Application Report
SLOA071 – September 2001

Using Texas Instruments Spice Models in Electronics Workbench

Bruce Carter *High-Performance Linear Products*

ABSTRACT

Texas Instruments provides Spice models for operational amplifiers. These Spice models, however, are in a generic text format. This note describes how to take these models and import them into a widely used simulation program—Electronics Workbench™.

Contents

List of Figures

1 Introduction

Spice models have been a mainstay of applications departments at semiconductor companies for decades. They have been provided for simulation purposes, and distributed in print, floppy disk—more recently through CDs and the World Wide Web. They are provided in ASCII text format, which is generic and allows importation into a variety of programs.

Unfortunately, the world has long since progressed beyond ASCII text entry Spice. The author remembers college classes where Spice was implemented on a mainframe. The schematic was reduced to numbered nodes, and each connection was painstakingly typed onto a punch card. Years later, the schematic was typed into an ASCII .cir file and PSpice™ was run from DOS, the resulting table of results seemed almost magical.

Microsoft Windows introduced a new element to Spice—graphical interface simulation programs. Designers, finally free of the need to meticulously type in ASCII files, could access part symbols and draw schematics on the screen—The designers produced schematics for documentation. The intuitive advantage is obvious: instead of a list of circuit connections, the designer can see the schematic. The chance of error transcribing a schematic into a list of connections is eliminated. There is a problem, however. The parts available to the designer are limited to the list available in the program's library. From the very beginning, simulation programs provided a method to enter new part symbols, but few designers have been able to understand the process, which is anything but intuitive. This document walks the designer through the process in Electronics Workbench with a THS4131.

2 Obtaining the Spice Model

Texas Instruments Spice models are available from the Texas Instruments Web Site: http://www.ti.com. The fastest way to get to a model is to enter the part number into the search box on the home page:

Figure 1. Searching for a Part on the Texas Instruments Home Page

Search results from the search above produces, among other things, product folders:

PSpice is a trademark of Cadence.

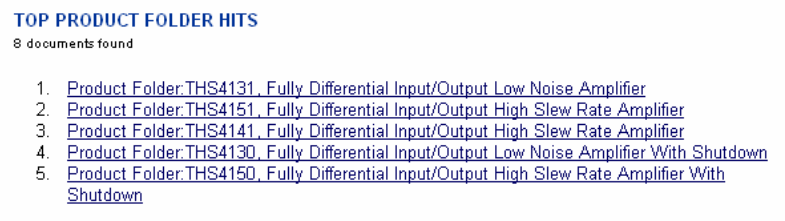

Figure 2. Search Results

The product folder conveniently places all documentation related to the part in one place. Click on the device product folder for the desired device. When the product folder comes up, the Spice models are found in the lower left hand side of the product folder in the *technical documentation* box, *related docs* link:

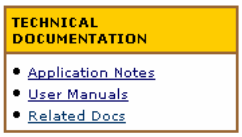

Figure 3. Technical Documentation Box

When the *related docs* link is clicked, the link to the Spice model comes up.

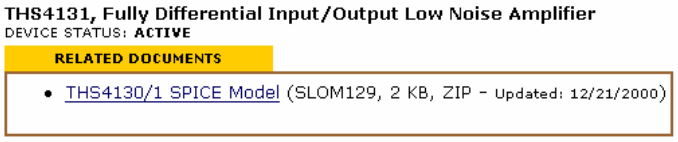

Figure 4. Spice Model Link

Different web browsers behave differently when the link is clicked. The designer needs to save the ASCII text file to the desired location on their hard drive:

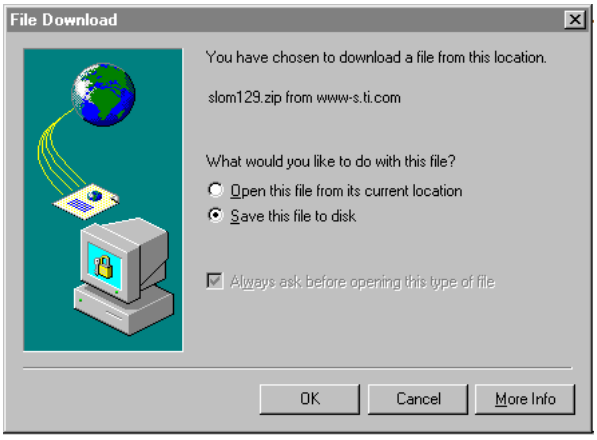

Figure 5. Example of How to Download File to Disk

This download example is from Internet Explorer 5.5. Take note of the filename, and the folder name where it is placed—both are needed in a moment.

