



John Miller

ABSTRACT

This document provides guidance on how to download and install the LDC5072 Sensor design tool. Additionally, the document gives a brief overview of the tool's capabilities and features, steps for the design tool, and shows how to use the tool to create and use a SPICE model of the design. Also included are instructions for creating a set of Gerber files for a sensor design and a layout and schematic of the resulting test board.

Table of Contents

1 Introduction	3
2 Gaining Access to the LDC5072 Sensor Design Tool	4
3 Installer Setup	5
4 Installing the Tool	8
5 Tool Overview	13
6 Using the Tool to Create a Design and Generate a Supporting SPICE Model and Gerber Files	21
7 Running Simulations in Pspice	28
8 Layout, Bill-of-Materials, and Schematic of the Test PC board	46
9 Troubleshooting	52
10 Revision History	53

List of Figures

Figure 3-1. LDC5072 Design Tool Installer Shortcut on Desktop.....	5
Figure 3-2. LDC5072 Sensor Design Tool Installer & Setup Start Panel.....	5
Figure 3-3. LDC5072 Sensor Design Tool License Agreement.....	6
Figure 3-4. Directory Path for Tool Installer & Setup.....	6
Figure 3-5. Ready to Download Installer Status Window.....	7
Figure 3-6. Installer Download & Setup Complete Status Window.....	7
Figure 4-1. Tool Installer Initial Splash Screen.....	8
Figure 4-2. Tool Installer <i>Start</i> Panel.....	9
Figure 4-3. Select/Configure Directory Path for Tool Installation.....	9
Figure 4-4. Required Software Panel - MATLAB Runtime Engine.....	10
Figure 4-5. Confirm Software and Directories for Tool Installation.....	10
Figure 4-6. Successful Tool Installation - Confirmation.....	11
Figure 4-7. Revising the Tool's Desktop Shortcut - Setting the <i>Start in:</i> Path.....	12
Figure 5-1. Starting the LDC5072 Sensor Designer - Splash Screen.....	13
Figure 5-2. LDC5072 Sensor Designer Start Screen.....	14
Figure 5-3. GUI Input Field Summary.....	14
Figure 5-4. GUI Front Panel <i>Help</i> Pop-up.....	15
Figure 5-5. LDC5072 Sensor Designer <i>Mechanical Design</i> Tab.....	16
Figure 5-6. LDC5072 Sensor Designer <i>Mechanical Design Help</i> Pop-up.....	16
Figure 5-7. LDC5072 Sensor Designer <i>PCB Design</i> Tab.....	17
Figure 5-8. LDC5072 Sensor Designer <i>PCB Design Help</i> Pop-Up.....	18
Figure 5-9. LDC5072 Sensor Designer <i>Manufacturing Tolerance</i> Tab.....	18
Figure 5-10. LDC5072 Sensor Designer <i>Manufacturing Tolerance Help</i> Pop-Up.....	19
Figure 5-11. LDC5072 Sensor Designer <i>Sensor Targets</i> Tab.....	19
Figure 5-12. <i>Sensor Targets Tab Help</i> Pop-Up.....	20
Figure 6-1. LDC5072 Sensor Designer Tool Start Panel.....	21
Figure 6-2. LDC5072 Sensor Designer <i>Tolerances</i> Tab.....	22
Figure 6-3. LDC5072 Sensor Designer <i>Sensor Parameters</i> Tab.....	22
Figure 6-4. Schematic Representation of the Sensor Spice Model.....	23

Figure 6-5. <i>Parameter Help</i> Pop-up for the Sensor Spice Model.....	24
Figure 6-6. LDC5072 Sensor Designer Front Panel: Completion of a Valid Design.....	25
Figure 6-7. LDC5072 Sensor designer <i>Custom Sensor</i> Tab.....	25
Figure 6-8. Spice Model Creation for the Sensor, LDC5072 and a Test Bench.....	26
Figure 6-9. Gerber File Creation for the Sensor, LDC5072 and a Test Bench.....	27
Figure 7-1. Starting the Cadence Allegro Pspice Simulator in the Windows Start Menu.....	28
Figure 7-2. Cadence Allegro Pspice Simulator Window.....	29
Figure 7-3. Opening the Spice Test Bench - Step 1.....	30
Figure 7-4. Opening the Spice Test Bench - Step 2.....	31
Figure 7-5. Sensor + LDC5072 Test Bench Open in Allegro Pspice Simulator.....	32
Figure 7-6. Test Bench Pspice Simulation Statements.....	33
Figure 7-7. Starting the Pspice Test Bench Transient Simulation from the <i>Simulation</i> Menu.....	34
Figure 7-8. Starting the Pspice Test Bench Transient Simulation from the <i>Run</i> Button.....	35
Figure 7-9. Status of a Running Pspice Simulation.....	36
Figure 7-10. Initial Allegro Pspice Plot Window After Completion of Simulation.....	37
Figure 7-11. Adding a Trace to the Allegro Pspice Plot Window.....	38
Figure 7-12. Adding the Trace for LDC5072 Channel 0 (COS) Differential Output Voltage.....	38
Figure 7-13. Plotted Curve for LDC5072 Channel 0 (COS) Differential Output Voltage.....	39
Figure 7-14. Adding the Trace for LDC5072 Channel 1 (SIN) Differential Output Voltage.....	39
Figure 7-15. Plotted Curve for LDC5072 Channel 0 (COS) and Channel 1 (SIN) Differential Output Voltages.....	40
Figure 7-16. Adding a Plot Window.....	41
Figure 7-17. New Empty Plotting Window.....	42
Figure 7-18. Add the Trace for the Target/Mechanical Angle.....	43
Figure 7-19. Addition of the Plotted Target/Mechanical Angle.....	44
Figure 7-20. Add Waveforms of Computed and Ideal Electrical Angle.....	44
Figure 8-1. Top Side of Sensor, LDC5072 and Target PC boards created by the LDC5072 Sensor Design Tool.....	47
Figure 8-2. Underside of Sensor, LDC5072 and Target PC boards created by the LDC5072 Sensor Design Tool.....	48
Figure 8-3. Evaluation PC Board Schematic Without Sensor.....	49
Figure 8-4. Detailed EVM Relative Part Placement for Soldered Components - Identical to Tool-Generated EVM.....	51
Figure 9-1. LDC5072 Sensor Designer State File.....	52

List of Tables

Table 8-1. LDC5072Q1EVM BOM.....	50
Table 9-1. Approximate Sensor Tool Elapsed Times for a Given Set of Design Parameters.....	52

Trademarks

All trademarks are the property of their respective owners.

1 Introduction

The LDC5072-Q1 is an inductive sensor interface that has been designed to provide accurate estimates of a motor's position using simple, inexpensive conductive targets and sense coils. These are usually implemented on PC boards with two or more layers. The LDC5072-Q1 contains internal fault-management circuitry that can alert a system monitor to opens or shorts in PC board traces or wire harnesses, or out-of-range signal levels. For more information, please see the LDC5072-Q1 device data sheet or functional safety manual.

While the system comprised of the target, sensors, and the LDC5072 is simple, designing the sensors to give the desired system behavior can be potentially challenging. Quantifying the electromagnetic interplay of the target, exciter coil, sense coils and the relationship to motor speed, position, and the resulting accuracy of the angle estimates can be difficult and complex. This is where this design tool can help.

The remainder of the document is divided into nine sections. The next section describes how to gain access to the tool and download the installer. The section *Installing the Tool* discusses the steps of installing the tool (based on the MATLAB run-time engine) with particular attention to the folder paths and operating system permissions. The section *Using the Tool to Create a Design* discusses the design steps with an example set of design criteria. The tool also creates a SPICE model of a design. *Using the Tool to Create a SPICE Model of Your Design with the LDC5072* provides background on creation of a SPICE model and gives an overview of the model files. *Running Simulations in Pspice* shows how use the SPICE model in the Cadence *Pspice for TI* simulation manager. The tool creates Gerber files of a design with a single button-click, and *Create Gerber Design Files* reviews the results of the sensor files, target and eval PC board. The next-to-last section provides the bill-of-materials, layout and schematic of the eval PC board. The final section is a brief troubleshooting guide.

2 Gaining Access to the LDC5072 Sensor Design Tool

There are two series of steps to access and install the tool. The first series is the installer setup: download the installer using your active myTI account, choosing the installation directory, approving the license agreement, and unpacking the installer files. The second series involves running the installer: choosing the directories, downloading and installing the MATLAB run-time environment (if not already installed), and completing and confirming the installation.

To gain access to the LDC5072 Sensor Design Tool you must have an active myTI account. Once you have created a myTI account go to the [LDC5072-Q1 product folder](#) and scroll down to the *Design Tools & Simulation* tab. Click on the link *LDC5072 Sensor Design Tool* or click the corresponding download button and view the export approval page. Once there, complete the form and transition to another page that confirms that you have been granted approval. Click the *Download* button to download the tool installer.

3 Installer Setup

After the download, a shortcut appears on your desktop.



Figure 3-1. LDC5072 Design Tool Installer Shortcut on Desktop

Double-click the icon and ignore any messages from Windows Defender. The *Setup* pop-up appears.

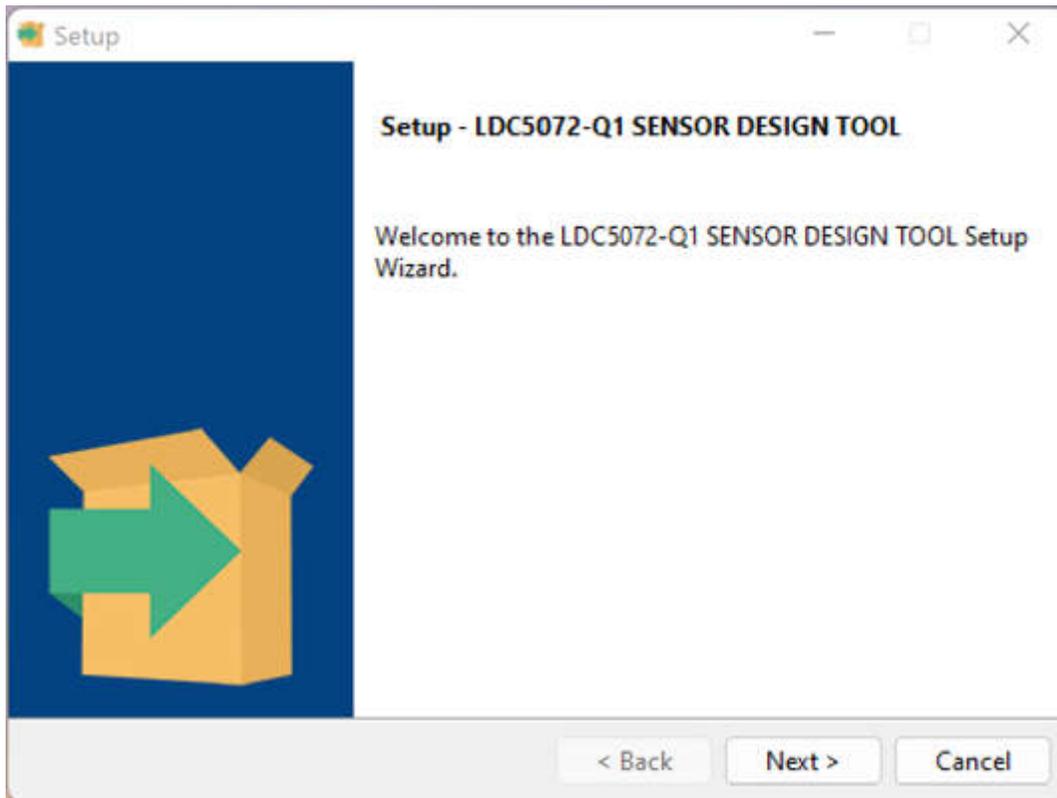


Figure 3-2. LDC5072 Sensor Design Tool Installer & Setup Start Panel

Click the *Next* button and the *License Agreement* appears.

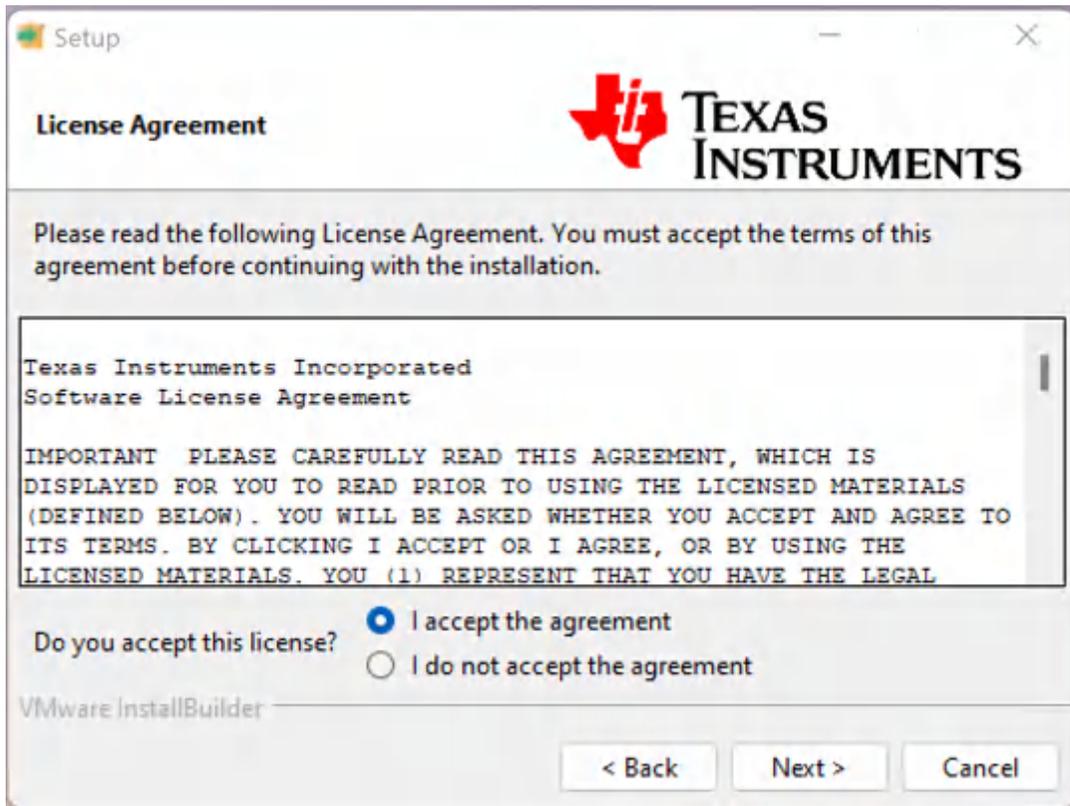


Figure 3-3. LDC5072 Sensor Design Tool License Agreement

Read the license agreement. If the terms are acceptable, click the *I accept the agreement* radio button and the *Next* button. The next pop-up shows a default installation path, but allows an alternative path to be entered.

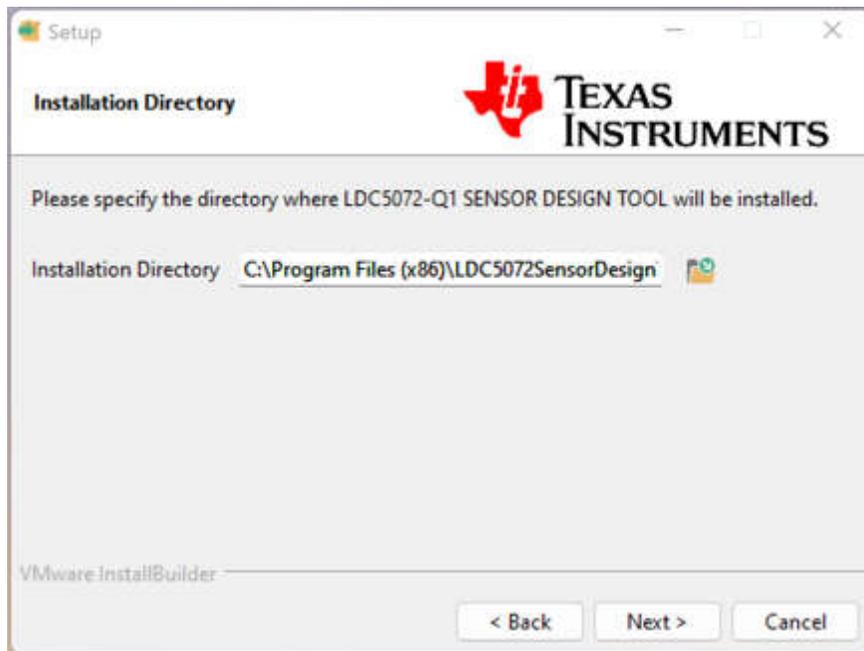


Figure 3-4. Directory Path for Tool Installer & Setup

Once the installation path is entered, click the *Next* button and a status window shows the setup is ready to install. Click *Next* to begin the setup and file unpacking.

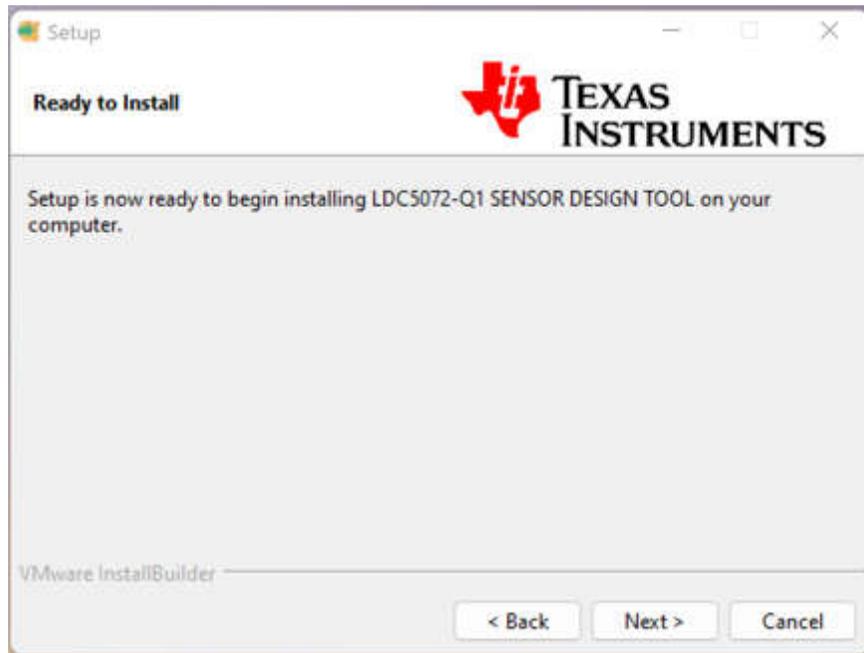


Figure 3-5. Ready to Download Installer Status Window

When the setup and unpacking are complete, a final pop-up appears.

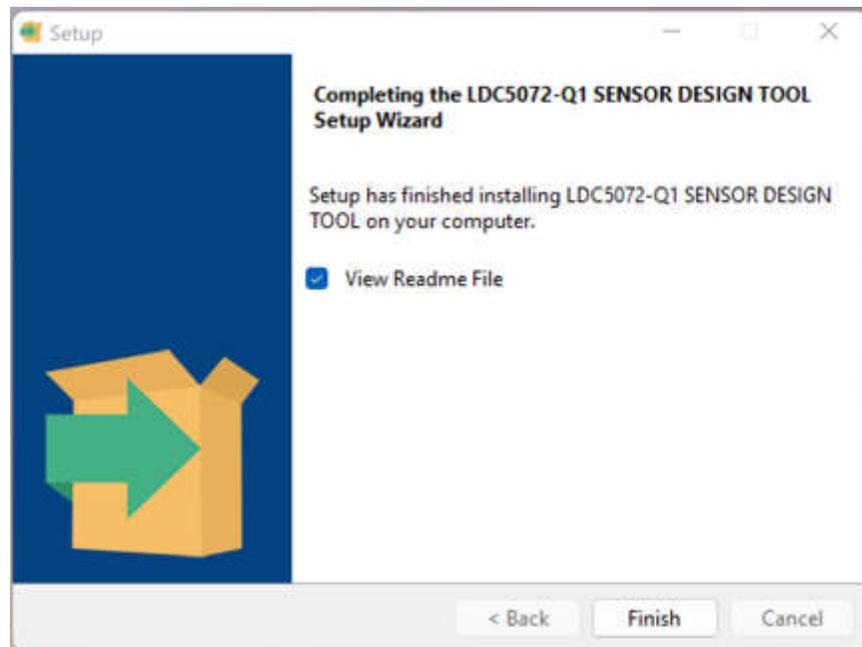


Figure 3-6. Installer Download & Setup Complete Status Window

Click *Finish* and the installer setup is complete.

Note

A *ReadMe* file displays that contains the location of the unpacked files. This information is needed for the next series of installation steps.

4 Installing the Tool

After the installer setup is complete, begin the tool installation by going to the directory listed in the ReadMe file shown at the end of the previous *Installer Setup* series and double-clicking on the file *MyAppInstaller_web_V#.exe*.

Click on the desktop icon and the splash screen briefly appears.

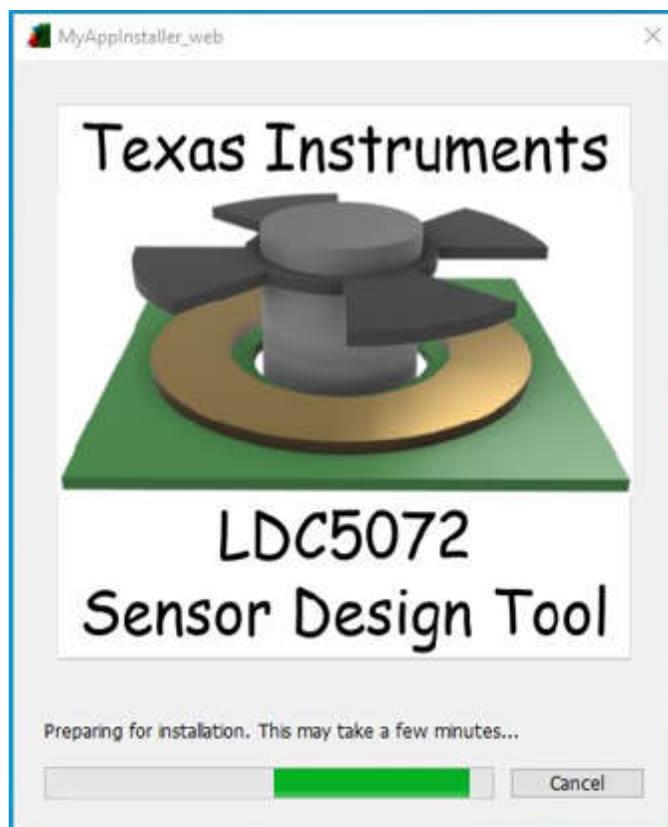


Figure 4-1. Tool Installer Initial Splash Screen

After the splash screen disappears, the first of the installer pop-ups appears. Click the *Next* button.

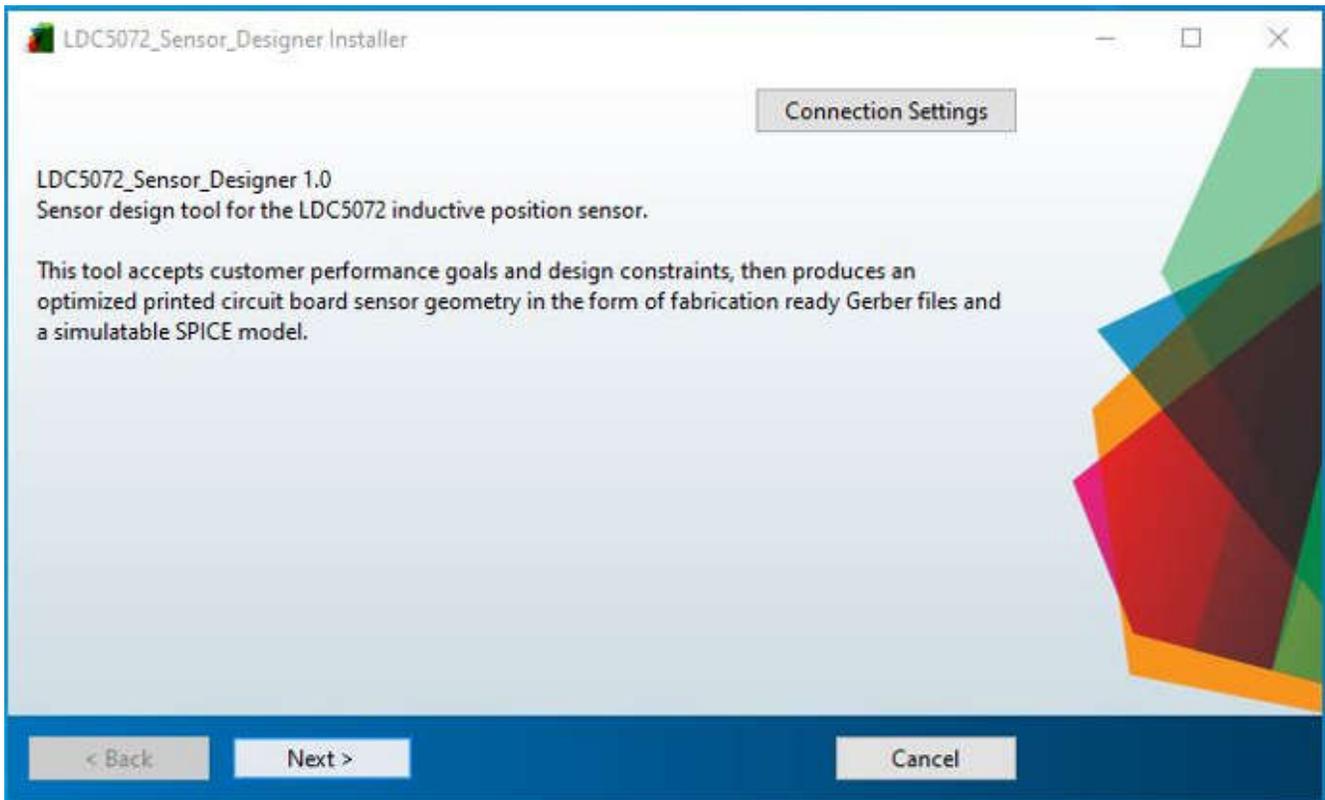


Figure 4-2. Tool Installer *Start Panel*

The *Installation Options* pop-up appears with a default installation path. Enter an alternate path if needed. Leave the check-box checked if you want a shortcut added to your desktop and click *Next*.

