Educational Demos for Resonant Converter Phenomena

USER MANUAL

Matthew Roberts

robertsmatt97@gmail.com
https://github.com/robertsmatt97/educational-resonant-demo

Contents

List of Figures

1 User Manual

This user manual will act as a manual for instructors to set up and operate the board to be used in demonstrations. Before building the converter, one should be aware of the expected hardware needed to use the full extent of the board. Subsection 1 lists the essential hardware used when demonstrating the board:

The Demonstration requires this set of hardware:

- DC power supply
- 2 Channel Oscilloscope
- Variable or Electronic Load (Rated at 5W minimum)
- At least 1 Passive Voltage Probes
- At least 1 Differential Voltage Probes
- Windows Computer (laptop preferably) with USB-A ports
- MATLAB 2019b or higher
- Latest Version of Code Composer Studio
- Screen to display results on for clear viewing

1.1 Demonstration Setup

This section covers every step needed to get the converter in operating conditions. This includes how to set up the GERBER files to order the PCB of the converter, explain the user variables within each script and simulation file, and to get Code Composer Studio to correctly build and debug the submitted CCS Project on your computer.

1.1.1 Project Files

The submitted bill of materials (BOM) Excel spreadsheet is to be used when ordering the board's components. The links to purchase them on Mouser are provided within the spreadsheet under the "URL" column. The submitted zip file named "gerber-files.zip" can be sent directly to most PCB manufacturers as your PCB files. If a manufacturer is not satisfied with the formatting, you can follow the steps shown in Section 1.1.1 to change the export settings through KiCAD. It is unnecessary to follow these steps unless your manufacturer needs a specific export.

How to export new GERBER files from KiCAD:

1. Install KiCAD 5.0+ onto your computer.
   — You can find the download link through KiCAD's official website, kicad.org.
2. Start KiCAD and open the PCB file, **"Resonant.kicad_pcb"**, by double-clicking the file name.
3. Navigate to **File** → **Plot...**.
4. Specify the file path of your plot files by pressing the folder icon next to the right of the **Output Directory** text box.
5. Check or Uncheck options based on what is specified by your PCB manufacturer.
   — If your manufacturer has not specified what must be changed, do not continue further. Send the manufacturer the GERBER files already given in the submission files.
   — If there are some options you are not seeing that the manufacturer has specified to have, continue onto the next step regardless.
6. Click on **Plot**, and then click on **Generate Drill Files...**

7. Specify the file path of your drill files by pressing the folder icon next to the right of the **Output Folder** text box.

8. Check or Uncheck options based on what is specified by your PCB manufacturer.

9. Click on **Generate Report File** and specify its name.

   — Make sure to save it in the same directory as the rest of your exported files.

10. Click on **Generate Drill File**, then click on **Generate Map File**.

1.1.2 MATLAB and LTSpice

There are two MATLAB scripts submitted: "Resonant_Design.m" and "Resonant_Design_Serial.m". There are variables seen at the top of the scripts, most of which will be changed during the demonstration. How to properly set these variables will be discussed in Section 1.2.2. However, to set up the MATLAB script "Resonant_Design_Serial" to receive data from the ADC, the correct serial port should be inserted into the script. Section 1.1.2 provides instructions to find the correct serial port for your computer and put it into the MATLAB script.

How to configure the MATLAB script to find the right serial port:

1. Plug in the launchpad into your computer with the USB-A cable.

   — It is not required to have CCS open, nor the converter attached to the launchpad.

2. Open up the Device Manager on your computer by pressing the Windows Key, typing in "Device Manager", and pressing Enter.

3. Expand the **Ports** drop down list, and look for "XDS110 Class Application/User UART".

4. Note the COM port number.

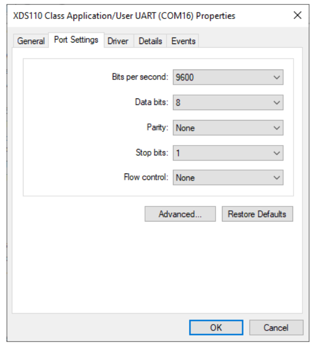

Figure 1.1: Port Settings

— Example: The COM port number for (COM3) is 3.

— If you do not see the port on the list, verify your launchpad's connection to the computer or that you have the latest drivers for the launchpad installed.

5. Right-Click the "XDS110 Class Application/User UART" port, and select **Properties**.
6. Navigate to the **Port Settings** tab and verify/change the settings to match those in Figure 1.1.
7. Close out of the **Properties** window.
8. Open up MATLAB and open the MATLAB script, **"Resonant_Design_Serial.m"**.
9. Uncomment line 29, %serialportlist, by deleting the "%" from the line.
10. Run the script with the MATLAB console on screen and wait.

