

# ADC Input Circuit Evaluation for C2000 MCUs (using TINA-TI simulation tool)



Devin Cottier

## ABSTRACT

The analog-to-digital converter (ADC) is a key module used to sense feedback and monitoring signals in a real-time control application. However, without careful design and evaluation of the circuits driving the ADC input, significant sample-and-hold circuit (S+H) settling errors can occur, resulting in poor performance of the ADC sensing and thus the system. This application report will identify the resources that can be used to understand ADC input settling and design appropriate ADC driving circuits as well as how to evaluate these resources in the specific context of TMS320F2837xD, TMS320F2837xS, TMS320F2838x, TMS320F2807x, TMS320F28004x, TMS320F28002x, TMS320F28003x, TMS320F280013x, and TMS320F280015x series microcontrollers.

Project collateral and source code discussed in this application report can be downloaded from the following URL: <http://www.ti.com/lit/zip/spract6>.

## Table of Contents

<b>1 Introduction</b> .....	3
1.1 Mechanism of ADC Input Settling.....	3
1.2 Symptoms of Inadequate Settling.....	3
1.3 Resources.....	3
<b>2 Input Settling Design Steps</b> .....	6
2.1 Select the ADC.....	7
2.2 Find the Minimum Op-Amp Bandwidth and RC Filter Ranges.....	7
2.3 Select an Op-Amp.....	9
2.4 Verify the Op-Amp Model.....	9
2.5 Build the ADC Input Model.....	9
2.6 Refine RC Filter Values Via Simulation.....	12
2.7 Perform Final Simulations.....	12
2.8 Input Design Worksheet.....	12
<b>3 Example Circuit Design</b> .....	13
3.1 Select the ADC.....	13
3.2 Find the Minimum Op-Amp Bandwidth and RC Filter Ranges.....	14
3.3 Verify the Op-amp Model.....	14
3.4 Build the ADC Input Model.....	15
3.5 DC Node Analysis.....	16
3.6 Refine RC Filter Values Via Simulation (Part 1).....	17
3.7 Refine RC Filter Values Via Simulation (Part 2).....	18
3.8 Refine RC Filter Values Via Simulation (Part 3).....	20
3.9 Further Refinement.....	20
3.10 Further Simulations.....	20
3.11 Completed Worksheet.....	21
<b>4 Working With Existing Circuits or Additional Constraints</b> .....	22
4.1 Existing Circuits.....	22
4.2 Pre-Selected Op-Amp.....	23
4.3 Pre-Selected $R_s$ and $C_s$ Values.....	24
<b>5 Summary</b> .....	25
<b>6 References</b> .....	25
<b>7 Revision History</b> .....	25

## List of Figures

Figure 1-1. Settling of the ADC S+H Capacitor.....	3
Figure 2-1. Screenshot of Analog Engineer's Calculator.....	7
Figure 2-2. F28004x ADC Input Model.....	10
Figure 2-3. Example Timings for $t_{acq}$ (Piecewise Linear).....	11
Figure 2-4. Example Timings for $t_{dis}$ (Piecewise Linear).....	12
Figure 3-1. F280049 Example Analog Engineer's Calculator Output.....	14
Figure 3-2. F280049 Example TINA-TI Circuit.....	15
Figure 3-3. F280049 Example $T_{acq}$ Source Timings.....	16
Figure 3-4. F280049 Example $T_{dis}$ Source Timings.....	16
Figure 3-5. F280049 Example Transient Analysis Configuration.....	17
Figure 3-6. F280049 Example Initial Transient Results.....	17
Figure 3-7. Sweep Button.....	18
Figure 3-8. F280049 Example Sweep Control Dialog.....	18
Figure 3-9. F280049 Example $R_s$ Initial Sweep Results.....	19
Figure 3-10. F280049 Example $R_s$ Fine Sweep Results.....	19
Figure 3-11. F280049 Example Final Transient Results.....	20
Figure 4-1. Example Circuit Utilizing Charge Sharing.....	22
Figure 4-2. $C_s$ Voltage With $R_s = 100 \Omega$ .....	23
Figure 4-3. $C_s$ Voltage With $R_s = 1k \Omega$ .....	23
Figure 4-4. Analog Engineer's Calculator Output for OPA2320.....	24

## List of Tables

Table 1-1. Series of Devices.....	5
Table 2-1. Calculator Input Type Map.....	8
Table 2-2. ADC Input Settling Design Worksheet.....	13
Table 3-1. ADC Input Settling Design Worksheet (F280049 Example).....	21

## Trademarks

TINA-TI™ is a trademark of Texas Instruments.

PSPICE® is a registered trademark of Cadence Design Systems, Inc.

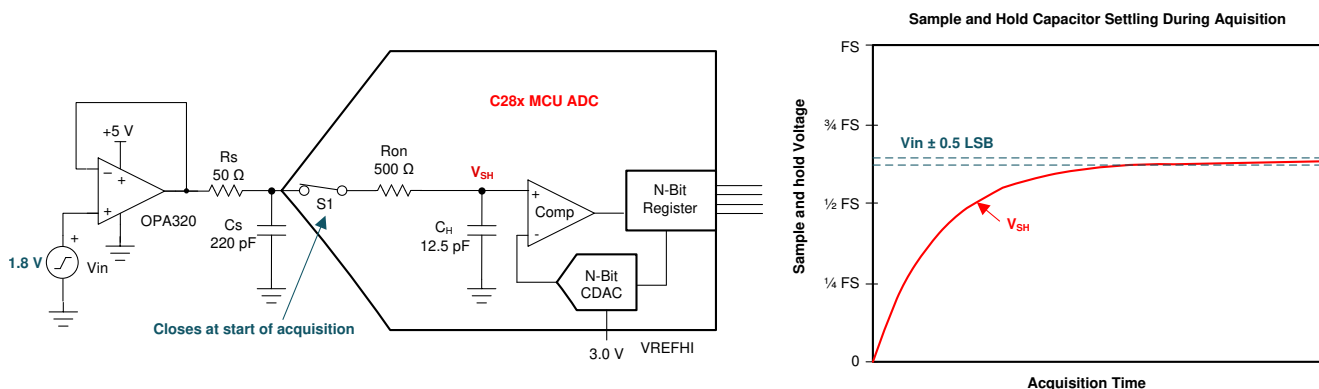
All trademarks are the property of their respective owners.

## 1 Introduction

When designing a system utilizing C2000 real-time MCUs, evaluating the input circuits that are driving the ADC for proper settling is a necessary step. Inadequate settling can lead to a variety of sampling issues, most commonly signal distortion and memory-crosstalk. This application report explores the methodology presented in the seven-part video series on ADC input settling by TI precision labs: [TI Precision Labs - SAR ADC Input Driver Design](#) in the specific context of the ADCs on C2000 real-time MCUs.

### 1.1 Mechanism of ADC Input Settling

To convert a sensed analog voltage to a digital conversion result, the ADC first must accurately capture the applied input voltage into its sample-and-hold circuit (S+H). As shown in [Figure 1-1](#), this entails charging the internal ADC S+H capacitor ( $C_h$ ) to within some acceptable tolerance (typically 0.5 LSBs) of the applied voltage within the configured acquisition window time (also referred to as the S+H time).



**Figure 1-1. Settling of the ADC S+H Capacitor**

Quickly charging  $C_h$  to the applied voltage is complicated by the finite bandwidth and settling time of the external ADC driver circuit and of the settling time of the internal ADC S+H circuit. In [Figure 1-1](#), the driver is shown as an op-amp (OPA320), which has a finite bandwidth, and the driver circuit also has intentionally placed source resistance ( $R_s$ ) and intentionally placed source capacitance ( $C_s$ ) which have a finite settling time determined by their RC time constant. Note that other circuit topologies are possible for driving the ADC, and these circuits may have additional components that need to be modeled to ensure appropriate settling time. These components could include unintentional parasitics such as the output impedance of a sensor or the effective source resistance of a voltage divider. [Figure 1-1](#) also shows that the ADC has an internal parasitic switch resistance ( $R_{on}$ ). This, along with  $C_h$ , provides an additional RC time constant that limits settling speed.

### 1.2 Symptoms of Inadequate Settling

Once a voltage has been captured into the S+H capacitor, the ADC will translate this voltage into a digital conversion result during the conversion phase. The CPU can then use this result to control or monitor the system. However, if the captured voltage does not accurately represent the applied voltage due to settling error, the final conversion result will have errors even if the ADC conversion process is perfect.

These settling errors will manifest differently depending on whether the ADC is sampling the same channel repeatedly or scanning through multiple channels in a sequence. The settling errors will also manifest differently depending on the starting voltage on the S+H capacitor at the beginning of the acquisition phase. Some ADC architectural implementations will have a starting S+H voltage close to the previously sampled voltage while other architectures will usually start the acquisition phase with a discharged S+H capacitor.

### 1.3 Resources

The following tools and materials are useful for ADC driver circuit design and evaluation for input settling.

### 1.3.1 TINA-TI SPICE-Based Analog Simulation Program

TI provides the TINA-TI application to TI customers to allow easy SPICE-based simulation of circuits utilizing TI products. Because this application report uses PSpice for TI instead of TINA-TI to evaluate circuit settling performance, it is not necessary to ensure TINA-TI is downloaded and installed before proceeding. However, the TINA-TI application and additional resources are still made available.

Link to Tool Folder: [TINA-TI™](#)

### 1.3.2 PSpice for TI Design and Simulation Tool

TI provides the PSpice for TI application to TI customers to allow easy SPICE-based simulation of circuits utilizing TI products. This application report utilizes this program to evaluate circuit settling performance. Ensure this application is downloaded and installed before proceeding.

Link to Tool Folder: [PSpice® for TI](#)

Link to PSpice for TI Training Series: [Explore PSpice for TI](#)

Link to TI E2E Support Forum: [Welcome to TI E2E Design Support Forums](#)

Link to Cadence PSpice for TI Forum: [Welcome to PSpice Forum](#)

### 1.3.3 TI Precision Labs - SAR ADC Input Driver Design Series

TI Precision Labs has provided an excellent eight-part video series that demonstrates how to design the input drivers for a SAR ADC. Reviewing this material will help to further understand the high-speed op-amp based signal conditioning design process and options.

Link to Video Training Series: [TI Precision Labs - SAR ADC Input Driver Design](#)

### 1.3.4 Analog Engineer's Calculator

The analog engineer's calculator tool provides a variety of very useful GUI-based calculation tabs to assist with common analog circuit design tasks. The high-speed op-amp design methodology takes advantage of the Data Converters → ADC SAR Drive calculator.

Link to Tool Folder: [Analog Engineer's Calculator](#)

### 1.3.5 Related Application Reports

TI provides several application reports that pertain to the design and evaluation of ADC input circuits on C2000 real-time MCU devices. These application reports are listed along with brief descriptions.

Application Report Title	Schematic Capture and Simulation Tool	Purpose
<a href="#">ADC Input Circuit Evaluation for C2000 MCUs (TINA-TI)</a>	TINA-TI	Describes how to design and evaluate a traditional high-speed ADC driving circuit. Input circuit includes an op-amp. Appropriate when sampling a high bandwidth signal near the maximum sample rate of the ADC.
<a href="#">ADC Input Circuit Evaluation for C2000 MCUs (PSpice for TI)</a>	PSpice for TI	
<a href="#">Charge-Sharing Driving Circuits for C2000 ADCs (TINA-TI)</a>	TINA-TI	Describes how to design and evaluate a charge-sharing ADC driving circuit. Input circuit may or may not include an op-amp. Appropriate when sampling a signal with a sufficiently low bandwidth and sample rate.
<a href="#">Charge-Sharing Driving Circuits for C2000 ADCs (PSpice for TI)</a>	PSpice for TI	
<a href="#">Methods for Mitigating ADC Memory Cross-Talk</a>	TINA-TI	Describes strategies to mitigate memory cross-talk error when the input circuit design does not achieve adequate settling. Applicable under special circumstances.

