From SPICE Netlist to Allegro Design Sub-Circuit

Introduction

Intersil provides a SPICE model for all our new precision Opamps. The SPICE model netlist is included in the data sheet, along with simulation vs characterization curves. Refer to AN1556 for more details about making SPICE models.

This application note will walk the user through the process of taking the netlist from the data sheet and creating a sub-circuit to drop into a Cadence Allegro Design simulator.

Copying the SPICE Netlist

Download the Intersil data sheet from the web. The data sheet will be in .pdf format. Open the .pdf document and right click to enable the select tool, if it is not already selected (Figure 1). This will enable you to then copy and paste the entire list into Notepad.

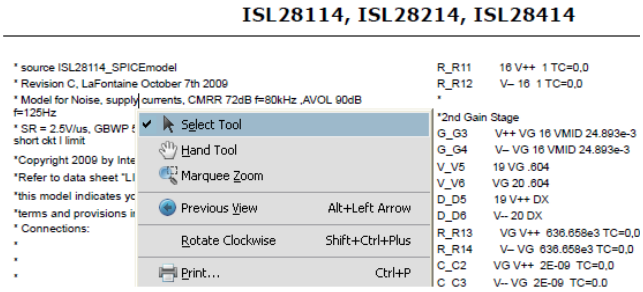

FIGURE 1. TURN ON THE SELECT TOOL IN THE PDF DATA SHEET

Scroll towards the end of the datasheet and find the SPICE netlist (Figure 3) and copy it into notepad. Name the file with the extension **.MOD** (not case sensitive) as shown in Figure 2. This file needs to be saved in a common directory with all the other SPICE files for this design.

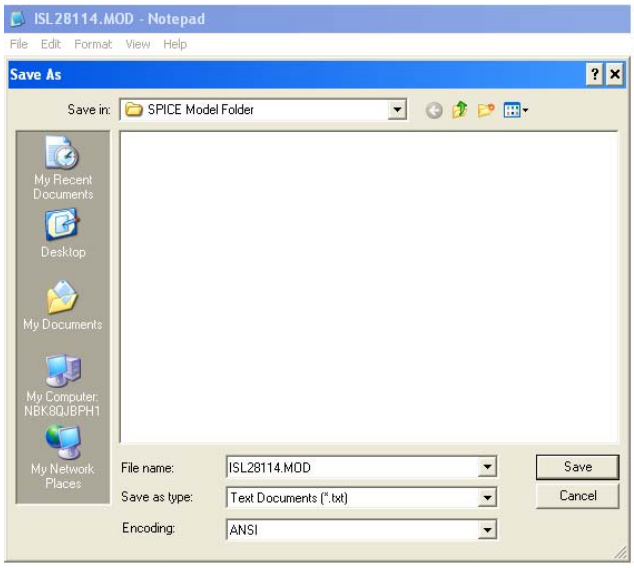

FIGURE 2. SAVING NOTE PAD FILE AS .MOD

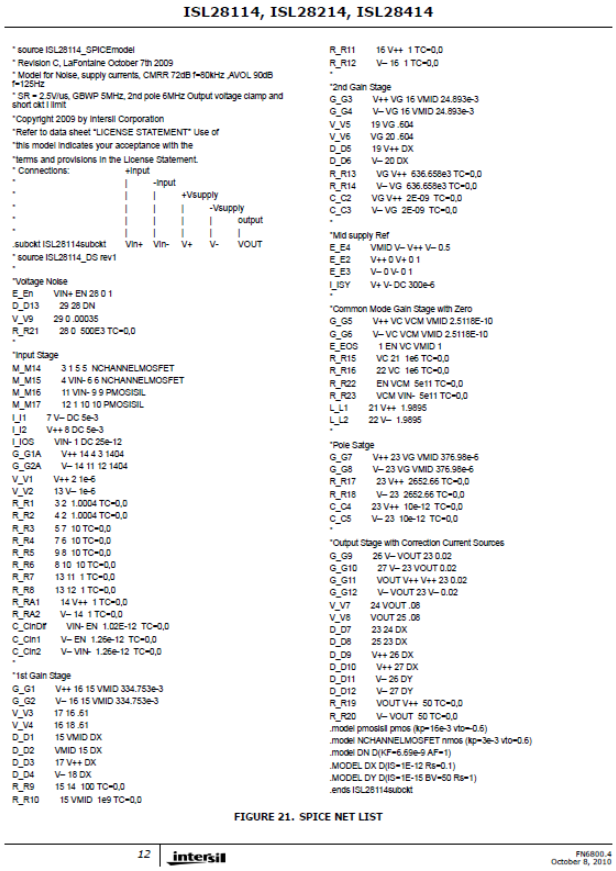

FIGURE 3. NETLIST FROM DATA SHEET

Model Editor

Open the Cadence model editor via the path shown in Figure 4 (Cadence SPB 16.2\AMS Simulator\Simulation Accessories\ Model Editor)