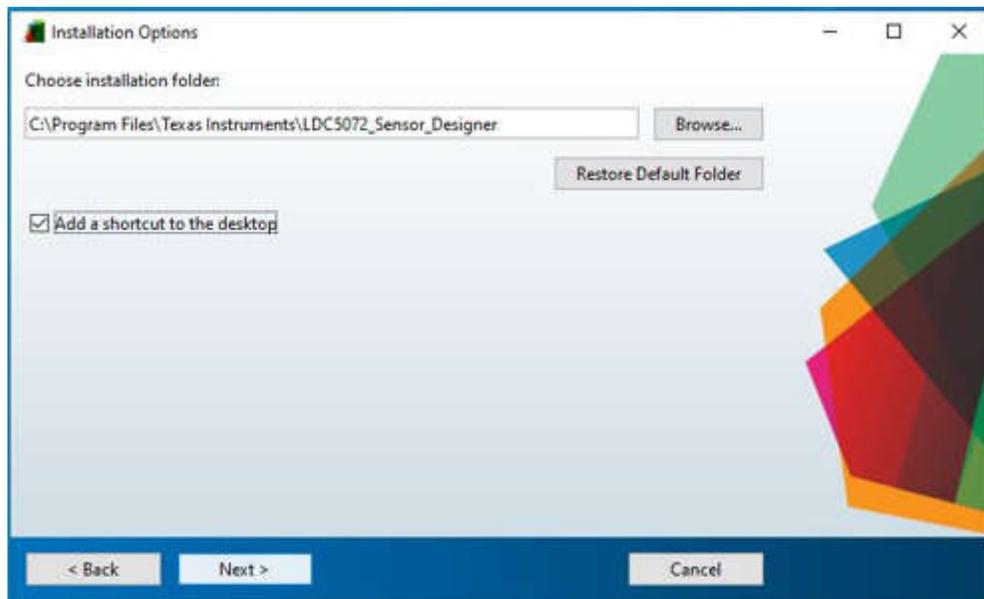


Figure 4-3. Select/Configure Directory Path for Tool Installation

The *Required Software* pop-up appears. Click *Next* to install the MATLAB Runtime Engine (if not already installed.)

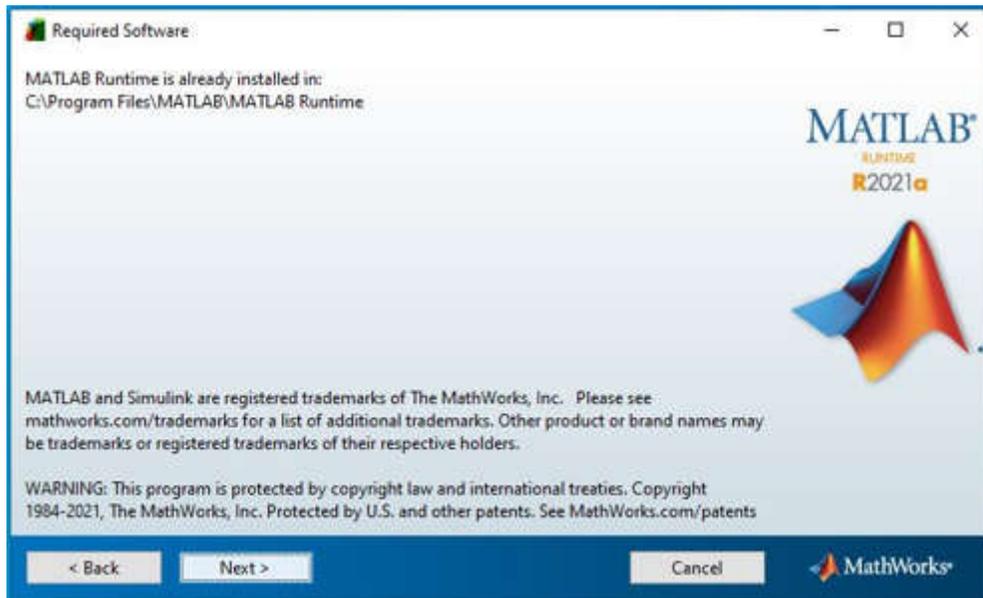


Figure 4-4. Required Software Panel - MATLAB Runtime Engine

The next two pop-ups confirms the installation directories, and the successful completion of the installation.

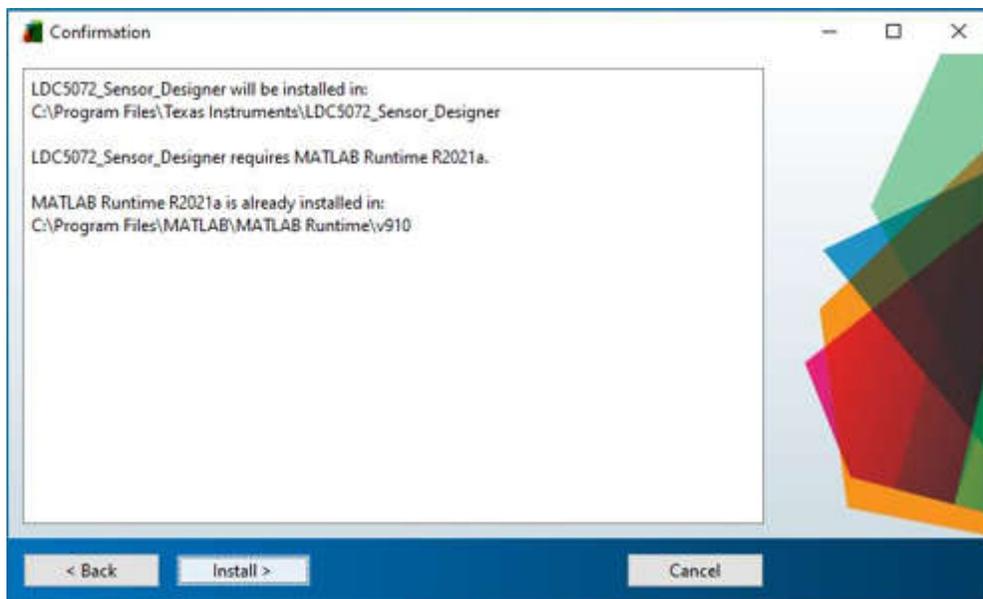


Figure 4-5. Confirm Software and Directories for Tool Installation

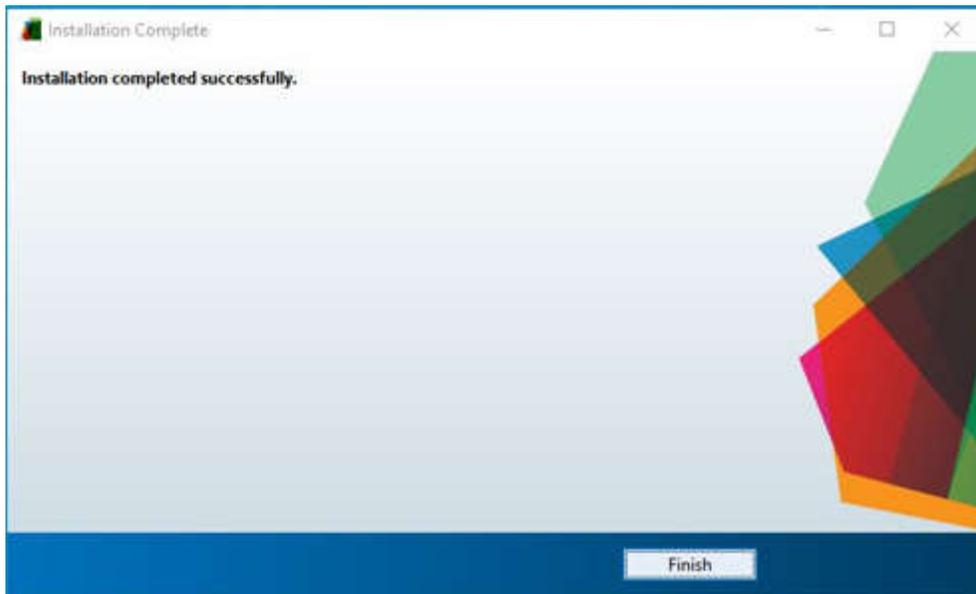


Figure 4-6. Successful Tool Installation - Confirmation

Note

The final step is to update the new desktop short-cut that was part of the installation.

Open the *Properties* of the desktop short-cut by right-clicking and selecting the last menu item. Change the *Start in:* directory to:
`C:\Users\USER_ID\AppData\Roaming\Texas_Instruments\LDC5072_Sensor_Designer\application`
otherwise the supporting .PNG files cannot be found and Windows security cannot allow the log file to be written.

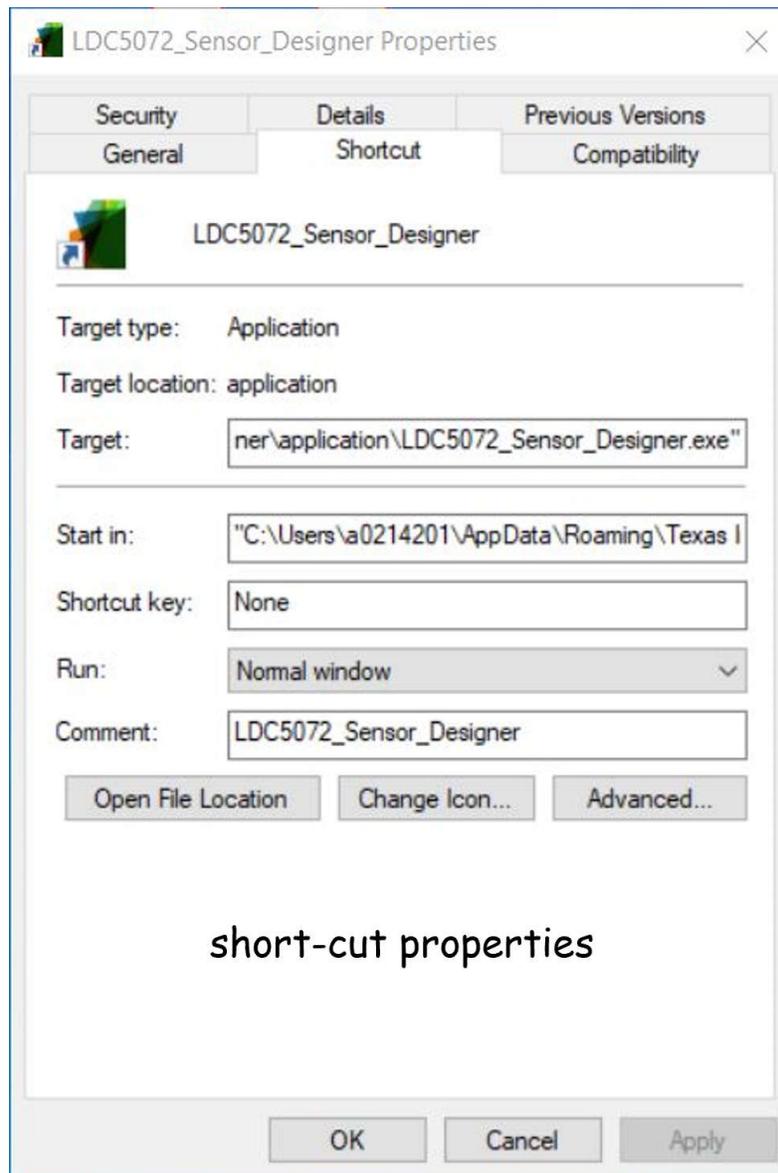


Figure 4-7. Revising the Tool's Desktop Shortcut - Setting the *Start in:* Path

5 Tool Overview

This section gives a user-oriented overview of the tool including the inputs and features.

To start the tool, click on the *LDC5072_Sensor_Designer* short-cut on the desktop (if that option was chosen) or from the Windows *Start* menu. The splash screen shown below appears for a few moments and then closes.



Figure 5-1. Starting the LDC5072 Sensor Designer - Splash Screen

A 60-second lapse between the splash screen closing and the appearance of the sensor design tool GUI (shown in [Figure 5-2](#)) is normal.

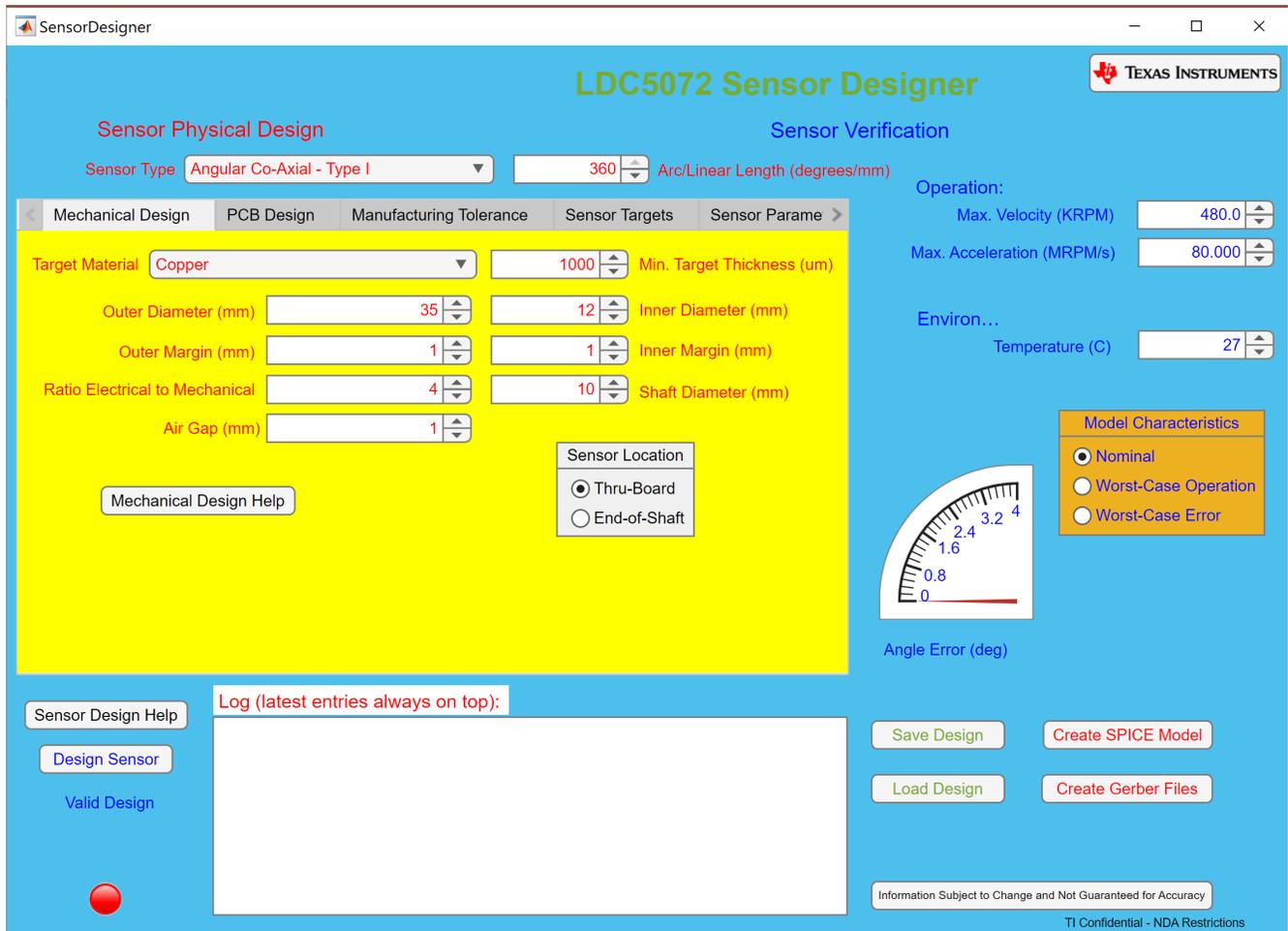


Figure 5-2. LD5072 Sensor Designer Start Screen

The main GUI front panel entries are summarized in Figure 5-3.

GUI

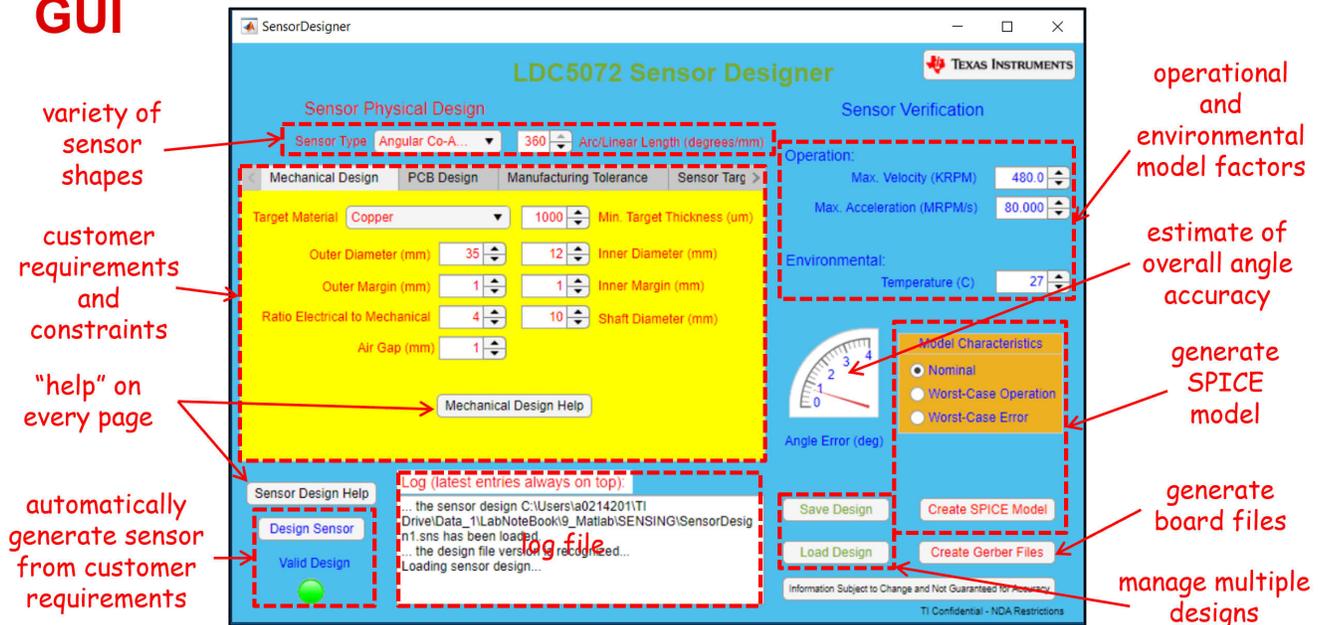


Figure 5-3. GUI Input Field Summary

The tool can support a variety of sensor shapes, as specified in the *Sensor Type* field near the top of the GUI. Inputs for operational and environmental factors (motor rotational speed and acceleration, ambient temperature) are on the upper right side of the GUI. The lower right side of the GUI contains buttons for creating a SPICE model and PC board Gerber files of the design. The *Angle Error* gauge gives an estimate of the overall angle accuracy of the design. The *Save Design* button saves the current design and the *Load Design* button allows the tool to import a saved design. Together these buttons allow a user to manage multiple designs and/or incrementally optimize a single design. The *Log* window, near the bottom of the GUI, displays the contents of the log file. Clicking the *Design Sensor* button in the lower left of the GUI starts a sensor design, which in turn causes the *Valid Design* indicator to turn red. The indicator turns green when the design is complete.

There is a *Help* button on each page. The *Sensor Design Help* button provides information about the buttons and entry fields of the GUI front panel as shown in [Figure 5-4](#).

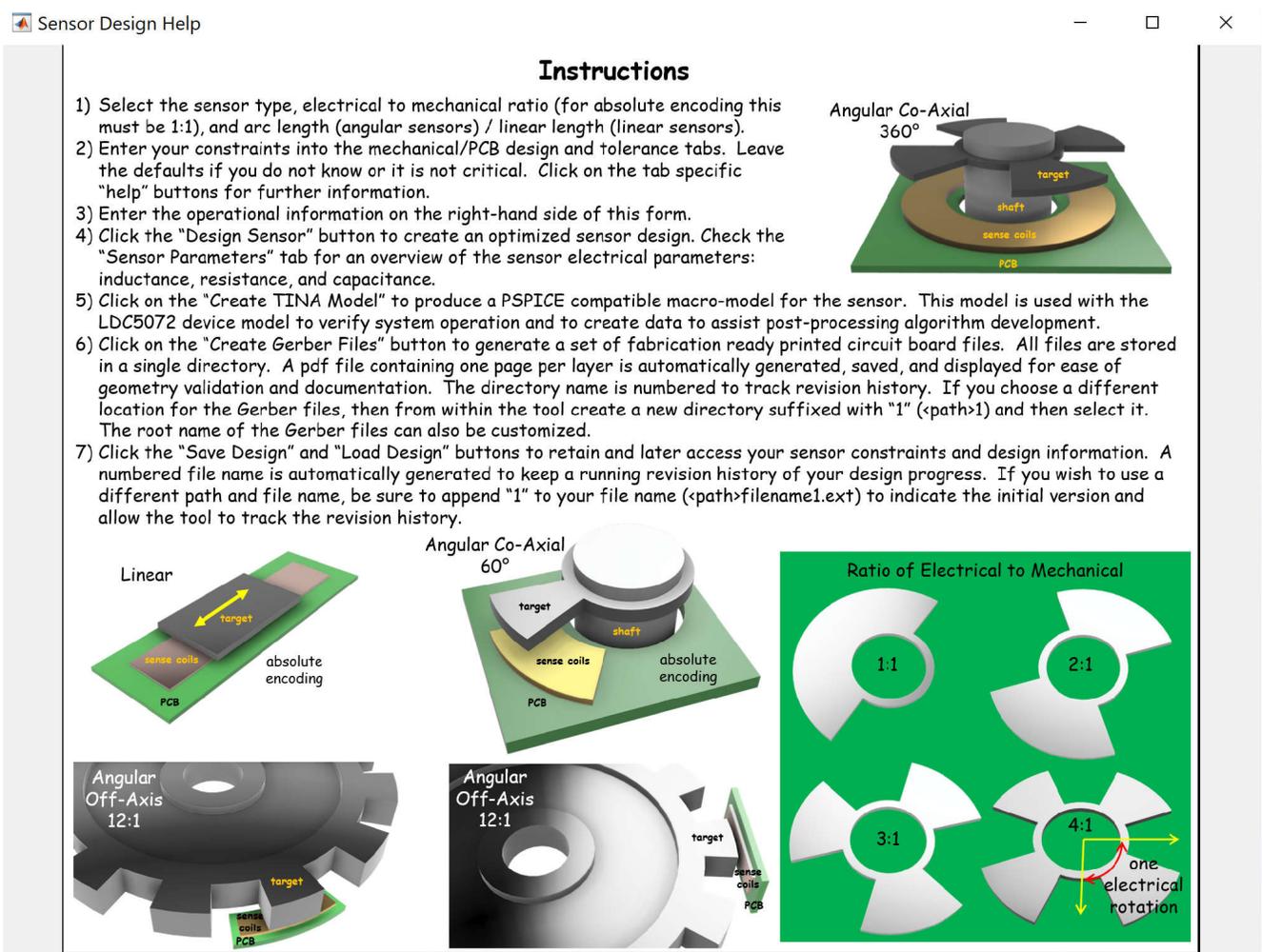


Figure 5-4. GUI Front Panel *Help* Pop-up

The *Mechanical Design* tab shown just below is one of seven tabs, each with individual input fields. Note that most tabs have a *Mechanical Design Help* button.