### 1.3.6 TINA-TI ADC Input Models

This application report is bundled with pre-generated ADC input models for several families of devices. The families of devices are listed alongside the ADC input models associated with each family and a brief description of each ADC input model.

For C2000 devices that are not listed in [Table 1-1](#), the input models can still be used as long as the signal mode matches (use a single-ended model for single-ended ADC operation or use a differential model for differential ADC operation). The  $R_{on}$  and  $C_h$  values also need to be updated to match the values specified in the device-specific data manual.

**Table 1-1. Series of Devices**

Series of Devices	ADC Input Model	Description
<a href="#">TMS320F2807x</a>	F2807x_Ext	F2807x ADC input model External reference of 3.0 V 12-bit resolution Single-ended input mode
<a href="#">TMS320F2837xD</a> , <a href="#">TMS320F2837xS</a>	F2837x_Ext_12BSE	F2837x ADC input model External reference of 3.0 V 12-bit resolution Single-ended input mode
	F2837x_Ext_16BDE	F2837x ADC input model External reference of 3.0 V 16-bit resolution Differential input mode
<a href="#">TMS320F2838xD</a> , <a href="#">TMS320F2838xS</a>	F2838x_Ext_12BSE	F2838x ADC input model External reference of 3.0 V 12-bit resolution Single-ended input mode
	F2838x_Ext_16BSE	F2838x ADC input model External reference of 3.0 V 16-bit resolution Single-ended input mode
	F2838x_Ext_16BDE	F2838x ADC input model External reference of 3.0 V 16-bit resolution Differential input mode
<a href="#">TMS320F28002x</a>	F28002x_Ext	F28002x ADC input model External reference of 3.0 V 12-bit resolution Single-ended input mode
	F28002x_Int25	F28002x ADC input model External reference of 2.5 V 12-bit resolution Single-ended input mode
	F28002x_Int33	F28002x ADC input model External reference of 3.3 V 12-bit resolution Single-ended input mode
<a href="#">TMS320F28004x</a>	F28004x_Ext	F28004x ADC input model External reference of 3.0 V 12-bit resolution Single-ended input mode
	F28004x_Int25	F28004x ADC input model Internal reference of 2.5 V 12-bit resolution Single-ended input mode
	F28004x_Int33	F28004x ADC input model Internal reference of 3.3 V 12-bit resolution Single-ended input mode

**Table 1-1. Series of Devices (continued)**

Series of Devices	ADC Input Model	Description
TMS320F28003x	F28003x_Ext	F28003x ADC input model External reference of 3.0 V 12-bit resolution Single-ended input mode
	F28003x_Int25	F28003x ADC input model Internal reference of 2.5 V 12-bit resolution Single-ended input mode
	F28003x_Int33	F28003x ADC input model Internal reference of 3.3 V 12-bit resolution Single-ended input mode
TMS320F280013x	F280013x_Ext	F280013x ADC input model External reference of 3.0 V 12-bit resolution Single-ended input mode
	F280013x_Int25	F280013x ADC input model Internal reference of 2.5 V 12-bit resolution Single-ended input mode
	F280013x_Int33	F280013x ADC input model Internal reference of 3.3 V 12-bit resolution Single-ended input mode
TMS320F280015x	F280015x_Ext	F280015x ADC input model External reference of 3.0 V 12-bit resolution Single-ended input mode
	F280015x_Int25	F280015x ADC input model Internal reference of 2.5 V 12-bit resolution Single-ended input mode
	F280015x_Int33	F280015x ADC input model Internal reference of 3.3 V 12-bit resolution Single-ended input mode

**CAUTION**

While the ADC input models for multiple C2000 devices are bundled with this application report, input model component values should always first be verified with the device-specific data manual before proceeding. For more information, see [Section 2.5](#).

## 2 Input Settling Design Steps

The [TI Precision Labs - SAR ADC Input Driver Design](#) methodology to design an ideal ADC input driver can be broken into the steps below. Subsequent sections will discuss how these steps can be applied to C2000 ADCs.

1. Select the ADC.
2. Find the minimum op-amp bandwidth and RC filter ranges.
3. Select an op-amp.
4. Verify the op-amp model.
5. Build the ADC input model.
6. Refine RC filter values via simulation.
7. Perform final simulations

## 2.1 Select the ADC

It is assumed that you have already selected a C28x based microcontroller with one or more integrated ADCs modules. The C28x product search can be used to select an MCU: [C2000 real-time control MCU - Products](#) and the C2000 peripheral guide enumerates the differences between peripherals on different devices (including the ADC): [C2000 Real-Time Control MCU Peripherals Reference Guide](#).

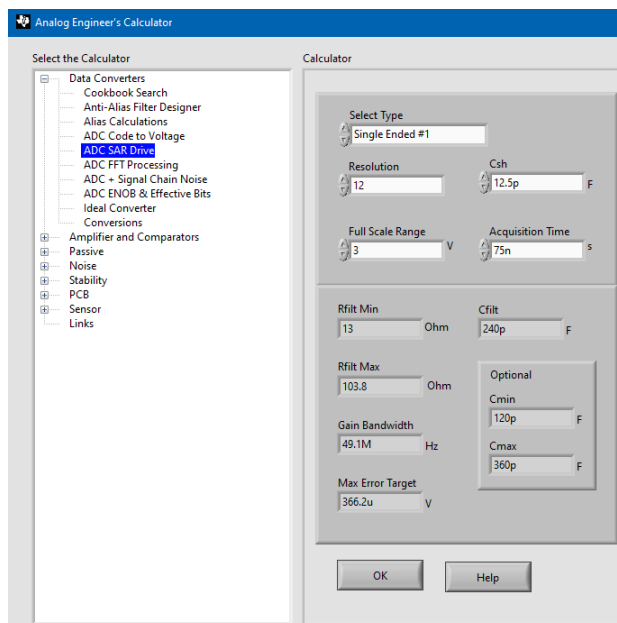
Once a C2000 device has been selected, you should locate the following information in the device-specific data manual before proceeding.

Data Manual Information	Location	Comments
Minimum sample window duration	"ADC Operating Conditions" table in the "ADC Electrical Data and Timing" section	Some devices support multiple resolutions modes and signaling modes; ensure that the parameter correspond to the mode you intend to use.
ADC input model	"Input Model Parameters" table in the "ADC Input Model" section	Some devices support multiple resolutions modes and signaling modes; ensure that the model correspond to the mode you intend to use.
Specific parasitic capacitance for selected ADC pin	"Per-Channel Parasitic Capacitance" table in the "ADC Input Model" section	If channel assignment is not yet known, use the value for ADCIN15
ADC timings	"ADC Timings" table in the "ADC Timing Diagrams" section	Some devices support multiple resolutions modes and signaling modes; ensure that the timings correspond to the mode you intend to use. You will also need to know your intended CPU clock (SYSCLK) and ADC clock (ADCCLK) rate. If not yet known, use the maximum ADCCLK value given in the "ADC Operating Conditions" table along with the maximum SYSCLK for the device.

## 2.2 Find the Minimum Op-Amp Bandwidth and RC Filter Ranges

Once you have located the detailed ADC information in the device-specific data manual, you can use the [Analog Engineer's Calculator](#) to determine the minimum required driving op-amp bandwidth as well as the potential ranges of the external source resistor and external source capacitor to put on the ADC input.

Open the Analog Engineer's Calculator application and select the calculator Data Converters -> ADC SAR Drive calculator. This will give a calculator window similar to that shown in [Figure 2-1](#).



**Figure 2-1. Screenshot of Analog Engineer's Calculator**

Once the correct calculator is open, enter the input information.

### 2.2.1 Select Type

C2000 ADCs can be configured as either single-ended input mode or differential input mode depending on the particular device. The calculator allows several input mode types. [Table 2-1](#) maps the C2000 input modes to calculator selection types.

**Table 2-1. Calculator Input Type Map**

C2000 Signal Mode	Calculator Input Type Selection
Single-ended	"Single Ended #1"
Differential	"Differential"

### 2.2.2 Resolution

Enter the resolution of the ADC mode to be used in bits. For C2000 ADCs this will usually be 12-bits or 16-bits.

#### Note

Usually the actual resolution of the ADC will be selected. However, in cases where an ADC channel does not need to utilize the full resolution of the ADC, a lower value can be entered to relax the input settling design requirements.

For example:

Resolution = 12 bits,  $C_{sh} = 12.5$  pF, Full-scale range = 3.0 V, Acquisition duration = 100 ns

- Full resolution of 12 bits requires minimum op-amp bandwidth of 37 MHz and settling to better than 0.366 mV
- Reduced resolution of 10 bits (approximately 0.1% settling) requires a minimum op-amp bandwidth of 28 MHz and settling to better than 1.465 mV

### 2.2.3 $C_{sh}$

Enter the value of the ADC S+H capacitor. This can be found as the parameter " $C_h$ " in the *Input Model Parameters* table in the device-specific data manual. Different resolutions and reference modes may have different input model parameters, so ensure that the parameter is selected that corresponds to the intended ADC use mode.

### 2.2.4 Full-Scale Range

Enter the voltage corresponding to the full-scale range of the ADC. When using external reference mode, this corresponds to the voltage supplied on the VREFHI pin of the ADC, usually 2.5 V or 3.0 V. When using internal reference mode (on supported devices), this corresponds to the voltage range of the selected reference mode (2.5 V or 3.3 V).

### 2.2.5 Acquisition Time

Enter the target ADC acquisition window time. For the lowest ADC conversion latency (corresponding to the maximum ADC sample rate), select the minimum value of the parameter "Sample window duration" in the *ADC Operating Conditions* table in the device-specific data manual.



---

**Note**

Selecting a target S+H time larger than the ADC minimum acquisition time can ease the input settling design requirements for the input driving op-amp.

For example:

Resolution = 12- bits,  $C_{sh} = 12.5$  pF, Full-scale range = 3.0 V

- Acquisition time = 75 ns requires a driving op-amp with around 50 MHz of bandwidth
- Acquisition time = 150 ns requires a driving op-amp with around 25 MHz of bandwidth
- Acquisition time = 750 ns requires a driving op-amp with around 5 MHz of bandwidth

These decisions can be made on a per-channel basis since the C2000 ADC wrapper allows per-SoC configuration of the S+H time (by using the ACQPS setting in the SoC control register).

---

### 2.2.6 Outputs

Once all inputs have been provided to the calculator, record the outputs for later analysis

- $R_{filt}$  and  $C_{filt}$  output ranges will be explored via TINA-TI simulation in order to determine an optimal driver circuit
- The max error target output corresponds to 1/2 LSBs at the specified resolution. The final TINA-TI simulation will be considered successful if the settling is within this bound.
- The gain bandwidth output will be used in the next step to select an appropriate op-amp. The selected op-amp should have a unity-gain bandwidth of this value or better for best settling performance.

### 2.2.7 Math Behind the Calculator

Additional information about analytically determining the required op-amp bandwidth and ranges of  $R_{filt}$  and  $C_{filt}$  can be found in the final TI Precision Labs video in the SAR ADC Input Driver Design series: [Math Behind the R-C Component Selection](#).

### 2.3 Select an Op-Amp

An appropriate op-amp next needs to be selected for the ADC driver circuit. This op-amp needs to meet the minimum unity gain bandwidth reported by the Analog Engineer's calculator.

Additional instructions for selecting the op-amp are given in the TI Precision Labs video: [Selecting and Verifying the Driver Amplifier](#).

---

**Note**

If selecting an op-amp to meet the specified requirements is proving difficult, remember that both the settling resolution and the target acquisition time can be relaxed (if the application requirements allow) to relax the driving op-amp requirements. These can both be adjusted on a per-channel basis to allow critical ADC inputs to have optimal performance while axillary ADC inputs can be allowed to have relaxed performance.

---

### 2.4 Verify the Op-Amp Model

Next, it is necessary to obtain the TINA-TI model for the op-amp selected in the previous step.