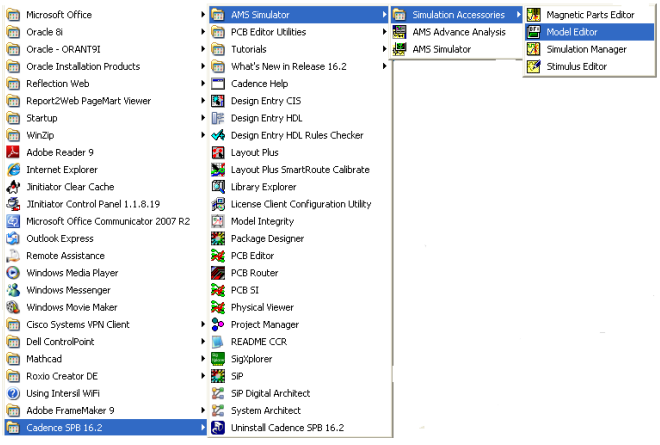

FIGURE 4. PATH TO CADENCE MODEL EDITOR

Note: This document is written using the SPB16.2 software. The look and feel may change with different revisions of the Cadence software, but the procedure will be the same.

After selecting the Model Editor, the screen in Figure 5 will open up. Select Capture and click DONE.

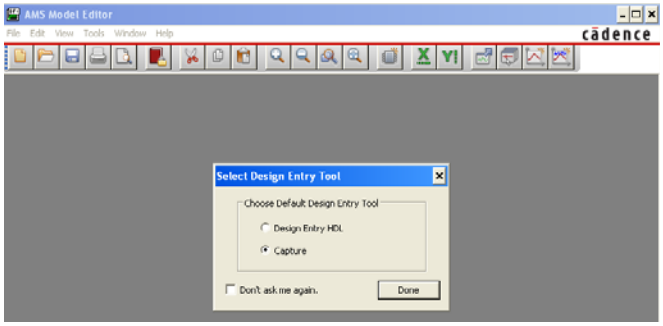

FIGURE 5. SELECT DESIGN ENTRY TOOL

Click on File in the tool bar and select New. Figure 6 will appear.

FIGURE 6. BEGINNING OF NEW MODEL

Click on Model in the tool bar and select Import. Then browse to the folder where you put the .MOD file. Figure 7 will appear. Select the desired .MOD file and click Open.

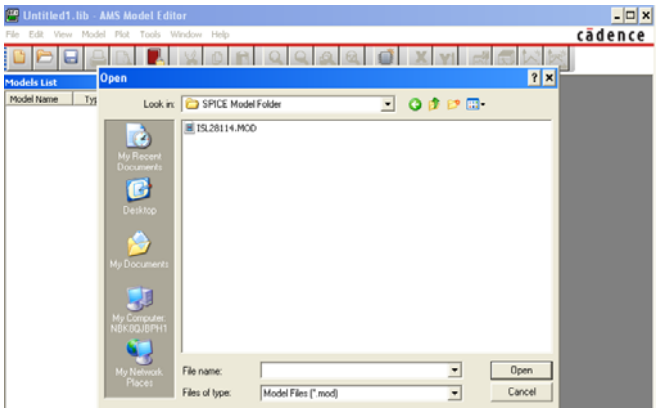

FIGURE 7. SELECT .MOD NETLIST

This will load the netlist into the Model editor tool as shown in Figure 8.

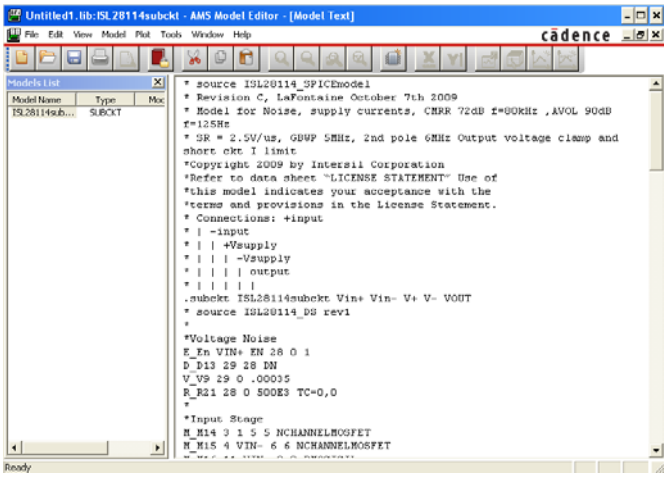

FIGURE 8. NETLIST LOADED INTO MODEL EDITOR

Click on File in the tool bar and select Save As. Then type the part name as the file name in Figure 9 and click Save. The file with the complete netlist is now saved as a .lib library file.

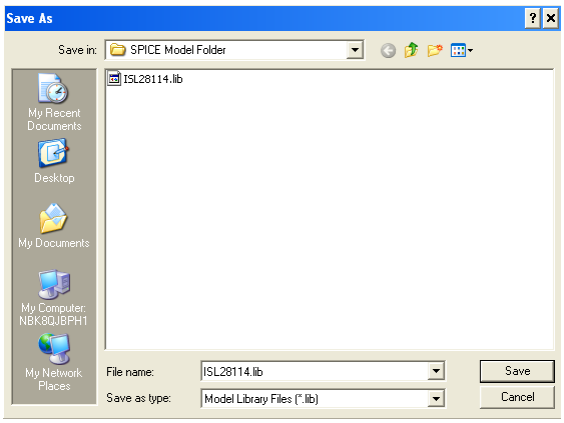

FIGURE 9. FILE SAVED AS .lib

Click on File in the tool bar and select Export to Capture Part Library. The Input Model Library path and the Output Part Library path will automatically be loaded as shown in Figure 10.

