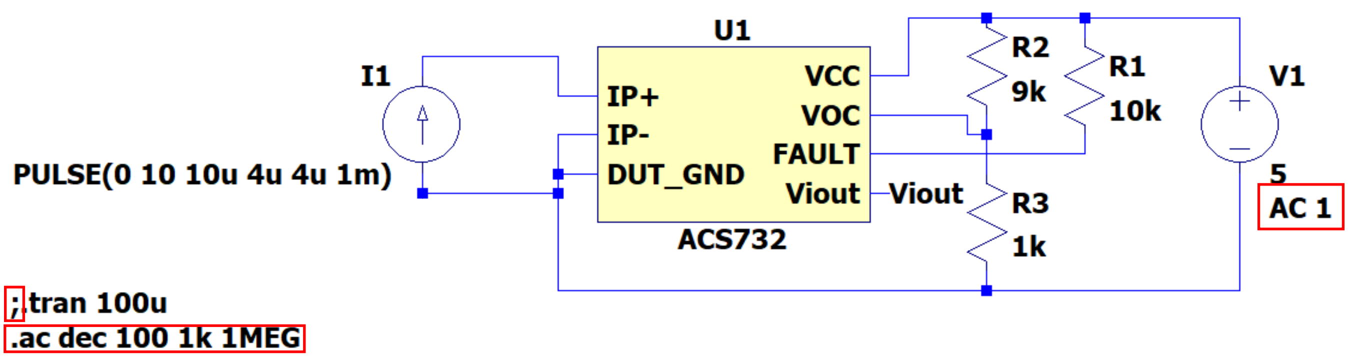
2. Ensure all other components pins of the symbol are given values. If an analysis has already been run, take the steps below. If no simulation has been run, right click on a blank portion of the schematic and select “Run”. This will open the “Edit Simulation Command” dialogue window where the user can 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.



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.



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.