

Precision DAC SPICE Models

Texas Instruments provides SPICE models for a number of parts, including [SPICE models for precision digital-to-analog converters \(DACs\)](#). These models come in two basic types: TINA-TI models and PSPICE models. The purpose of this document is to introduce the user to both types of models, and explain how to begin simulating with them in their respective environments.

1 TINA-TI Models

TINA-TI is a free software from Texas Instruments that allows for circuit simulation with both passive components and semiconductors. In order to use the models associated with TINA-TI, download the software from <http://www.ti.com/tool/tina-ti> and install it on your PC. Learn the basics of simulating with TINA-TI using either the user's guide found in the help menu of the program or *Getting Started with TINA-TI: A Quick Start Guide* ([SBOU052](#)).

1.1 Importing a TINA-TI Model

There are two ways to import a model into TINA-TI: through a .tsm macro file or through a .tsc schematic file. Both of these methods are described in [Section 1.1.1](#) and [Section 1.1.2](#).

1.1.1 Macro Import via .tsm File

Start by creating a new or opening an existing TINA-TI schematic. Open the "Insert" menu and select "Macro...". In the file explorer, navigate to and select the .tsm macro file for the precision DAC that you would like to import, then click the "Open" button. Finally, clicking on the schematic will place the symbol for the DAC.

1.1.2 Schematic Import via .tsc File

After opening TINA-TI, go to the "File" menu and select "Open". In the file explorer, navigate to and select the .tsc schematic file for the precision DAC that you would like to import, then click the "Open" button. The schematic file that loads will include the model for the chosen DAC as well as some circuit components connected to it to form a simple test-bench.

1.2 Using the TINA-TI Models

1.2.1 Parallel and Serial Interfaces

Depending on the type of model you downloaded, the precision DAC model will have either a parallel or serial interface. An example of a parallel interface precision DAC model and surrounding test circuitry is pictured in [Figure 1](#).

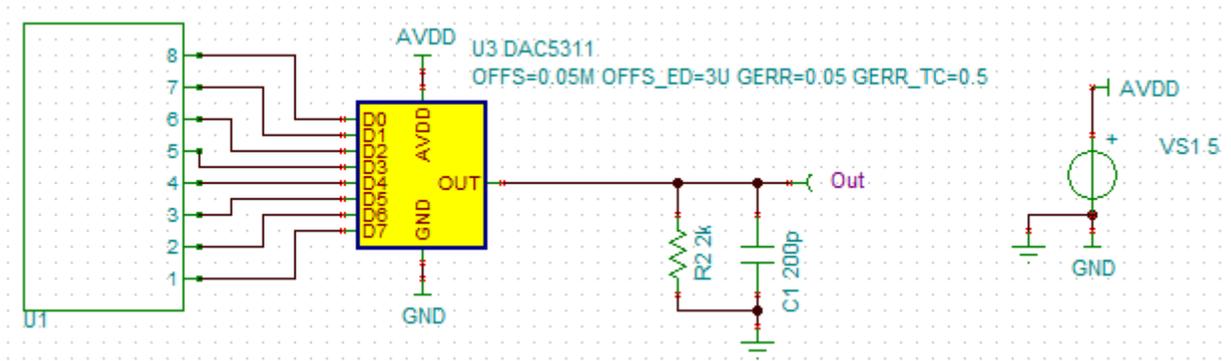


Figure 1. Parallel Precision DAC Model in TINA-TI Schematic

To communicate to this model, you can use a data generator from the "Sources" tab in TINA-TI, also pictured in [Figure 1](#) on the left side. The example shown here uses an 8-bit data generator to set the inputs to the [DAC5311](#), an 8-bit DAC. Double clicking on the data generator symbol and clicking on the "..." on the "Pattern" line will allow you to change the inputs to the DAC as well as control how quickly the inputs change during a simulation.

An example of a serial interface model and surrounding test circuitry is pictured in [Figure 2](#).

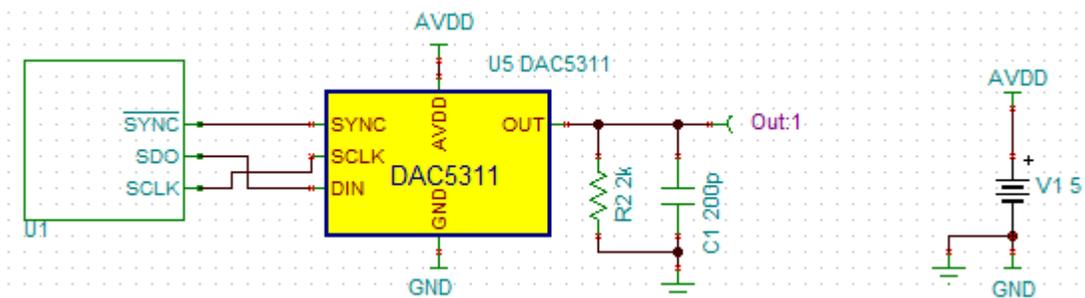


Figure 2. Serial Precision DAC Model in TINA-TI Schematic

Communication with this model can be done via an SPI generator, also available from the "Sources" tab in TINA-TI. This SPI generator is likewise pictured in [Figure 2](#) on the left side. The SPI generator output is configured in the same fashion as the parallel data generator from the parallel model example.

