

## README: TIDA-01228 Simulation models (TINA-TI and SIMPLIS)

December 2018

This ZIP file contains two models for TINA-TI and SIMPLIS simulator tools for the TIDA-01228 LC-sensing circuitry and a brief README document (this document).

The goal of providing these files is to enable easy modifications of the components or their parameters and simulate the performance of the circuitry.

<http://www.ti.com/tool/tina-ti> (TINA is a product of DesignSoft exclusively for Texas Instruments. This complimentary version is fully functional but does not support some other features available with the full version of TINA.)

<https://www.simetrix.co.uk/> Fast analog and switching power electronics simulators, providing professional level support, reliable convergence and a fully featured development environment.

The LC-sensing circuitry described in TIDA-01228 utilizes the FemtoFET CSD23285F5 device. Two SPICE models for this FemtoFET device can be downloaded here:

[https://webench.ti.com/webench5/spicemodels/?DCMP=hpa\\_hpa\\_elabs&HQS=spicerack#](https://webench.ti.com/webench5/spicemodels/?DCMP=hpa_hpa_elabs&HQS=spicerack#)

The PSpice Model for CSD23285F5 is unencrypted and can be imported as easy as per “drag and drop” into the SIMETRIX simulator tool.

The dedicated TINA-TI FemtoFET model file must be imported into TINA\_TI as described in the Application report: [Importing a SPICE NetList into TINA9-TI](#)

These models are used to simulate the waveforms for the LC-oscillation, generated by a negative pulse as the trigger to the FemtoFET device gate pin.

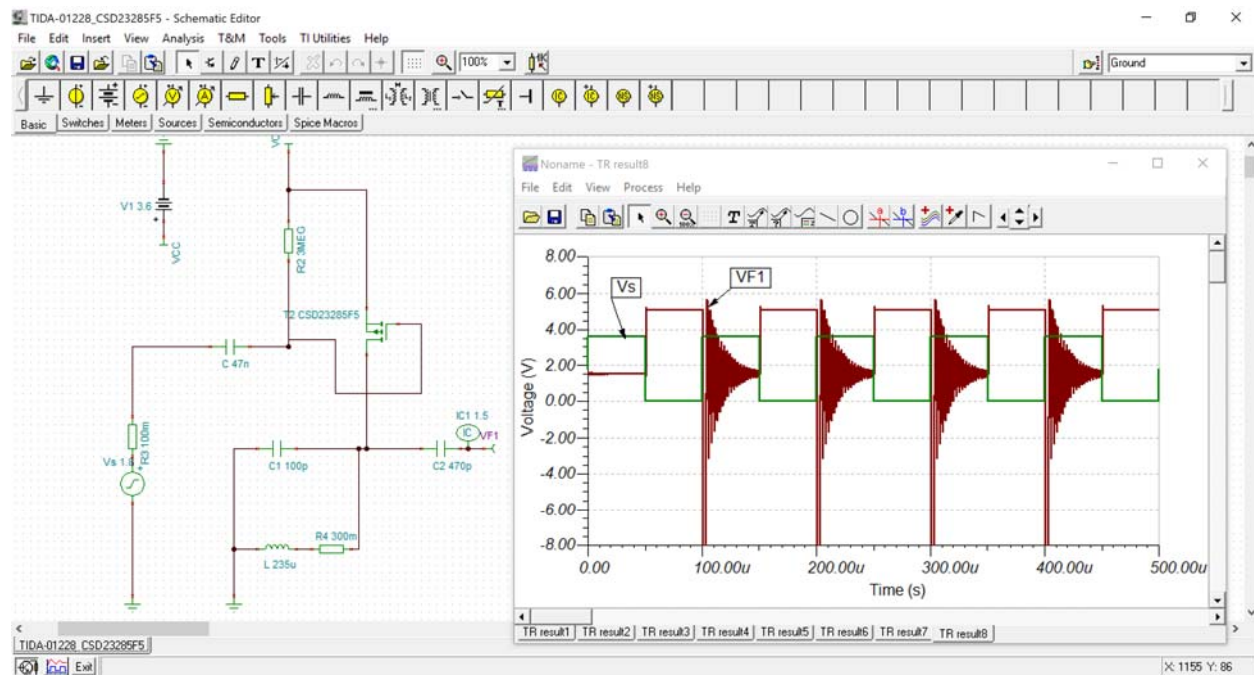


Figure 1: TINA-TI simulation of the external LC-sensing circuitry in TIDA-01228

Figure 1 shows the Vs (the Trigger pulse) and VF1 simulation waveform obtained with the simulation file and the conditions used for the calculations in this section.