Texas Instruments Spice models are stored in the .zip format. The designer needs to have some type of decompression software, such as WinZip, to extract the actual Spice model file from the .zip archive. Once that has been done, the designer will have the file shown in Appendix A, slom129_1.txt.

A quick examination shows that it is essentially a .cir file—a legacy from Spice's past. This document describes how to create an Electronics Workbench symbol. These techniques are similar in many CAD programs, so hopefully this note provides some general guidance for the designer who owns a different simulation program.

3 Create a New Part Symbol

Open the Electronics Workbench Program. Figure 6 shows the Icon:

Figure 6. Electronics Workbench Icon

When the program opens, a new schematic window is created automatically.

The new part symbol editor is available from the schematic entry window. Click, *tools*, *create new component*. This opens a dialog box that starts a six-step process of creating a new component for simulation:

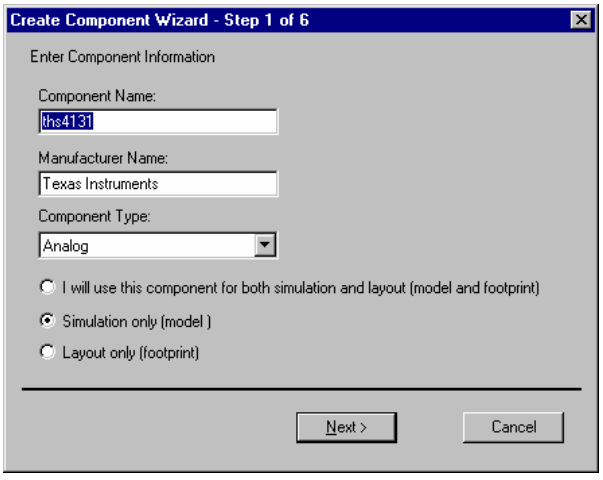

Figure 7. Component Creation Wizard, Step 1

Enter a component name, manufacturer name, and component type (analog, from a drop down list). Next, select a radio button for the use of the part. This part will only be used for simulation, so the *simulation only* button is selected.

Click on *Next*, and step 2 appears:

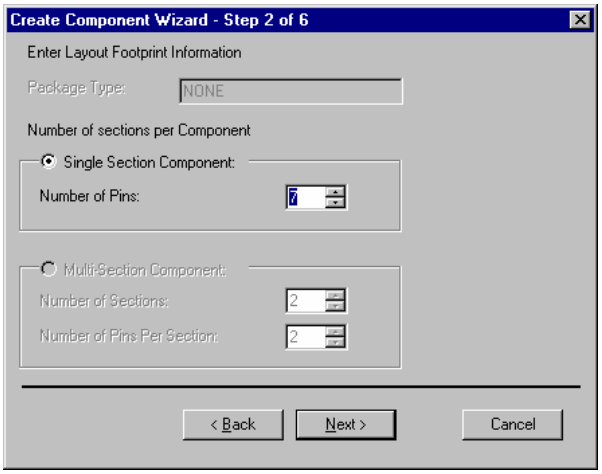

Figure 8. Component Creation Wizard, Step 2

The only option available for a *simulation only* component is *single section*. Select the number of pins in the model. From Appendix A, seven pins.

Click on *Next*, and step 3 appears:

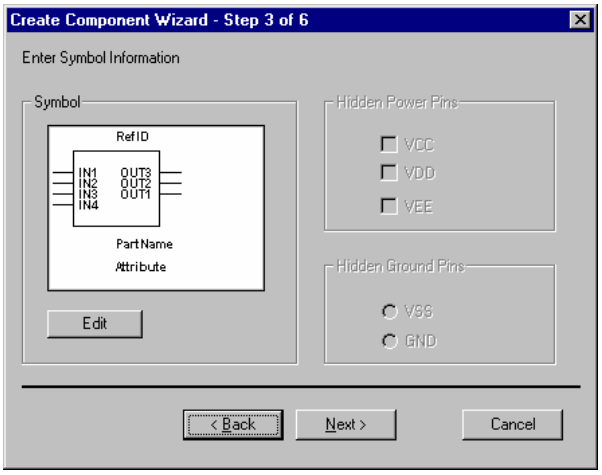

Figure 9. Component Creation Wizard, Step 3

This is where the real work begins. The generic symbol supplied is unsatisfactory. Select *Edit*. The symbol editor window opens:

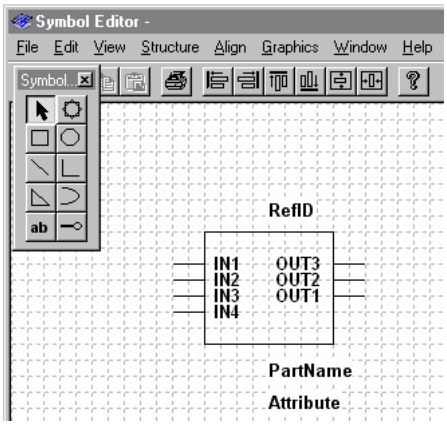

Figure 10. Symbol Editor

It is helpful to choose *view*, *zoom*, and that makes the part easier to work with. Successive steps create a more acceptable component symbol:

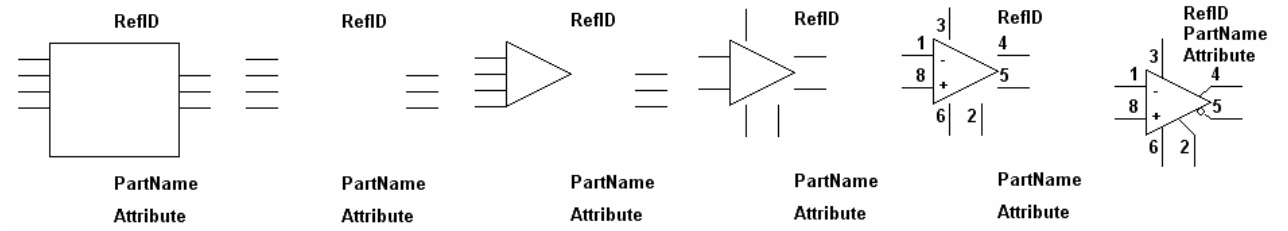

Figure 11. Symbol Creation

There is some artistic license involved in the creation of a part symbol. The process described here creates a symbol, although not necessarily the best symbol. The steps used to create the part symbol above were:

- Edit the pins. Highlight the pin, then edit the properties. The pin edit window comes up:

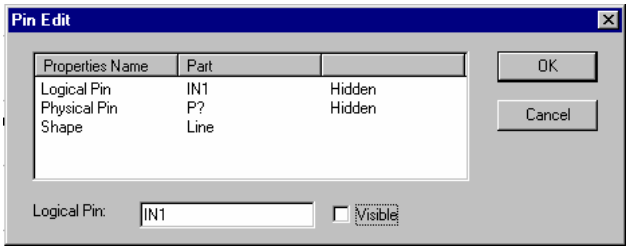

Figure 12. Pin Edit Window

Highlight the logical pin *IN1*, and uncheck the *visible* check box. This turns off the name *IN1* on the symbol. Do this with all the pins.

- Select and delete the box. The proper outline for an operational amplifier is a triangle.

- Resize the part outline. A lot of operational amplifiers are drawn on a 0.4 by 0.4 grid.
- Use the line tool to create the triangle outline of an operational amplifier body.
- Move the pins and labels to the desired positions. In this case, the inverting input is on the top left, the + and – power are on the top and bottom, respectively, the outputs is on the right, and $V_{(OCM)}$ is on the bottom. A couple of very important hints:
  - Be sure to keep track of the logical pin name and where it is moved. This information is needed later.
  - When a pin is moved to the top or bottom, it can be rotated 90° clockwise or counter clockwise in the *edit* menu. It does not matter which end of the pin symbol is connected to the part.
- Use the text tool to add the - and + to the inputs and the pin numbers to the pins. Be careful where text boxes are placed, because text boxes are not transparent and can hide portions of the body or pin symbol.

  Pin number assignments can be obtained from the data sheet:

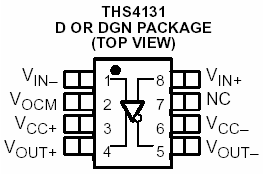

Figure 13. Pin Number Assignments for the THS 4131

- Use the line and circle tool to make cosmetic changes, and move the component text closer to the part.

When the appearance of the component is acceptable, do a *file save*, and then *file*, *exit*. This returns to the component creation wizard, Step 3. Click *next*, and to go to step 4:

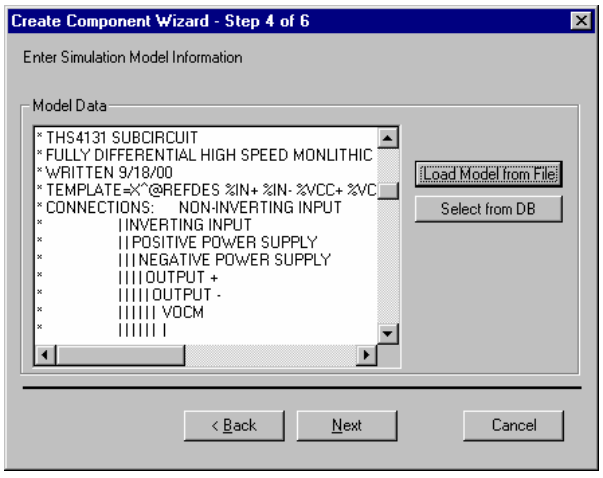

Figure 14. Component Creation Wizard, Step 4

Click *load model from file*, then browse to the location of the model file. The component wizard defaults to the .cir extension, which is a legacy from the Spice days. Select *all files* from the drop down box to overcome this and select the .txt model file. It appears in the window.