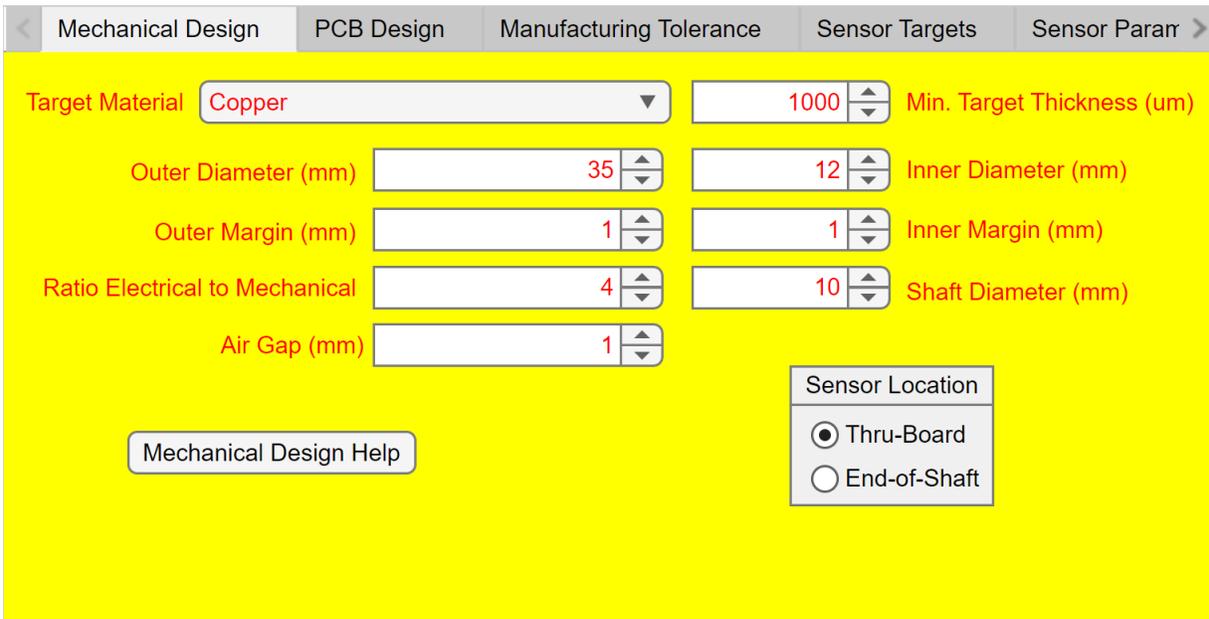


Figure 5-5. LDC5072 Sensor Designer *Mechanical Design* Tab

The Mechanical Design Help pop-up can be seen by clicking that tab's *Mechanical Design Help* button and is shown in Figure 5-6.

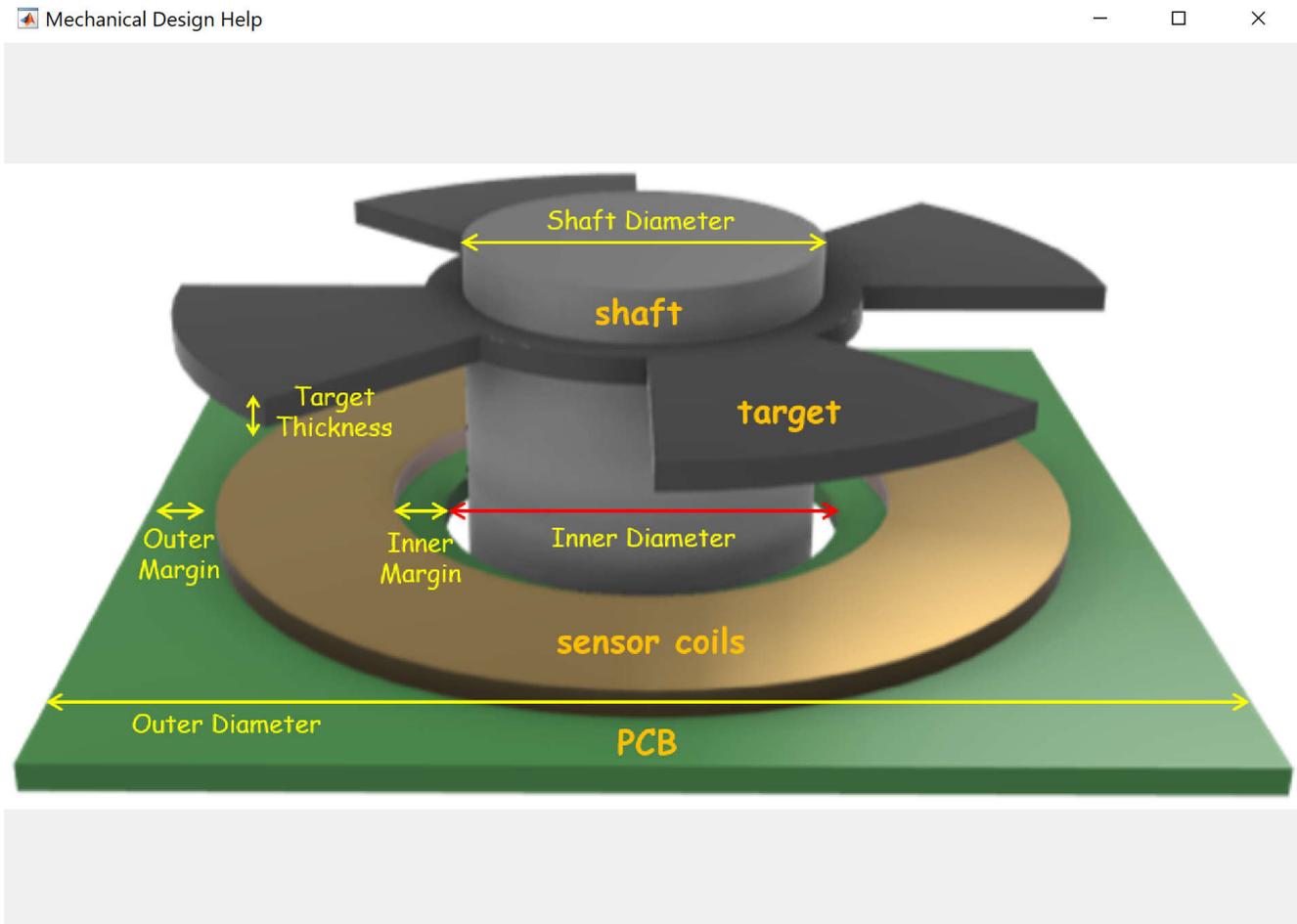


Figure 5-6. LDC5072 Sensor Designer *Mechanical Design Help* Pop-up

As the design help graphic shows above, this tab's input fields specify the mechanical dimensions of the sensor, target, and motor shaft. The field *Ratio Electrical to Mechanical* specifies the number of petals on the target; each petal represents a partial mechanical rotation and a complete electrical rotation through 360 degrees. As an example, a four-petal target rotating through 360 degrees represents a single mechanical rotation and four electrical rotations.

The *PCB Design* tab shown just below contains input fields for the widths and thicknesses of the PC board traces, vias and layers, with the meaning of these parameters shown in the *PCB Design Help* image in the second figure.

Figure 5-7. LDC5072 Sensor Designer *PCB Design* Tab

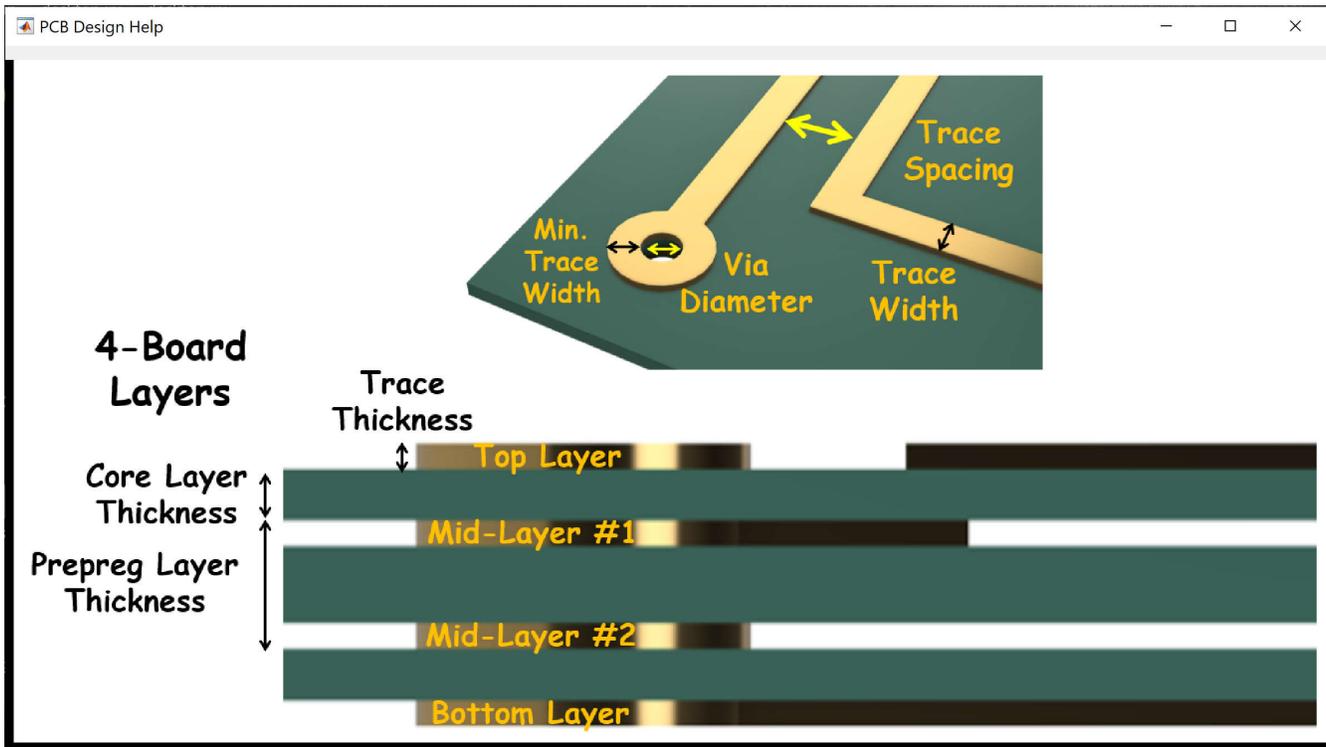


Figure 5-8. LDC5072 Sensor Designer *PCB Design Help* Pop-Up

Design trade-offs between the electrical and mechanical fabrication constraints impact the cost and reliability of the PC board. The PC board parameters can also negatively affect performance if traces are made too thin, thereby increasing parasitic resistance.

The *Manufacturing Tolerance* tab's input fields specify the relative spacing and tilt between the sensor and target, as well as the relative angle of the target with respect to the sense coils. *Variation Rate*, *Runout* and *Coil and Target Tilt* are some of the parameters that can be specified in this tab.

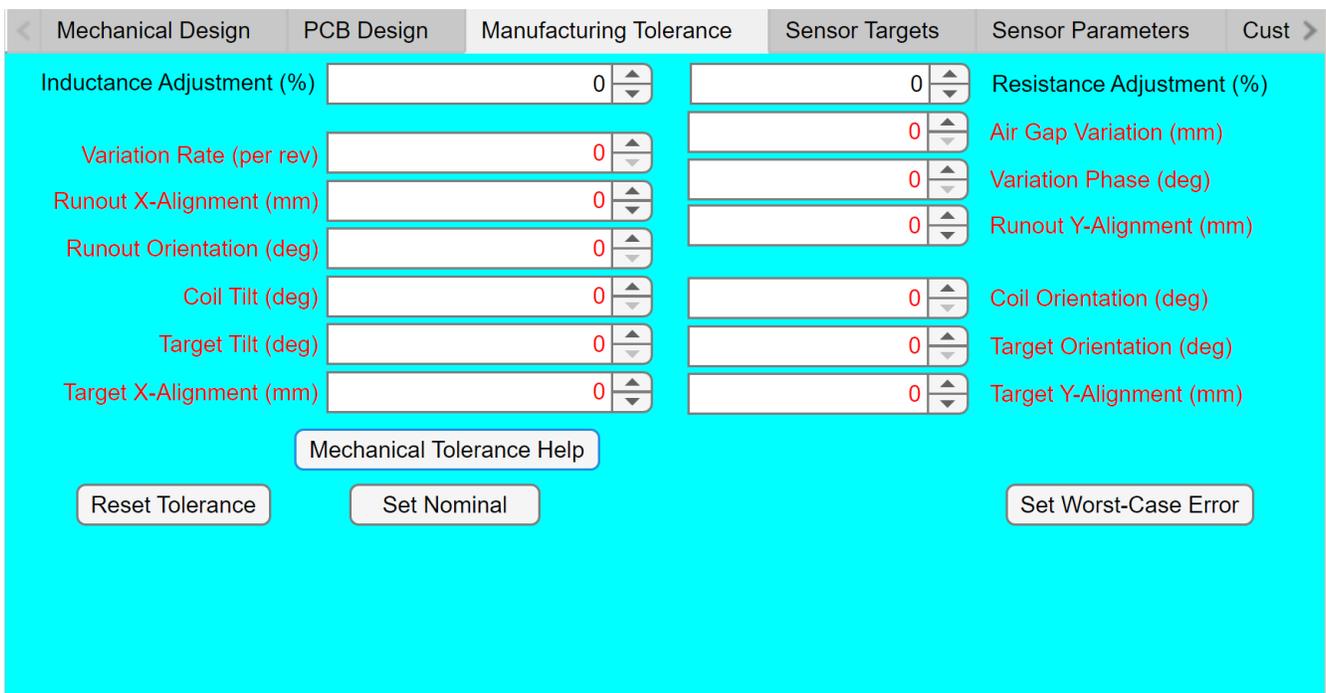


Figure 5-9. LDC5072 Sensor Designer *Manufacturing Tolerance* Tab

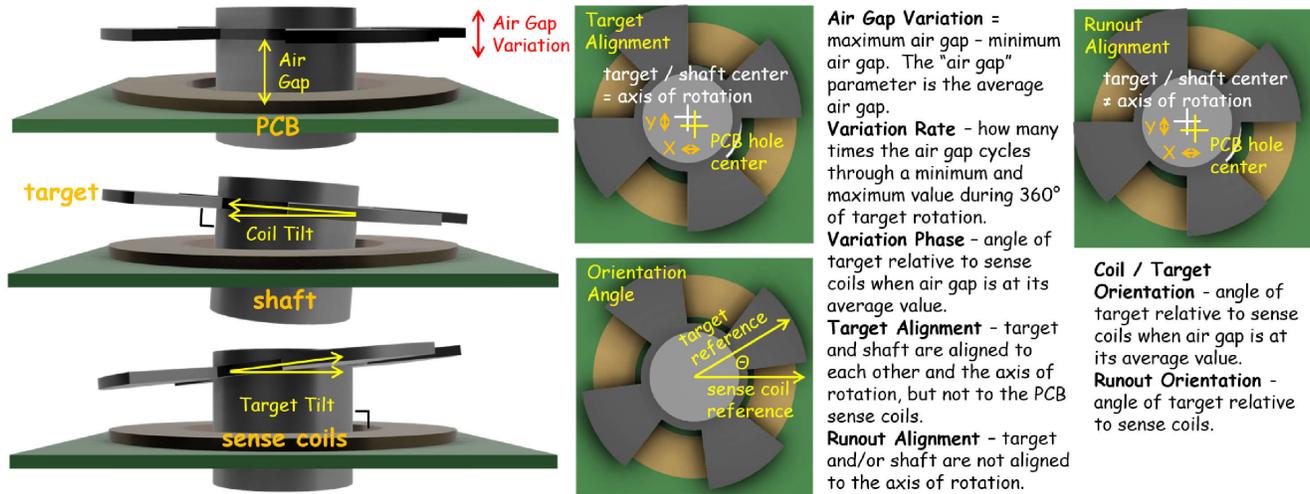


Figure 5-10. LDC5072 Sensor Designer *Manufacturing Tolerance Help* Pop-Up

The *Sensor Targets* tab allows advanced users to design sensors that are outside the guidelines of the data sheet. Not all parameters in this tab can be changed for all sensor types. Clicking the *Optimizer Help* button gives a high-level explanation for the optimizer parameters, as shown in the second figure just below.

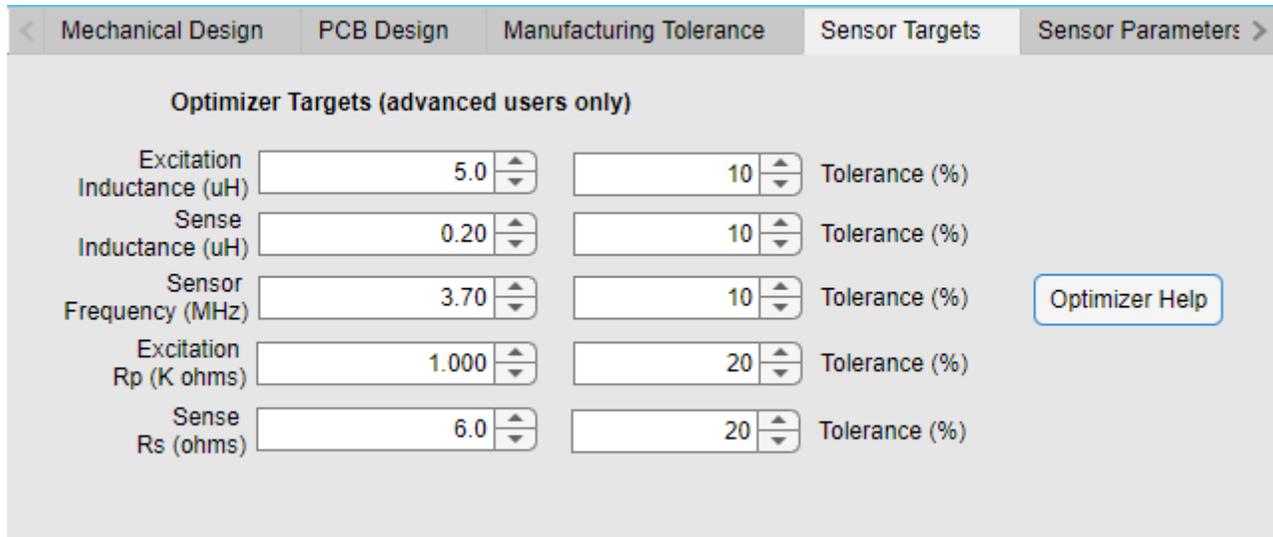


Figure 5-11. LDC5072 Sensor Designer *Sensor Targets* Tab

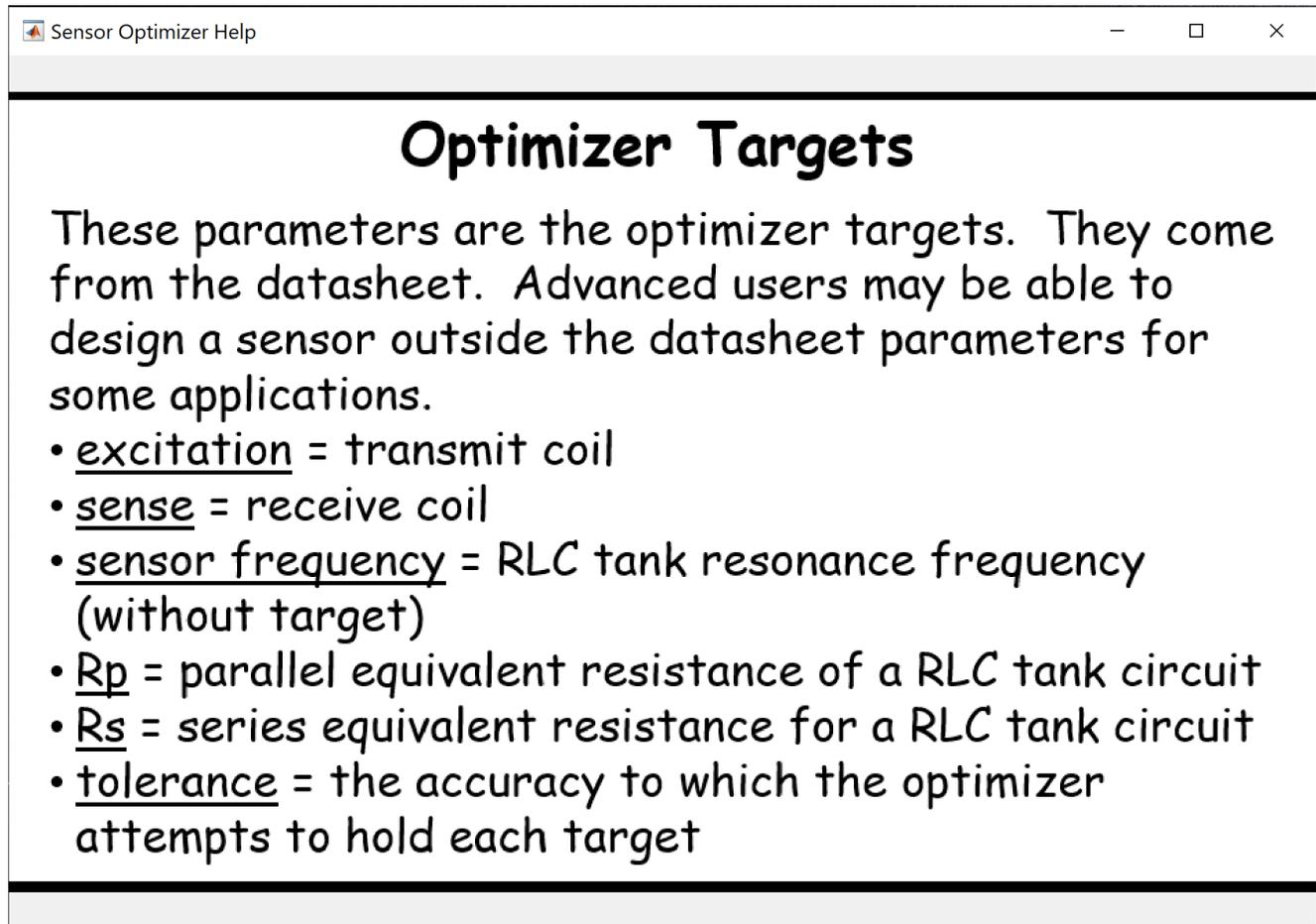


Figure 5-12. Sensor Targets Tab Help Pop-Up

The *LDC5072 Tolerances* tab is used to modify the typical performance of the LDC5072 as described in the *Electrical Characteristics* section of the device data sheet. Changes to this tab modify the LDC5072 Spice model to reflect the values entered in this tab. The Spice model is covered in more detail in [Using the Tool to Create a Design and Generate a Supporting SPICE Model and Gerber Files](#).

6 Using the Tool to Create a Design and Generate a Supporting SPICE Model and Gerber Files

Upon starting the tool, the *Log* window is empty and the *Valid Design* indicator is red as shown in Figure 6-1.



Figure 6-1. LDC5072 Sensor Designer Tool Start Panel

The example illustrated here assumes the parameters across all of the tabs stay at the default values. Note the parameters *Max Velocity* and *Max Acceleration* on the right side of the front panel, and which impacts Spice simulations. In simulations, the *Max Acceleration* controls the speed-up from zero to *Max Velocity*. If the acceleration is set to zero, then the target starts and operates at *Max Velocity*, which is recommended for short run times. If *Max Acceleration* is non-zero, the target starts at zero rotational speed and accelerates to *Max Velocity*, then decelerates back to zero velocity. This can result in very long run times. The *Temperature* parameter sets the ambient temperature of the Spice simulation.

The Spice model can be made to include tolerances from the LDC5072 data sheet via the *LDC5072 Tolerances* tabs. The table in the *Sensor Parameters* tab provides summary tables of the mutual inductances and resistance, as well as the sensor coefficients of capacitance and inductance as calculated by the optimizer.