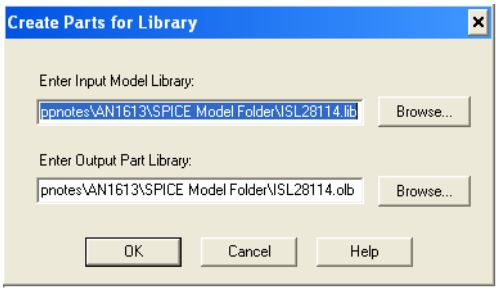

FIGURE 10. LIBRARY CREATION

Verify that the files paths are the same with the only difference being the .lib and .olb extensions.

Click OK and verify no Error messages or Warning messages as shown in Figure 11. Click OK.

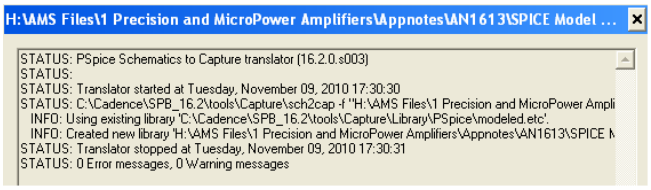

FIGURE 11. SCHEMATIC TO CAPTURE TRANSLATOR CHECK

Click on File in the tool bar and select Import Wizard [Capture]. Like before, both path names will load automatically and should have the same file paths with the only difference being the .lib and .olb extensions as shown in Figure 12.

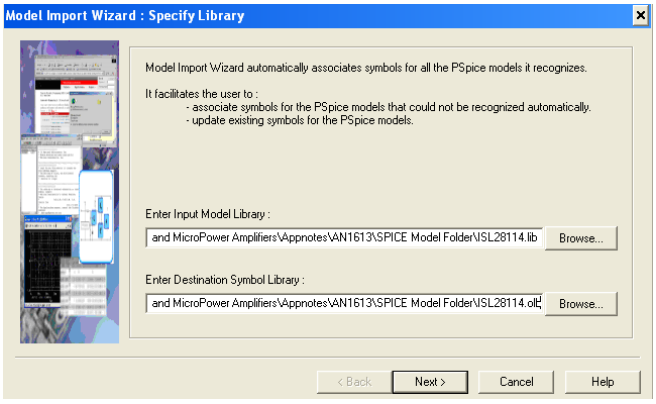

FIGURE 12. ASSOCIATE SYMBOL WITH SPICE MODEL

Click Next and the screen shown in Figure 13 will appear. This is the screen in which we will associate the pins of our SPICE model to the pins of the sub-circuit model. The symbol shown is a generic 5 pin device. We want our Opamp symbol to look like an Opamp. To do this click on the Replace Symbol button and select from the list of symbols provided with the Cadence program. This list is located at the following location on your C drive. C:\Cadence\SPB.16.2\tools\capture\libary\OPAmp.olb

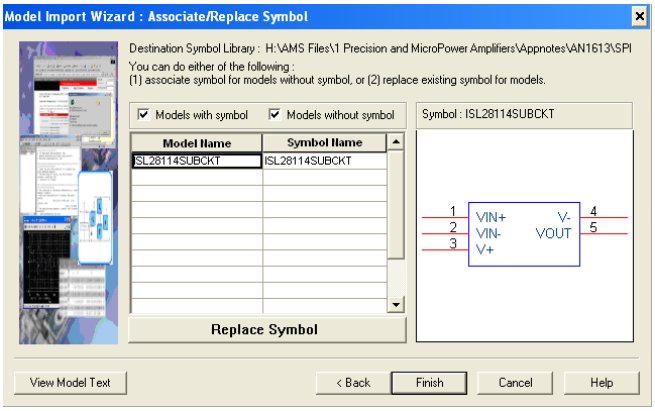

FIGURE 13. REPLACE GENERIC SYMBOL

If the location of your Cadence software was loaded in a different location, then search for Cadence\SPB.

When selecting your symbol, all that matters is the pin count. The numbers assigned to the symbol pins can be changed later. Just scroll through the list to find a symbol that matches a desired pinout and pin count of your device. In this example, we selected the TLC2201. Click Next.

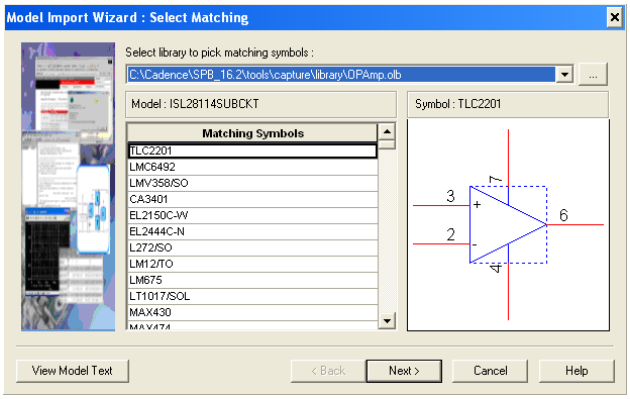

FIGURE 14. ASSOCIATE OP AMP SYMBOL WITH MODEL

Then click on the row under the Symbol Pin column to activate pull down menu box under the symbol column. Now pick the associated pin to match the Model Terminal function in the model terminal column. As shown in Figure 15.