Click *next* to go to step 5:

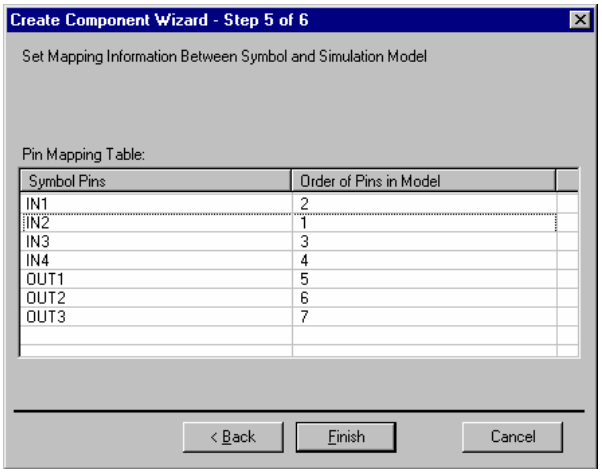

Figure 15. Component Creation Wizard, Step 5

This is the step where the notes made during component editing come in handy. The raw component made four inputs and three outputs, which were moved around the outline to the desired location. The order in which they were added when the raw component was edited is important, because the order corresponds to the order in the model:

Table 1. Mapping Raw Pin Names to Model

Order	Model #	Model Name	Raw Symbol Name
(1st)	1	Noninverting input	IN3
(2nd)	2	Inverting input	IN1
(3rd)	3	Positive supply	IN2
(4th)	4	Negative supply	IN4
(5th)	5	Noninverting output	OUT3
(6th)	5B	Inverting output	OUT2
(7th)	17	$V_{(OCM)}$	OUT1

Click *finish* to go to step 6, which is back to the component properties window. Click *save*, and a *component destination* box comes up:

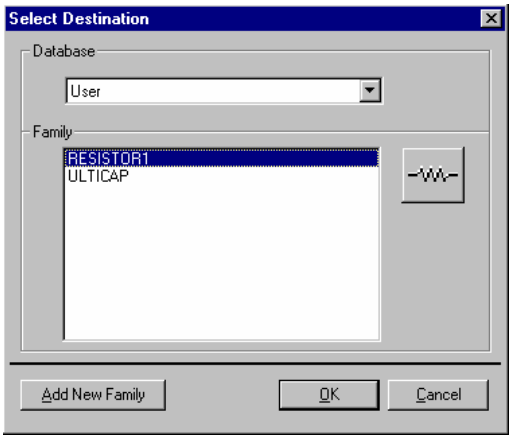

Figure 16. Component Destination Dialog Box

Neither choice is particularly good—the component is not a resistor or an *ulticap*, so click *on add new family*. Note that these choices may be different, depending on the installation of the program and what components have been previously added. When *add new family* is clicked, a *new family name* box comes up:

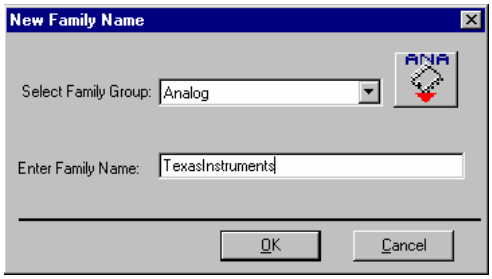

Figure 17. New Family Name Box

The family group is selected from a drop-down box, and *analog* is where all the operational amplifiers are, so it is a good choice. Then enter the desired family name—no spaces allowed.

Click *OK* to return to the *select destination* box:

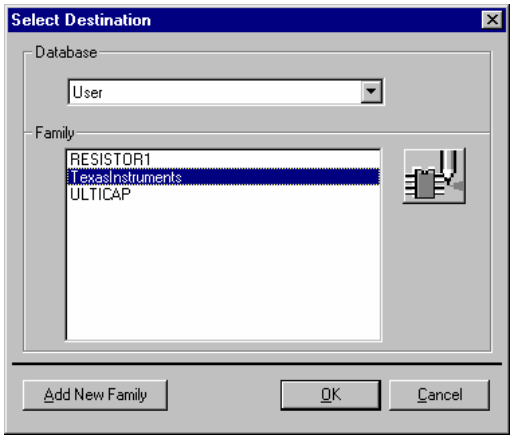

Figure 18. Component Destination Box With New Family

The new Texas Instruments family is now a convenient place where other TI parts can be added. Click *ok*, and a confirmation box appears to confirm the addition of the new part. Click *ok* to close the confirmation box, and *exit* (not *save*) on the component properties box.