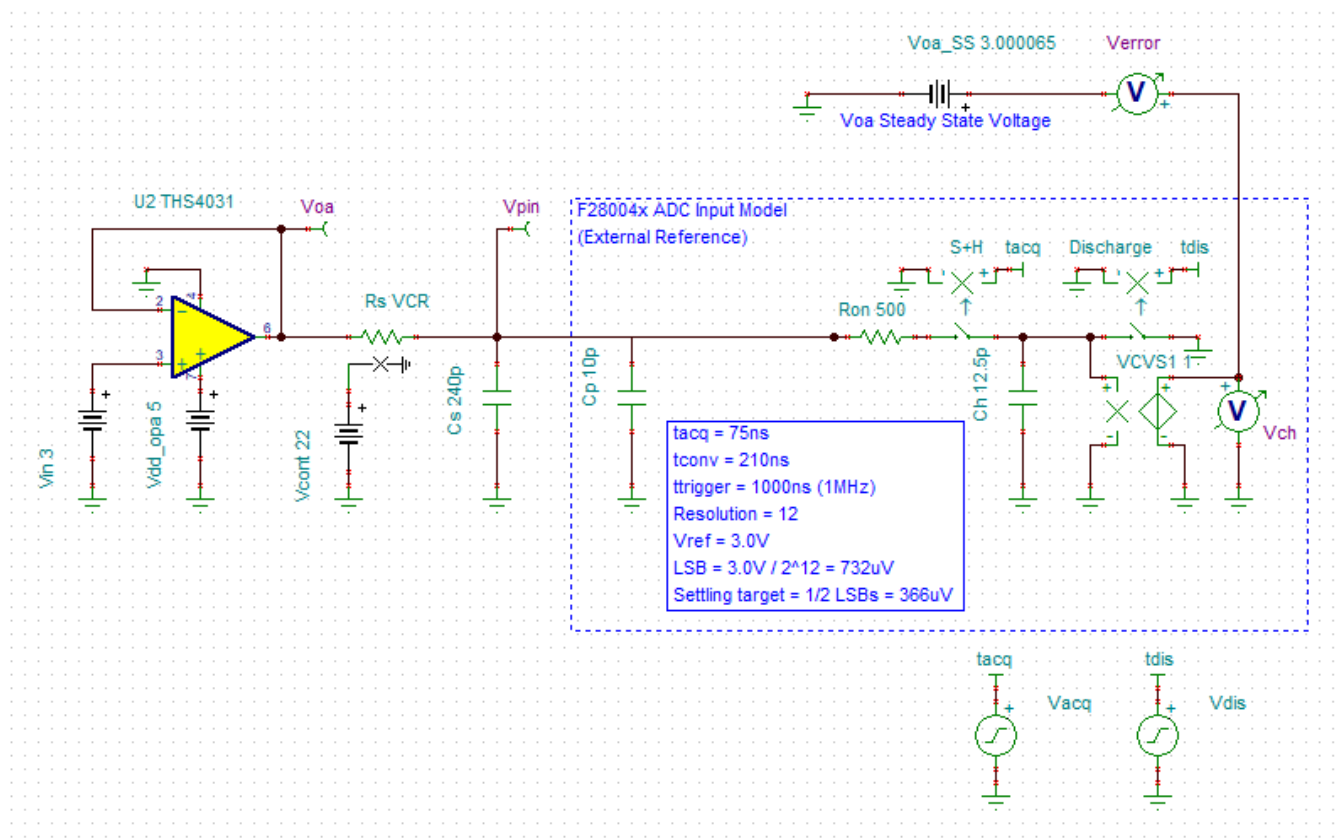
The TI Precision Labs video [Selecting and Verifying the Driver Amplifier](#) provides detailed instructions on how to obtain the TINA-TI model for the selected op-amp as well as how to verify the model against the information provided in the device-specific data manual.

### 2.5 Build the ADC Input Model

To simulate and refine the ADC input driver circuit in TINA-TI, it is first necessary to build a TINA-TI input model for the ADC. The TI Precision Labs video [Building the SAR ADC Input Model](#) outlines how to perform this process based on the information provided in the device-specific data manual for an ADC device. This application report is bundled with pre-generated ADC input models for TMS320F2837xD, TMS320F2837xS, TMS320F2838x, TMS320F2807x, TMS320F28004x, and TMS320F28002x family devices, so manual creation of the models is not necessary. However, the video does show how to configure your TINA-TI simulation parameters (via the slides titled "Optimizing Simulation Results").

**CAUTION**

The first time a settling simulation is run, do not skip the instructions to configure the simulation parameters in the [Building the SAR ADC Input Model](#) video. These instructions can be found in the "Optimizing Simulation Results" section.



**Figure 2-2. F28004x ADC Input Model**

An example of one of the provided ADC input models can be seen in [Figure 2-2](#). The following sections briefly describe the primary functions of each model component.

### 2.5.1 $V_{in}$

$V_{in}$  is the applied voltage that is used to charge the S+H capacitor. While any input can be simulated, applying the full-scale input voltage provides the worst-case DC settling conditions because the S+H capacitor always starts discharged in this model.

### 2.5.2 $V_{oa}$ , $V_{oa\_SS}$ , and $V_{error}$

TINA-TI op-amp models incorporate various non-idealities, which results in the DC steady state output of the op-amp at  $V_{oa}$  not exactly matching  $V_{in}$ . The steady state output at  $V_{oa}$  can be simulated and entered into  $V_{oa\_SS}$  such that  $V_{error}$  is an accurate representation of only the input settling error.

### 2.5.3 $R_s$ , $C_s$ , and $V_{cont}$

$R_s$  and  $C_s$  are the source resistance and source capacitance. These values correspond to  $R_{fit}$  and  $C_{fit}$  in the TI Precision Labs training terminology.  $R_s$  resistance is controlled by the control voltage  $V_{cont}$ .  $R_s$  is a voltage-controlled resistor with gain of 1 (1 V = 1  $\Omega$ , 10 V = 10  $\Omega$ , and so forth) instead of a simple resistor to allow parameter sweeping when there are two  $R_s$  resistors in the differential models.

### 2.5.4 $C_h$ , $R_{on}$ , and $C_p$

Component values for the ADC input model.

$C_h$ , the S+H capacitor, corresponds to  $C_{sh}$  in the TI Precision Labs training terminology while  $R_{on}$ , the S+H switch on resistance, corresponds to  $R_{sh}$ .

$C_p$  is the pin-specific input capacitance. This is not modeled in the TI Precision Labs training, but can be optionally added to the simulation to improve simulation accuracy. This is particularly helpful for pins that are multiplexed with VDAC (alternate DAC voltage reference pin) that have a value of  $C_p$ , which is greater than 100 pF on most devices.

Pins that are multiplexed with the buffered DAC output on F2837x and F2807x devices have a parasitic 50k  $\Omega$  pull-down resistor present on these pins. It is recommended to add this resistor to the model (not shown in Figure 2-2) to better represent behavior of these pins.

**CAUTION**

While the ADC input models for multiple C2000 devices are bundled with this application report, input model component values should always first be verified with the device-specific data manual before proceeding. Values for  $C_h$  and  $R_{on}$  can be obtained from the *Input Model Parameters* table in the device-specific data manual while values for  $C_p$  can be obtained from the *Per-Channel Parasitic Capacitance* table. In any cases where the device-specific data manual values disagree with the values in this document or in the provided models, the device-specific data manual values should always be considered the correct values.

**2.5.5 S+H Switch, Discharge Switch,  $t_{acq}$ , and  $t_{dis}$**

Switch "S+H" controls when the S+H capacitor,  $C_h$ , is being charged by the driving circuit. The timings for this switch are controlled by  $t_{acq}$ . Sample  $t_{acq}$  timing configurations for a 75 ns acquisition window are shown in Figure 2-3.

Switch "Discharge" controls when the S+H capacitor,  $C_h$ , is being reset by being discharged to ground. The timings for this switch are controlled by  $t_{dis}$ . Sample  $t_{dis}$  timing configurations for a 1  $\mu$ s trigger period are show in Figure 2-4.

When neither  $t_{acq}$  or  $t_{dis}$  are active, the value of the S+H capacitor is held at its final settled values.

Note that the timings are configured to emulate conversions triggered at a set 1 MHz frequency instead of back-to-back continuous conversions. This more closely emulates the typical C2000 ADC use case of being periodically triggered by the ePWM module. The triggering rate can be modified to match the actual application ePWM trigger rate by modifying the period of the  $t_{acq}$  and  $t_{dis}$  timing sources. Do, however, ensure that the trigger rate is slower than the total S+H plus acquisition time for the ADC conversion. The ADC conversion time can be found in the device-specific data manual table "ADC Timings".

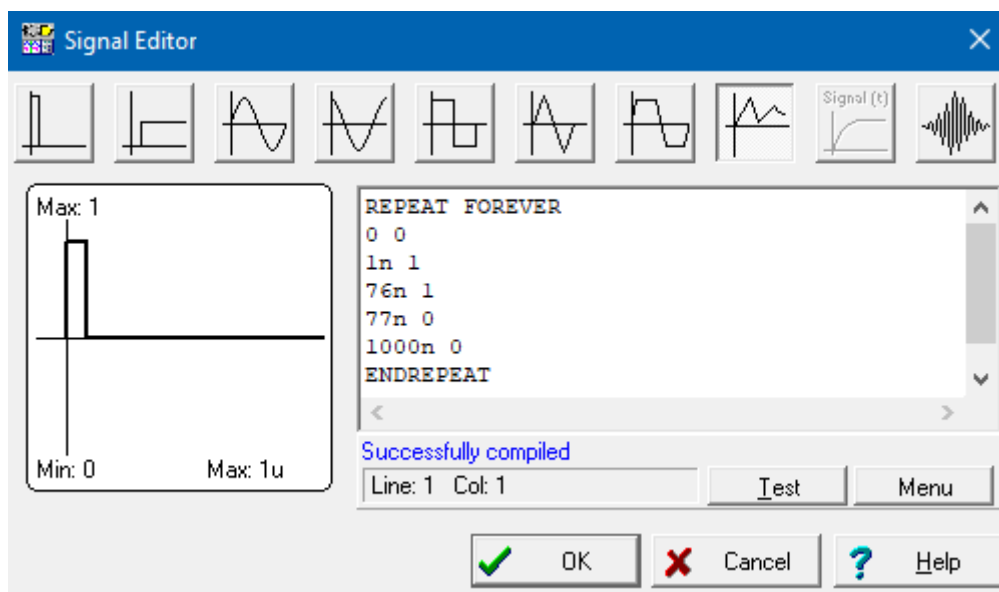
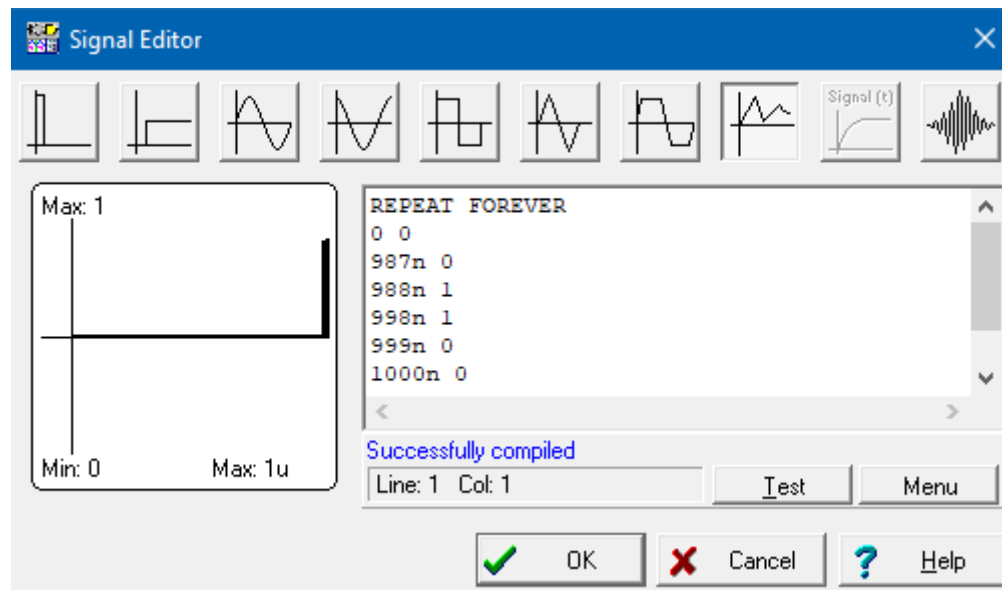


Figure 2-3. Example Timings for  $t_{acq}$  (Piecewise Linear)



**Figure 2-4. Example Timings for  $t_{dis}$  (Piecewise Linear)**

## 2.6 Refine RC Filter Values Via Simulation

Once the ADC input model has been combined with the op-amp simulation model, simulation can be used to select the best values of  $R_s$  and  $C_s$ . In-depth instructions for performing the component refinement simulation can be found in TI Precision labs video [Refine the Rfilt and Cfilt Values on ADC Drive](#) or by following the input circuit design example in [Section 3](#).