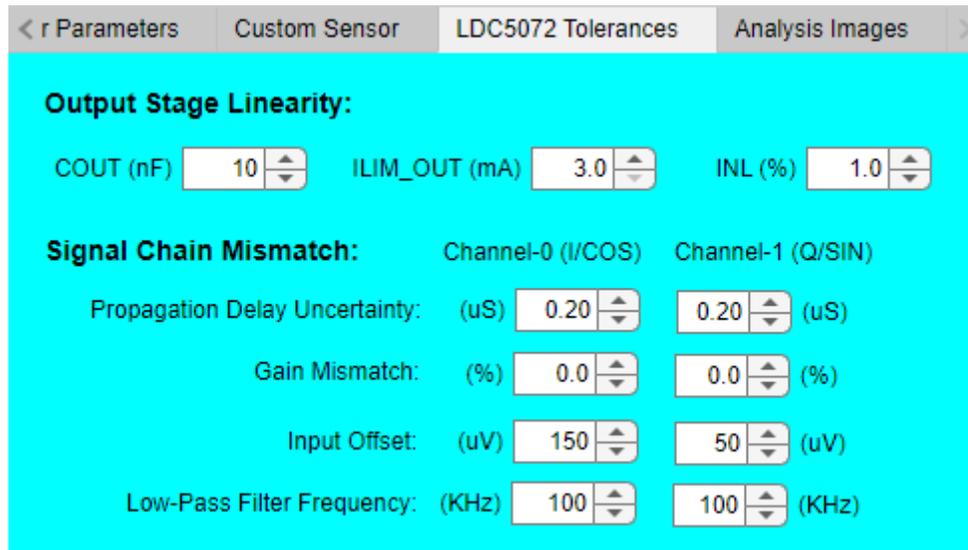


Figure 6-2. LDC5072 Sensor Designer *Tolerances* Tab

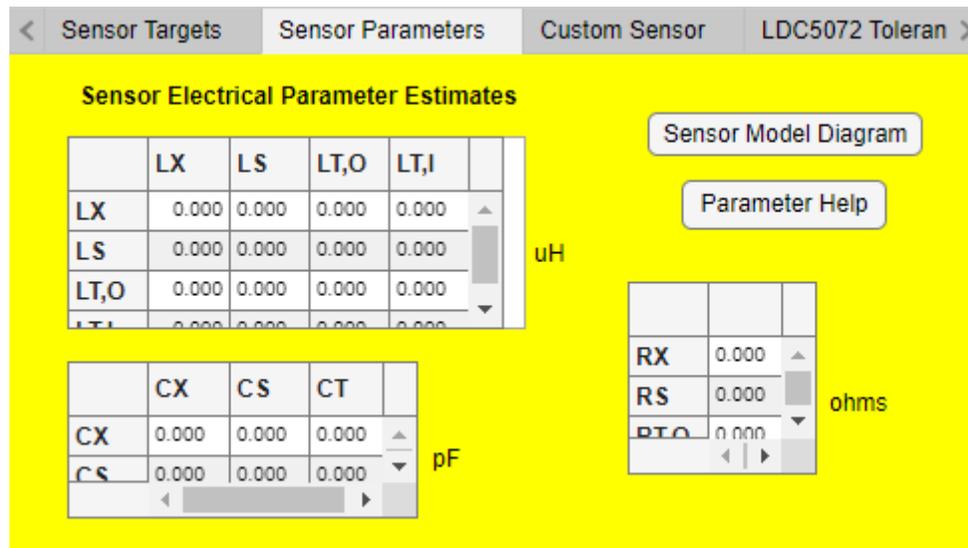


Figure 6-3. LDC5072 Sensor Designer *Sensor Parameters* Tab

A schematic representation of the sensor Spice model can be seen by clicking the *Sensor Model Diagram* button on the *Sensor Parameters* tab. An explanation of the sensor electrical parameters can be seen by clicking the *Parameter Help* button.

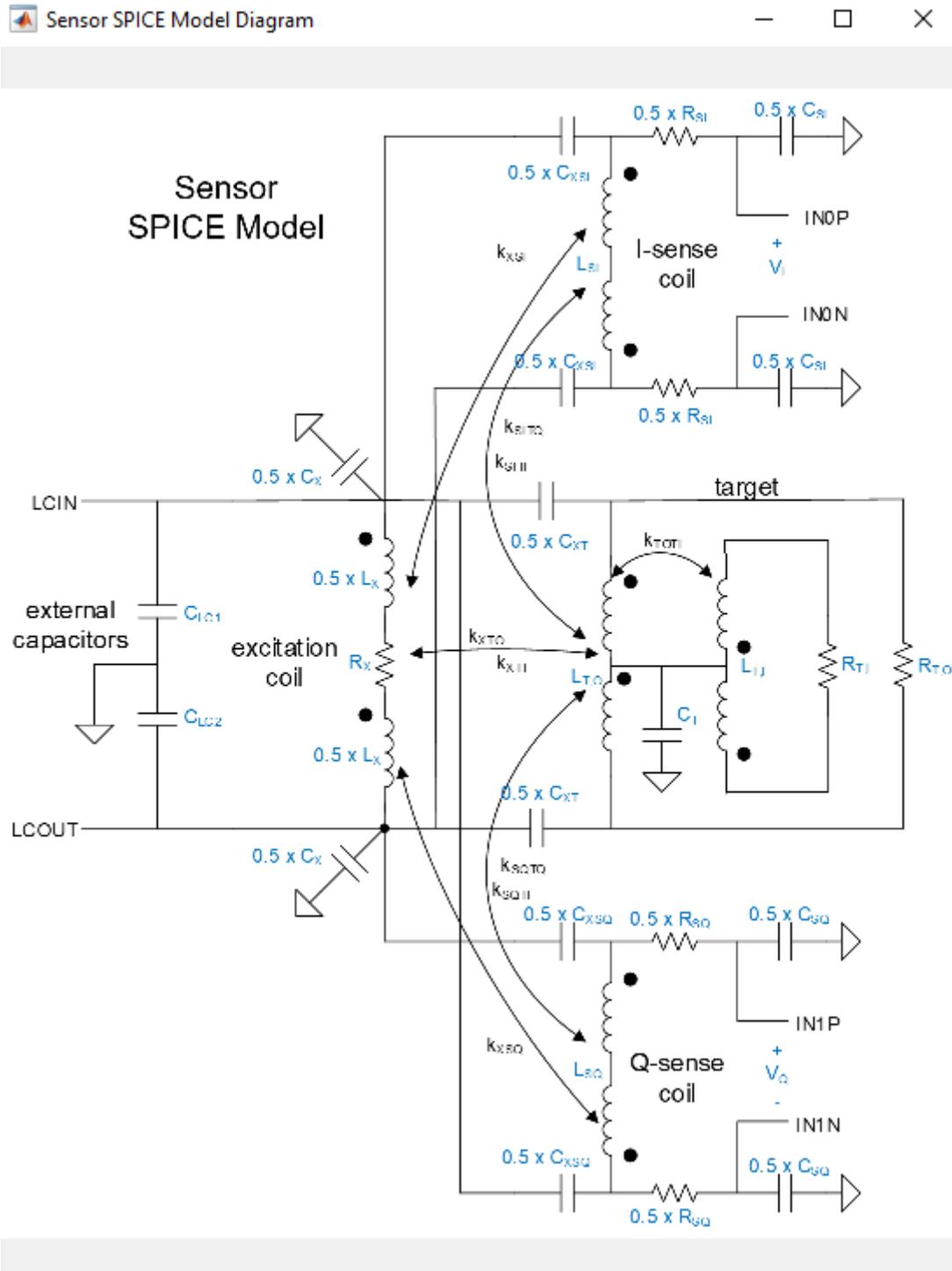


Figure 6-4. Schematic Representation of the Sensor Spice Model

Sensor Electrical Parameters

These parameters are the calculated mutual and self inductances and capacitances for the sensor structure. They are used to generate the SPICE electrical model. The diagonal entries represent self-inductances and capacitances. The off-diagonal entries represent the mutual-inductances and capacitances.

- L_X = excitation coil inductance
- L_S = sense coil inductance
- $L_{T,O}$ = target outer inductance
- $L_{T,I}$ = target inner inductance
- R_X = excitation coil resistance
- R_S = sense coil resistance
- C_X = excitation coil capacitance
- C_S = sense coil capacitance
- C_T = target outer capacitance
- $R_{T,O}$ = target outer resistance
- $R_{T,I}$ = target inner resistance

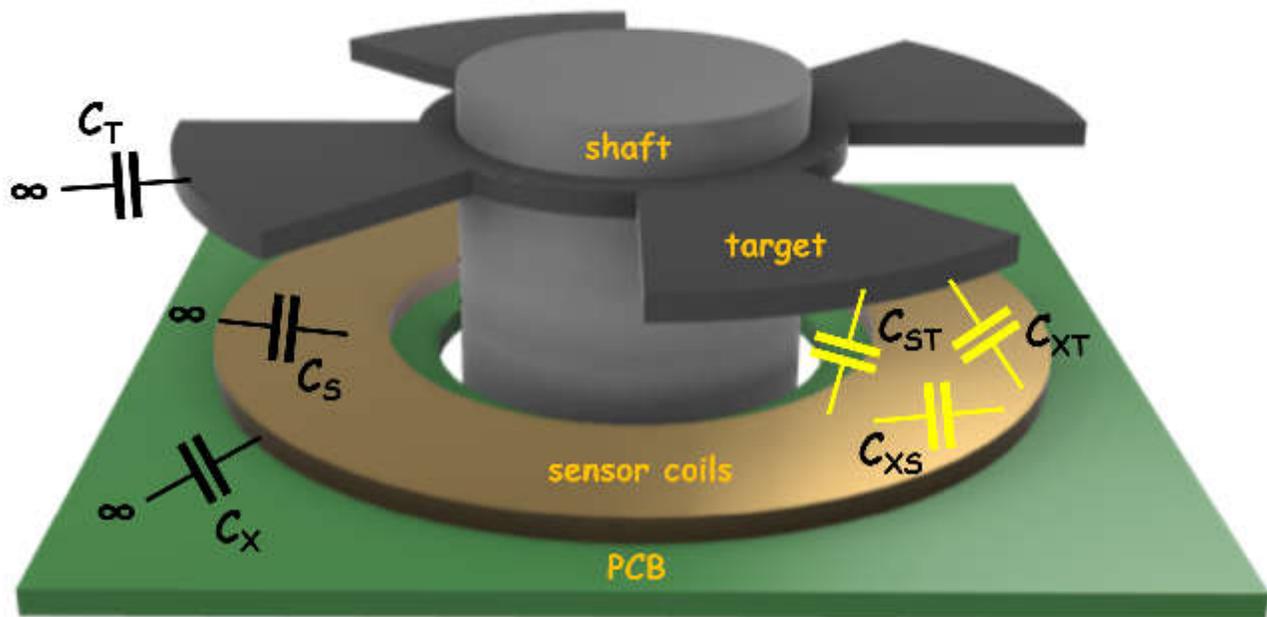


Figure 6-5. Parameter Help Pop-up for the Sensor Spice Model

Clicking the *Design Sensor* button starts the tool optimizer, with *Log* window entries scrolling through different status messages as the design progresses. When the design is complete, the *Valid Design* indicator turns green and the *Log* window displays "...completed design process." as shown in the figure below.

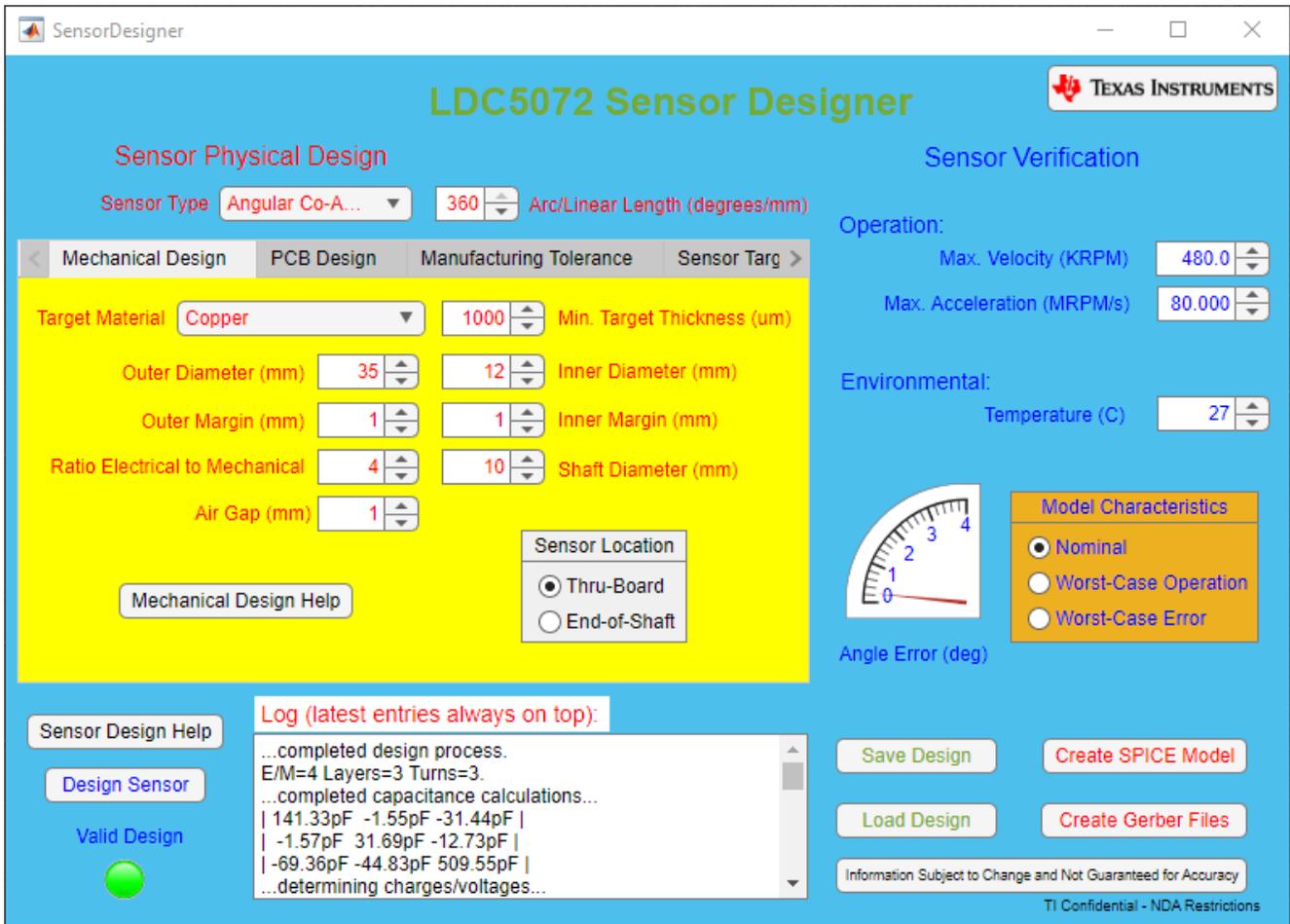


Figure 6-6. LDC5072 Sensor Designer Front Panel: Completion of a Valid Design

Some of the physical characteristics of the sensor designed by the tool's optimizer can be varied. After the optimizer produces a design, click the *Import* button on the *Custom Sensor* tab shown in the figure below to import the physical characteristics. Adjustments to the optimizer-created design can now be made. When the box *Use this custom design for Save Design, Create Gerber Files, and Create TINA Model functions* is checked, then the tool generates Gerber files and Spice models based on the adjusted design.

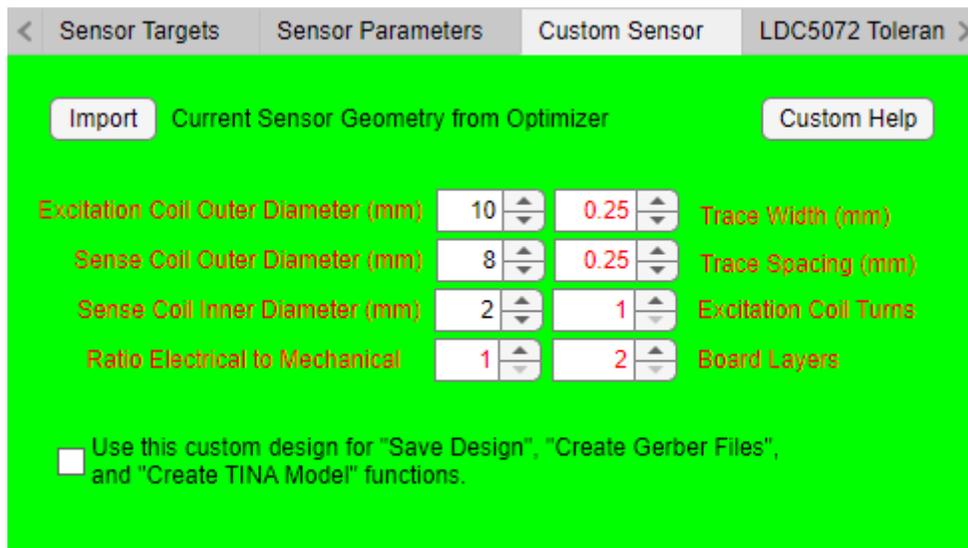


Figure 6-7. LDC5072 Sensor designer Custom Sensor Tab

The tool can create three SPICE model files by clicking the *Create Spice Model* button after completion of a design. The tool can also create Gerber files by clicking the *Create Gerber Files* button after the completion of the design.

The SPICE models are in the form of text files with .cir extensions. One file is for the sensor and target model (as shown in the [Figure 6-4](#) above). One file is for the LDC5072 model, and one is a test bench that can run a transient simulation of the sensor+target+LDC5072. Details on how to best run the test bench simulation are provided in the section [Running Simulations in Pspice](#)

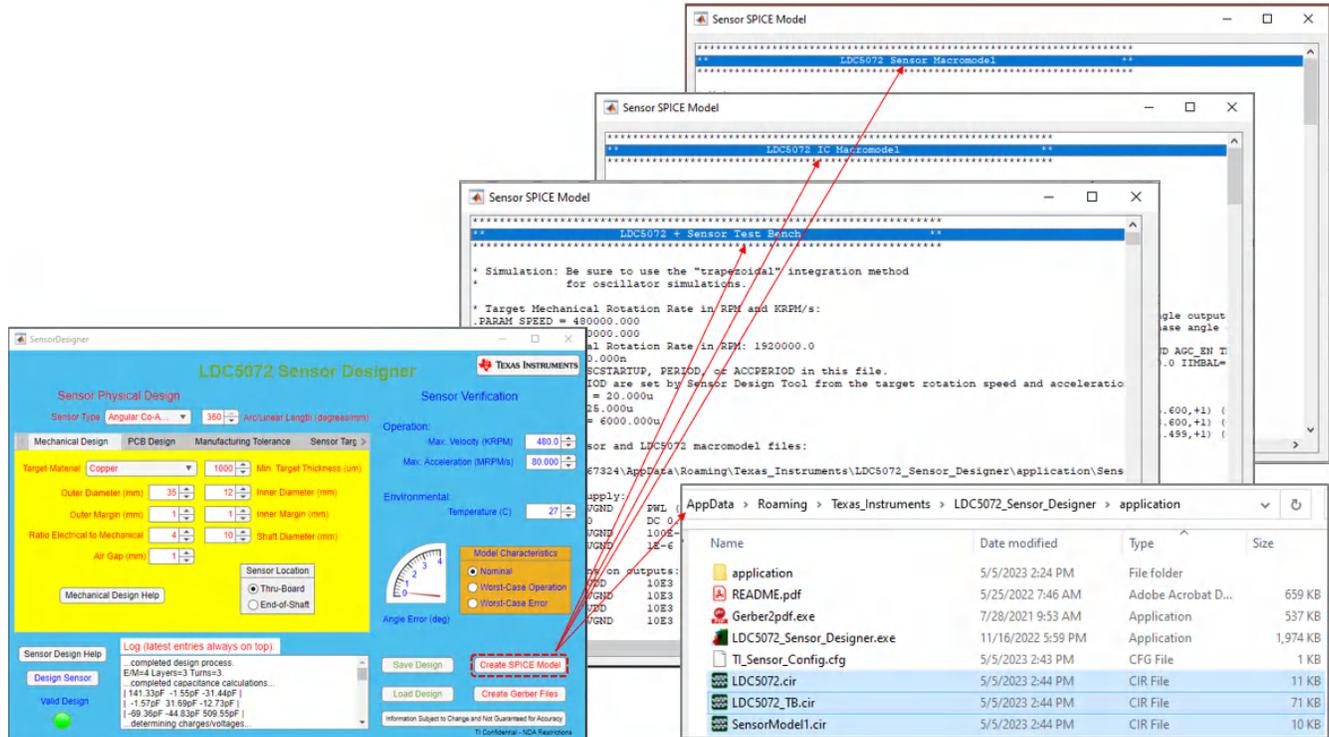


Figure 6-8. Spice Model Creation for the Sensor, LDC5072 and a Test Bench

Note

When creating the SPICE models, please choose the default folder instead of navigating to an alternate folder. If an alternate directory is chosen, not all of the three target models can be saved.

Clicking the *Create Gerber Files* button on the front panel makes the tool create a set of Gerber files for the target, sense coils and an evaluation PC board with the sense coils and an LDC5072-Q1 device. This also creates *.pdf files of those assemblies, as well as schematics, and part placement diagrams and displays these as a pop-up. All files are saved to the application folder when created (e.g. C:\Users\UserID\AppData\Roaming\Texas_Instruments\LDC5072_Sensor_Designer\application\SensorGerber#)

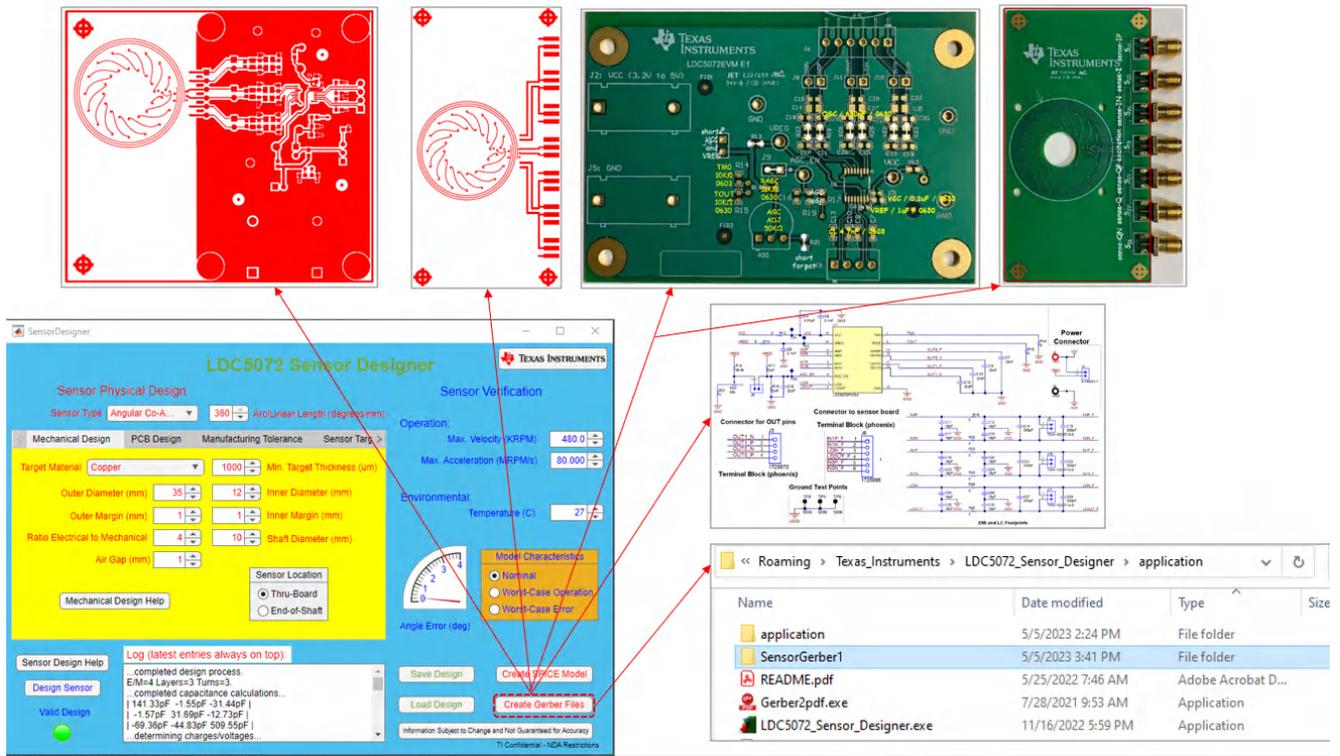


Figure 6-9. Gerber File Creation for the Sensor, LDC5072 and a Test Bench

7 Running Simulations in Pspice

This section shows how the Spice models created by the tool can be used in a Spice transient simulation. These instructions assume the Spice models for the design have been created as described in the previous section.

All three models are usable in many Spice simulators, including [Pspice for TI](#). The sensor and LDC5072 device models (*SensorModel.cir*, *LDC5072.cir*) can be imported and used as components in a larger schematic capture and/or circuit simulation, while the test bench file *LDC5072_TB.cir* can support a stand-alone Spice simulation. The remainder of this section gives an overview of how to use the test bench in a transient simulation using the *Cadence Allegro Pspice Simulator*.