4 Drawing the Schematic

This document assumes that the designer is familiar with how to use the Electronics Workbench program to draw schematics. The location of the new component, however, needs explanation.

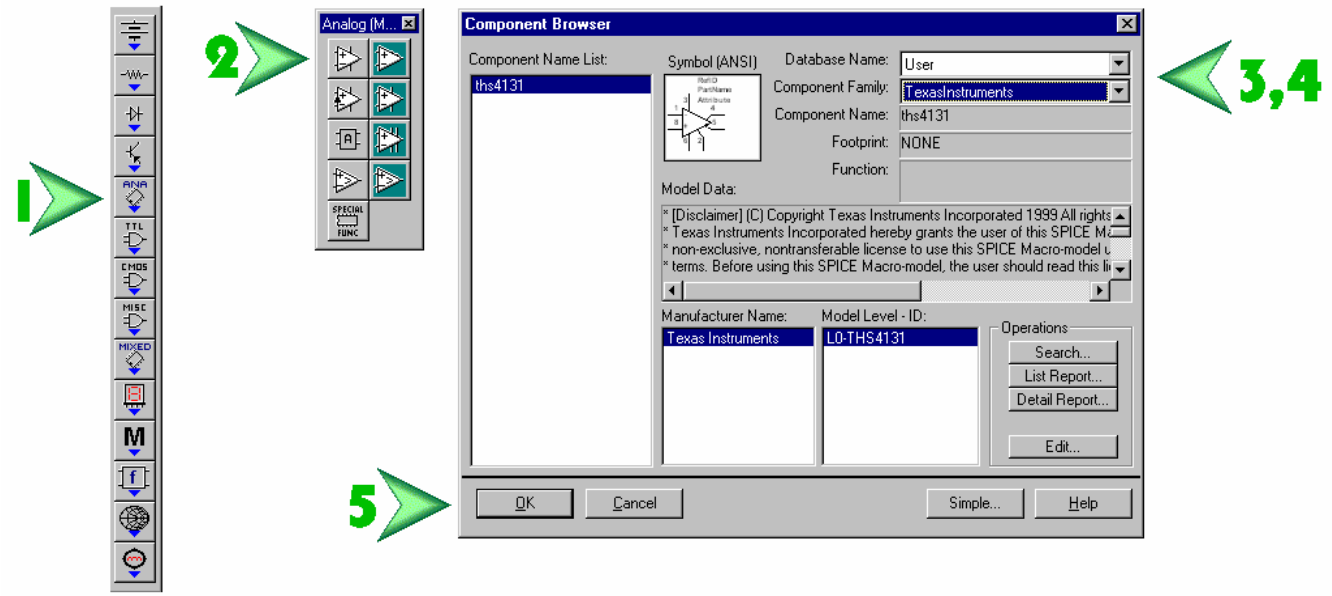

Figure 19. Location of the New Component

To use the new component, go to the part toolbar, and select *ANA* (analog). Next, select the generic op amp symbol on the upper left. Select the *user* database name from the drop-down menu, then *Texas Instruments* from the component family drop-down menu. There is only one part in the family, if more are present—highlight the desired part. Finally, click on *ok*, and the outline of the new component is available for placement on the schematic window.

The schematic that was drawn for this example is a one-pole low pass filter with a breakpoint at 10 kHz. The colors have been adjusted for display in this document.