Once optimal values of  $R_s$  and  $C_s$  have been determined, a final simulation should be performed using the closest available standard component values. This establishes the final settling error, which can be compared to the settling error target obtained from the Analog Engineer's Calculator.

If the settling error does not meet the target, additional iteration is required that will entail evaluating a different op-amp, evaluating different  $R_s$  and  $C_s$  values, or relaxing the S+H time or settling resolution targets.

## 2.7 Perform Final Simulations

Additional simulations to verify the robustness of the design can also optionally be performed. These include:

- Simulate with a longer S+H duration to ensure that input is consistently settling
- Observe settling at the op-amp's output node. This node should also settled to within 1/2 LSBs (or the selected settling target) at the end of the acquisition period
- Observe settling over multiple cycles
- Perform settling simulations with an AC input

Information on performing these simulations can be found in the TI Precision Labs video [Final SAR ADC Drive Simulations](#).

## 2.8 Input Design Worksheet

[Table 2-2](#) lists the required inputs needed to evaluate an ADC input driving circuit and provides a place to summarize the outputs. Completing this worksheet for each unique circuit in your real-time control application is recommended to ensure good settling performance. For some circuits, it may be desirable to use the alternate design methodology presented in the application report [Charge-Sharing Driving Circuits for C2000 ADCs \(using TINA-TI simulation tool\)](#). That report also provides a worksheet that can be evaluated using the alternate design methodology.

**Table 2-2. ADC Input Settling Design Worksheet**

Symbol	Description	Value	Comments
Vfs	Full scale voltage range		In external reference mode, this is the voltage supplied to the VREFHI pin (usually 3.0V or 2.5 V) In internal reference mode, this is the effective input range based on the selected reference mode (usually 3.3 V or 2.5 V)
N	Target settling resolution (bits)		Usually the same as the resolution of the ADC Lower resolution can be targeted to relax the input design requirements
V <sub>errmax</sub>	Maximum error target		$V_{fs} / 2^{N+1}$ Obtain using Analog Engineer's Calculator: ADC SAR Drive
t <sub>sh</sub>	S+H time		Enter target S+H time if known Longer S+H times will result in less stringent BW requirements for the driving op-amp. Can be solved for given a pre-determined op-amp selection or a pre-determined R <sub>S</sub> and C <sub>S</sub>
R <sub>on</sub>	ADC switch resistance		Provided in the <i>Input Model Parameters</i> table in the device-specific data manual TI precision labs training refers to this as "R <sub>sh</sub> "
C <sub>h</sub>	ADC S+H capacitance		Provided in the <i>Input Model Parameters</i> table in the device-specific data manual TI precision labs training refers to this as "C <sub>sh</sub> "
C <sub>p</sub>	ADC pin parasitic capacitance		Provided in the <i>Per-Channel Parasitic Capacitance</i> table in the device-specific data manual
C <sub>S</sub> (range)	Range of source capacitance		Obtain using Analog Engineer's Calculator: ADC SAR Drive. TI precision labs training refers to this as "C <sub>filt</sub> "
R <sub>S</sub> (range)	Range of source resistance		Obtain using Analog Engineer's Calculator: ADC SAR Drive. TI precision labs training refers to this as "R <sub>filt</sub> "
BW <sub>OPA</sub>	ADC driver op-amp minimum bandwidth		Obtain using Analog Engineer's Calculator: ADC SAR Drive.
Op-amp	Selected Op-amp part number		Record selected op-amp here
V <sub>oa_ss</sub>	Steady state op-amp output voltage		Generated from DC nodal analysis of the V <sub>oa</sub> node Copy to Voa_ss before proceeding with other simulations
C <sub>S</sub> (final)	Final source capacitance		Final selected C <sub>S</sub> from simulation. TI precision labs training refers to this as "C <sub>filt</sub> "
R <sub>S</sub> (final)	Final source resistance		Final selected R <sub>S</sub> from simulation. TI precision labs training refers to this as "R <sub>filt</sub> "
BW <sub>R<sub>S</sub>C<sub>S</sub></sub>	Filter bandwidth from C <sub>S</sub> and R <sub>S</sub>		$1 / (2\pi \cdot C_S \cdot R_S)$  <b>Note</b> For proper settling, the filter bandwidth will be necessarily higher than the ½ the sampling frequency, thus the combination of C <sub>S</sub> and R <sub>S</sub> generally will not function as an anti-aliasing filter.
V <sub>err</sub>	Actual settling error		Ensure V <sub>err</sub> < V <sub>errmax</sub> Otherwise, additional iteration on selection of C <sub>S</sub> , R <sub>S</sub> , or the driving amplifier is needed.

### 3 Example Circuit Design

The following sections work through and discuss designing an example C2000 circuit for input settling.

#### 3.1 Select the ADC

This example assumes an F280049 device in external reference mode (VREFHI = 3.0 V) using channel A0. Triggering will occur at 1 MHz. This ADC supports 12-bit single-ended sampling only.

From the [TMS320F28004xMicrocontrollers Data Manual](#), the following key values are obtained:

- Minimum sample window duration = 75 ns
- Ch = 12.5 pF
- Ron = 500 Ω
- Cp = 12.7 pF

- ADC conversion time = 210 ns ( $t_{eoc}$  in the *ADC Timings* table from the [TMS320F28004xMicrocontrollers Data Manual](#)) = 21 SYSCLK cycles, SYSCLK = 100 MHz)

### 3.2 Find the Minimum Op-Amp Bandwidth and RC Filter Ranges

Figure 3-1 shows the inputs and outputs to the Analog Engineer's Calculator for this example. From this, we obtain the following key values:

- $C_s = 240$  pF (120 pF to 360 pF range, if needed)
- $R_s$  range = 17  $\Omega$  to 138  $\Omega$
- Op-amp minimum bandwidth = 37 MHz
- Settling error target (1/2 LSBs) = 366  $\mu$ V

Note that, in conjunction with the op-amp selection step, it was found that slightly relaxing the S+H duration from the ADC minimum of 75 ns to 100 ns reduced the required op-amp bandwidth from 50 MHz to 37 MHz. This allowed for an op-amp selection that better met the project constraints (OPA2350) while still meeting the application requirements for sample latency.

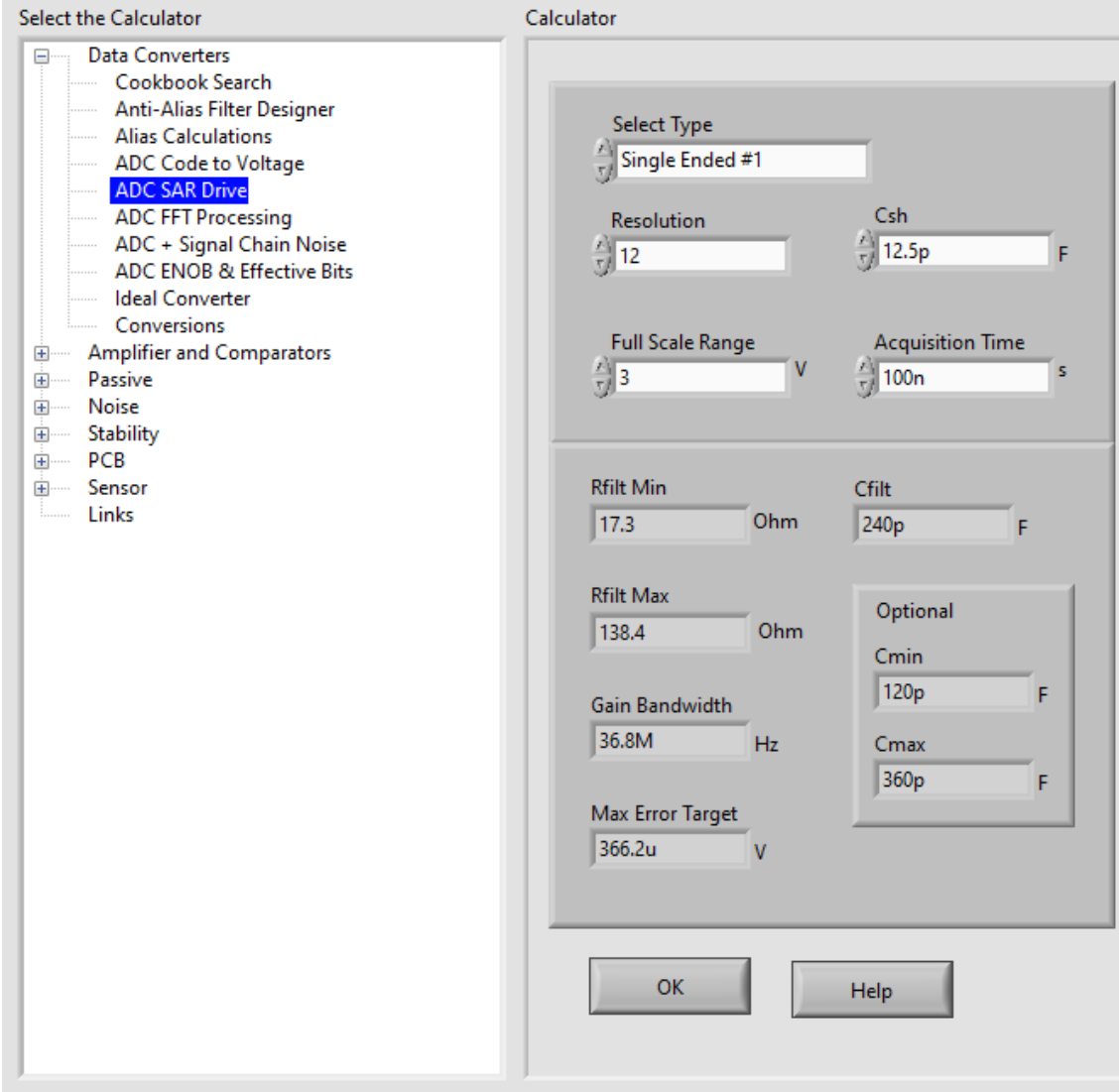


Figure 3-1. F280049 Example Analog Engineer's Calculator Output

### 3.3 Verify the Op-amp Model

The latest OPA2350 model was obtained from ti.com.

### 3.4 Build the ADC Input Model

The OPA2350 model was combined with the provided F28004x\_Ext.TSC ADC input model. In addition to setting the driving amplifier to OPA2350, the following were performed on the TINA-TI file to prepare for simulation:

- Op-amp power supplies set to an appropriate voltage (5.0 V and ground)
- $V_{in}$  set to full-scale voltage (3.0 V)
- $C_s$  and  $C_p$  initial value set to full-scale voltage (3.0 V)
- $V_{oa\_ss}$  set to the voltage obtained from DC nodal voltage analysis (demonstrated in the next section)
- $T_{acq}$  and  $T_{dis}$  source period set to 1000 ns (1 MHz trigger frequency)
- $T_{acq}$  source S+H time set to 100 ns
- $C_p$ ,  $R_{on}$ , and  $C_h$  values set based on values obtained from the device-specific data manual
- $C_s$  set to the nominal value provided by calculator
- $R_s$  set to a value in the calculator provided range (by setting  $V_{cont}$ )

The initial circuit is shown in Figure 3-2 and the timings for  $t_{acq}$  and  $t_{dis}$  are shown in Figure 3-3 and Figure 3-4.