The *Cadence Allegro Pspice Simulator* is started by selecting the PSpice AD application in the Windows Start menu as shown in the figure below. The number 17.4 refers to the version number of the tool used in this example.

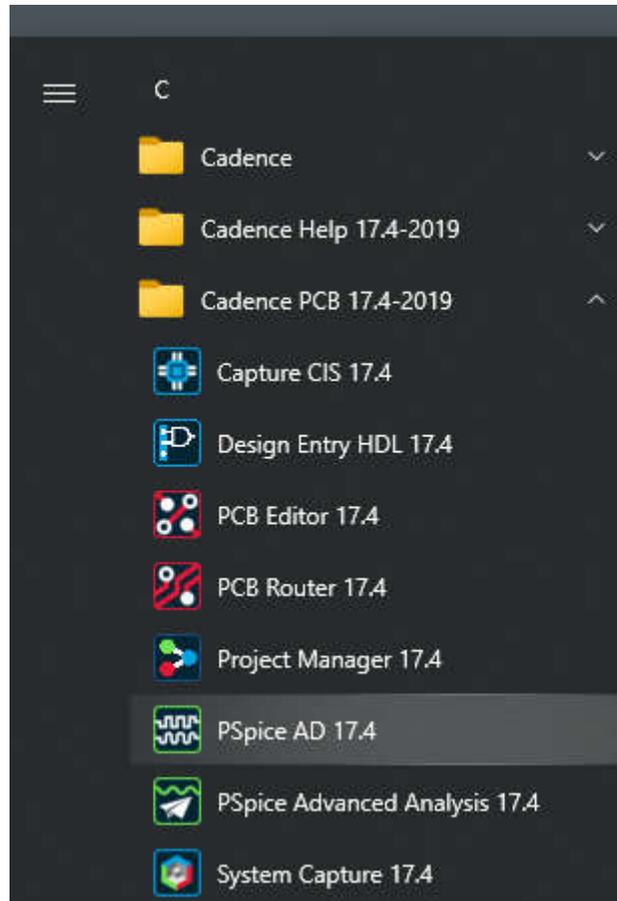


Figure 7-1. Starting the Cadence Allegro Pspice Simulator in the Windows Start Menu

The simulator window appears and resembles [Figure 7-2](#).

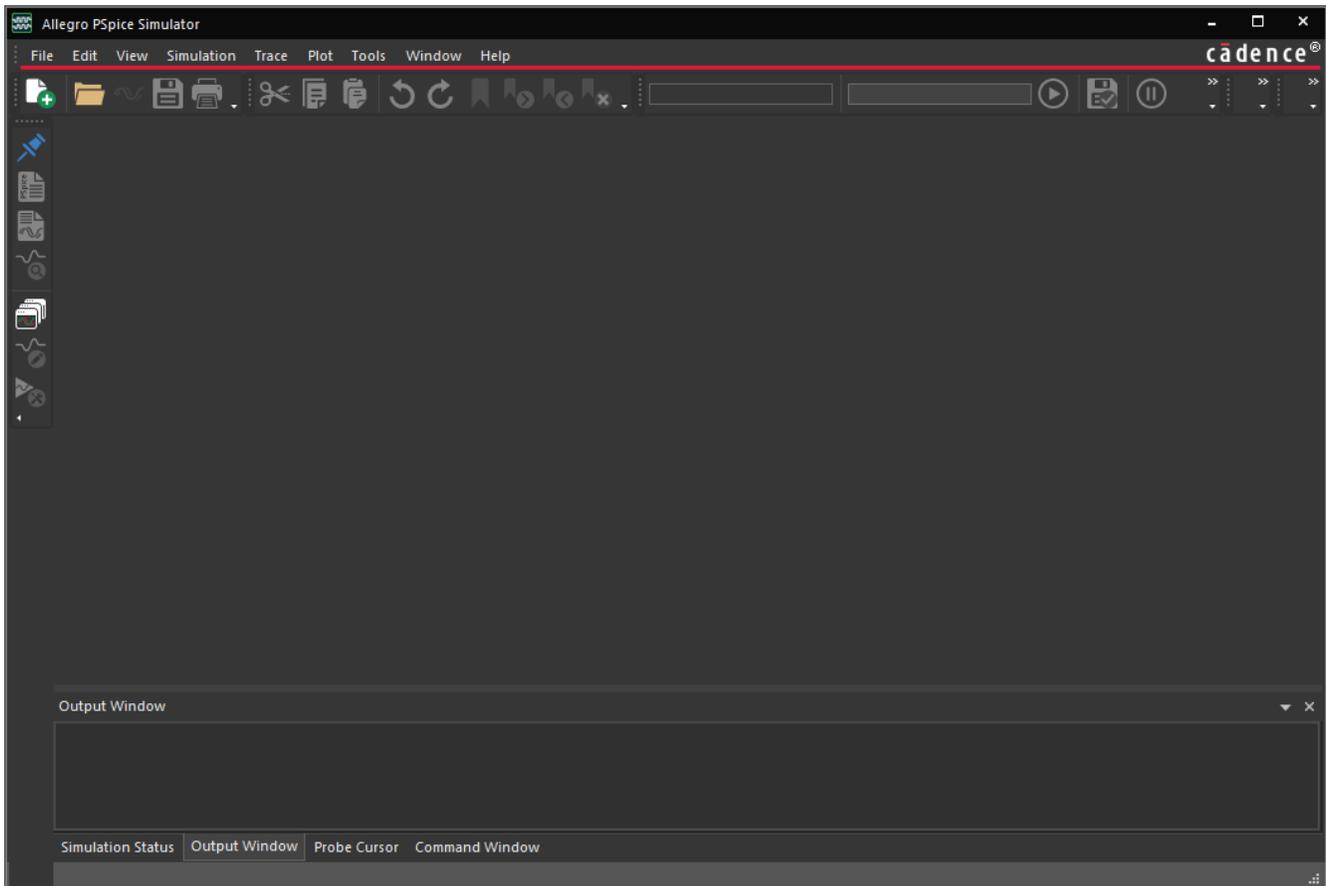


Figure 7-2. Cadence Allegro Pspice Simulator Window

As shown in [Figure 7-3](#) and [Figure 7-4](#), after the window appears the test bench is opened by selecting the *File > Open* command in the upper menu, and navigating to the application folder. The *File Type* field in the window needs to be changed to *.cir to be able to see the test bench file *LDC5072_TB.cir*. Select the file and click *Open*.

Note

For the test bench simulation to run, the LDC5072 model (LDC5072.cir) and the sensor model (SensorModel.cir) must be in the same folder as the test bench file.

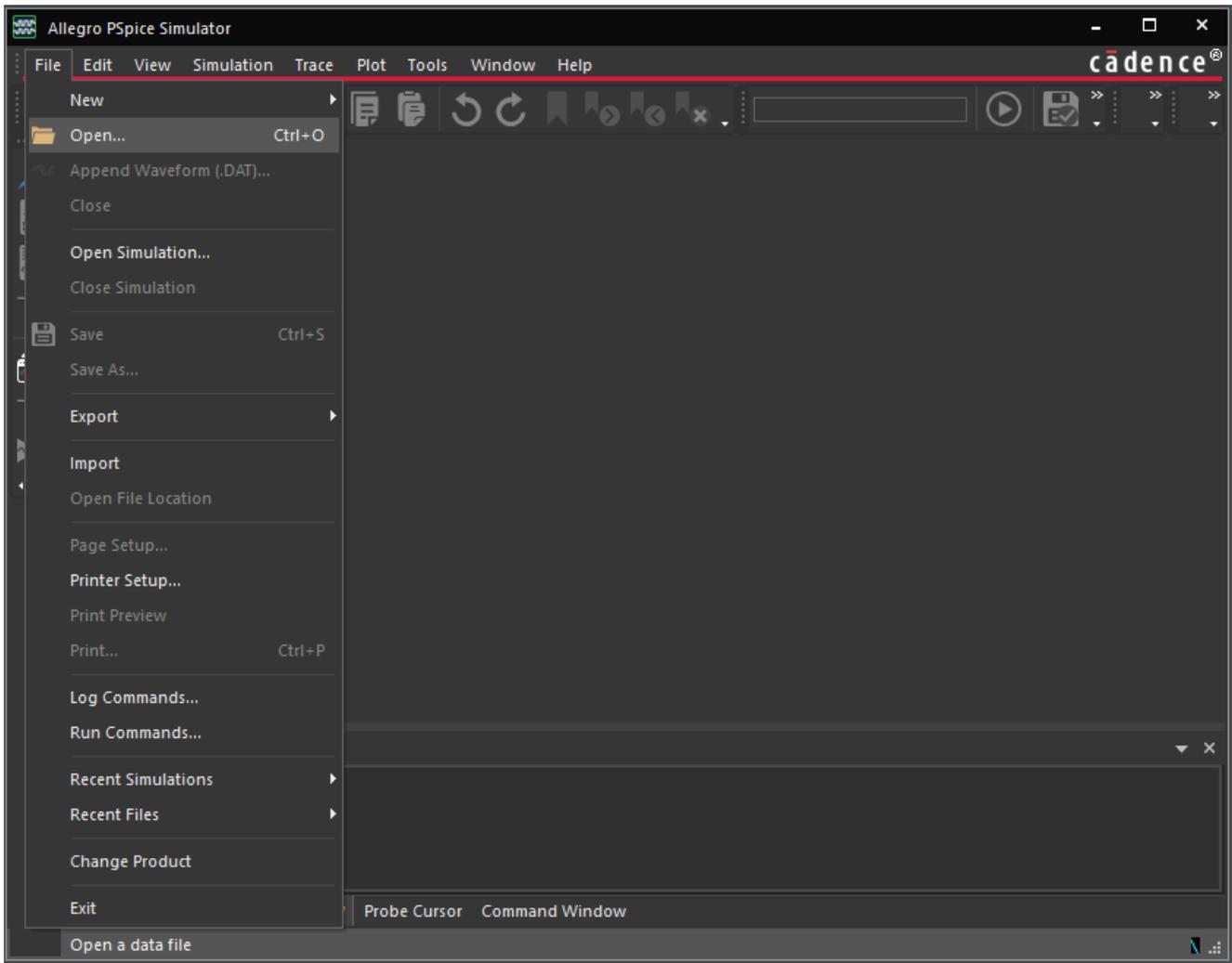


Figure 7-3. Opening the Spice Test Bench - Step 1

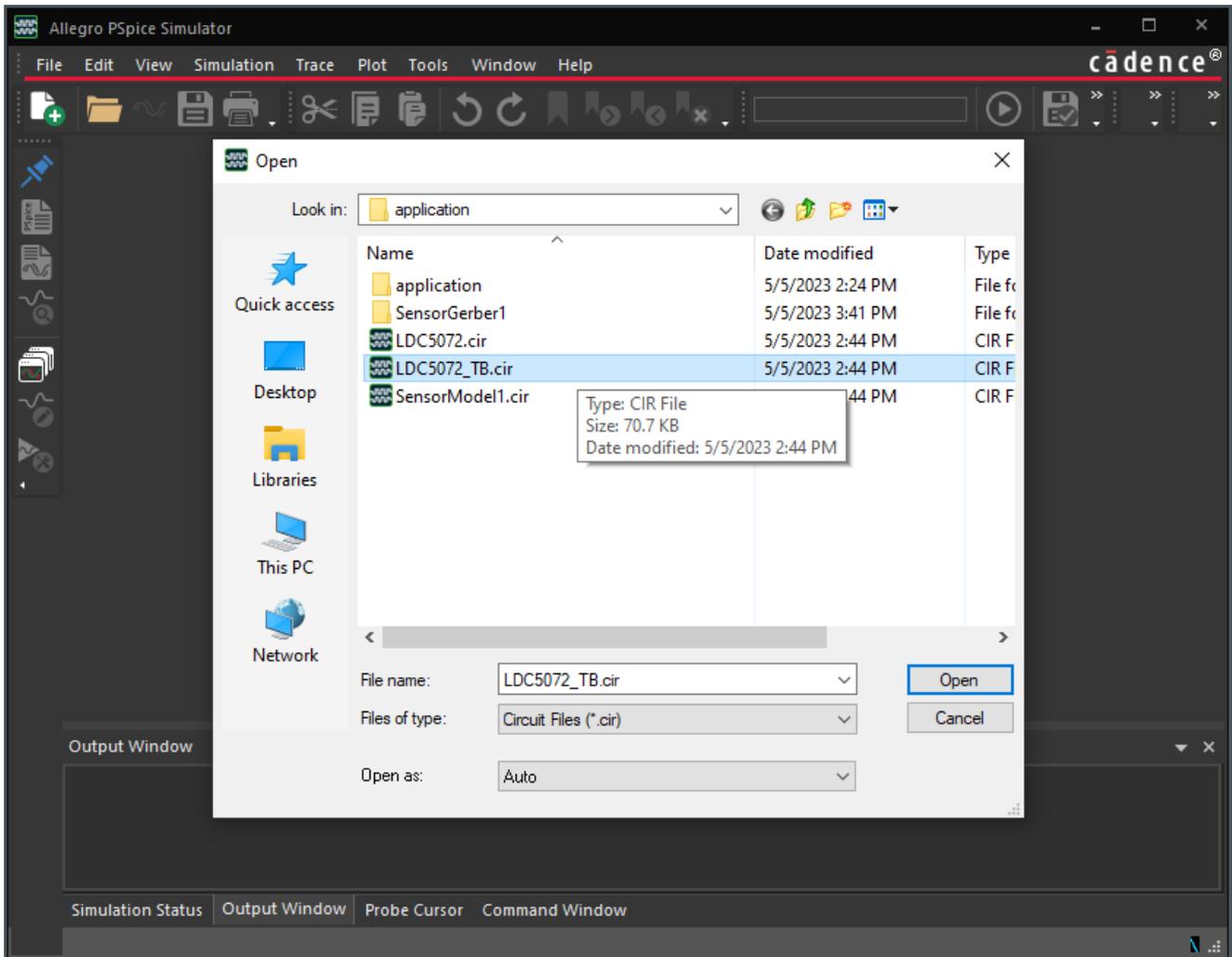


Figure 7-4. Opening the Spice Test Bench - Step 2

If you are using *Pspice for TI* you do not have the option of starting the *Cadence Allegro Pspice Simulator* directly from the Windows Start menu. You can easily access the simulator window by opening *Pspice for TI* and running a simulation of one of the built-in internal test benches. The *Cadence Allegro Pspice Simulator* window appears during or upon completion of the test bench simulation. After the completion of the simulation, close any plot windows and open your sensor design's test bench via the *File > Open* command in the upper menu and following the steps in the previous paragraph.

Opening the *LDC5072_TB.cir* file in Allegro can appear similar to the image shown just below. The upper parameters and comments reflect values for the sensor design. The first two *.PARAM* statements reflect the motor's rotational *Max Velocity* and *Max Acceleration* values chosen for this sensor design. For this example, the value in the comment "Target Electrical Rotation Rate in RPM = 1920000.0" results from the product of the parameters on the *Mechanical Design* tab *Max Velocity* (= 480 kRPM) and the *Ratio Electrical to Mechanical* (= 4). The *.PARAM CLOAD* = 10.000 n reflects the nominal load capacitance of the LDC5072 analog outputs. The parameters *OSCSTARTUP*, *PERIOD*, *ACCPERIOD* remain at their assigned values and do not change. The *TEMP* parameter was chosen in the *Mechanical Design* tab and reflects the modeled ambient temperature. The two *.LIB* statements include the sensor and LDC5072 device models in the test bench. Without those statements the test bench circuit is not complete.

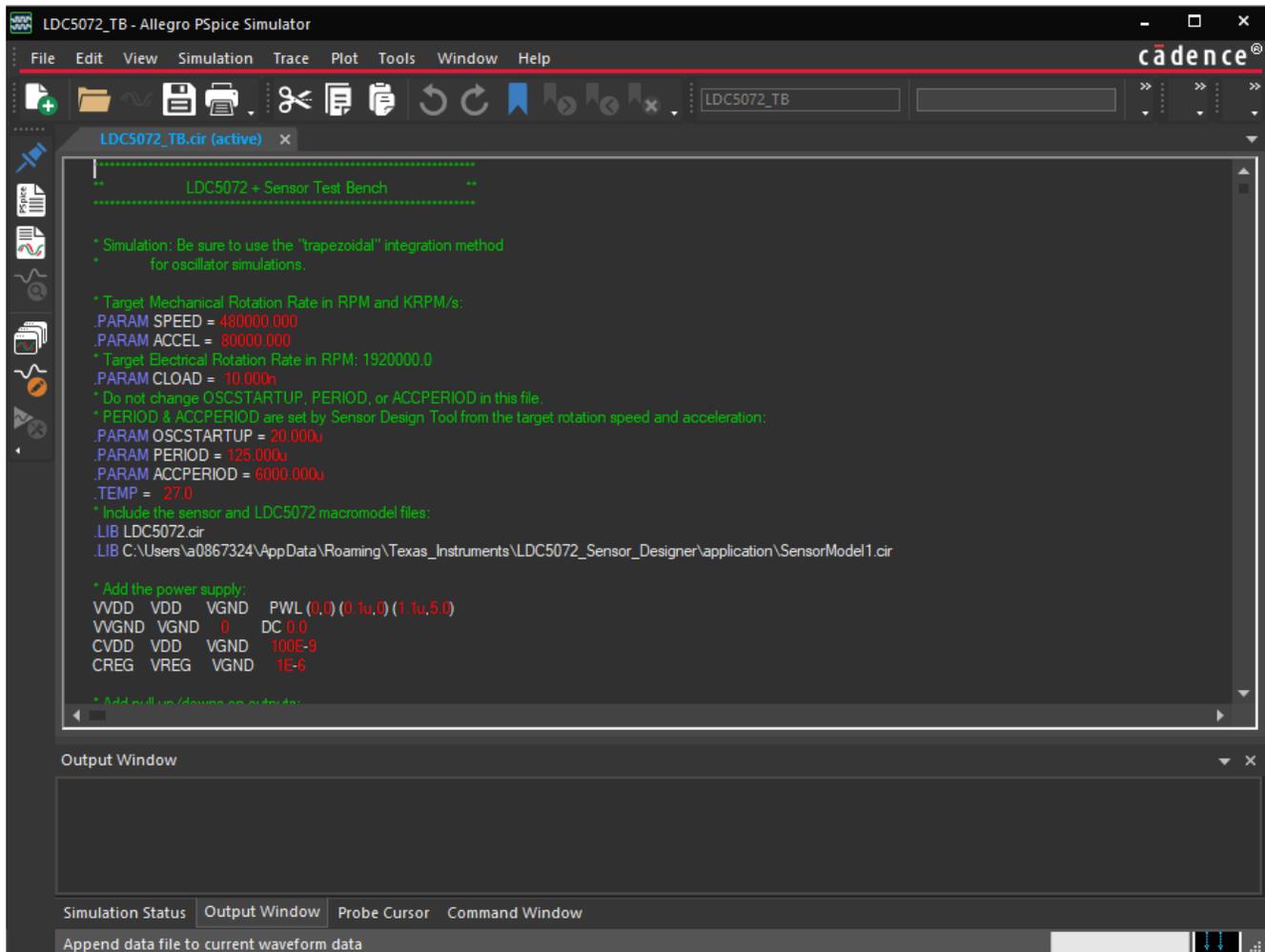


Figure 7-5. Sensor + LDC5072 Test Bench Open in Allegro Pspice Simulator

At the bottom of the test bench file (see [Figure 7-6](#)) are the simulation analysis statements (.TRAN and .FOUR) and the uncommented .OPTIONS statements that set the convergence behavior (trade-off between simulation speed and accuracy) of the simulator. The .PROBE statements specify data-gathering for post-simulation plotting. Note that the last .PROBE statement has voltages that represent the mechanical angle, the computed electrical angle, and the ideal electrical angle (e.g. V(MECHANICAL), V(COMPUTED) and V(IDEAL)). The next-to-last .PROBE statement has differential voltages that represents the differential outputs of the LDC5072 output pins - V([XLDC5072.OUT01P],[XLDC5072.OUT01N]) and V([XLDC5072.OUT11P],[XLDC5072.OUT11N]).

Note

For additional details on the meanings of the SPICE statements and declarations, please consult some of the many references about Spice simulation. Please note the last six comments of the file that describe the waveform-dependent requirements for credible results and also advise against modifying the simulation time.

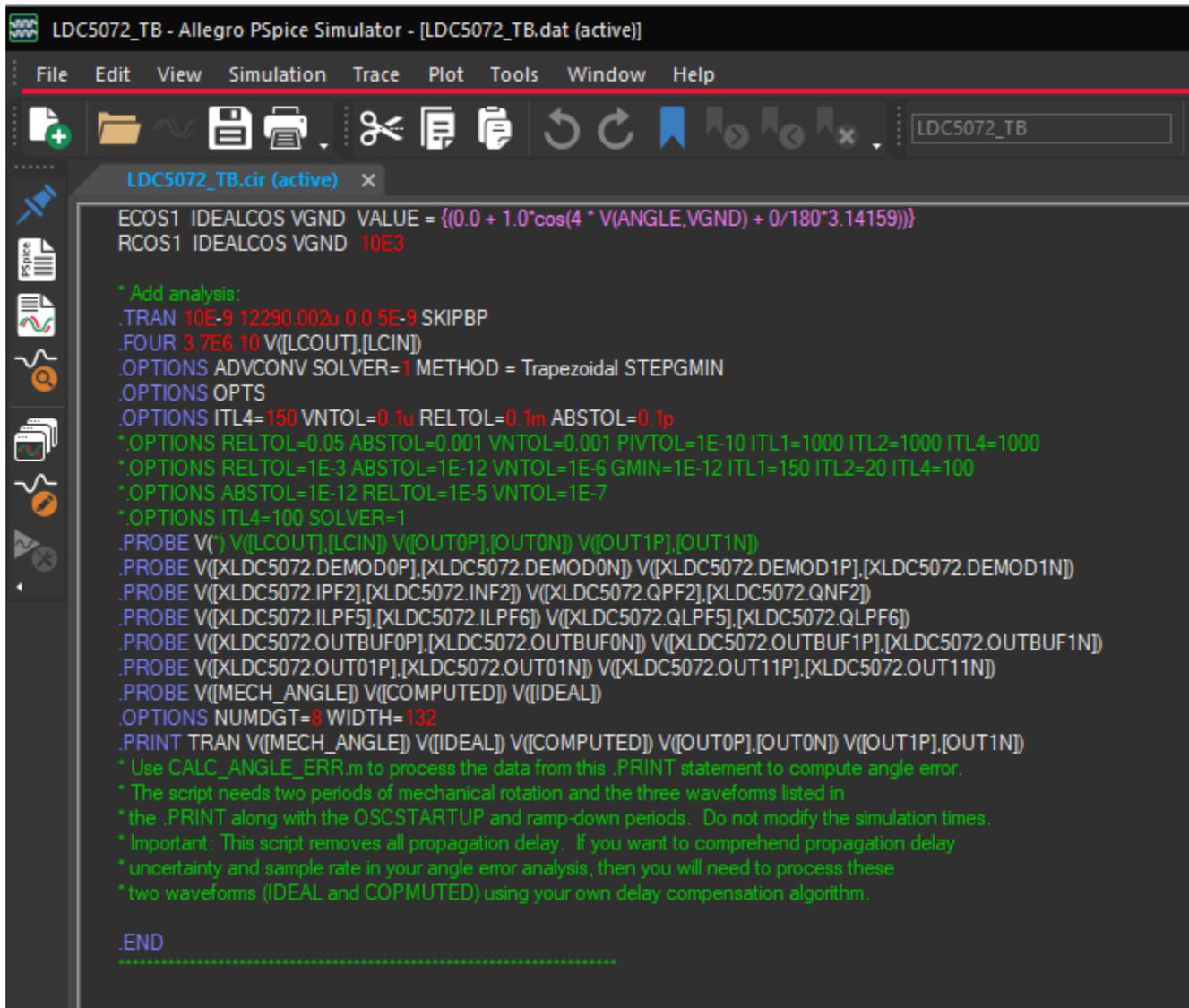


Figure 7-6. Test Bench Pspice Simulation Statements

The simulation is started by selecting *Simulation > Run LDC5072_TB* in the upper menu, or by clicking the right-pointing arrow in the upper banner as shown in the figures below. The simulation time depends on the design, the analysis parameters (.TRAN), and the .OPTIONS parameter values. If you are unfamiliar with Spice simulations, please be aware that some combinations of parameter values can result in long simulation times.