1.2.2 TINA-TI Parameters

The precision DAC SPICE models have user-definable parameters that will adjust the behavior of the DAC to closely replicate the errors in a real device. In order to change these parameters, start by double-clicking on the DAC symbol in the schematic. Click in the SubCkt-Parameters section in the window that pops up. Then click on the "..." button that appears on the right side. In the window that appears (shown in [Figure 3](#)) you can edit the parameters pertaining to gain and offset errors.

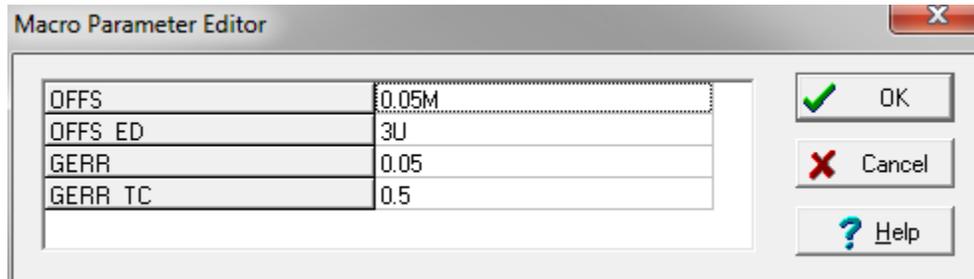


Figure 3. Macro Parameter Editor Window

When using a serial model, you will need to right click on the DAC symbol and select "Enter Macro", then do the same steps as in the first paragraph of [Section 1.2.2](#), in the schematic that appears.

All of the values use standard units, but you can use the modifiers "M" or "U" to denote milli- and micro-values, respectively. Additionally, some common parameter coefficients are:

- "_ED" - Error drift of spec in [unit]/°C
- "_TC" - Temperature coefficient in ppm/°C

For more information, consult the data sheet for the chosen precision DAC as the parameters for the model correspond to values in the specification table.

Click "OK" twice to get back to the schematic, and your parameters will be updated and reflected on the schematic. The changes to the parameters will then affect all subsequent simulations.

2 PSPICE Models

Texas Instruments supplies PSPICE models for some precision DACs for those designers who are more accustomed to PSPICE simulation software. The instructions for use of these models assume the use of the Cadence OrCAD suite of products.

2.1 Importing a PSPICE Model Into OrCAD

Start the *PSpice Model Editor* software. Then go to the "File" menu and choose "Open". In the file browser, navigate to and select the .lib file that you downloaded for the precision DAC PSPICE model, then click "Open". You can now view the contents of each of the models included in the library. Next, start the generation of the .olb file by going to the "File" menu and selecting "Export to Capture Part Library". On the next screen, the default selections will generate the .olb file in the same directory and with the same name as the .lib file you are generating from. Click "OK" and the .olb file will be generated. You can now close out of the *PSpice Model Editor*.

Open the OrCAD schematic capture program, then either create a new project or open an existing one. On the schematic sheet you wish to place the precision DAC on, press the "P" key to place a part. The "Place Part" toolbar will appear to the right. Click the button highlighted in [Figure 4](#) to add a library.

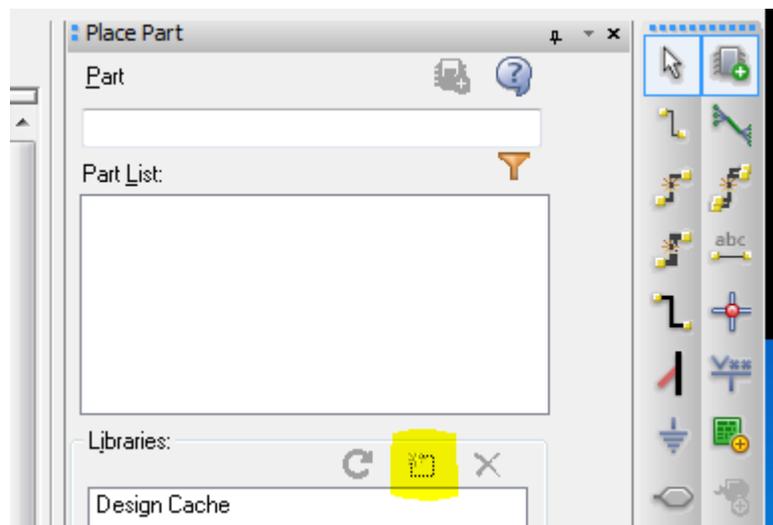


Figure 4. Place Part Window

In the file explorer, navigate to the .olb file you just created, select and open it. The PSPICE model is now available to be placed into the design. Adding the .lib file to the project *Design Resources* library is also recommended.