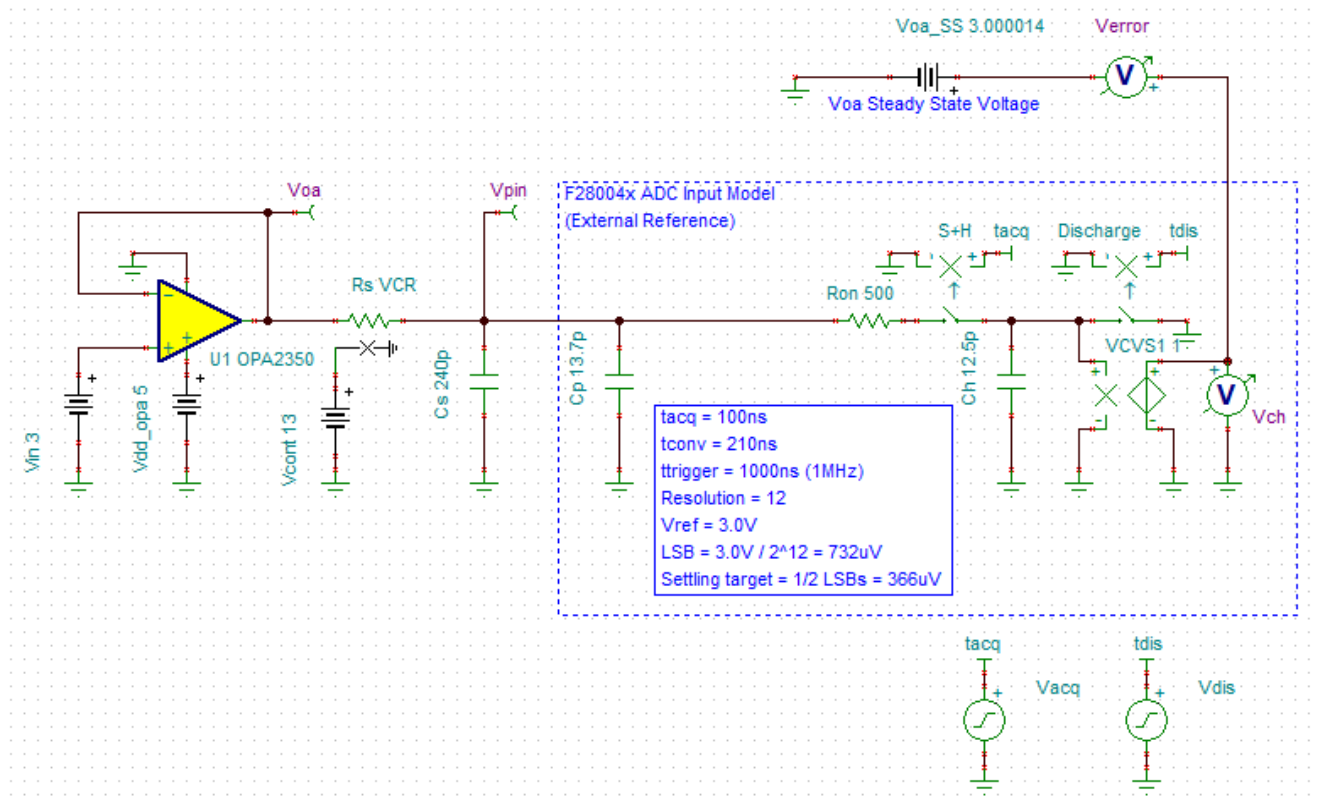


Figure 3-2. F280049 Example TINA-TI Circuit

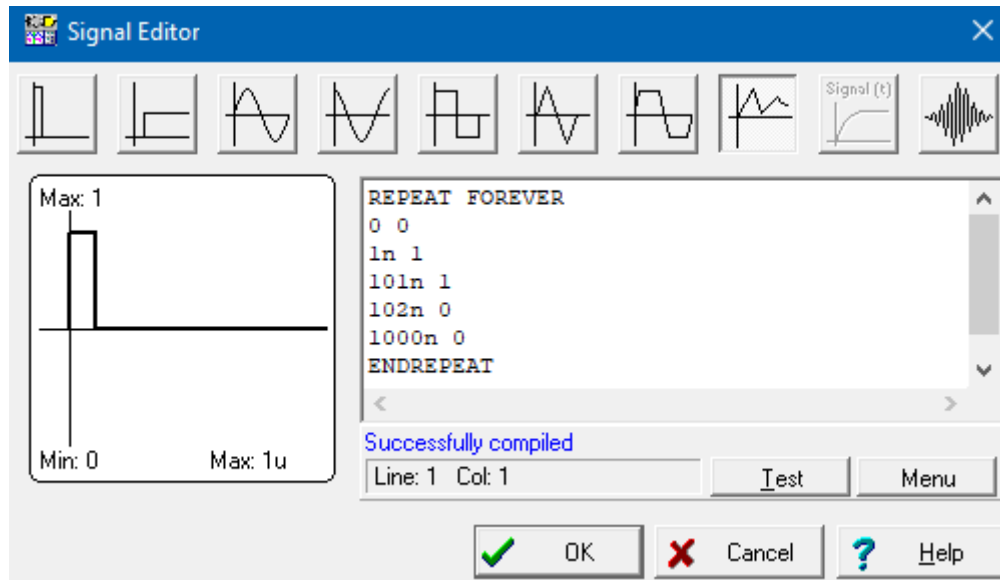


Figure 3-3. F280049 Example  $T_{acq}$  Source Timings

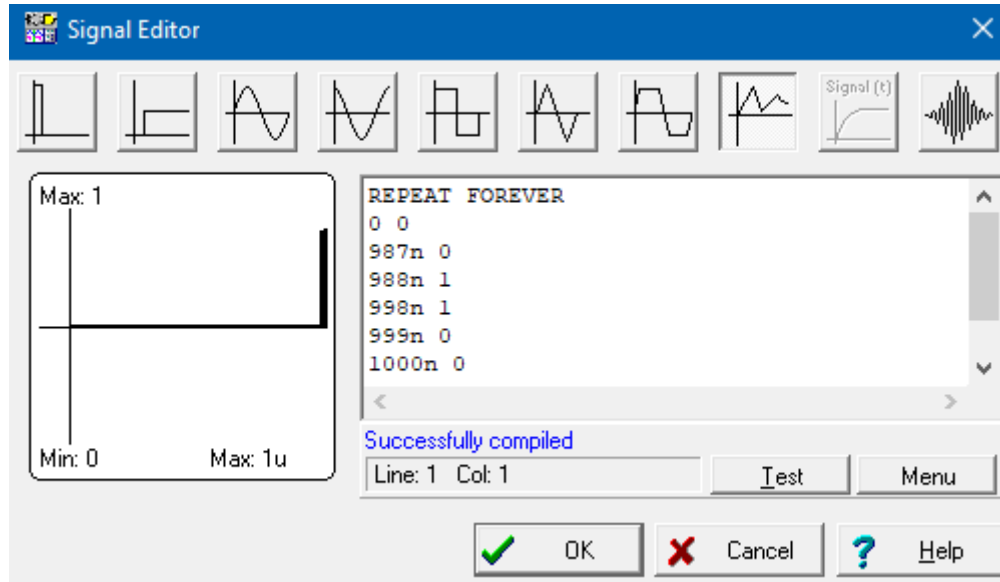


Figure 3-4. F280049 Example  $T_{dis}$  Source Timings

### 3.5 DC Node Analysis

To finalize the TINA-TI input model, a DC node analysis is performed to determine the steady-state output value for  $V_{oa}$  given the specific op-amp being evaluated ( Analysis → DC Analysis → Calculated nodal voltages). For OPA2350 in this circuit, this gives:

- $V_{oa} = 3.000014$  V

Which is then copied to the  $V_{oa\_ss}$  voltage source.

#### CAUTION

Ensure that the simulation parameters are configured per the "Optimizing Simulation Results" slides from the [Building the SAR ADC Input Model](#) video.



### 3.6 Refine RC Filter Values Via Simulation (Part 1)

Now that the circuit is setup for simulation, the first step is to run a basic transient analysis to ensure everything is functioning and to check the initial settling. The transient analysis can be run via the TINA-TI menu options: Analysis → Transient...

The settling should be captured after allowing the circuit to stabilize for a couple sampling cycles, so a time period of 2.5  $\mu$ s to 5.5  $\mu$ s is selected as shown in Figure 3-5. This allows capture of the full 3rd and 4th sampling cycles (each cycle is 1 $\mu$ s since the trigger frequency is 1 MHz).

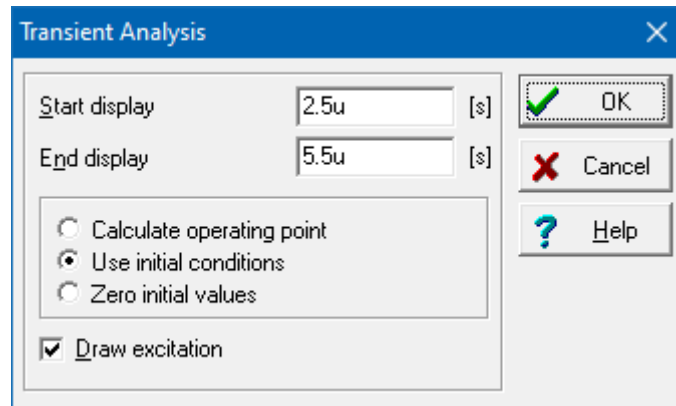


Figure 3-5. F280049 Example Transient Analysis Configuration

Figure 3-6 shows the output waveforms from the transient simulation after separating the outputs (View → Separate outputs). The error voltage scale has also been adjusted to show a range of +10 mV to -10 mV (double-click on  $V_{error}$ , then enter -10m in the "Lower limit" field and 10m in the "Upper limit" field, then click "OK").