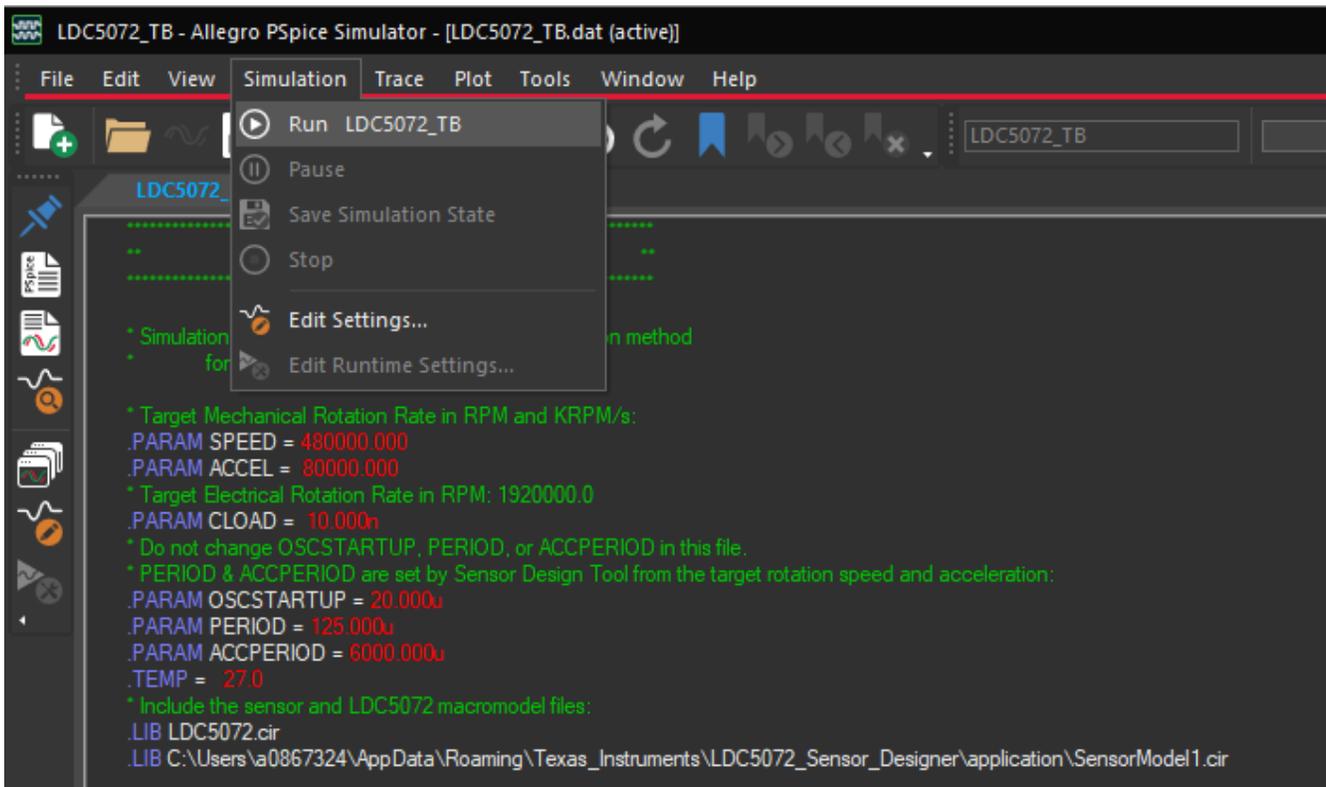


Figure 7-7. Starting the Pspice Test Bench Transient Simulation from the *Simulation* Menu

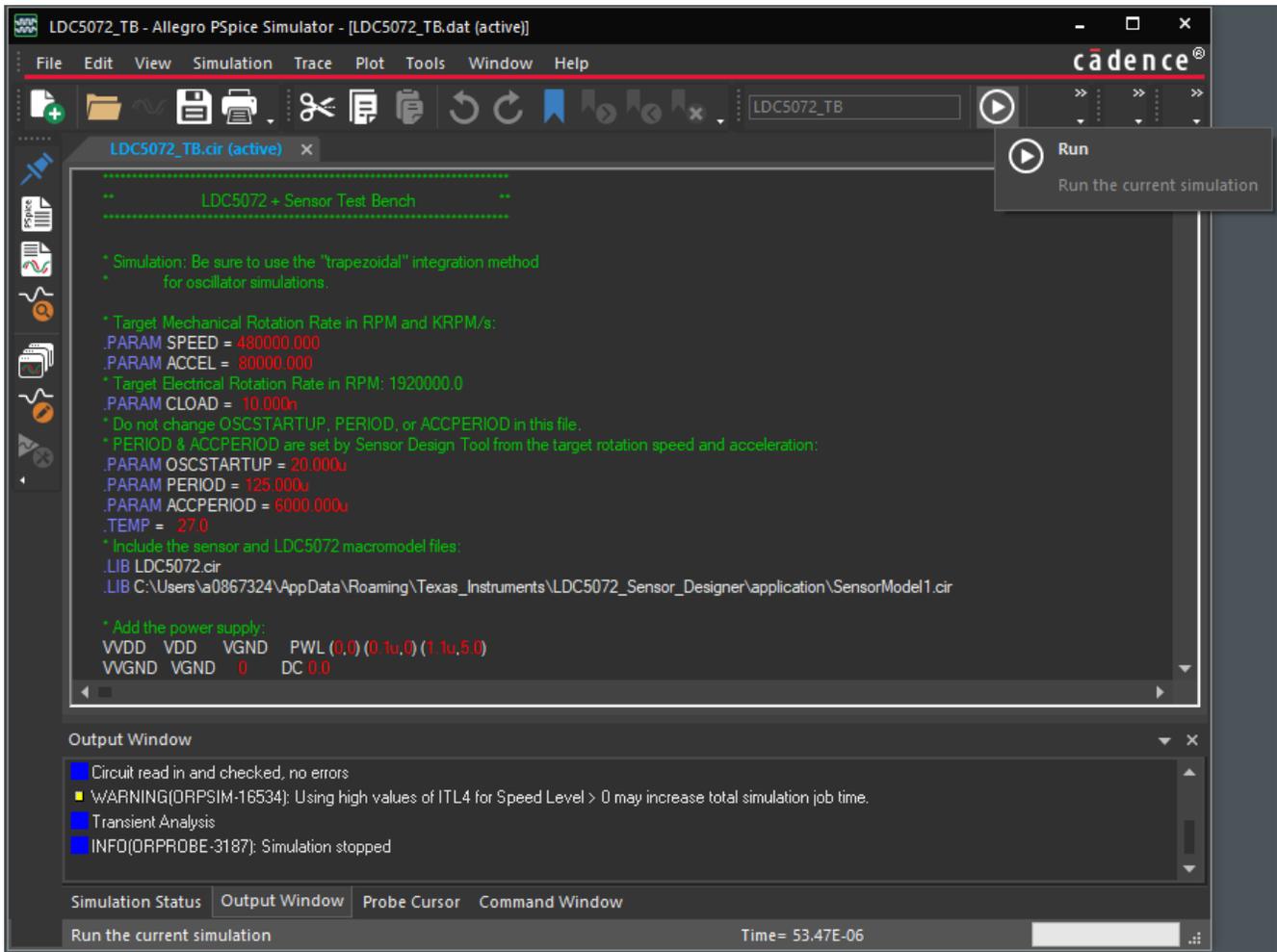


Figure 7-8. Starting the Pspice Test Bench Transient Simulation from the *Run* Button

While the simulation is running, you can see status messages in the *Output Window*, and the completion status as a percentage in the lower right-hand status bar.

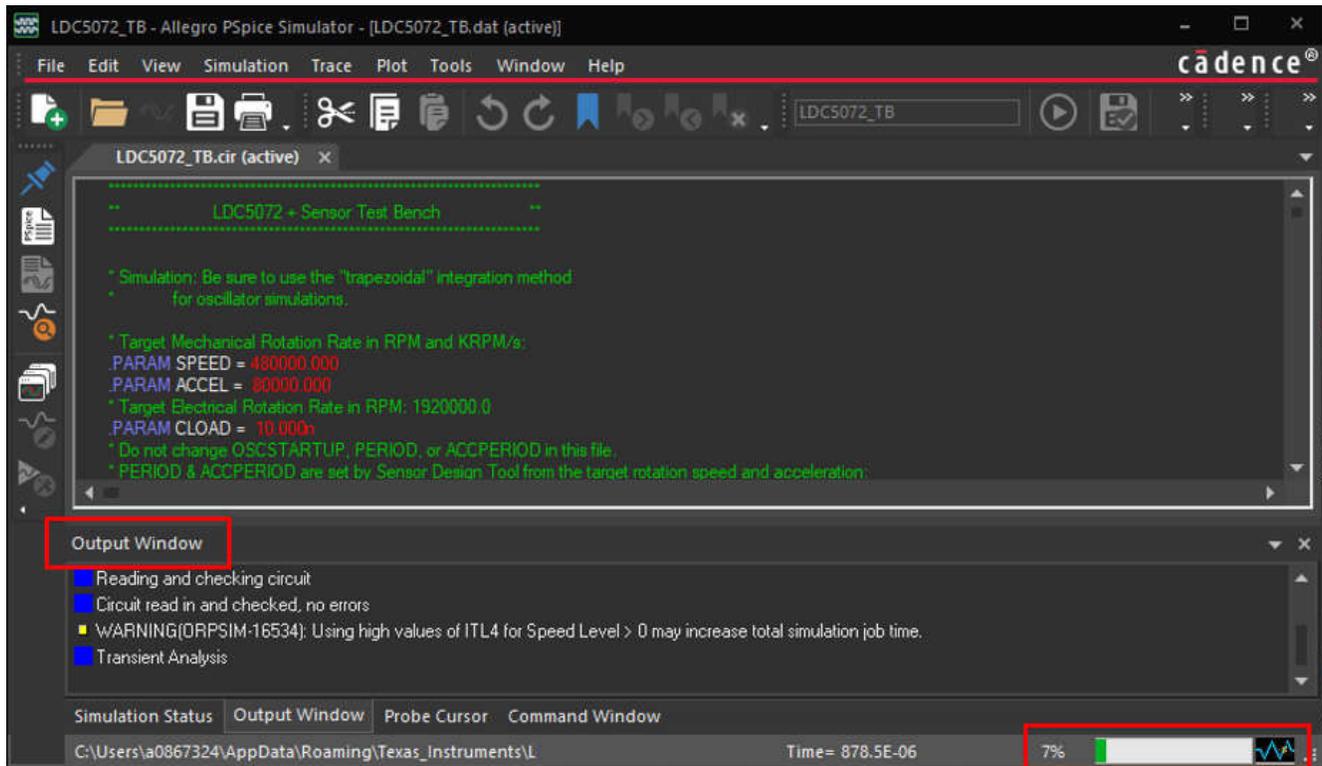


Figure 7-9. Status of a Running Pspice Simulation

For this example, the simulation is stopped after approximately 880 us, and the plot window appears as an additional tab in the simulator panel as shown in [Figure 7-10](#). The next few steps illustrate how to plot the LD5072 output pins' voltages, as well as angle estimates that model the target angular rotation. You can plot the results by selecting either *Trace/Add Trace*, or by clicking the *Add Trace* button in the icon ribbon as shown in [Figure 7-11](#).

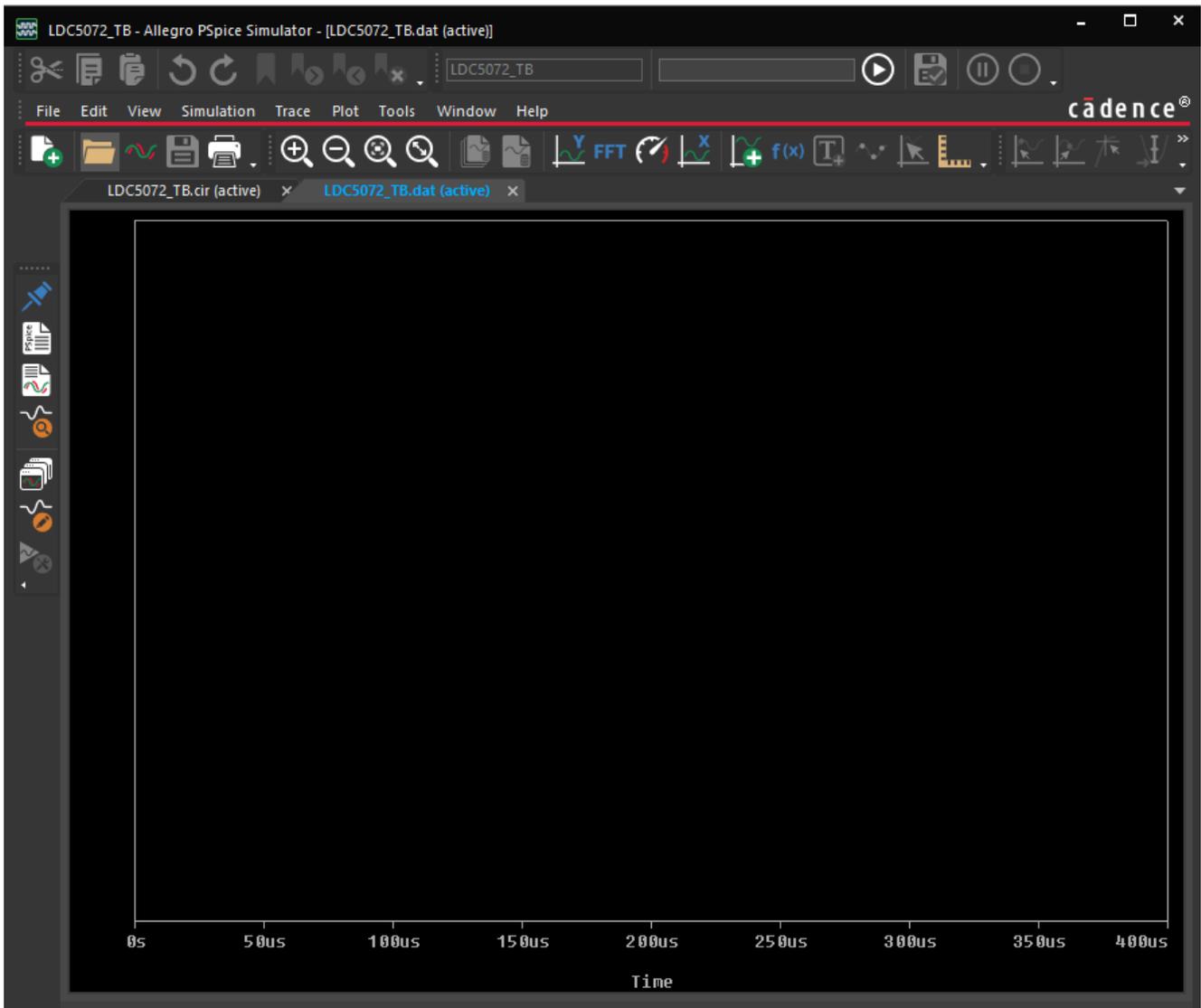


Figure 7-10. Initial Allegro Pspice Plot Window After Completion of Simulation

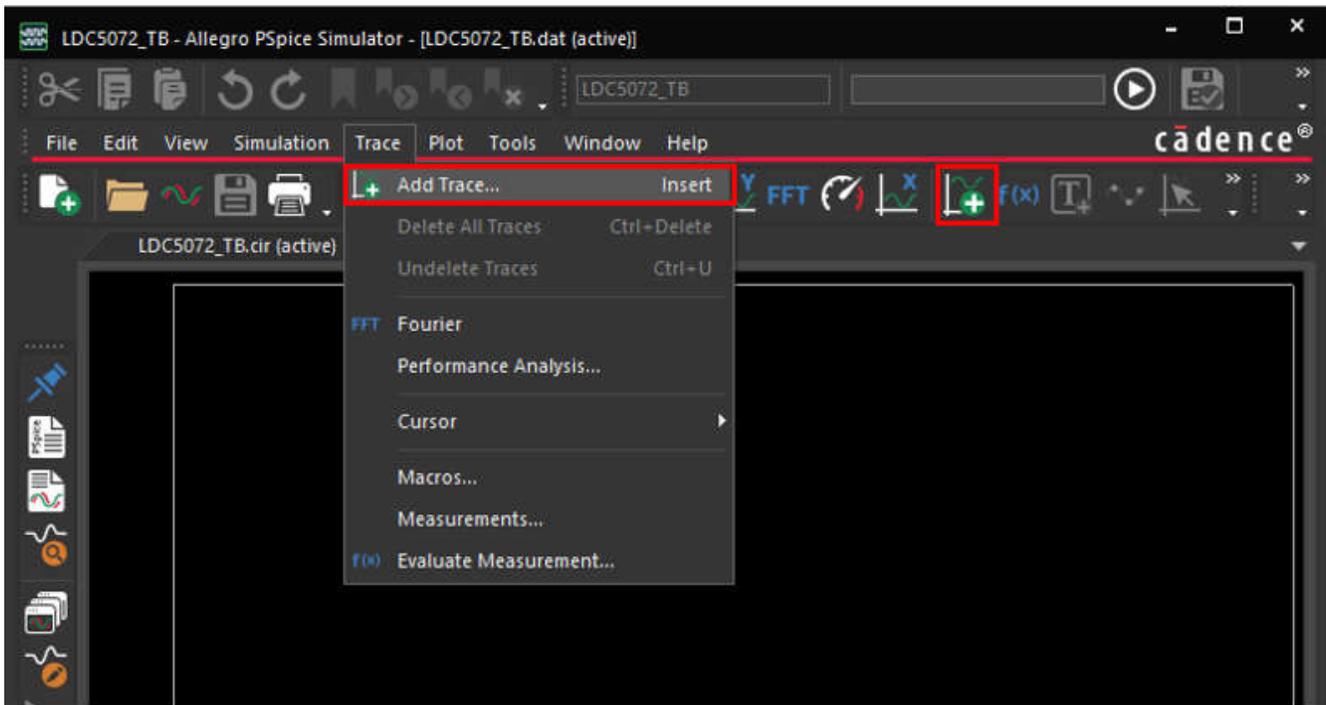


Figure 7-11. Adding a Trace to the Allegro Pspice Plot Window

The first step is to select the differential voltage of the LDC5072 channel 0 (COS) output pins and by first clicking *Add Trace* and selecting *V(XLDC5072.OUT01P,XLDC5072.OUT01N)* as shown in Figure 7-12.

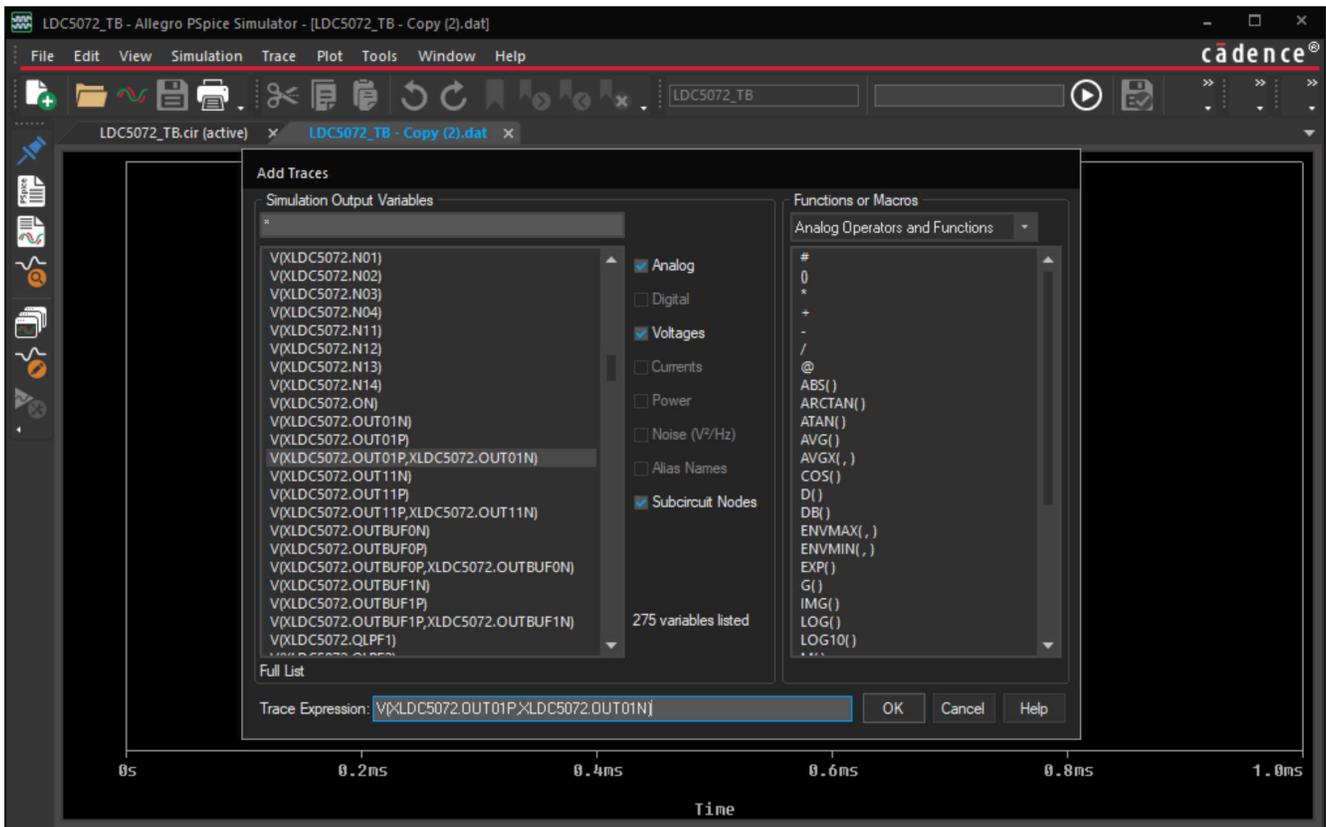


Figure 7-12. Adding the Trace for LDC5072 Channel 0 (COS) Differential Output Voltage

This shows a plot similar to Figure 7-13.

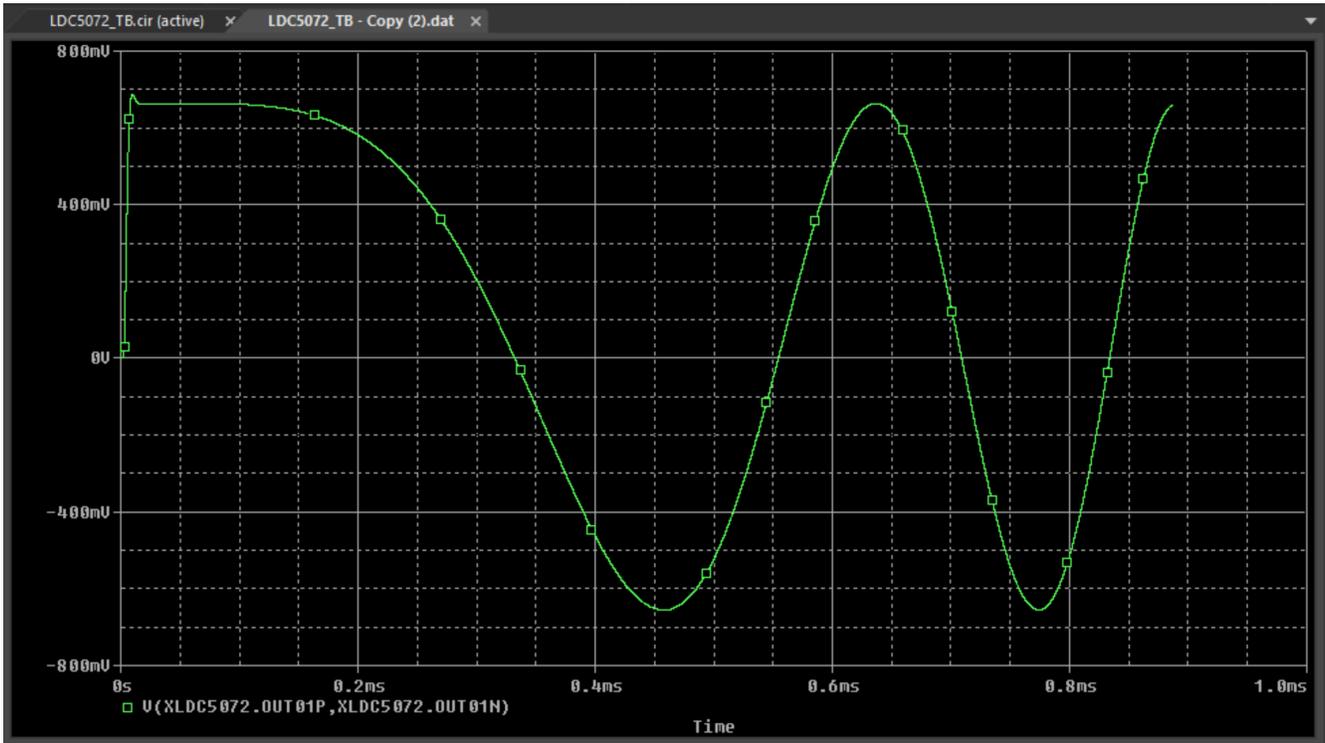


Figure 7-13. Plotted Curve for LDC5072 Channel 0 (COS) Differential Output Voltage

The second step is to add and select the differential voltage of the LDC5072 channel 1 (SIN) output pins by clicking *Add Trace* and selecting `V(XLDC5072.OUT11P,XLDC5072.OUT11N)`.