    — You may encounter an error regarding serial transmission after some time. This is intended.

11. Scroll up on the MATLAB console till you see COM ports. Verify that the COM port number you previously noted is on the list.
12. Edit near the top of the script the variable "port" to match the COM port you noted in the Device Manager and MATLAB console.

Unlike the MATLAB scripts, it is recommended to setup LTSpice waveform data before your demonstration. Both MATLAB scripts import LTSpice data through a text file. Since gathering the data from LTSpice takes at least five minutes, having the text files generated beforehand will minimize downtime during demonstrations. There are three LTSpice files: a full-bridge configuration, a half-bridge configuration, and an FHA simulation. Section 1.1.2 demonstrates how to run the LTSpice simulation and then export its data into a text file.

How to set up LTSpice simulations for your demonstration:

1. Install LTSpice XVII onto your computer.
   — You can find the download link through the Analog Devices website, analog.com, under **Design Center**.
2. Open LTSpice and open the desired simulation file.
3. Edit the configurable parameters by right-clicking the black ".param" line between the blue "Configure Parameters here:" lines.
4. Click **Cancel** and type in the desired parameters in the second new window in the format initially present. Click OK when finished.
   — You can denote milli, micro, and nano units by adding an "m", "u", or "n" at the end of the value.
   — Example: "200m" is 200 milli, or 0.2.
5. Run the simulation by clicking on the "running man" icon and patiently wait.
   — This process may take a long time depending on your computer's specs. It may take around 20 minutes on lower-end computers.
   — Simulation progress can be tracked on the bottom left corner of the window.
6. After completion, press *Ctrl+l* to open the Error Log.

7. Right-click the open white space in the Error Log and click **Plot .step'ed .meas data**. A graph of gain versus switching frequency will appear.

8. Right-click the open black space in the Waveform viewer and go to **File** → **Export data as Text**.

9. Click on **Browse** and navigate to the "Sim Text Files" folder under the MATLAB folder from the submitted files.

10. Change the file name, click **Save**, and then press **OK**.

1.1.3 Code Composer Studio

Code Composer Studio (CCS) requires many steps to build imported projects properly. It is essential to use the latest version of the selected compiler and the software, or else the project may not build. Section 1.1.3 is an extensive set of instructions to set up the CCS project for running the microcontroller script onto the launchpad.

How to set up Code Composer Studio for demonstration use:

1. Install the latest version of Code Composer Studio, version 3.03.00 of C2000Ware, and version 20.8.0.STS of the C2000 Compiler (version C2000-CGT-21 of C2000 Code Generation Tools).

   — All the listed software can be downloaded from Texas Instruments' website, ti.com.

2. Open CCS and define the workspace in the "CCS" folder of the submitted files.

3. Click the **Copy Settings** drop-down and check all three boxes. Afterwards, click **Launch**.

4. Navigate to **File** → **Import...** and click on it.

5. Click the **Code Composer Studio** drop down, click on **CCS Projects**, and finally click **Next**.

6. Click **Browse**, and choose the directory of the submitted CCS Project **Resonant_Demo** under "CCS". Afterwards, click **Finish**.
   - It is best to click the folder "Resonant Demo", and click "Select Folder".
7. Right-click the Project "Resonant_Demo", and go to **Properties**.
8. Verify that you have no warning signs in the CCS General tab. Click **Apply and Close**.
   - If you see errors, you should make sure you have chosen the right compiler version from the drop-down list. If the version does not show up, you do not have the compiler installed properly. Reinstall the compiler and restart your computer.
   - If you encounter linker errors, be sure that your workspace directory does not contain spaces or non-ASCII characters.
9. Right-click the project again and click **Rebuild Project**, and wait.
   - Alternatively, you can press the "bug" icon, located towards the top right of the window, to debug the script. Pressing the debug icon will always rebuild the project.
10. If you see "Build Finished" in the console, typically located at the bottom of the window, you have successfully set up your CCS Project.
11. Plug in your launchpad into your computer using the USB-A cable.
12. Click the debug icon and wait until the "play" icon, towards the top of the window, turns from a clear gray to a green and yellow color.
13. You can start the script by clicking the play icon.

1.1.4 Assembly

After the converter has had its components soldered onto it, it needs to attach to the launchpad and connect itself to a power supply and load. Section 1.1.4 briefly describes setting up the physical converter to ready for operation.