From this output, it is clear that:

- Discharge and acquisition times are functioning as expected
- The transients on  $V_{oa}$  and  $V_{pin}$  are of reasonable magnitude (less than 100 mV, so the signal settling will be considered a small signal event)
- $V_{error}$  is settling to about 14  $\mu$ V, which is well within the error target of 366  $\mu$ V

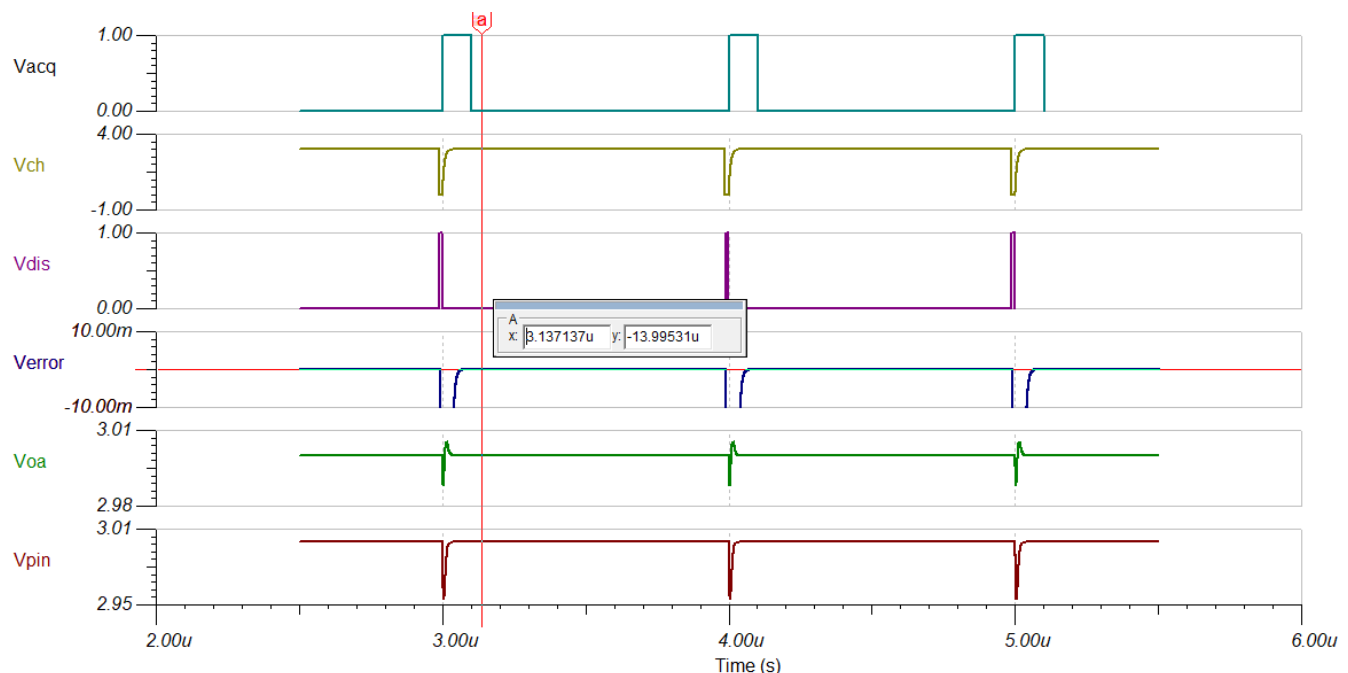


Figure 3-6. F280049 Example Initial Transient Results

### 3.7 Refine RC Filter Values Via Simulation (Part 2)

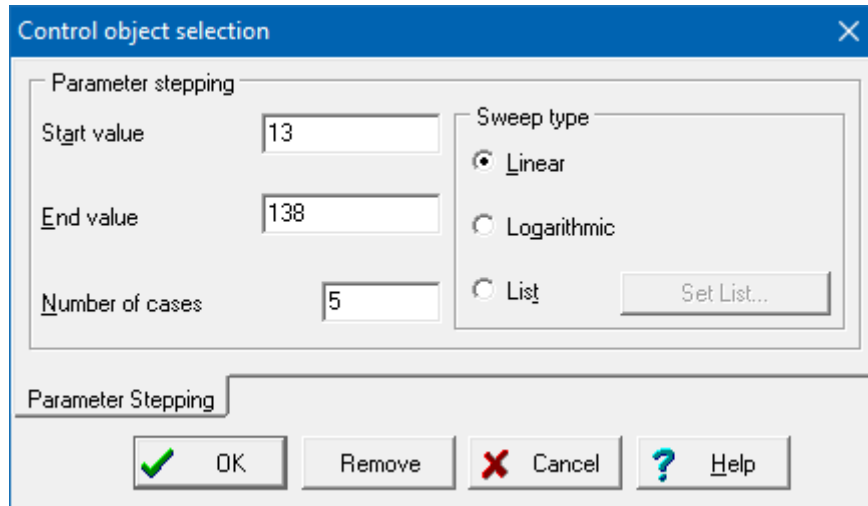
Now that the basic simulation has shown that the simulation is fundamentally working, sweeps can be performed to refine the  $R_s$  component selection.

To perform a sweep, click on the sweep button, then click on the  $V_{cont}$  voltage source (which controls the value of  $R_s$ ).



**Figure 3-7. Sweep Button**

This will bring up a dialog box. Select the "..." button next to the Voltage [V] parameter and configure the sweep to select 5 points on a linear scale from 13  $\Omega$  to 138  $\Omega$  as shown in [Figure 3-8](#).



**Figure 3-8. F280049 Example Sweep Control Dialog**

The output of the transient simulation with a sweep of  $R_s$  is shown in [Figure 3-9](#). Note that the outputs have again been separated, and all outputs other than  $V_{pin}$  and  $V_{error}$  have been deleted.  $V_{error}$  range has been set to -50 mV to +50 mV and the waveform has been limited to better observe the settling.

From this output, it can be concluded:

- Higher resistance values in the  $R_s$  range do not provide quick enough settling. For instance, 106  $\Omega$  only settles to 6.6 mV in the allocated S+H time while the settling target is 366  $\mu$ V
- A good range for further investigation would be 10  $\Omega$  to 50  $\Omega$

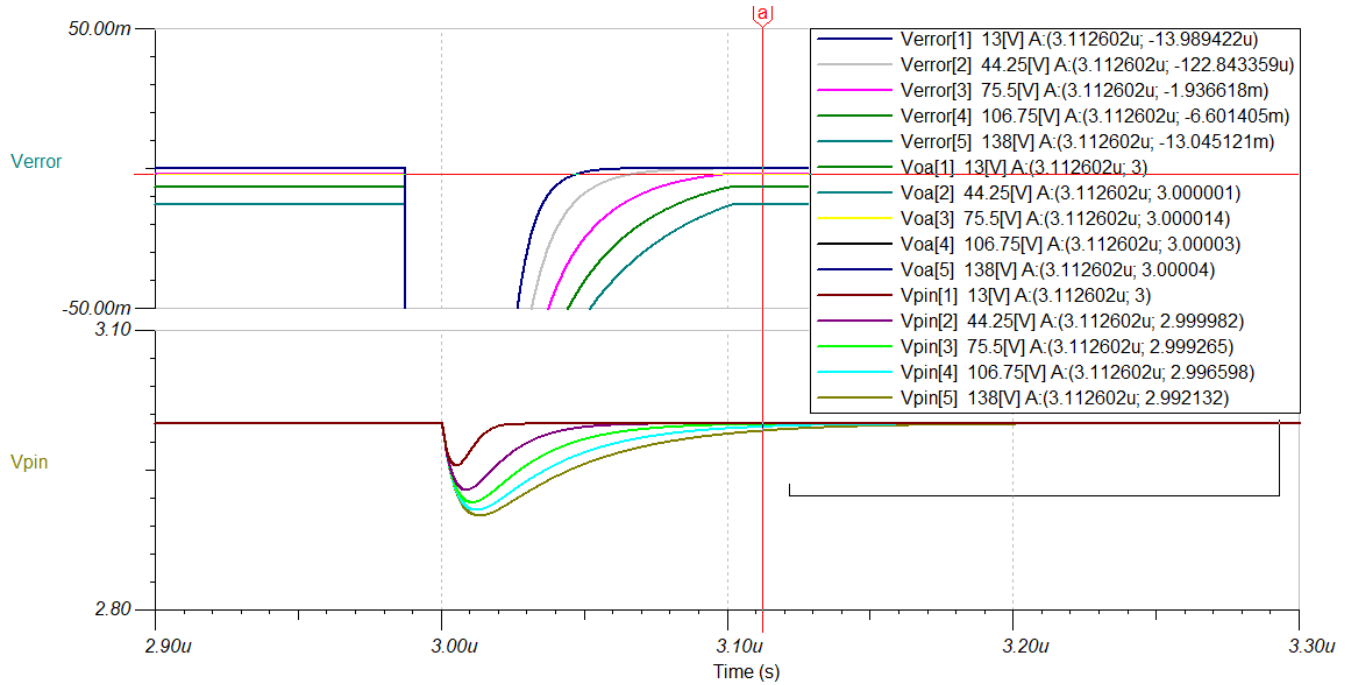


Figure 3-9. F280049 Example  $R_s$  Initial Sweep Results

Figure 3-10 shows the results of a sweep of  $R_s$  from  $10\ \Omega$  to  $50\ \Omega$ . From this set of waveforms, it appears that values of  $R_s$  from  $10\ \Omega$  to  $30\ \Omega$  all provide settling that greatly exceeds the design target. A final value of  $R_s$  was thus selected as  $27\ \Omega$ .

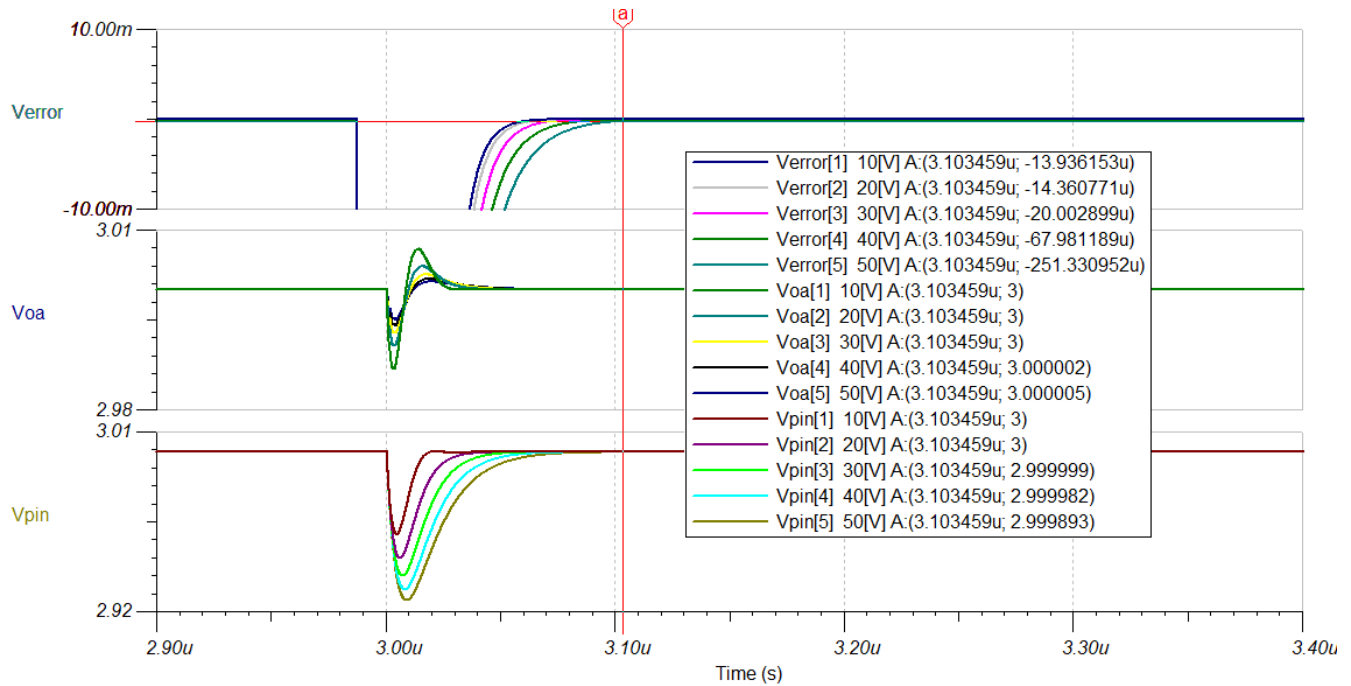


Figure 3-10. F280049 Example  $R_s$  Fine Sweep Results

### 3.8 Refine RC Filter Values Via Simulation (Part 3)

With the final  $R_s$  and  $C_s$  values determined, a final transient simulation can be performed to validate the component selections. The results of this final simulation can be seen in Figure 3-11. From these results, it appears that the settling crosses the settling target of  $366\ \mu\text{V}$  somewhere around  $67\ \text{ns}$  and the final settling at  $100\ \text{ns}$  is around  $17\ \mu\text{V}$ . The final application circuit should have excellent ADC input settling.