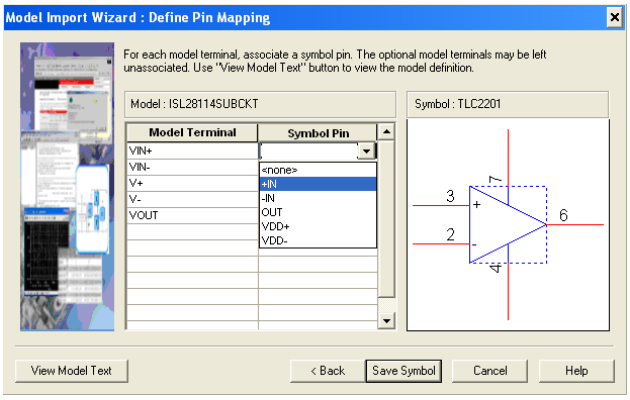

FIGURE 15. DEFINE PINS OF SYMBOL TO PINS OF MODEL

Repeat for all Model Terminal pins as shown in Figure 16.

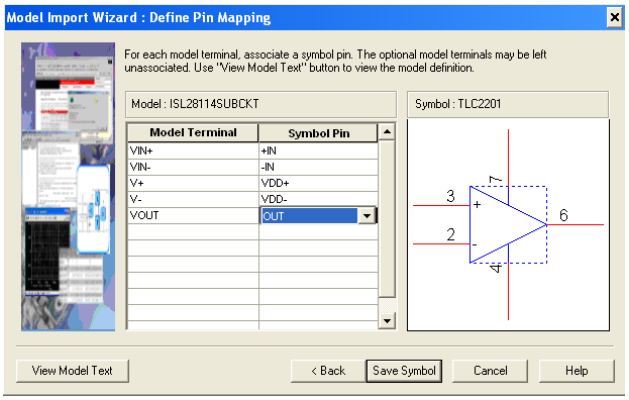

FIGURE 16. ALL PINS ASSOCIATED TO SYMBOL

Click Save Symbol, and Figure 17 will appear. Verify no Error messages or Warning messages appear. Click OK and then close the Model Editor.

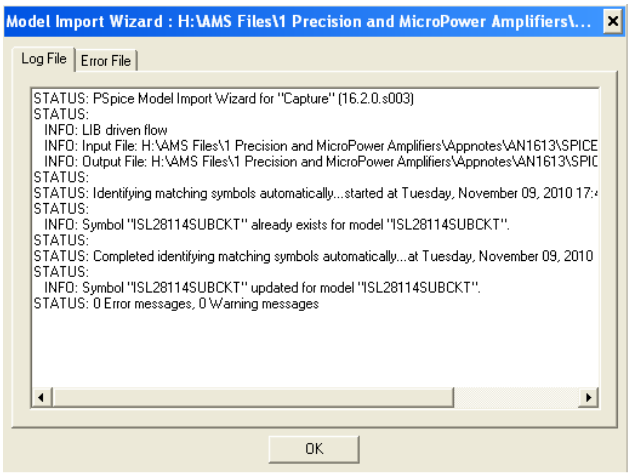

FIGURE 17. MODEL IMPORT WIZARD CHECK

Using the New Sub-Circuit to Run Simulations

Open the Cadence Software. Figure 18 shows the path to select the Design Entry CIS.

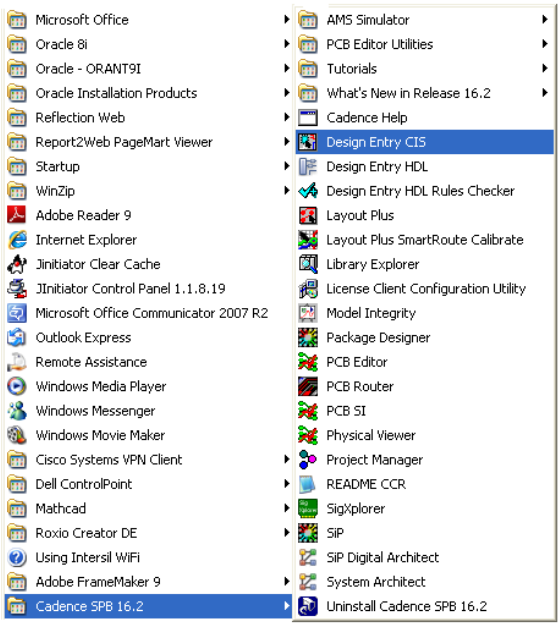

FIGURE 18. PATH TO DESIGN ENTRY CIS

Figure 19 shows the Cadence Product Choices. Select Allegro Design Entry CIS and Click OK.