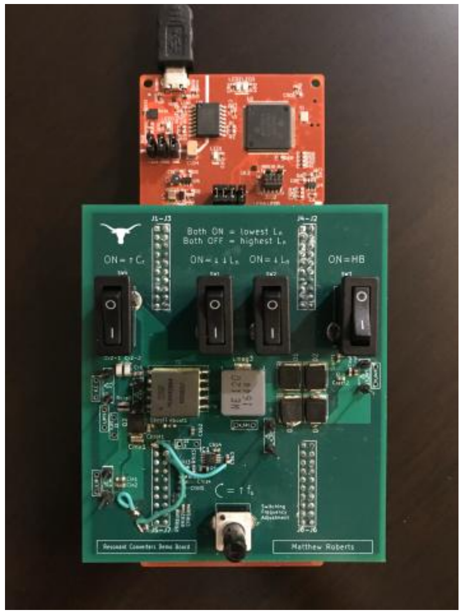

Figure 1.2: Converter on top of Launchpad to create the Board

How to set up the converter for operation:

1. Take your completed converter and attach it onto the launchpad from above, matching the orientation seen in Figure 1.2.

   — The potentiometer should be on the opposite side of the USB port of the launchpad.

2. Slowly insert the pin sockets of the converter onto the microcontroller's pin headers, making sure all pins are correctly inserted.

3. Plug in the USB-A cable into the microcontroller, and then into the computer

4. Clip onto the converter's input and output voltage pins with the DC Power Supply and the Electronic/Variable Load.

   — Your bench should look similar to what is seen in Figure 1.3.

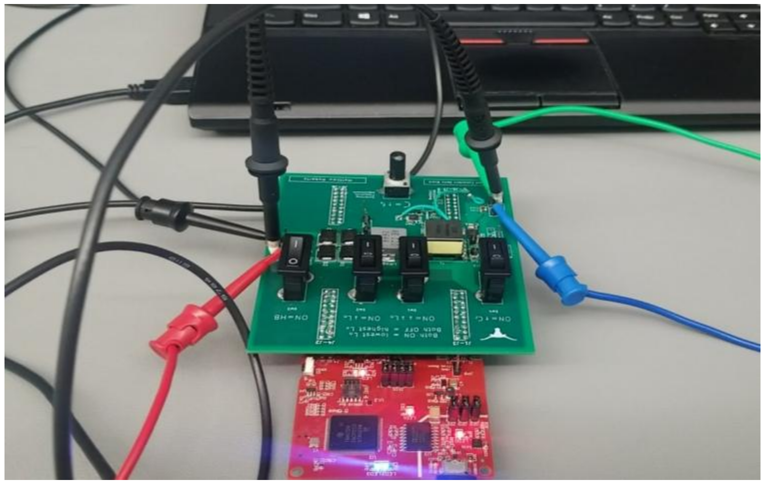

Figure 1.3: Bench Setup for Demonstration with Input on the Right, Output on the Left

1.2 How to use the Board

During the demonstration, the MATLAB and CCS script variables may need to be changed to show different results for different design parameters or configurations. This section will briefly explain how to change these variables based on operation mode and change between the modes.

1.2.1 Variable Switching Frequency Operation

This mode allows the user to change the switching frequency in real-time. The switching frequency, input voltage, and output voltage can be seen through CCS's expression viewer. Section 1.2.1 describes how to set the CCS script to run in this mode and shows how to view ADC inputs.

How to set up Variable Switching Frequency Mode (No Sweeping):

1. Open Code Composer Studio and open the project up by double-clicking the Project under the “Project Explorer”.

   — This should list all the files in the project directory.

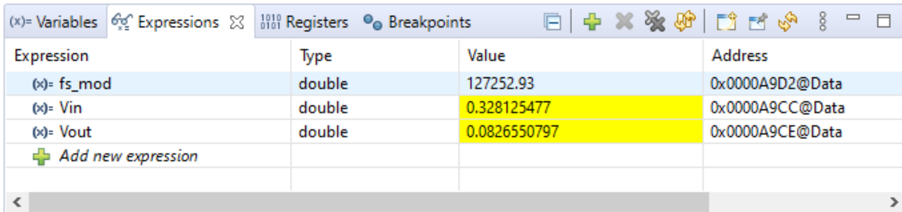

Figure 1.4: Expressions for Normal Operation

2. Double click the file "Main_Resonant.c" to open the file in the "Shared Area".

3. Under the "Operation Mode" comment section starting on line 20, set both flags to 1.

4. Configure the desired options under the "Normal Operation Parameters" comment section starting on line 30.

5. Save the file and rebuild the project.

6. Plug in the launchpad with the converter attached and debug the project.

7. Open the expressions tab located in the top right corner and insert the expressions in the "Name" column shown in Figure 1.4.

   — You may need to open up the expressions window by navigating to **View** → **Expressions**.