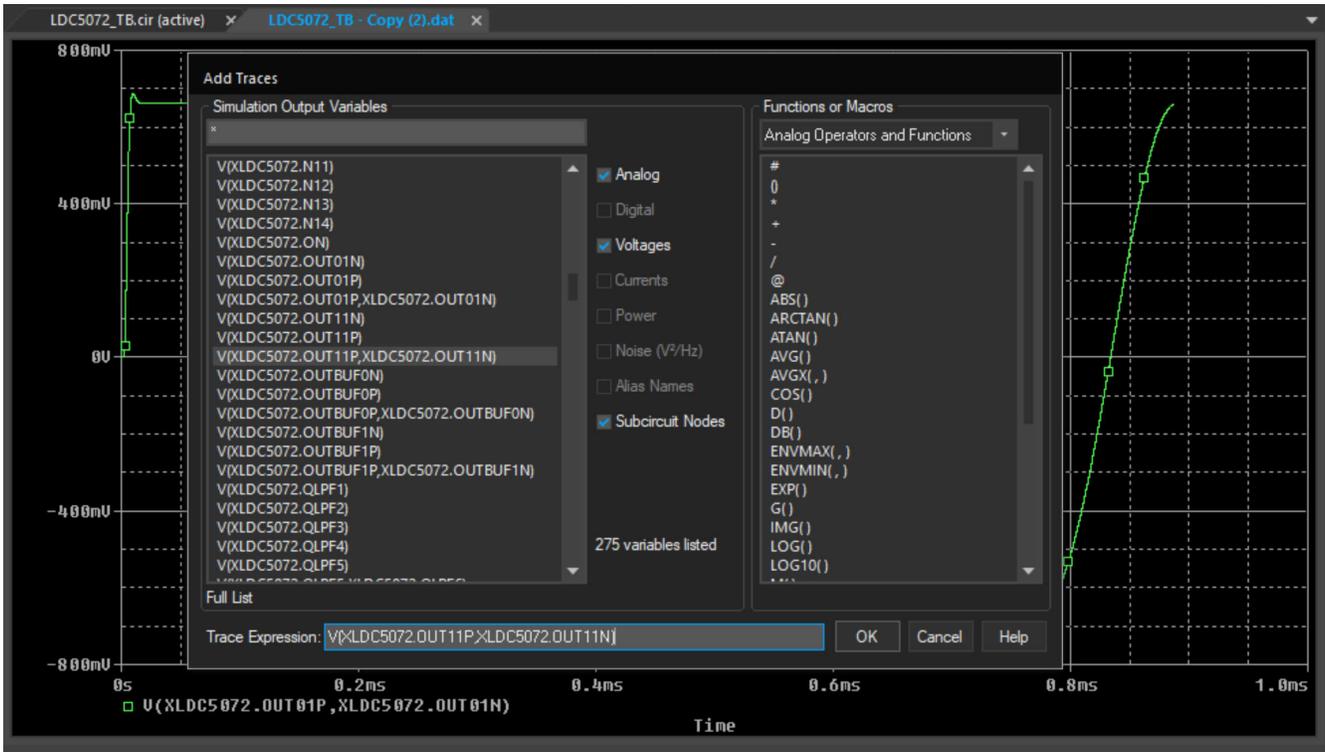


Figure 7-14. Adding the Trace for LDC5072 Channel 1 (SIN) Differential Output Voltage

This shows a plot similar to [Figure 7-15](#).

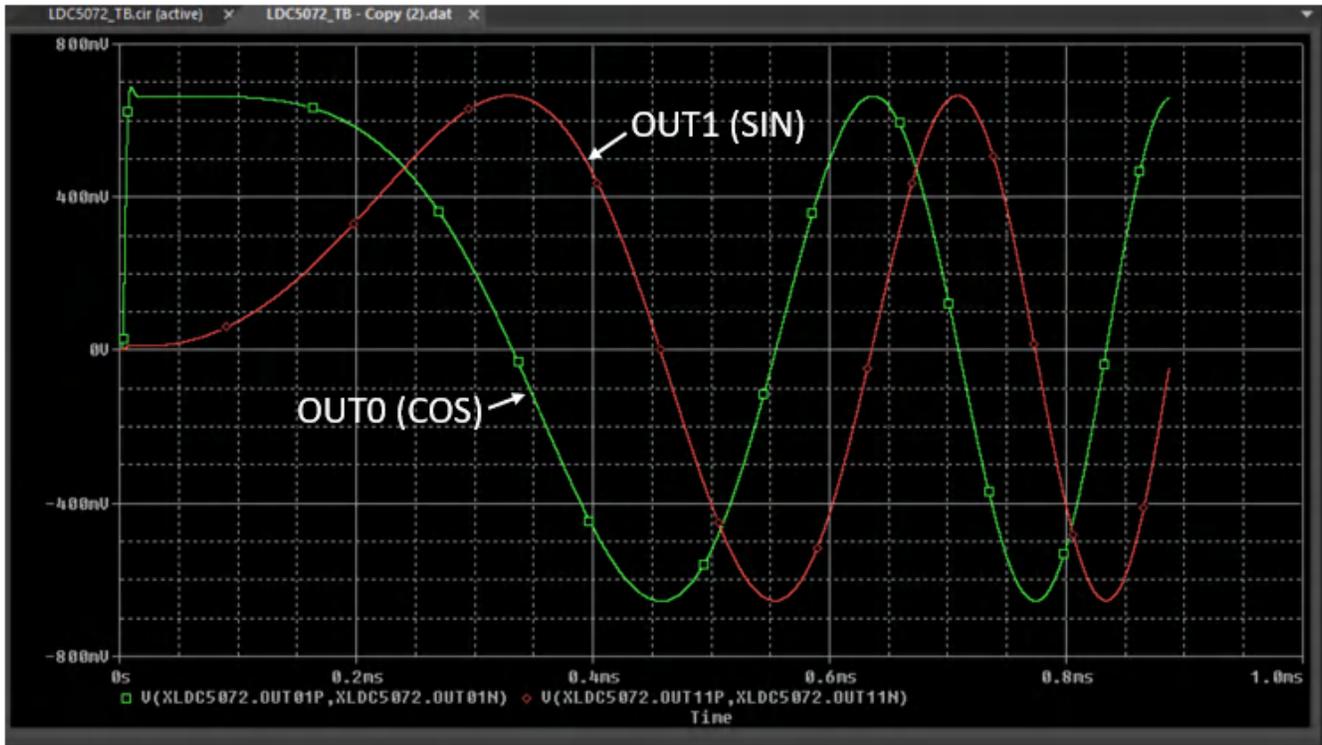


Figure 7-15. Plotted Curve for LDC5072 Channel 0 (COS) and Channel 1 (SIN) Differential Output Voltages

The next steps add an additional plot to the window to separate the LDC5072 pin voltage curves that were just plotted from the curves that show estimates of the target rotational angle. The first step is to add an additional plot window by selecting *Plot/Add Plot to Window* from the menu.

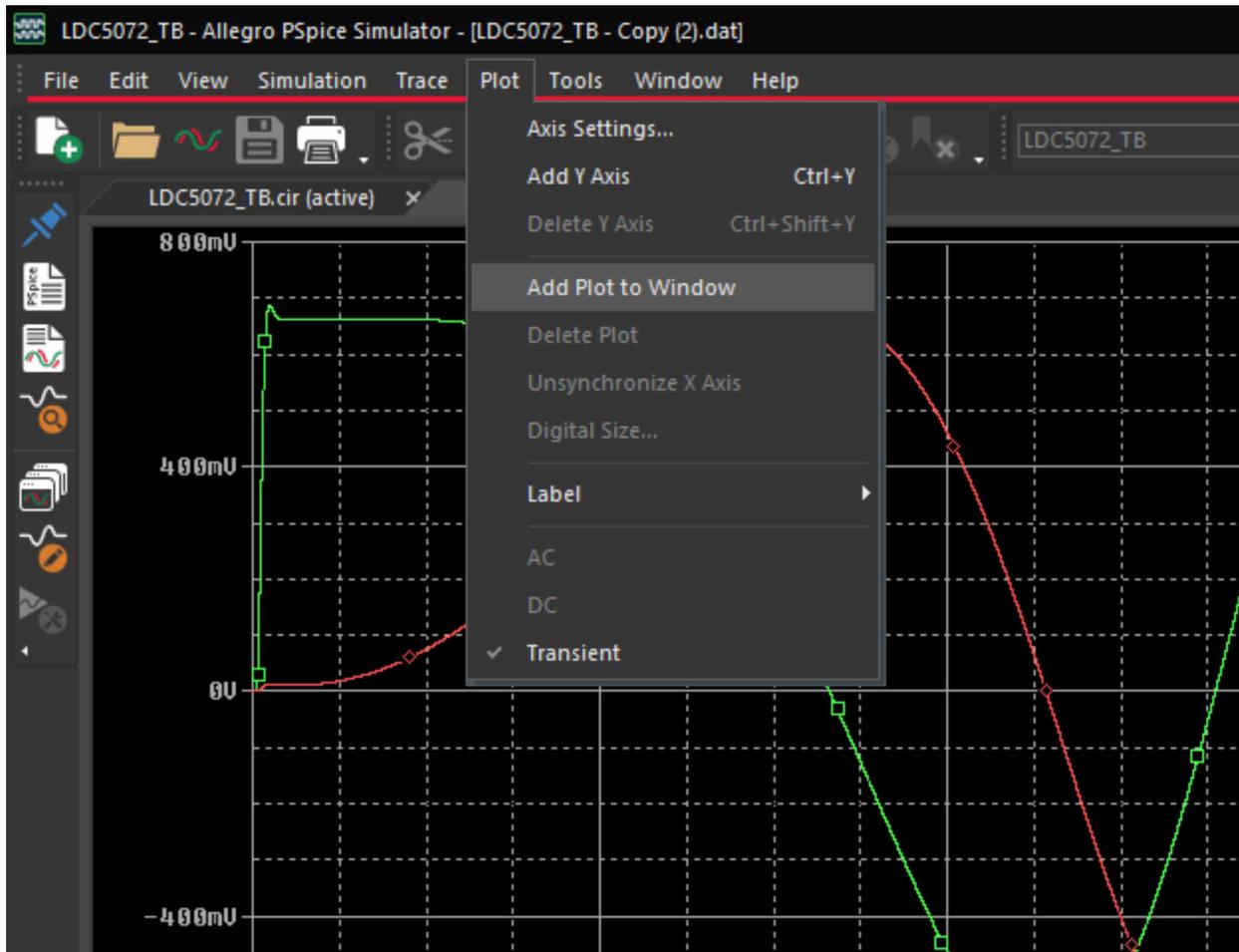


Figure 7-16. Adding a Plot Window

This places a new, empty plotting grid in the upper half of the plotting window.

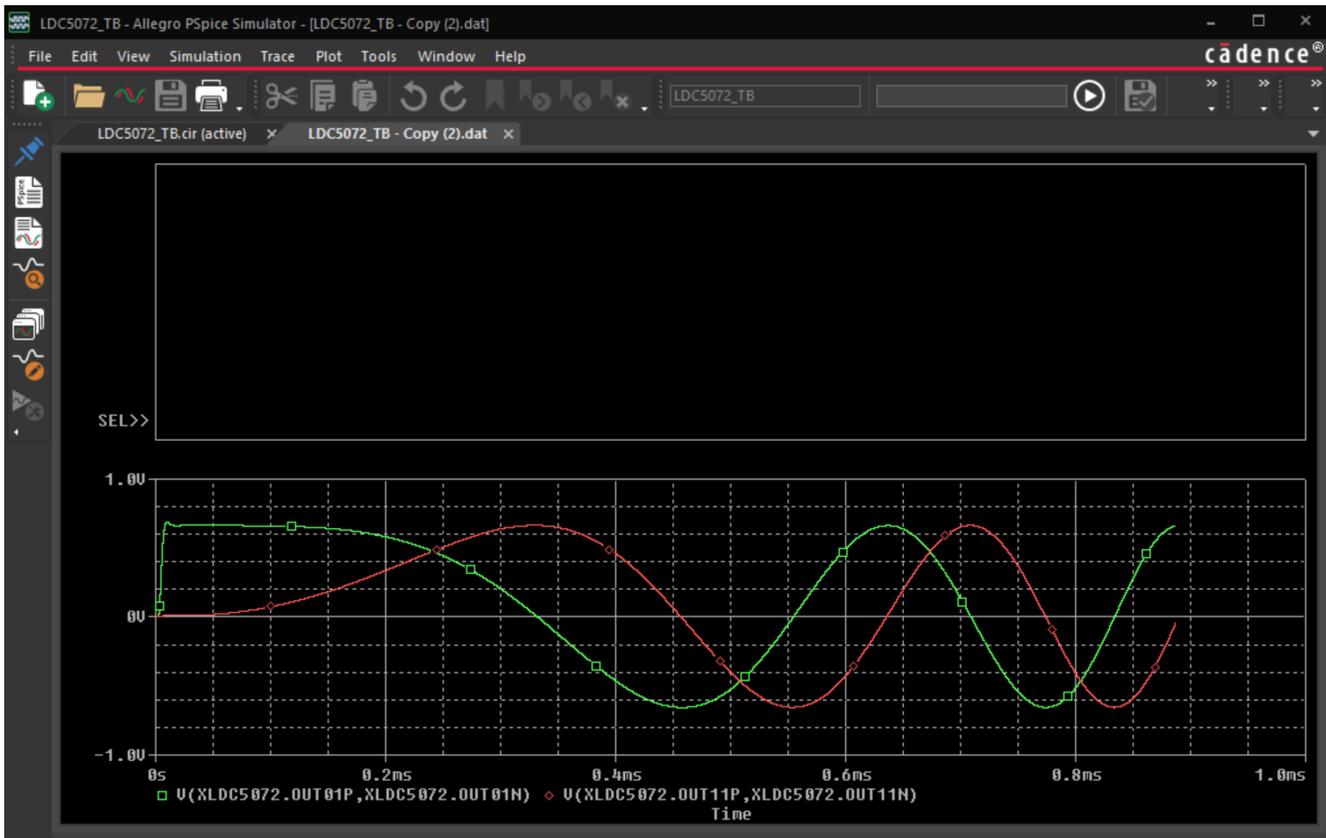


Figure 7-17. New Empty Plotting Window

Next add and plot the waveform that represents the physical angle of the target, which is represented by the the curve V(MECH_ANGLE) as shown in [Figure 7-18](#) and [Figure 7-19](#).

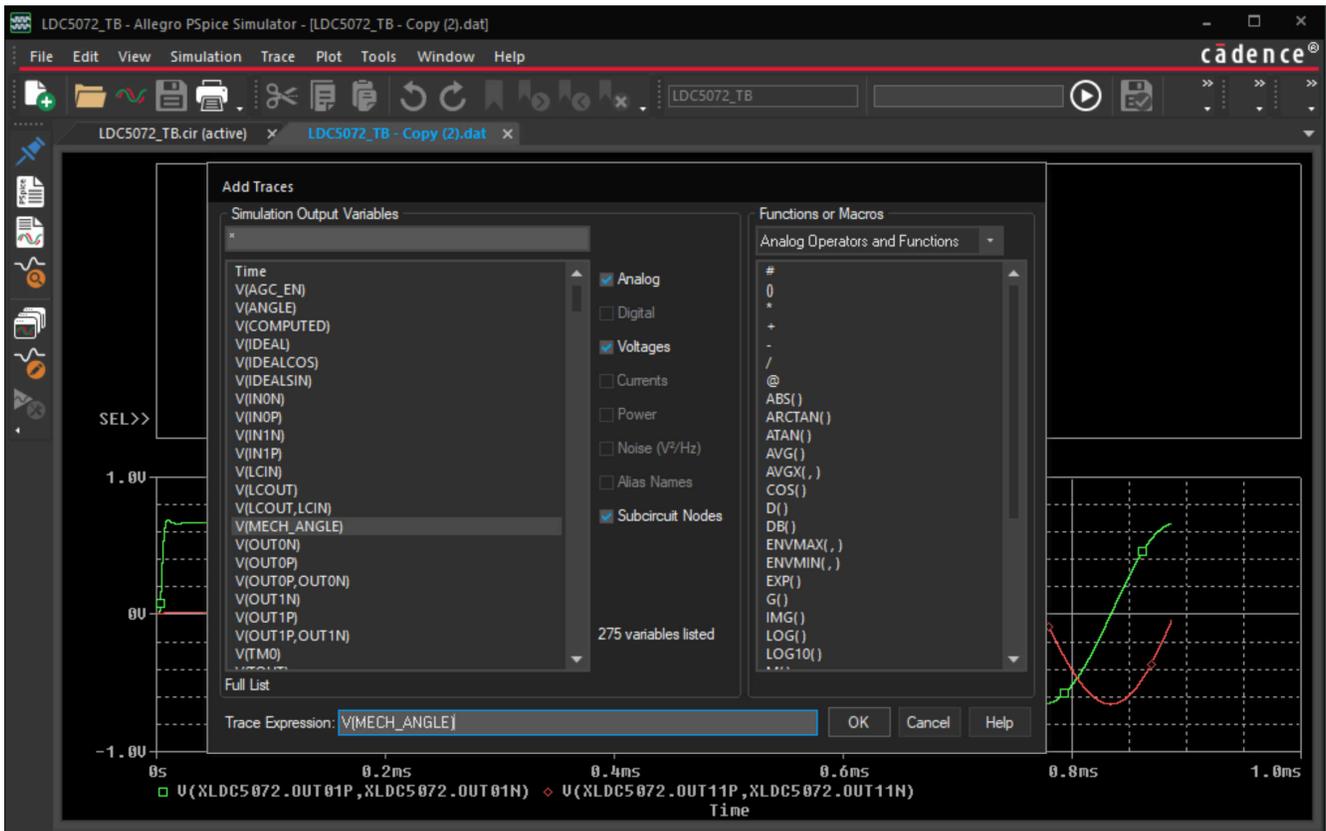


Figure 7-18. Add the Trace for the Target/Mechanical Angle

The plotted mechanical angle has an initial value of zero at the start of the simulation, but then moves between -180 degrees to 180 degrees as the target moves through full mechanical rotations. This is shown in [Figure 7-19](#) where the units are displayed in volts, and represent the target position in degrees where one volt represents one degree of target rotation.

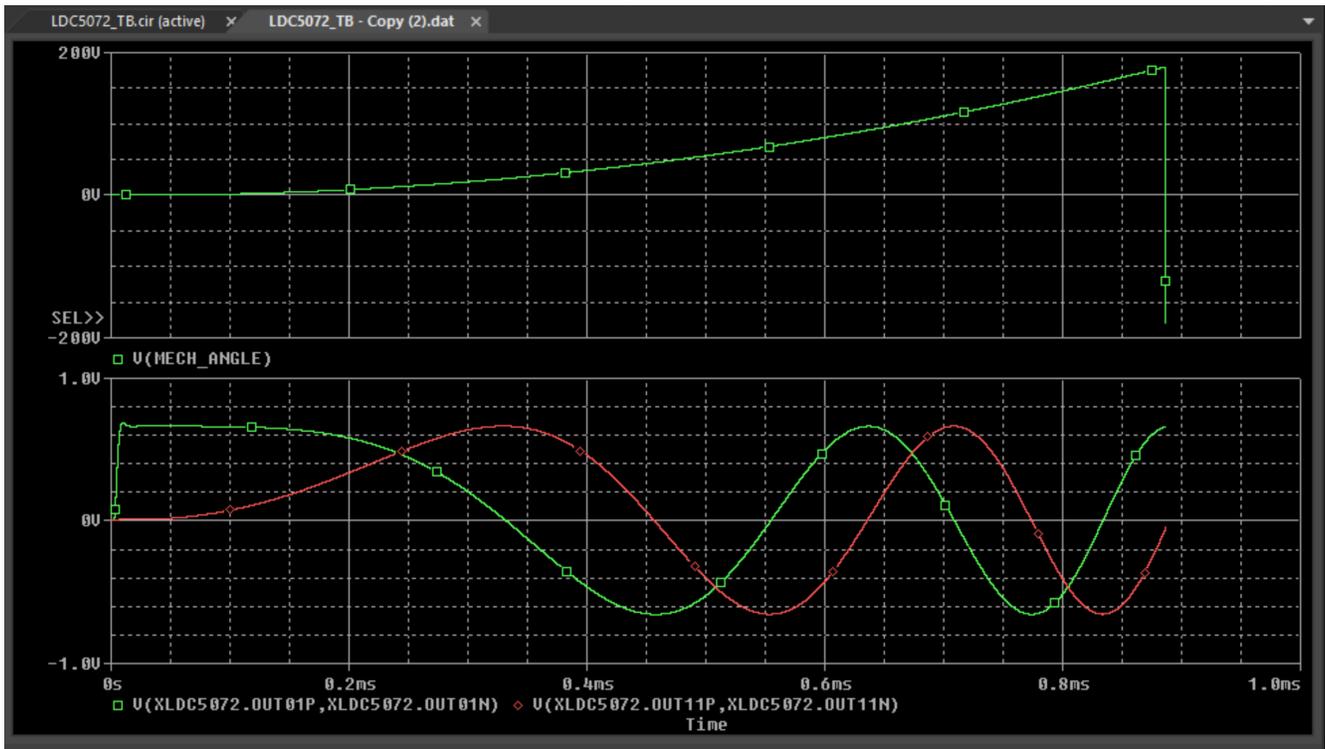


Figure 7-19. Addition of the Plotted Target/Mechanical Angle

Using similar steps, additional traces can be added which represent the computed electrical angle (V(COMPUTED)) and the angle (V(IDEAL)). The angle (V(COMPUTED)) is equal to the arctangent of the COS and SIN output channels and includes the effects of the time delays. The ideal electrical angle (V(IDEAL)) is also computed as an arctangent, but is based on the actual angle of the target and does not consider channel delays.

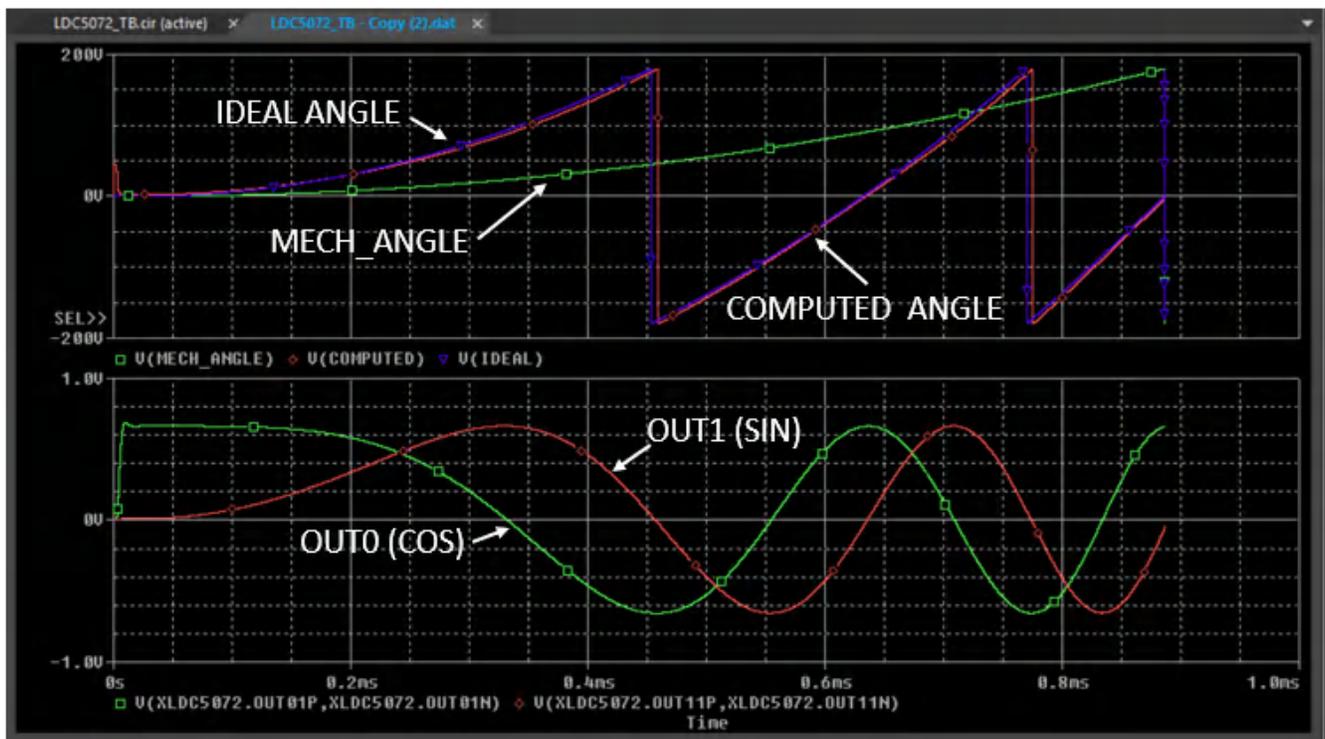


Figure 7-20. Add Waveforms of Computed and Ideal Electrical Angle

Fixed Gain Control

The gain of the LDC5072 Spice model can be changed by adjusting the values of **RAGC1** and **RAGC2** in the test bench file **LDC5072_TB.cir**. In the circuit file, the two resistors form a potentiometer with one end connected to VREG, and the other connected to GND and the mid-point (wiper) is connected to the AGC_EN pin of the device model (in file **LDC5072.cir**). The lines from the test bench netlist are shown just below.