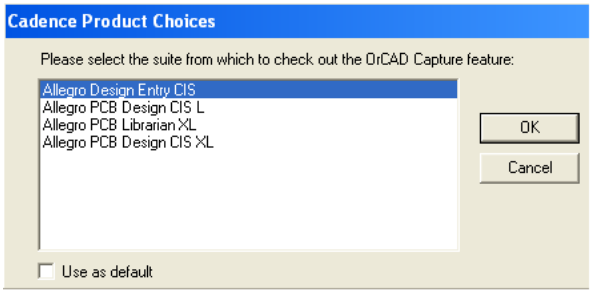

FIGURE 19. CADENCE PRODUCT CHOICES

Click on File in the tool bar and select New, and then Project. The screen shown in Figure 20 will appear. Type in the name of the project and select Analog of Mixed A/D button. Browse to where you saved the Netlist in the common directory (you must have all the files located in the same directory) and click OK.

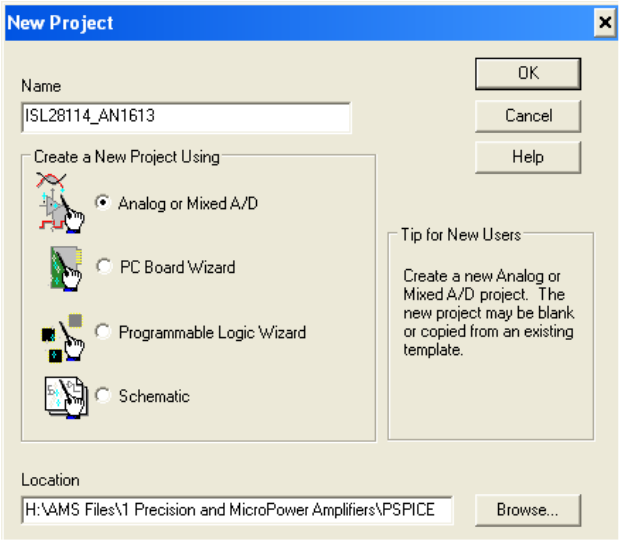

FIGURE 20. SCREEN TO SETUP NEW PROJECT

The screen shown in Figure 21 will appear. The user can select to base their new project on an existing project or start a new one. Selecting to base upon an existing project will carry over the existing project with all the simulation profiles and schematics. This can be a real time saver if the new project is very similar to an old project. In this example, we will chose to create a new project.

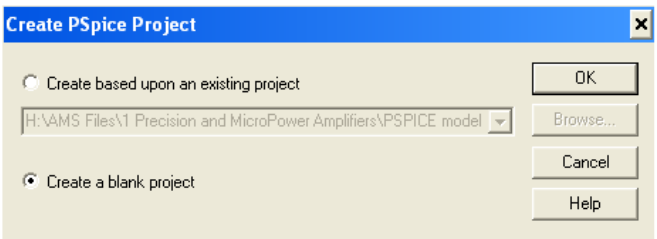

FIGURE 21. CREATING A NEW PROJECT OPTIONS

Click OK, and the screen in Figure 22 will appear. Click on the SCHEMATIC1 to open the Page tab and then Click on the page tab to open the Schematic1 page. This is where the new sub-circuit will be placed to run the simulations.

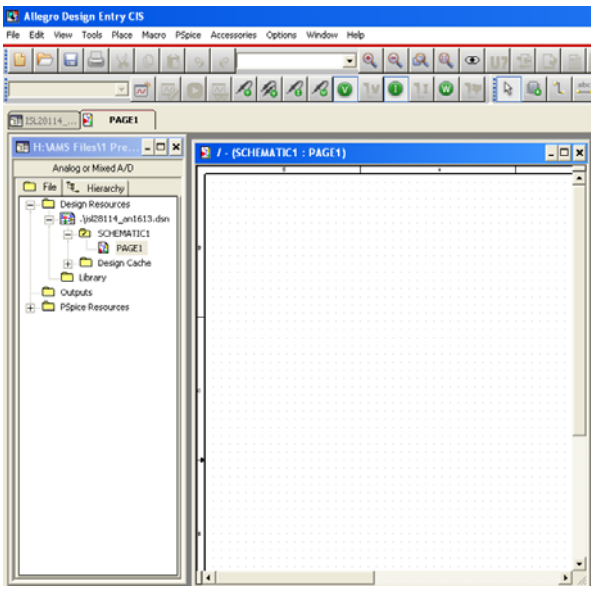

FIGURE 22. SIMULATION SCHEMATIC PAGE

Before we can place the new sub-circuit model and run a simulation, we need to set-up the simulation profile and add the library. Click on PSpice in the tool bar and select New Simulation Profile. Figure 23 will appear. Then type in any name that will help you keep track of the different simulations. Then click Create and Figure 24 will appear.

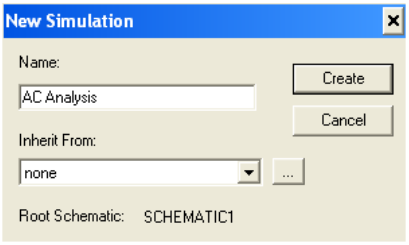

FIGURE 23. NAMING SIMULATION PROFILE

Click the Configuration Files tab. Then click on Library in the Category field (highlighted in blue). Browse to where to saved the Library file. Then click the Add to Design button. The Simulation Settings screen should look like that shown in Figure 24 with the file path name being the location of the common directory. Click the Apply button.