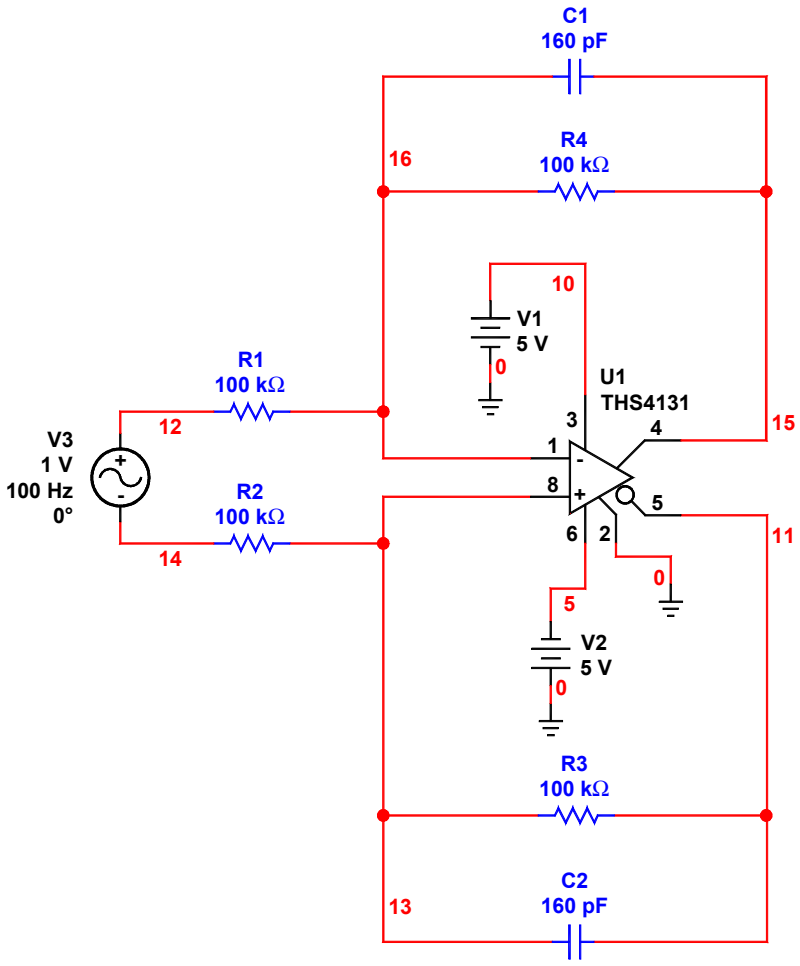

Figure 20. Electronics Workbench Schematic, Using New Symbol

5 Simulating With the Part Symbol

To simulate, select *simulation*, *analyses*, and *AC Analysis*. The following window opens:

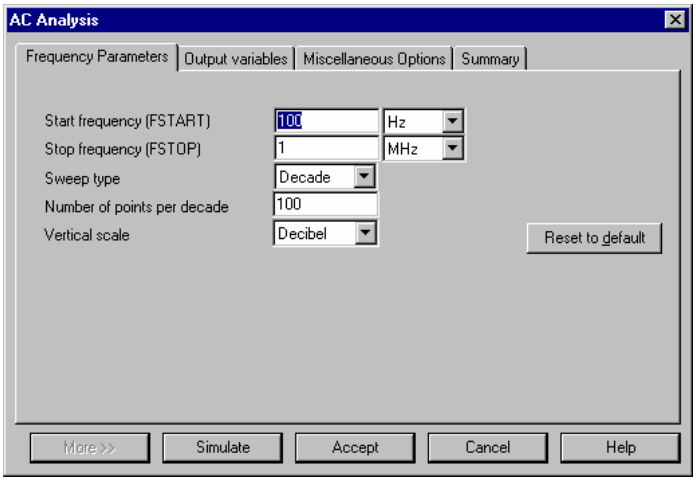

Figure 21. AC Analysis Window

Set the start and stop frequencies, select the sweep type—decade or linear, select the number of points per decade, and select the vertical scale type. It is seldom necessary to select more than 100 points per decade, except for the highest Q bandpass and notch filters.

Next, select which output variables to display on the *output variables* tab:

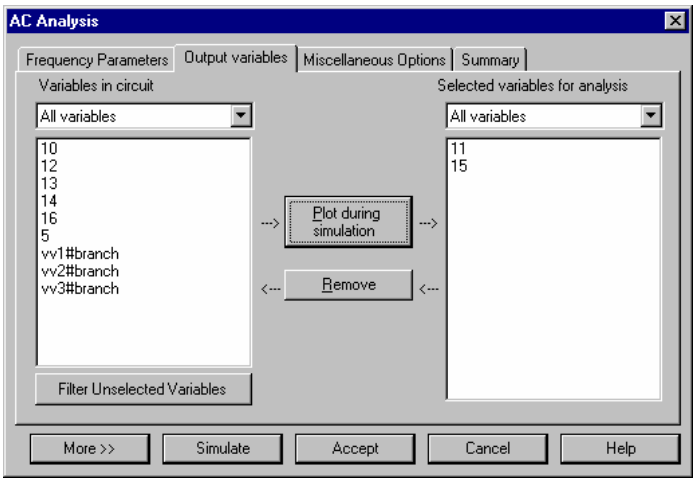

Figure 22. Output Variables Window

To move a node to the right, highlight the node number in the list on the left, and then click on *plot during simulation*. Then click on *simulate*.