8. Connect the input and output sides of the converter to the power supply and load if you have not done so already.

9. Turn on the power supply at the desired voltage.

10. Click the green and yellow play button.

11. Turn the potentiometer knob and observe "fs_mod" change.

Keep in mind that no voltage gain data will be plotted in this operation, so it is recommended that gain be shown on an oscilloscope over the expression viewer for easier visualization of trends.

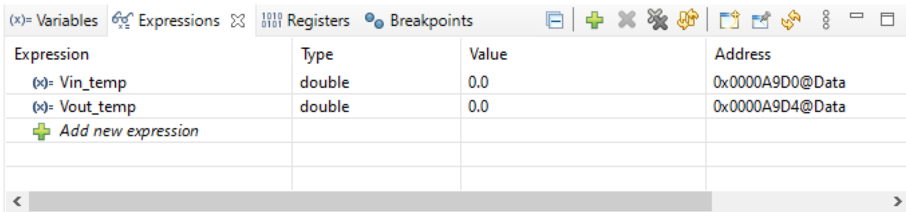

Figure 1.5: Expressions for Calibration

1.2.2 Sweeping across Frequency Operation

This mode enables the converter to sweep over a user-defined frequency range, recording voltage gain for every step. The results will be plotted into MATLAB when both scripts are running in a particular order. Firstly, the ADC must be calibrated to give accurate readings. Section 1.2.2 explains the process of how to calibrate the ADCs.

How to calibrate the ADCs:

1. Follow the steps in Section 1.2.1, but have the expressions match those in Figure 1.5.

2. Debug the project and start the script.

3. Attach probes onto the input and output voltage test points, either simultaneously or one at a time.

4. Turn the power supply on and observe the readings on the oscilloscope.

   — It is recommended to use the DC measurement on your oscilloscope to get a more accurate reading.

   — These measurements will be referred to as "Vin_measurement" and "Vout_measurement".

5. Following the formulas at lines 26 and 27 of the "Main_Resonant.c" code, use the expressions viewer's values and the readings from the oscilloscope to calculate and insert the voltage calibrations.

6. Turn off the power supply and stop the script.

Now the data can be read correctly. It is crucial that both the CCS script and MATLAB script have certain variables changed to match the converter's configuration. Section 1.2.2 describes how to set the CCS script to run in this mode and achieve juxtaposed results on the MATLAB plots.

How to set up Sweeping Frequency Mode:

1. Open Code Composer Studio and open the project up by double-clicking the Project under the "Project Explorer".
2. Double click the file "Main_Resonant.c" to open the file in the "Shared Area".
3. Under the "Operation Mode" comment section starting on line 20, set both flags to 0.
4. Configure the desired sweeping options under the "Sweeping Parameters" comment section starting on line 35.
5. Open the MATLAB script file "Resonant_Design_Serial.m"
6. Plug in the launchpad with the converter attached and debug the project.
7. Connect the input and output sides of the converter to the power supply and load if you have not done so already.
8. Turn on the power supply at the desired voltage.
9. Click the green and yellow play button.
10. Run the MATLAB script and wait patiently.
    - This process may take a minute or two. The "pause" variable in the MATLAB script can be changed to lengthen or shorten the time MATLAB ends its transmission.
    - It is suggested not to change this value unless one is getting errors in the MATLAB script.
11. Once the MATLAB plots have popped up, the CCS script can be stopped, and the power supply turned off unless one wants to remain in regular operation.

After the sweeping has finished, the board will default back into the variable switching frequency mode. If one plans on showing another plot, both scripts must be rerun by following the steps in Figure 1.2.2.

1.3 Demonstration Warnings

1.3.1 Warnings

To prevent equipment or board damage, one must follow the warnings covered in this section. It is recommended to reread the warnings section right before a demonstration. If multiple people are demonstrating the device or if there are any assistants in the demonstration, it is highly advised that every person involved reads this section. Figure 1.3.1 details these warnings:

Every person involved in the demonstration must read the following:

1. Do not put a connected load higher than 50 Ω. Higher, unintended loads will quickly destroy on-board components.

2. Pay attention to the orientation of passive probes for the test points.

3. Open loads are fine and will not break the equipment.

    — The positive side of the test point will have a plus sign located close to it.

4. Do not use passive probes on the differential probe test points.

    — Differential probe test points will have a caution symbol located in the middle of the label.

5. Do not leave the board and the supply unattended, even when the CCS script is not running.

6. Do not alter the CCS code without complete knowledge of what you are doing.

7. Do not substitute the resonant component values with different ones.

8. Be gentle with inserting and removing the converter shield onto the launchpad.