* Bias for AGC_EN pin:

- **RAGC1 VREG AGC_EN 7.0E3**
- **RAGC2 AGC_EN VGND 93.0E3**

* Instantiate the sensor and LDC5072:

- XSENSOR LCIN LCOU IN0P IN0N IN1P IN1N ANGLE VGND SensorModel
- XLDC5072 LCIN LCOU IN0P IN0N IN1P IN1N OUT0P OUT0N OUT1P OUT1N VDD VREG VGND
AGC_EN TM0 TOUT LDC5072
- + PARAMS: LPFI = 0.000 LPFQ = 0.000 DLYI = 0.190 DLYQ = 0.190 COUTLOAD = {CLOAD} ILIM = 3.000
m
- + IOFFSET = 0.000 QOFFSET = 0.000 IIMBAL = 0.000 QIMBAL = 0.000 INL = 0.01000

The values of RAGC1 and RAGC2 can be calculated via equation (7) in the device data sheet.

$$Gain = 0.375 + 0.759 \times \left(10^{(1.903 \times (\%VREG - 4.55) / 90.9)} - 1 \right) \quad (1)$$

Where:

- Gain is the effective gain set by the gain block.
- %VREG is the voltage on the AGC_EN pin expressed as a percentage of the voltage on the VREG pin:

$$\%VREG = 100 \times RAGC2 / (RAGC1 + RAGC2) \quad (2)$$

8 Layout, Bill-of-Materials, and Schematic of the Test PC board

As mentioned previously, the tool creates Gerber files of the sensor, target, and an evaluation PC board layout with a *.pdf file representation of the design. The images below show examples of PC boards created from the tool's default settings.

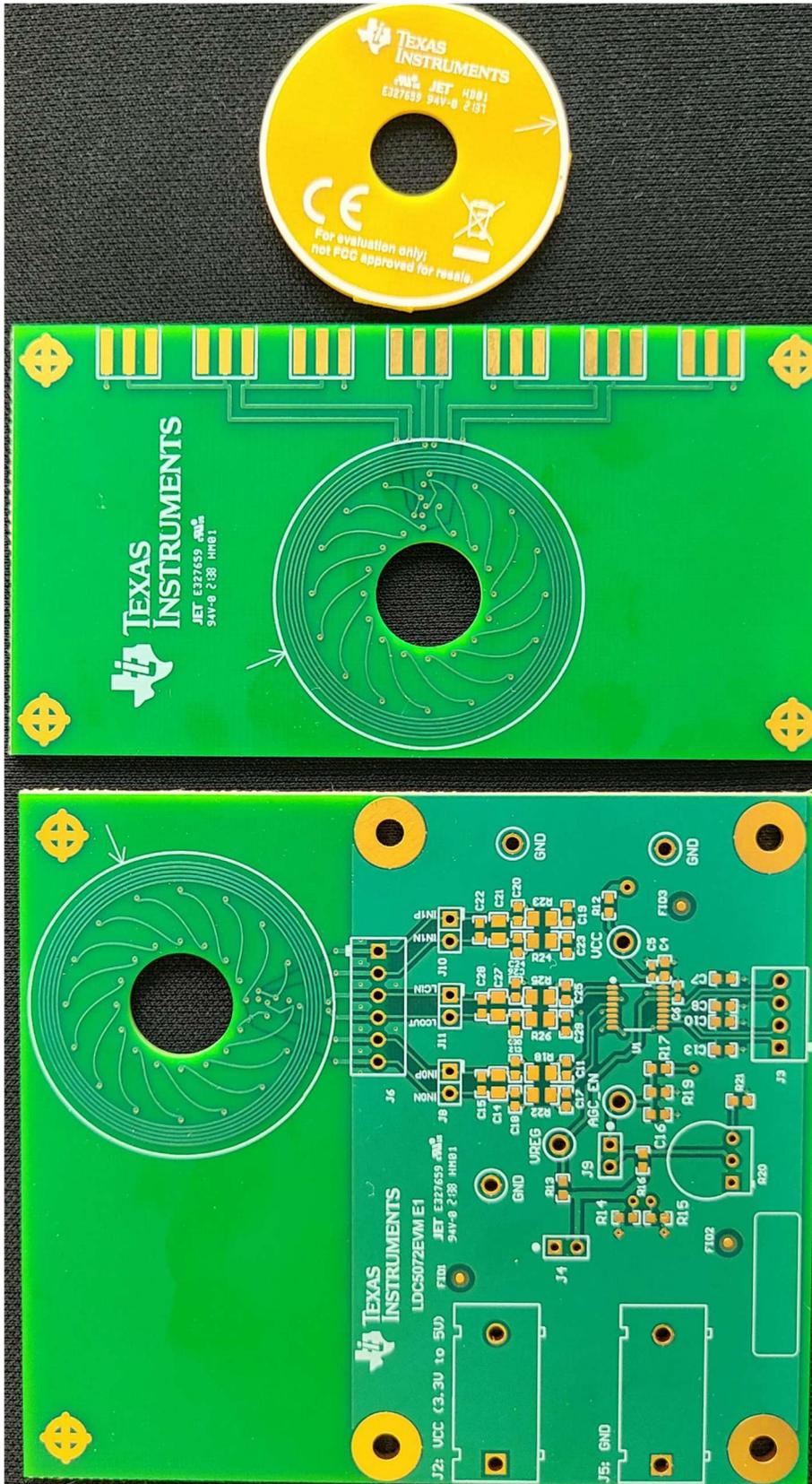


Figure 8-1. Top Side of Sensor, LDC5072 and Target PC boards created by the LDC5072 Sensor Design Tool



Figure 8-2. Underside of Sensor, LDC5072 and Target PC boards created by the LDC5072 Sensor Design Tool

The schematic of the evaluation PC board created by the design is shown in the figure just below, and the parts list is shown in [Table 8-1](#). Part placement on the PC board is shown in [Figure 8-4](#).

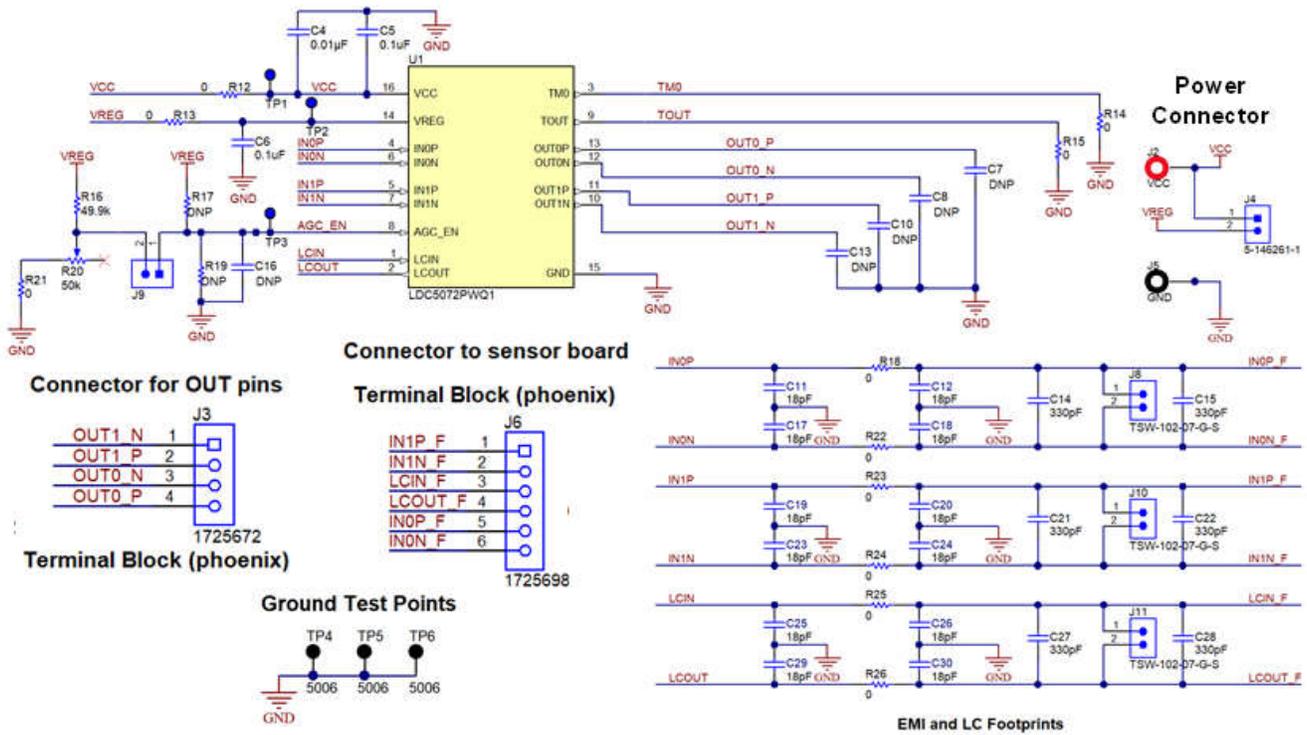


Figure 8-3. Evaluation PC Board Schematic Without Sensor

Table 8-1. LDC5072Q1EVM BOM

Designator	Quantity	Value	Description	Package	PartNumber	Manufacturer
C1	1	0.01uF	CAP, CERM, 0.01 uF, 25 V, +/- 10%, X7R, 0603	0603	GRM188R71E103KA01D	MuRata
C2	1	0.1uF	CAP, CERM, 0.1 uF, 25 V, +/- 10%, X7R, 0603	0603	C1608X7R1E104K080AA	TDK
C3	1	1uF	CAP, CERM, 1 uF, 35 V, +/- 10%, X7R, AEC-Q200 Grade 1, 0603	0603	CGA3E1X7R1V105K080AC	TDK
C4, C5, C6, C9	4	6800 pF	CAP, CERM, 6800 pF, 25 V, +/- 10%, X7R, 0603	0603	GRM188R71E682KA01D	MuRata
H9, H10, H11, H12	4		Bumpon, Hemisphere, 0.44 X 0.20, Clear	Transparent Bumpon	SJ-5303 (CLEAR)	3 M
J1	1		Standard Banana Jack, insulated, 10 A, red	571-0500	571-0500	DEM Manufacturing
J2	1		Standard Banana Jack, insulated, 10 A, black	571-0100	571-0100	DEM Manufacturing
J4	1		FFC/FPC Connector, 6 POS, 0.5mm, R/A, Tin, SMT	FFC/FPC Connector, 6 POS, 0.5mm, R/A, SMT	52745-0633	Molex
J8, J9	2		Header, 2.54mm, 2x1, Tin, TH	Header, 2.54mm, 2x1, TH	22284023	Molex
LBL1	1		Thermal Transfer Printable Labels, 0.650" W x 0.200" H - 10,000 per roll	PCB Label 0.650 x 0.200 inch	THT-14-423-10	Brady
R1, R2, R3, R4, R10	5	0	RES, 0, 5%, 0.1 W, AEC-Q200 Grade 0, 0603	0603	CRCW06030000Z0EA	Vishay-Dale
R5	1	49.9k	RES, 49.9 k, 1%, 0.1 W, AEC-Q200 Grade 0, 0603	0603	ERJ-3EKF4992V	Panasonic
R7, R11, R12, R13, R14, R15	6	0	RES, 0, 5%, 0.125 W, AEC-Q200 Grade 0, 0805	0805	CRCW08050000Z0EA	Vishay-Dale
R9	1	50k	Trimming Potentiometer, 50K, 0.5W, TH	9.53x8.89mm	3352T-1-503LF	Bourns
R16, R17, R18, R19	4	10.0k	RES, 10.0 k, 1%, 0.1 W, AEC-Q200 Grade 0, 0603	0603	CRCW060310K0FKEA	Vishay-Dale
SH-J1	1		Shunt, 2.54mm, Gold, Black	Shunt, 2.54mm, Black	60900213421	Wurth Elektronik
TP1, TP2, TP3	3		Test Point, Compact, Blue, TH	Blue Compact Testpoint	5122	Keystone
TP4, TP5, TP6	3		Test Point, Compact, Black, TH	Black Compact Testpoint	5006	Keystone
U1	1		Inductive Position Sensor with Sin/Cos Interface, PW0016A (TSSOP-16)	PW0016A	LDC5072PWQ1	Texas Instruments
C7, C8, C13, C14, C15, C16, C19, C20, C21, C22, C25, C26	0	18 pF	CAP, CERM, 18 pF, 100 V, +/- 5%, COG/NPO, 0603	0603	GRM1885C2A180JA01D	MuRata
C10, C17, C23	0	330 pF	CAP, CERM, 330 pF, 50 V, +/- 5%, COG/NPO, 0805	0805	08055A331JAT2A	AVX
C11, C18, C24	0	330 pF	CAP, CERM, 330 pF, 50 V, +/- 1%, COG/NPO, 0603	0603	C1608C0G1H331F080AA	TDK
C12	0	0.1uF	CAP, CERM, 0.1 uF, 25 V, +/- 10%, X7R, 0603	0603	C1608X7R1E104K080AA	TDK
FID1, FID2, FID3	0		Fiducial mark. There is nothing to buy or mount.	N/A	N/A	N/A

Table 8-1. LDC5072Q1EVM BOM (continued)

Designator	Quantity	Value	Description	Package	PartNumber	Manufacturer
J3	0		Terminal Block, 4x1, 2.54 mm, Green, TH	Terminal Block, 4x1, 2.54 mm, TH	1725672	Phoenix Contact
J5, J6, J7	0		Header, 100mil, 2x1, Gold, TH	2x1 Header	TSW-102-07-G-S	Samtec
R6	0	10k	RES, 10 k, 5%, 0.1 W, 0603	0603	RC0603JR-0710KL	Yageo
R8	0	1.0k	RES, 1.0 k, 5%, 0.1 W, AEC-Q200 Grade 0, 0603	0603	CRCW06031K00JNEA	Vishay-Dale

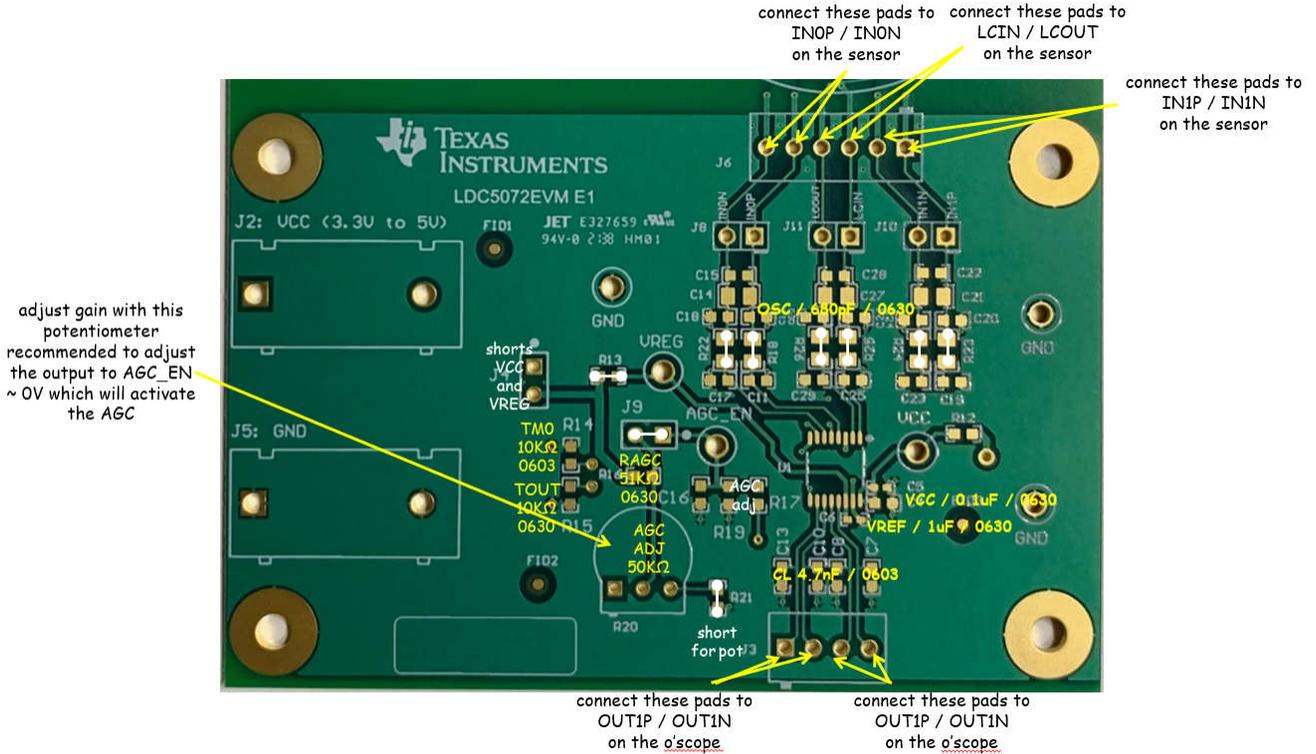
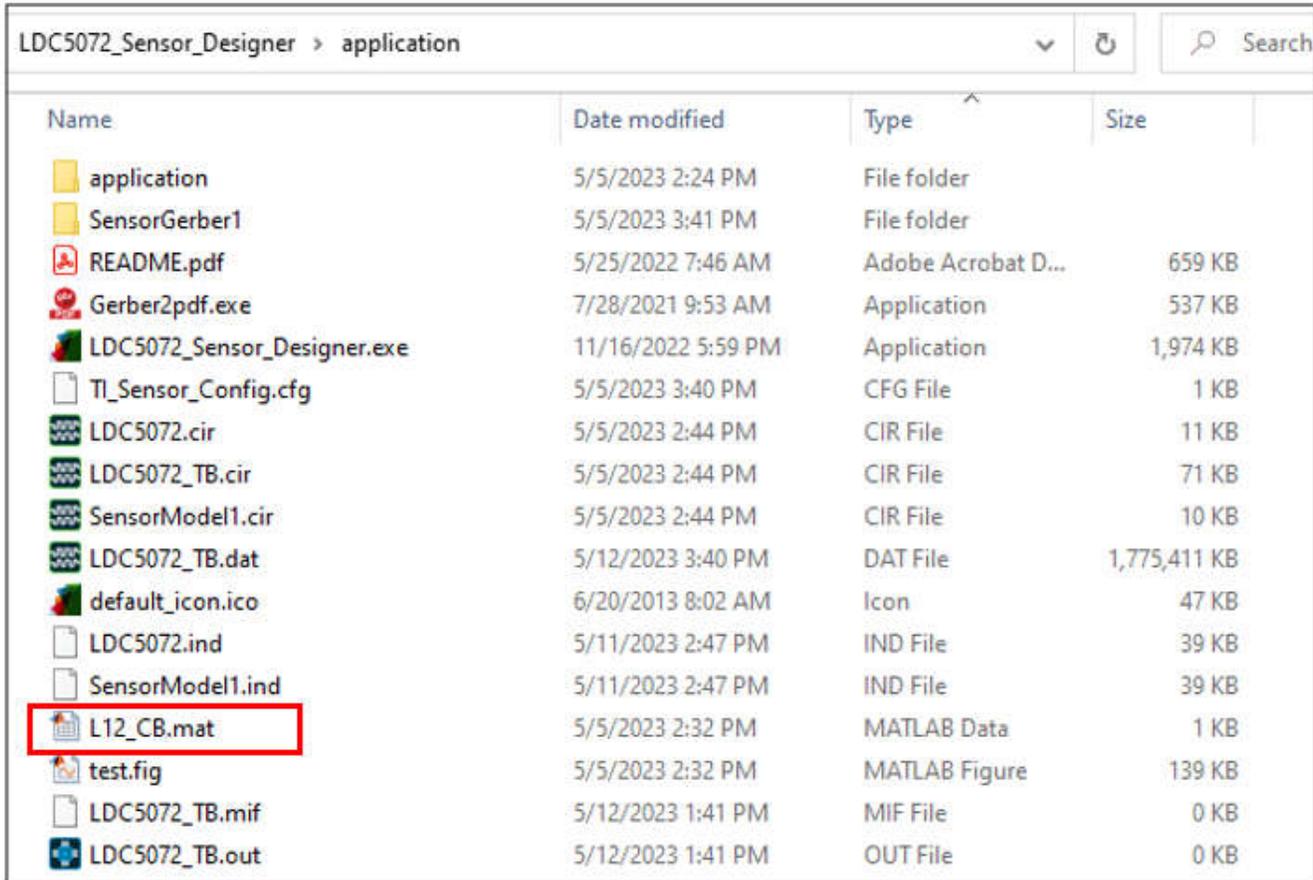


Figure 8-4. Detailed EVM Relative Part Placement for Soldered Components - Identical to Tool-Generated EVM

9 Troubleshooting

If the tool does not seem to be creating a design based on GUI input, then the tool state file located at folder application/L12_CB.mat is possibly corrupted. Try deleting this file and start again.



Name	Date modified	Type	Size
application	5/5/2023 2:24 PM	File folder	
SensorGerber1	5/5/2023 3:41 PM	File folder	
README.pdf	5/25/2022 7:46 AM	Adobe Acrobat D...	659 KB
Gerber2pdf.exe	7/28/2021 9:53 AM	Application	537 KB
LDC5072_Sensor_Designer.exe	11/16/2022 5:59 PM	Application	1,974 KB
TI_Sensor_Config.cfg	5/5/2023 3:40 PM	CFG File	1 KB
LDC5072.cir	5/5/2023 2:44 PM	CIR File	11 KB
LDC5072_TB.cir	5/5/2023 2:44 PM	CIR File	71 KB
SensorModel1.cir	5/5/2023 2:44 PM	CIR File	10 KB
LDC5072_TB.dat	5/12/2023 3:40 PM	DAT File	1,775,411 KB
default_icon.ico	6/20/2013 8:02 AM	Icon	47 KB
LDC5072.ind	5/11/2023 2:47 PM	IND File	39 KB
SensorModel1.ind	5/11/2023 2:47 PM	IND File	39 KB
L12_CB.mat	5/5/2023 2:32 PM	MATLAB Data	1 KB
test.fig	5/5/2023 2:32 PM	MATLAB Figure	139 KB
LDC5072_TB.mif	5/12/2023 1:41 PM	MIF File	0 KB
LDC5072_TB.out	5/12/2023 1:41 PM	OUT File	0 KB

Figure 9-1. LDC5072 Sensor Designer State File

The time to complete the design process varies with size and complexity of the sensor – typical times are two to eight minutes. If the design time is longer than 30 minutes, then the optimizer is probably stuck or the meshing cannot satisfy the error tolerances. Close the tool and start again with a different set of parameters. [Table 9-1](#) shows some examples of elapsed times to generate a design for different conditions. Note these times can vary depending on the user's hardware and operating system.

Table 9-1. Approximate Sensor Tool Elapsed Times for a Given Set of Design Parameters

Design	Diameter (mm)	E/M	Layers	Turns/Layer	Design Time (min:sec)
1	75	3	1	6	2:04
2	102	2	4	1	15:03
3	51	3	2	4	6:08
4	32	4	5	2	5:29
5	35.7	5	3	3	3:18
6	64.2	1	6	1	1:20
7	35	4	3	3	1:36

If the Gerber files show strange routing or shorts, then the trace width, spacing, or VIA dimensions are insufficient for the sensor size or electrical-to-mechanical ratio.

If you are having difficulty with the tool please feel free to post an inquiry on the [Texas Instruments Sensing E2E Forum](#). As part of the inquiry, please attach the following to your E2E thread:

1. The *.sns design file created by clicking the “Save Design” button on the front panel of the tool (see first figure below).
2. A text file containing a copy of the text in the tool’s “log” window (second figure below).
3. The “LDC5072_Sensor_Designer.log” data file in the application directory of the install folder.

10 Revision History

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

DATE	REVISION	NOTES
July 2023	*	Initial Release

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