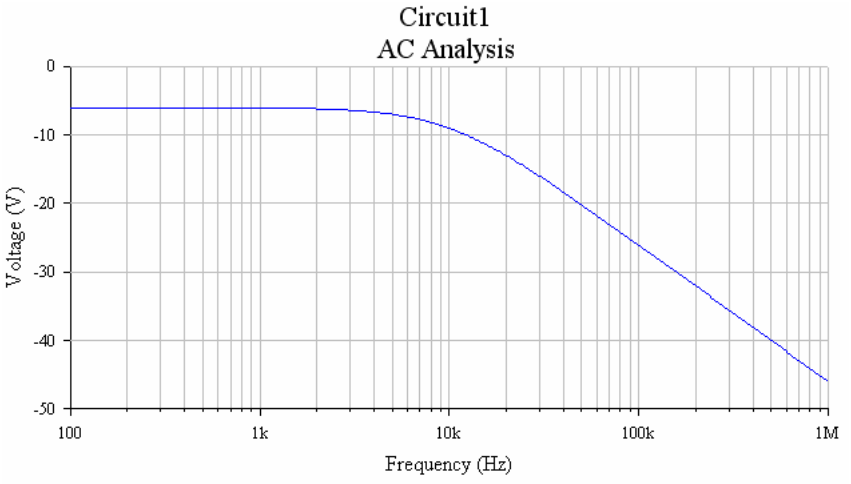

Figure 23. Simulation Results

This is the result that is expected, although it is not immediately apparent. Electronics Workbench has no command to mathematically add the two outputs, one of which displays in red, and the other in blue. The blue overwrites the red, both of which have an amplitude –6 dB in the passband. On the phase plot, they are both shown 180° out of phase. Therefore, the differential output would have an amplitude of 0 dB in the passband.

The Spice model has been successfully translated into an Electronics Workbench simulation.

6 Conclusions

Texas Instruments Spice models, in their downloaded form, are not directly usable in Electronics Workbench. They can be used successfully, following the procedure given in this note.

Appendix A: Example Spice Model

```
* THS4131 SUBCIRCUIT
* FULLY DIFFERENTIAL HIGH SPEED MONLITHIC OPERATIONAL AMPLIFIER
* WRITTEN 9/18/00
* TEMPLATE=X^@REFDES %IN+ %IN- %VCC+ %VCC- %OUT+ %OUT- %VOCM @MODEL
* CONNECTIONS:   NON-INVERTING INPUT
*           | INVERTING INPUT
*           | | POSITIVE POWER SUPPLY
*           | | | NEGATIVE POWER SUPPLY
*           | | | | OUTPUT +
*           | | | | | OUTPUT -
*           | | | | | | VOCM
*           | | | | | | |
.SUBCKT THS4131   1 2 3 4 5 5b 17
*
* INPUT *
 Q1     27 1 28 NPN_IN 2
 Q2     25 2 29 NPN_IN 2
 R2     26 29 15
 R1     26 28 15

* SECOND STAGE *
 Q3     07 Vref 27 PNP 2
 Q4     09 Vref 25 PNP 2
 Q5     07 105 06 NPN 1
 Q7     09 105 08 NPN 1
 R3      4 06 333
 R4      4 08 333
 Cc      0 09 25.5p
 Ccb     0 07 25.5p

* HIGH FREQUENCY SHAPING *
 Ehf     34 0 09 0 1
 Lhf     34 35 7n
 Rhf     102 35 25
 Chf     0 102 23p
 Ehfb     24 0 07 0 1
 Lhfb     24 33 7n
 Rhfb     12 33 25
 Chfb     0 12 23p

* OUTPUT *
 Q8     102 102 103 PNP 1
 Q9     102 102 104 NPN 1
 Q10     3 103 30 NPN 5
 Q11     4 104 31 PNP 7.5
 R5     5 30 4
 R7     31 5 4
 Q8b     12 12 13 PNP 1
 Q9b     12 12 15 NPN 1
 Q10b     3 13 22 NPN 5
 Q11b     4 15 23 PNP 7.5
 R5b     5b 22 4
 R7b     23 5b 4

* Vcm ERROR AMP *
 Gcm     0 105 16 17b 1e-4
 Vcm       17 17b 285e-3
 Rtop     17 3 30k
```

```
 Rbot     17 4 30k
 Rcm     16 5 10k
 Ccm     16 5b 100p
 Rcm2     16 5b 10k
 Ccm2     16 5 100p

* BIAS SOURCES *
 V1     3 Vref 1.85
 I1     3 27 DC 2.1e-3
 I2     3 25 DC 2.1e-3
 I3     0 103 DC 1.225e-3
 I4     26 4 DC 2.1e-3
 I5     104 0 DC 1.86e-3
 I6     0 13 DC 1.225e-3
 I7     15 0 DC 1.86e-3

.MODEL NPN_IN NPN
+ IS=170E-18 BF=400 NF=1 VAF=100 IKF=0.0389 ISE=7.6E-18
+ NE=1.13489 BR=1.11868 NR=1 VAR=4.46837 IKR=8 ISC=8E-15
+ NC=1.8 RB=25 RE=0.1220 RC=20 CJE=120.2E-15 VJE=1.0888 MJE=0.381406
+ VJC=0.589703 MJC=0.265838 FC=0.1 CJC=133.8E-15 XTF=272.204 TF=12.13E-12
+ VTF=10 ITF=0.147 TR=3E-09 XTB=1 XTI=5 KF=7.5E-14

.MODEL NPN NPN
+ IS=170E-18 BF=100 NF=1 VAF=100 IKF=0.0389 ISE=7.6E-18
+ NE=1.13489 BR=1.11868 NR=1 VAR=4.46837 IKR=8 ISC=8E-15
+ NC=1.8 RB=250 RE=0.1220 RC=200 CJE=120.2E-15 VJE=1.0888 MJE=0.381406
+ VJC=0.589703 MJC=0.265838 FC=0.1 CJC=133.8E-15 XTF=272.204 TF=12.13E-12
+ VTF=10 ITF=0.147 TR=3E-09 XTB=1 XTI=5

.MODEL PNP PNP
+ IS=296E-18 BF=100 NF=1 VAF=100 IKF=0.021 ISE=494E-18
+ NE=1.49168 BR=0.491925 NR=1 VAR=2.35634 IKR=8 ISC=8E-15
+ NC=1.8 RB=250 RE=0.1220 RC=200 CJE=120.2E-15 VJE=0.940007 MJE=0.55
+ VJC=0.588526 MJC=0.55 FC=0.1 CJC=133.8E-15 XTF=141.135 TF=12.13E-12
+ VTF=6.82756 ITF=0.267 TR=3E-09 XTB=1 XTI=5

.ENDS
```