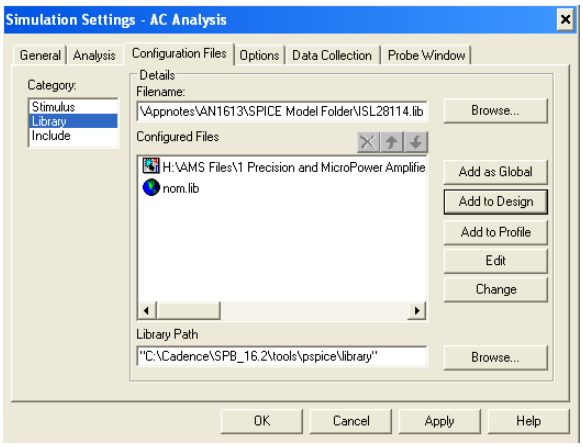

FIGURE 24. CONFIGURATION FILE TO ADD LIBRARY

Now click the analysis tab and configure the simulation as shown in Figure 25. The analysis selected for this example is an AC Sweep/Noise. Other types of analysis are: Time Domain (Transient), DC Sweep and Bias Point. Just click the down arrow in the analysis type section to access the different Analysis options. When done, click OK.

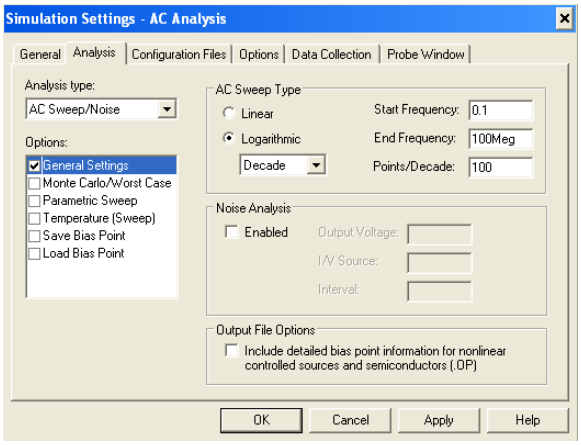

FIGURE 25. SCREEN TO SET-UP THE ANALYSIS PROFILE CONFIGURED

Add the Library .olb to the simulator. Click Place in the tool bar and select Part. This will bring up the part placement tool at the far right of the simulator as shown in Figure 26.

To add the library, click on the tab where the arrow is pointing to in Figure 26.

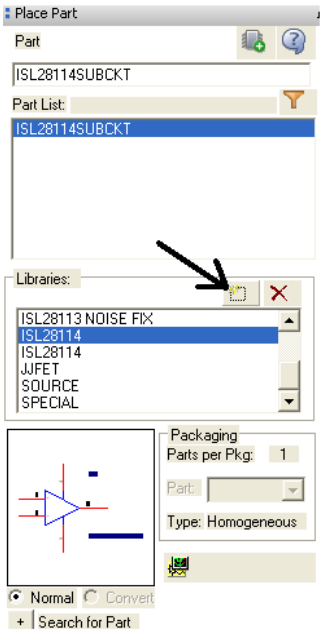

FIGURE 26. PART PLACEMENT TOOL

This will bring up the screen shown in Figure 27. Browse to where you saved the Netlist in the common directory and click Open.

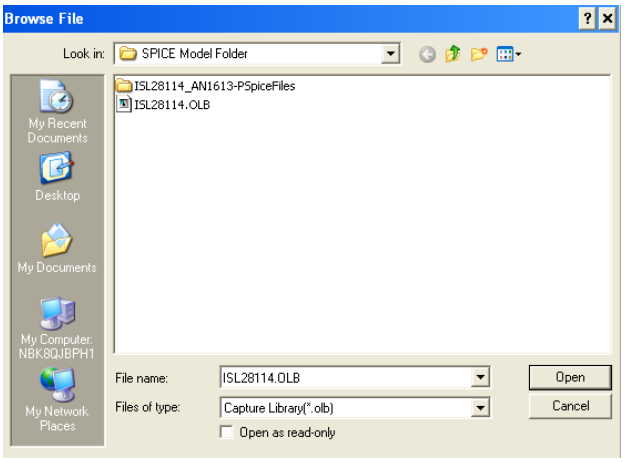

FIGURE 27. CONNECTING THE SYMBOL LIBRARY TO SIMULATOR

Now you are ready to add the sub-circuit to your simulation schematic and start your simulations.

Adding the Sub-Circuit to Your Simulation Schematic

With the .lib file added to the simulation profile and the .olb file added to the Part placement tool, your are now ready to place the Opamp sub-circuit into your simulation schematic. Figure 28 shows the part placement tool after the .olb has been added to it. Under the Libraries section, find the new .olb symbol you added in the previous step (highlighted in blue). Double click the file to add the sub-circuit to the Part list section (also highlighted in blue). Double click the Part in the part list section to add the sub-circuit to the simulation schematic.

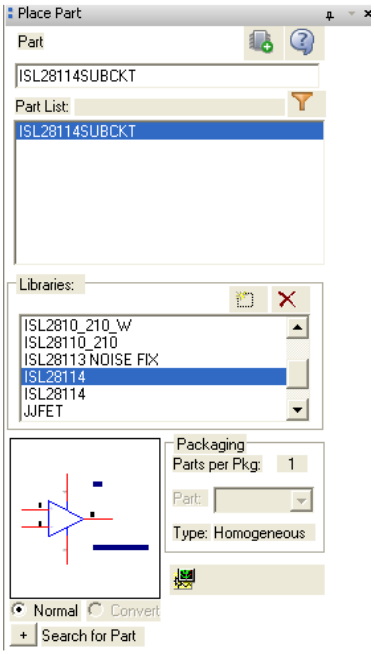

FIGURE 28. ADDING A SUB-CIRCUIT TO A SIMULATION SCHEMATIC

Figure 29 shows sub-circuit in a basic non-inverting application circuit. The simulation result showing AVOL (green trace) and Phase (pink) are shown in Figure 30.