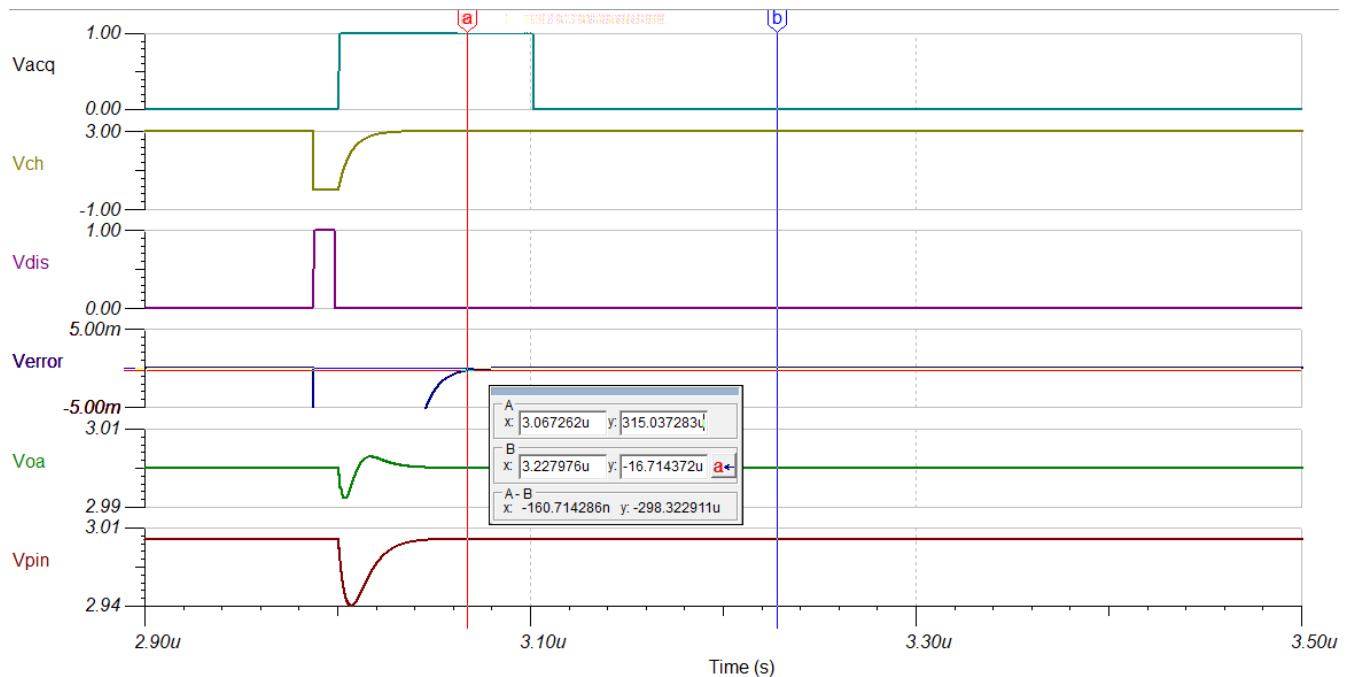


Figure 3-11. F280049 Example Final Transient Results

### 3.9 Further Refinement

Overall, it appears that this design has quite a bit of settling margin. If further optimization is desired, the designer could explore using an op-amp with slightly lower bandwidth. Alternately, the simulation supports using a S+H window that is shorter than  $100\ \text{ns}$ , if desired.

Another area refinement is to resolve a circuit with intentionally higher  $R_s$  and  $C_s$ . As designed, the low-pass filter bandwidth of the  $R_s$  and  $C_s$  circuit is about  $2.5\ \text{MHz}$ , so this circuit provides some rejection of high-frequency noise. By intentionally increasing  $R_s$  and  $C_s$ , the filter bandwidth can be lowered, resulting in additional noise rejection. However, note that the  $R_s$  and  $C_s$  circuit can generally not provide a true anti-aliasing filter, as this would necessarily result in a settling time constant that would be much too large to allow proper settling. If a true anti-aliasing filter is needed, this should be built as a separate filter stage before the ADC drive stage.

#### CAUTION

Even if the simulations indicate that the input could settle faster than the minimum acquisition window specified in the device-specific data manual, do not select an acquisition window that violates the device-specific data manual minimum acquisition window time.

### 3.10 Further Simulations

As indicated in Section 2.7, once the final components are selected, it is possible to perform additional simulations to further validate the design. This design was validated over multiple cycles and with increased S+H duration (not shown). No issues were identified in these additional simulations.

### 3.11 Completed Worksheet

Table 3-1 shows the completed worksheet for the F280049 with OPA2350 example.

**Table 3-1. ADC Input Settling Design Worksheet (F280049 Example)**

Symbol	Description	Value	Comments
$V_{fs}$	Full scale voltage range	3.0 V	In external reference mode, this is the voltage supplied to the VREFHI pin (usually 3.0 V or 2.5 V) In internal reference mode, this is the effective input range based on the selected reference mode (usually 3.3 V or 2.5 V)
N	Target settling resolution (bits)	12 bits	Usually the same as the resolution of the ADC Lower resolution can be targeted to relax the input design requirements
$V_{errmax}$	Maximum error target	366 $\mu$ V	$V_{fs} / 2^{N+1}$ Obtain using Analog Engineer's Calculator: ADC SAR Drive
$t_{sh}$	S+H time	100 ns	Enter target S+H time if known Longer S+H times will result in less stringent BW requirements for the driving op-amp. Can be solved for given a pre-determined op-amp selection or a pre-determined $R_S$ and $C_S$
$R_{on}$	ADC switch resistance	500 $\Omega$	Provided in the data manual table "Input Model Parameters" TI precision labs training refers to this as " $R_{sh}$ "
$C_h$	ADC S+H capacitance	12.5 pF	Provided in the data manual table "Input Model Parameters" TI precision labs training refers to this as " $C_{sh}$ "
$C_p$	ADC pin parasitic capacitance	12.7 pF	Provided in the data manual table "Per-Channel Parasitic Capacitance"
$C_S$ (range)	Range of source capacitance	240 pF (120pF to 360pF)	Obtain using Analog Engineer's Calculator: ADC SAR Drive. TI precision labs training refers to this as " $C_{fit}$ "
$R_S$ (range)	Range of source resistance	17 to 138 $\Omega$	Obtain using Analog Engineer's Calculator: ADC SAR Drive. TI precision labs training refers to this as " $R_{fit}$ "
$BW_{OPA}$	ADC driver op-amp minimum bandwidth	37 MHz	Obtain using Analog Engineer's Calculator: ADC SAR Drive.
Op-amp	Selected Op-amp part number	OPA2350	Record selected op-amp here
$V_{oa\_ss}$	Steady state op-amp output voltage	3.000014 V	Generated from DC nodal analysis of the $V_{oa}$ node Copy to $V_{oa\_ss}$ before proceeding with other simulations
$C_S$ (final)	Final source capacitance	240 pF	Final selected $C_S$ from simulation. TI precision labs training refers to this as " $C_{fit}$ "
$R_S$ (final)	Final source resistance	27 $\Omega$	Final selected $R_S$ from simulation. TI precision labs training refers to this as " $R_{fit}$ "
$BW_{R_S C_S}$	Filter bandwidth from $C_S$ and $R_S$	2.5 MHz	$1 / (2\pi \cdot C_S \cdot R_S)$  <b>Note</b>  For proper settling, the filter bandwidth will be necessarily higher than the $\frac{1}{2}$ the sampling frequency, thus the combination of $C_S$ and $R_S$ generally will not function as an anti-aliasing filter.
$V_{err}$	Actual settling error	17 $\mu$ V	Ensure $V_{err} < V_{errmax}$ Otherwise, additional iteration on selection of $C_S$ , $R_S$ , or the driving amplifier is needed

## 4 Working With Existing Circuits or Additional Constraints

The TI Precision Labs design methodology explored in this report start by assuming a S+H window is known and that the op-amp and  $R_s$  and  $C_s$  have not yet been determined. The following sections provide some guidance if one or more of these components has already been selected or otherwise needs to be constrained.

### 4.1 Existing Circuits

Fully determined input driving circuits can of course be simulated to check their settling using the provided ADC input models and simulation methodology.

Note that it is especially important to verify the performance over many sampling cycles for existing circuits that were not designed using the TI Precision Labs methodology. One common case where this is important is when the input was designed using a "charge-sharing" design methodology.

#### 4.1.1 Brief Overview of Charge Sharing

Normally, as the size of  $C_s$  increases, the settling time will also increase. It is generally not advantageous to make  $C_s$  very large. However, if  $C_s$  becomes sufficiently large, it can fully provide all the charge needed to drive  $C_h$  to within the settling error target, regardless of the value of  $R_s$ .

The external source still needs to recharge  $C_s$  through  $R_s$  in the time between samples, otherwise each sample will slowly deplete the voltage on  $C_s$ . This results in a sample-rate limitation based on the magnitude of the  $R_s$  resistance.

For 1/2 LSBs settling, the critical value of  $C_s$  occurs at  $C_s = 2^{(N+1)} \times C_h$ .

#### 4.1.2 Charge Sharing Example

In the example circuit in [Figure 4-1](#),  $C_s$  has been set to approximately  $2^{(N+1)} \times C_h$  and the sample trigger rate is set to 100 kHz.

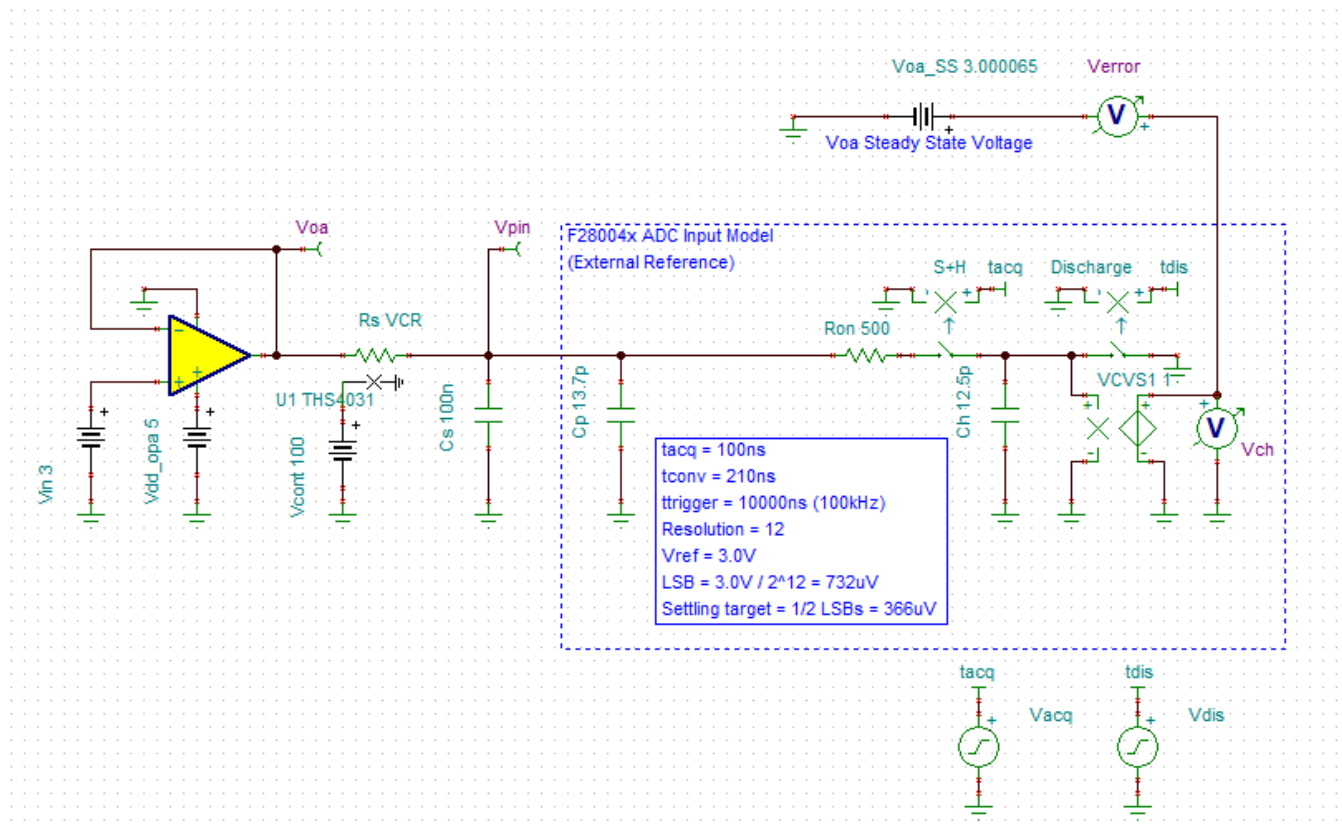
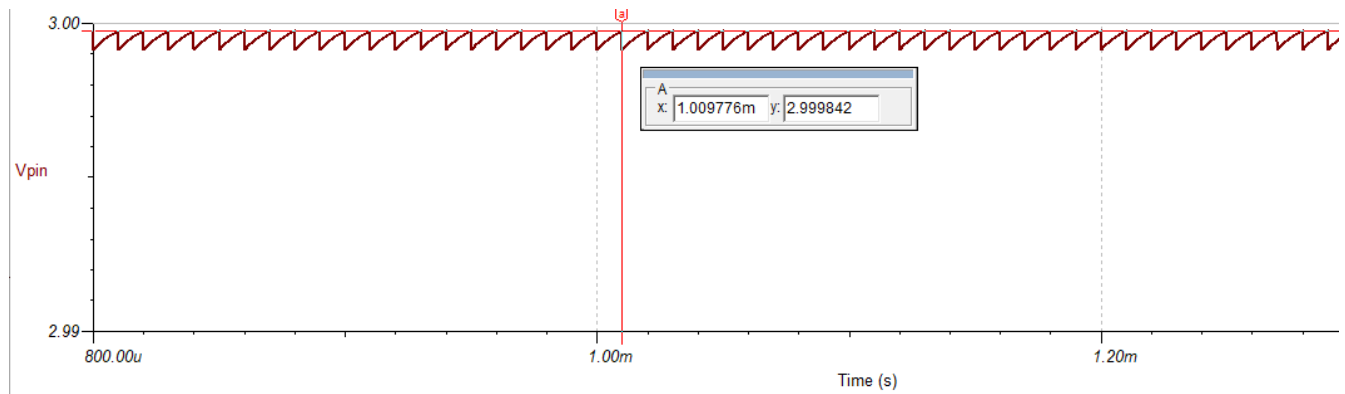