IMPORTANT NOTICE

Texas Instruments Incorporated and its subsidiaries (TI) reserve the right to make corrections, modifications, enhancements, improvements, and other changes to its products and services at any time and to discontinue any product or service without notice. Customers should obtain the latest relevant information before placing orders and should verify that such information is current and complete. All products are sold subject to TI's terms and conditions of sale supplied at the time of order acknowledgment.

TI warrants performance of its hardware products to the specifications applicable at the time of sale in accordance with TI's standard warranty. Testing and other quality control techniques are used to the extent TI deems necessary to support this warranty. Except where mandated by government requirements, testing of all parameters of each product is not necessarily performed.

TI assumes no liability for applications assistance or customer product design. Customers are responsible for their products and applications using TI components. To minimize the risks associated with customer products and applications, customers should provide adequate design and operating safeguards.

TI does not warrant or represent that any license, either express or implied, is granted under any TI patent right, copyright, mask work right, or other TI intellectual property right relating to any combination, machine, or process in which TI products or services are used. Information published by TI regarding third–party products or services does not constitute a license from TI to use such products or services or a warranty or endorsement thereof. Use of such information may require a license from a third party under the patents or other intellectual property of the third party, or a license from TI under the patents or other intellectual property of TI.

Reproduction of information in TI data books or data sheets is permissible only if reproduction is without alteration and is accompanied by all associated warranties, conditions, limitations, and notices. Reproduction of this information with alteration is an unfair and deceptive business practice. TI is not responsible or liable for such altered documentation.

Resale of TI products or services with statements different from or beyond the parameters stated by TI for that product or service voids all express and any implied warranties for the associated TI product or service and is an unfair and deceptive business practice. TI is not responsible or liable for any such statements.

Following are URLs where you can obtain information on other Texas Instruments products & application solutions:

Products		**Applications**	
Amplifiers	amplifier.ti.com	Audio	www.ti.com/audio
Data Converters	dataconverter.ti.com	Automotive	www.ti.com/automotive
DSP	dsp.ti.com	Broadband	www.ti.com/broadband
Interface	interface.ti.com	Digital Control	www.ti.com/digitalcontrol
Logic	logic.ti.com	Military	www.ti.com/military
Power Mgmt	power.ti.com	Optical Networking	www.ti.com/opticalnetwork
Microcontrollers	microcontroller.ti.com	Security	www.ti.com/security
		Telephony	www.ti.com/telephony
		Video & Imaging	www.ti.com/video
		Wireless	www.ti.com/wireless

Mailing Address: Texas Instruments
Post Office Box 655303 Dallas, Texas 75265