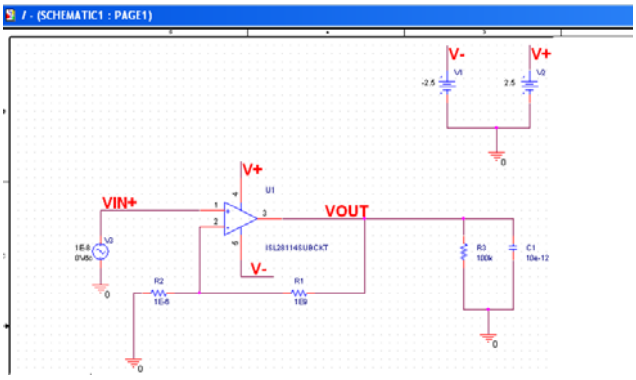

FIGURE 29. SIMULATION SCHEMATIC

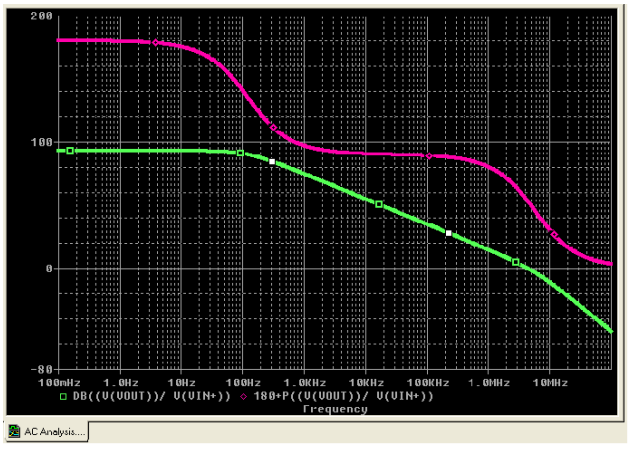

FIGURE 30. AVOL GAIN/PHASE SIMULATION RESULTS

Notice

1. Descriptions of circuits, software and other related information in this document are provided only to illustrate the operation of semiconductor products and application examples. You are fully responsible for the incorporation or any other use of the circuits, software, and information in the design of your product or system. Renesas Electronics disclaims any and all liability for any losses and damages incurred by you or third parties arising from the use of these circuits, software, or information.
2. Renesas Electronics hereby expressly disclaims any warranties against and liability for infringement or any other claims involving patents, copyrights, or other intellectual property rights of third parties, by or arising from the use of Renesas Electronics products or technical information described in this document, including but not limited to, the product data, drawings, charts, programs, algorithms, and application examples.
3. No license, express, implied or otherwise, is granted hereby under any patents, copyrights or other intellectual property rights of Renesas Electronics or others.
4. You shall not alter, modify, copy, or reverse engineer any Renesas Electronics product, whether in whole or in part. Renesas Electronics disclaims any and all liability for any losses or damages incurred by you or third parties arising from such alteration, modification, copying or reverse engineering.
5. Renesas Electronics products are classified according to the following two quality grades: “Standard” and “High Quality”. The intended applications for each Renesas Electronics product depends on the product’s quality grade, as indicated below.

   "Standard": Computers; office equipment; communications equipment; test and measurement equipment; audio and visual equipment; home electronic appliances; machine tools; personal electronic equipment; industrial robots; etc.

   "High Quality": Transportation equipment (automobiles, trains, ships, etc.); traffic control (traffic lights); large-scale communication equipment; key financial terminal systems; safety control equipment; etc.