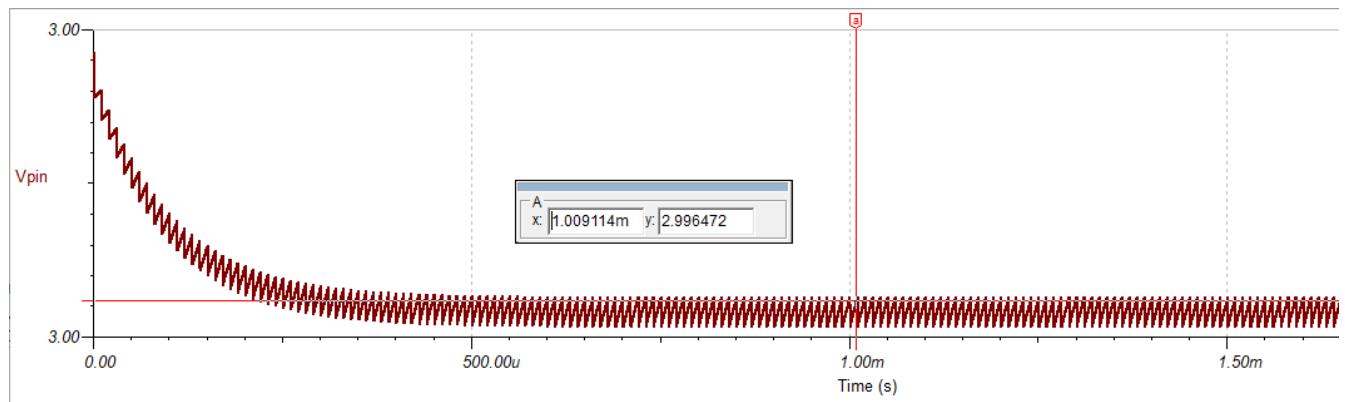
Figure 4-1. Example Circuit Utilizing Charge Sharing

Figure 4-2 shows the pin transient over many sampling cycles when  $R_s$  is  $100\ \Omega$ . It can be seen that the voltage on the pin (which is the voltage directly on  $C_s$ ) recovers to within 1/2 LSBs in the time between conversions. This results in good ADC input settling using a very short S+H window as long as the sample-rate remains at or below 100 ksp/s.



**Figure 4-2.  $C_s$  Voltage With  $R_s = 100\ \Omega$**

Now consider the results in Figure 4-3.  $R_s$  has been set to  $1\text{ k}\Omega$ . In this case, the voltage on  $C_s$  is slowly depleted sample-by-sample until an equilibrium is reached. At equilibrium, the pin voltage has an error of about 3.5 mV. Even though  $C_h$  will quickly match the  $C_s$  voltage during the S+H phase, the voltage on  $C_s$  has a significant error.



**Figure 4-3.  $C_s$  Voltage With  $R_s = 1\text{ k}\Omega$**

To resolve this error, either  $R_s$  would need to be decreased or the sample rate would have to be reduced.

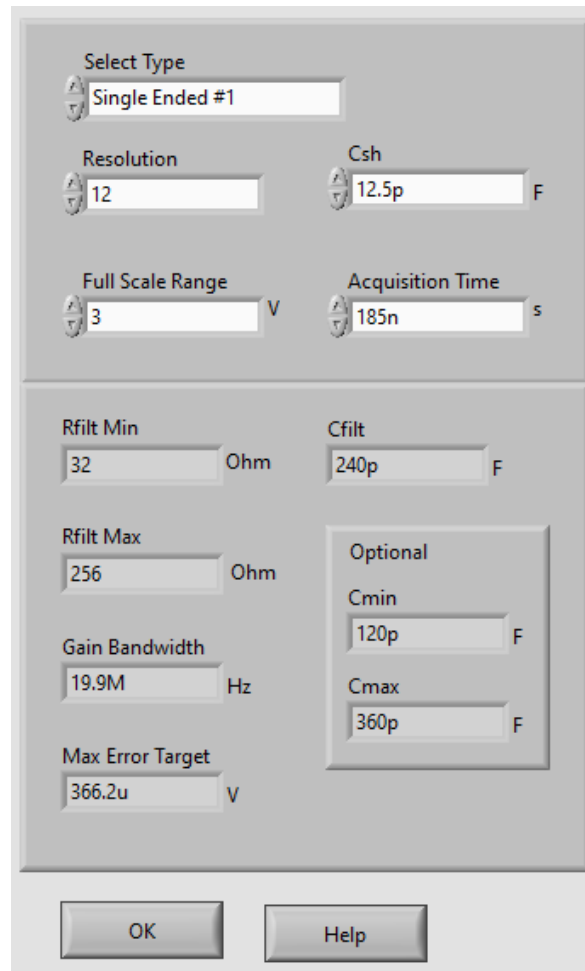
This example underscores the importance of simulating circuits with large  $C_s$  or  $R_s$  over many cycles.

## 4.2 Pre-Selected Op-Amp

If the op-amp has already been selected for the input driver, the easiest method for proceeding with the design is to iterate the acquisition time setting in the Analog Engineer's Calculator until the necessary bandwidth matches the bandwidth of the pre-selected op-amp. The designer can then record the  $C_s$  and  $R_s$  component ranges and proceed with the design per the existing methodology (assuming that the predicted S+H meets the application's latency or sample rate requirements).

### 4.2.1 Pre-Selected Op-Amp Example

The designer wants to use OPA2320 as the driving op-amp for an ADC input (perhaps there is a spare channel from an OPA2320 performing another function on the PCB). OPA2320 has a bandwidth of 20 MHz and the ADC in question has a  $C_h$  value of 12.5 pF. By iterating the acquisition window input of the calculator, the solution shown in Figure 4-4 is found and thus a S+H window of 185 ns can be targeted.



The screenshot shows the following parameters and values:

- Select Type: Single Ended #1
- Resolution: 12
- Csh: 12.5p F
- Full Scale Range: 3 V
- Acquisition Time: 185n s
- Rfilt Min: 32 Ohm
- Cfilt: 240p F
- Rfilt Max: 256 Ohm
- Gain Bandwidth: 19.9M Hz
- Optional Cmin: 120p F
- Optional Cmax: 360p F
- Max Error Target: 366.2u V

Buttons for OK and Help are visible at the bottom.

Figure 4-4. Analog Engineer's Calculator Output for OPA2320

### 4.3 Pre-Selected $R_s$ and $C_s$ Values

If both external source components have been pre-selected, a target acquisition time can be determined by simulating the circuit with an ideal source or with an op-amp with very high bandwidth. This initial target acquisition time can then be used to provide some guidance in op-amp selection, and then design refinement can proceed as normal.

#### 4.3.1 Analytical Solution for ADC Acquisition Time

A quick method for estimating the ADC S+H time given selections of  $R_s$  and  $C_s$  (along with the input model parameters from the device-specific data manual) is to use the formulas shown in [Equation 1](#) and [Equation 2](#):

$$T = (R_s + R_{on})C_h + R_s (C_s + C_p)$$

$$k = \ln\left(\frac{2^N}{\text{target settling error}}\right) - \ln\left(\frac{C_s + C_p}{C_h}\right)$$

$$\text{acquisition time} = Tk$$

(1)

#### 4.3.2 Example Analytical Solution for ADC Acquisition Time

- $n = 12$  bits
- $R_{on} = 500 \Omega$
- $C_h = 12.5$  pF
- $C_p = 12.7$  pF
- Target settling error = 1/2 LSBs



- $R_s = 180 \Omega$
- $C_s = 150\text{pF}$

$$T = (180 \Omega + 500 \Omega)12.5 \text{ pF} + 180 \Omega(150 \text{ pF} + 12.7 \text{ pF}) = 37.8 \text{ ns}$$

$$k = \ln\left(\frac{4069 \text{ LSBs}}{0.5 \text{ LSBs}}\right) - \ln\left(\frac{(150 \text{ pF} + 12.7 \text{ pF})}{12.5 \text{ pF}}\right) = 6.4 \text{ time constants}$$

$$\text{acquisition time} = 37.8 \text{ ns} \times 6.4 = 242 \text{ ns} \tag{2}$$

## 5 Summary

Input settling is an important design consideration for ADC driving circuits. Without proper evaluation, distortion and memory cross-talk errors could occur, resulting in degraded sensing performance and accuracy.

By using the excellent resources provided by in the [TI Precision Labs - SAR ADC Input Driver Design](#) video series, the [Analog Engineer's Calculator](#), and the guidance in this application report, input settling can be systematically evaluated to ensure that ADC driving circuits have appropriate settling performance.

## 6 References

- [TI Precision Labs - SAR ADC Input Driver Design](#)
- [Analog Engineer's Calculator](#)
- [TINA-TI](#)
- Texas Instruments: [TMS320F28004xMicrocontrollers Data Manual](#)

## 7 Revision History

NOTE: Page numbers for previous revisions may differ from page numbers in the current version.

Changes from Revision * (October 2020) to Revision A (March 2023)	Page
• Updated the Abstract of this document.....	1
• Updated the numbering format for tables, figures and cross-references throughout the document.....	3
• Updated <a href="#">Section 1.3.5</a> .....	4
• Updated <a href="#">Section 1.3.6</a> .....	5

## IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATA SHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES "AS IS" AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, regulatory or other requirements.

These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you will fully indemnify TI and its representatives against, any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI's products are provided subject to [TI's Terms of Sale](#) or other applicable terms available either on [ti.com](https://www.ti.com) or provided in conjunction with such TI products. TI's provision of these resources does not expand or otherwise alter TI's applicable warranties or warranty disclaimers for TI products.

TI objects to and rejects any additional or different terms you may have proposed.

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265  
Copyright © 2023, Texas Instruments Incorporated