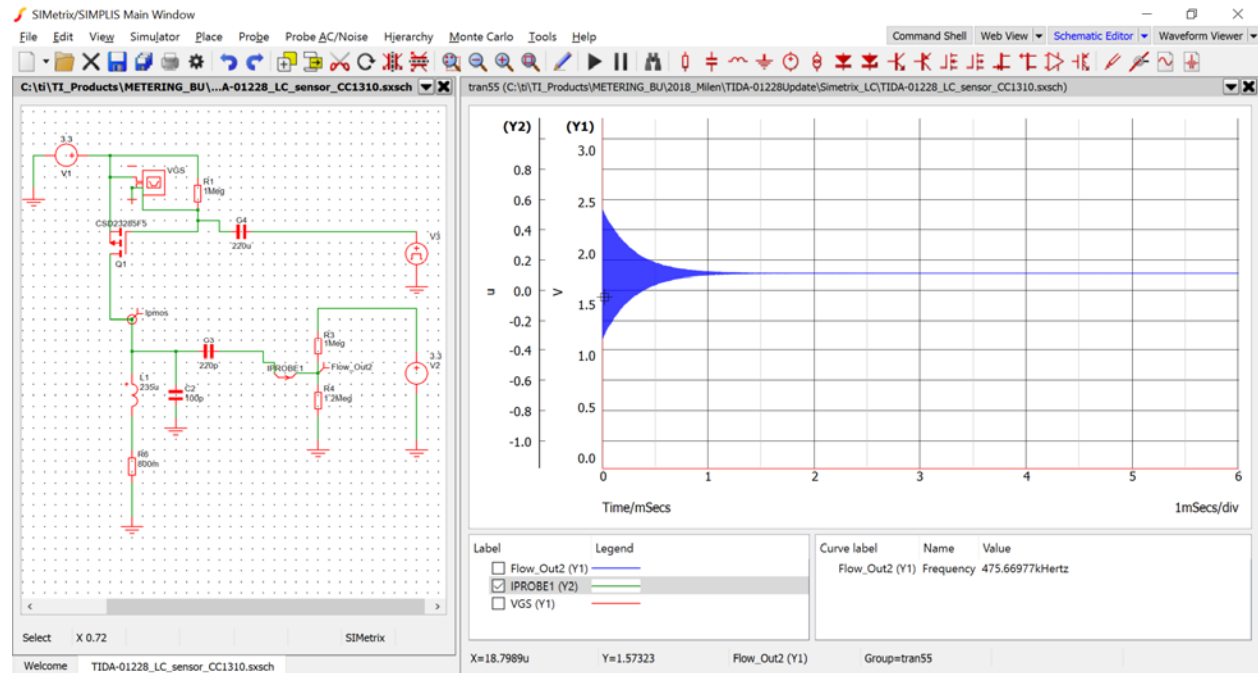


Figure 2: SIMETRIX PSpice simulation

Figure 2 shows the SIMETRIX model schematics to the left and the simulation results to the right.

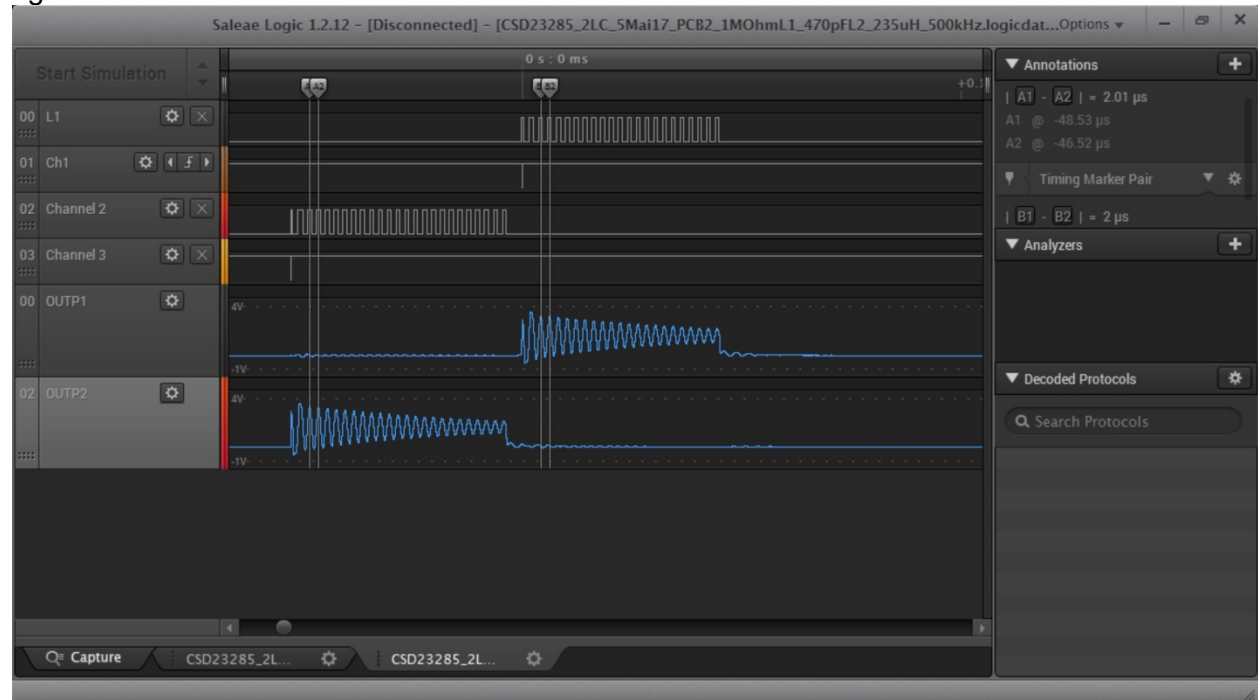


Figure 3: Undamped LC-oscillation of TIDA-01228 Rev2 with 497.5 kHz

**Simulation results:**

Figure 2 with the SIMETRIX simulated waveform shows 475.67 kHz which is close to the 497.5 kHz frequency (2.01  $\mu$ s period), measured with the Oscilloscope on TIDA-01228 Rev2 hardware (see Figure 3).

The simulation files use R4 (TINA-TI) and R6 (SIMETRIX) to represent the resistance of the Inductor used. In both files a short negative pulse (active LOW) on Vs or V3 (SIMETRIX) to the Gate of CSD23285F5 device is used to start the oscillation.

The accuracy of the simulated waveforms (frequency and voltage amplitude) can be further improved by using a SPICE model for the Inductor (If there is one available) and fine-tuning the trigger input waveform to the FemtoFET.

In addition further components can be added to the model for more accurate simulation of the CC1310 DIO pins FLOW\_OUTx and FLOW\_TRIGx behavior.