   Unless expressly designated as a high reliability product or a product for harsh environments in a Renesas Electronics data sheet or other Renesas Electronics document, Renesas Electronics products are not intended or authorized for use in products or systems that may pose a direct threat to human life or bodily injury (artificial life support devices or systems; surgical implantations; etc.), or may cause serious property damage (space system; undersea repeaters; nuclear power control systems; aircraft control systems; key plant systems; military equipment; etc.). Renesas Electronics disclaims any and all liability for any damages or losses incurred by you or any third parties arising from the use of any Renesas Electronics product that is inconsistent with any Renesas Electronics data sheet, user’s manual or other Renesas Electronics document.
6. When using Renesas Electronics products, refer to the latest product information (data sheets, user’s manuals, application notes, “General Notes for Handling and Using Semiconductor Devices” in the reliability handbook, etc.), and ensure that usage conditions are within the ranges specified by Renesas Electronics with respect to maximum ratings, operating power supply voltage range, heat dissipation characteristics, installation, etc. Renesas Electronics disclaims any and all liability for any malfunctions, failure or accident arising out of the use of Renesas Electronics products outside of such specified ranges.
7. Although Renesas Electronics endeavors to improve the quality and reliability of Renesas Electronics products, semiconductor products have specific characteristics, such as the occurrence of failure at a certain rate and malfunctions under certain use conditions. Unless designated as a high reliability product or a product for harsh environments in a Renesas Electronics data sheet or other Renesas Electronics document, Renesas Electronics products are not subject to radiation resistance design. You are responsible for implementing safety measures to guard against the possibility of bodily injury, injury or damage caused by fire, and/or danger to the public in the event of a failure or malfunction of Renesas Electronics products, such as safety design for hardware and software, including but not limited to redundancy, fire control and malfunction prevention, appropriate treatment for aging degradation or any other appropriate measures. Because the evaluation of microcomputer software alone is very difficult and impractical, you are responsible for evaluating the safety of the final products or systems manufactured by you.
8. Please contact a Renesas Electronics sales office for details as to environmental matters such as the environmental compatibility of each Renesas Electronics product. You are responsible for carefully and sufficiently investigating applicable laws and regulations that regulate the inclusion or use of controlled substances, including without limitation, the EU RoHS Directive, and using Renesas Electronics products in compliance with all these applicable laws and regulations. Renesas Electronics disclaims any and all liability for damages or losses occurring as a result of your noncompliance with applicable laws and regulations.
9. Renesas Electronics products and technologies shall not be used for or incorporated into any products or systems whose manufacture, use, or sale is prohibited under any applicable domestic or foreign laws or regulations. You shall comply with any applicable export control laws and regulations promulgated and administered by the governments of any countries asserting jurisdiction over the parties or transactions.
10. It is the responsibility of the buyer or distributor of Renesas Electronics products, or any other party who distributes, disposes of, or otherwise sells or transfers the product to a third party, to notify such third party in advance of the contents and conditions set forth in this document.
11. This document shall not be reprinted, reproduced or duplicated in any form, in whole or in part, without prior written consent of Renesas Electronics.
12. Please contact a Renesas Electronics sales office if you have any questions regarding the information contained in this document or Renesas Electronics products.

(Note 1) “Renesas Electronics” as used in this document means Renesas Electronics Corporation and also includes its directly or indirectly controlled subsidiaries.

(Note 2) “Renesas Electronics product(s)” means any product developed or manufactured by or for Renesas Electronics.

(Rev.4.0-1 November 2017)

RENESAS

SALES OFFICES

Renesas Electronics Corporation

http://www.renesas.com

Refer to "http://www.renesas.com/" for the latest and detailed information.

Renesas Electronics America Inc.
1001 Murphy Ranch Road, Milpitas, CA 95035, U.S.A.
Tel: +1-408-432-8888, Fax: +1-408-434-5351

Renesas Electronics Canada Limited
9251 Yonge Street, Suite 8309 Richmond Hill, Ontario Canada L4C 9T3
Tel: +1-905-237-2004

Renesas Electronics Europe Limited
Dukes Meadow, Millboard Road, Bourne End, Buckinghamshire, SL8 5FH, U.K
Tel: +44-1628-651-700, Fax: +44-1628-651-804

Renesas Electronics Europe GmbH
Arcadiastrasse 10, 40472 Düsseldorf, Germany
Tel: +49-211-6503-0, Fax: +49-211-6503-1327

Renesas Electronics (China) Co., Ltd.
Room 1709 Quantum Plaza, No.27 ZhichunLu, Haidian District, Beijing, 100191 P. R. China
Tel: +86-10-8235-1155, Fax: +86-10-8235-7679

Renesas Electronics (Shanghai) Co., Ltd.
Unit 301, Tower A, Central Towers, 555 Langao Road, Putuo District, Shanghai, 200333 P. R. China
Tel: +86-21-2226-0888, Fax: +86-21-2226-0999

Renesas Electronics Hong Kong Limited
Unit 1601-1611, 16/F., Tower 2, Grand Century Place, 193 Prince Edward Road West, Mongkok, Kowloon, Hong Kong
Tel: +852-2265-6688, Fax: +852 2886-9022

Renesas Electronics Taiwan Co., Ltd.
13F, No. 363, Fu Shing North Road, Taipei 10543, Taiwan
Tel: +886-2-8175-9600, Fax: +886 2-8175-9670

Renesas Electronics Singapore Pte. Ltd.
80 Bendemeer Road, Unit #06-02 Hyflux Innovation Centre, Singapore 339949
Tel: +65-6213-0200, Fax: +65-6213-0300

Renesas Electronics Malaysia Sdn.Bhd.
Unit 1207, Block B, Menara Amcorp, Amcorp Trade Centre, No. 18, Jln Persiaran Barat, 46050 Petaling Jaya, Selangor Darul Ehsan, Malaysia
Tel: +60-3-7955-9390, Fax: +60-3-7955-9510

Renesas Electronics India Pvt. Ltd.
No.777C, 100 Feet Road, HAL 2nd Stage, Indiranagar, Bangalore 560 038, India
Tel: +91-80-67208700, Fax: +91-80-67208777

Renesas Electronics Korea Co., Ltd.
17F, KAMCO Yangjae Tower, 262, Gangnam-daero, Gangnam-gu, Seoul, 06265 Korea
Tel: +82-2-558-3737, Fax: +82-2-558-5338

Colophon 7.0