2.2 PSPICE Parameters

The precision DAC SPICE models have user-definable parameters that will adjust the behavior of the DAC to closely replicate the errors in a real device. In order to change these parameters, start by double-clicking on the DAC symbol in the schematic. The parameters for the model appear horizontally across the screen. From here you can edit the parameter values for the model. Refer to [Section 1.2.2](#) on TINA-TI SPICE models for an explanation of the parameters.

IMPORTANT NOTICE FOR TI DESIGN INFORMATION AND RESOURCES

Texas Instruments Incorporated ("TI") technical, application or other design advice, services or information, including, but not limited to, reference designs and materials relating to evaluation modules, (collectively, "TI Resources") are intended to assist designers who are developing applications that incorporate TI products; by downloading, accessing or using any particular TI Resource in any way, you (individually or, if you are acting on behalf of a company, your company) agree to use it solely for this purpose and subject to the terms of this Notice.

TI's provision of TI Resources does not expand or otherwise alter TI's applicable published warranties or warranty disclaimers for TI products, and no additional obligations or liabilities arise from TI providing such TI Resources. TI reserves the right to make corrections, enhancements, improvements and other changes to its TI Resources.

You understand and agree that you remain responsible for using your independent analysis, evaluation and judgment in designing your applications and that you have full and exclusive responsibility to assure the safety of your applications and compliance of your applications (and of all TI products used in or for your applications) with all applicable regulations, laws and other applicable requirements. You represent that, with respect to your applications, you have all the necessary expertise to create and implement safeguards that (1) anticipate dangerous consequences of failures, (2) monitor failures and their consequences, and (3) lessen the likelihood of failures that might cause harm and take appropriate actions. You agree that prior to using or distributing any applications that include TI products, you will thoroughly test such applications and the functionality of such TI products as used in such applications. TI has not conducted any testing other than that specifically described in the published documentation for a particular TI Resource.

You are authorized to use, copy and modify any individual TI Resource only in connection with the development of applications that include the TI product(s) identified in such TI Resource. NO OTHER LICENSE, EXPRESS OR IMPLIED, BY ESTOPPEL OR OTHERWISE TO ANY OTHER TI INTELLECTUAL PROPERTY RIGHT, AND NO LICENSE TO ANY TECHNOLOGY OR INTELLECTUAL PROPERTY RIGHT OF TI OR ANY THIRD PARTY IS GRANTED HEREIN, including but not limited to any patent right, copyright, mask work right, or other intellectual property right relating to any combination, machine, or process in which TI products or services are used. Information regarding or referencing third-party products or services does not constitute a license to use such products or services, or a warranty or endorsement thereof. Use of TI Resources 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.

TI RESOURCES ARE PROVIDED "AS IS" AND WITH ALL FAULTS. TI DISCLAIMS ALL OTHER WARRANTIES OR REPRESENTATIONS, EXPRESS OR IMPLIED, REGARDING TI RESOURCES OR USE THEREOF, INCLUDING BUT NOT LIMITED TO ACCURACY OR COMPLETENESS, TITLE, ANY EPIDEMIC FAILURE WARRANTY AND ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, AND NON-INFRINGEMENT OF ANY THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

TI SHALL NOT BE LIABLE FOR AND SHALL NOT DEFEND OR INDEMNIFY YOU AGAINST ANY CLAIM, INCLUDING BUT NOT LIMITED TO ANY INFRINGEMENT CLAIM THAT RELATES TO OR IS BASED ON ANY COMBINATION OF PRODUCTS EVEN IF DESCRIBED IN TI RESOURCES OR OTHERWISE. IN NO EVENT SHALL TI BE LIABLE FOR ANY ACTUAL, DIRECT, SPECIAL, COLLATERAL, INDIRECT, PUNITIVE, INCIDENTAL, CONSEQUENTIAL OR EXEMPLARY DAMAGES IN CONNECTION WITH OR ARISING OUT OF TI RESOURCES OR USE THEREOF, AND REGARDLESS OF WHETHER TI HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

You agree to fully indemnify TI and its representatives against any damages, costs, losses, and/or liabilities arising out of your non-compliance with the terms and provisions of this Notice.

This Notice applies to TI Resources. Additional terms apply to the use and purchase of certain types of materials, TI products and services. These include; without limitation, TI's standard terms for semiconductor products (<http://www.ti.com/sc/docs/stdterms.htm>), [evaluation modules](#), and [samples](http://www.ti.com/sc/docs/sampterm.htm) (<http://www.ti.com/sc/docs/sampterm.htm>).

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265
Copyright © 2017, Texas Instruments Incorporated