



Power Engineering Laboratory (MSc)

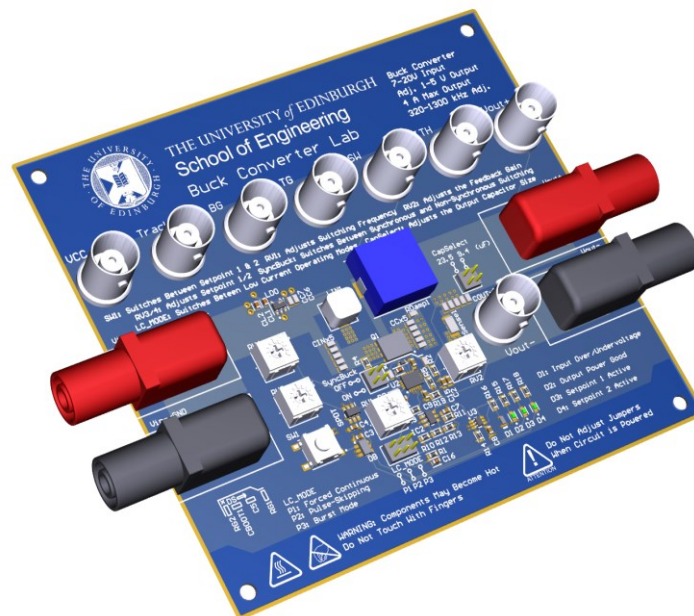
Buck Converter Lab – Dr. Paul Judge

Week 1

Welcome to the first week (of three) of the power electronics components of this module.

This is a lab using a buck converter; however, it is not primarily about the buck converter itself. Rather the key learning outcomes are:

- Development of your abilities to use an oscilloscope to take measurements of power electronic circuits.
- Development of your ability to interpret Printed Circuit Board (PCB) schematics and component data sheets.
- Experience with using a modern switched mode power supply.
- Development of experience with using LTSpice to simulate a power electronic system.



Examination:

The marks for this lab overall lab are split between a results presentation that must be submitted at the end of 4th week of the lab schedule (20% of marks) and a 25-minute viva examination that will take place in the 5th week of the lab schedule (80% of marks).

The results presentation is for students to record results from different exercises that take place in weeks 2 & 3 of the lab. The first week's session will focus purely on skills and understanding development. There is no need to keep a lab book or save any measurements or results from this session, it is simply preparation for the following two weeks. You may however be asked questions about different board functions that you explore during this lab during the viva examination.

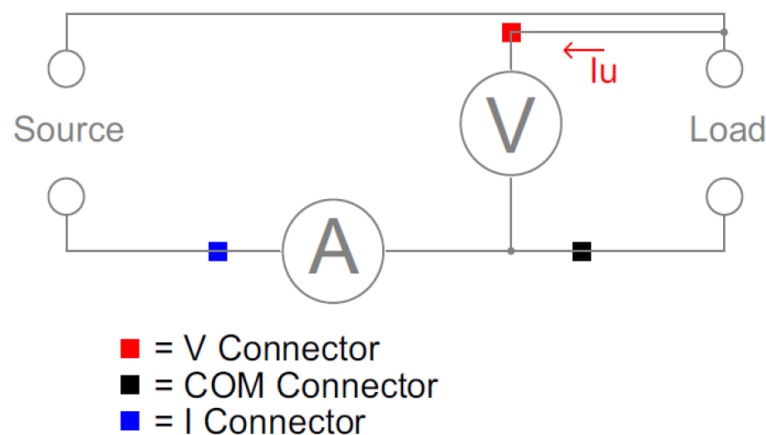
Part 1: Development of Board Familiarity (Target time – 15 Minutes)

Note: The board has been designed to be very robust and has been subjected to numerous tests (overvoltage's, inverse input voltages, short-circuited outputs etc). If for any reason you suspect a board has become damaged, please let a T&D or myself know. We've tried breaking them ourselves first and weren't successful, so will be impressed rather than annoyed.

WHEN CHANGING JUMPERS ON THE BOARD PLEASE TURN THE POWER SUPPLY OFF FIRST!

Connecting the board to the power supply, power analyser and load resistor

Connect the power supply to the input connectors on the buck converter board using the banana connectors available. You should then connect the output of the buck converter board to the 10 Ohm load resistor through the power analyser using the below figure as reference.



With this done you should be able to turn on the converter for the first time. Set the output voltage setpoint of the power supply to 10 V and set the current limit of the power supply to 3.00 A (if you need help doing this you can refer to pg. 24 of the power supply's (NGE100B) manual or ask a T&D for help).

With the power supply's setpoints set you can now enable the output of the power supply. Using the power supply you should now have a measure of:

- The input voltage.
- The input current.
- The input power.

Using the power analyser you should now have measurements of:

- The output voltage.
- The output current.
- The output power.

The board and 10 Ohm load rheostat should have been pre-configured so that the output voltage is approximately 3.3 V and the load current is 2 A – If this is not the case please ask a T&D to assist you. You should now be able to calculate the efficiency of the buck converter under these input and output conditions. Double check your value with a T&D.

Part 2: Oscilloscope Skills (Target Time 45 Minutes)

The buck converter board contains 8 different BNC connector sockets that provide easy access to measure different voltages within the board. **All of the measurements give you a voltage measurement with respect the ground of the board.** This ground is connected to the negative input from the power supply.

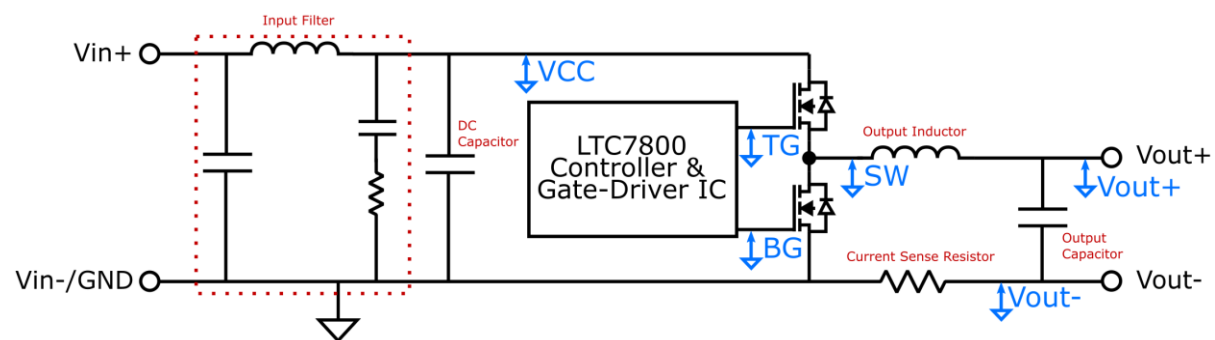


Figure 1: Simplified Electrical Schematic of Buck Converter with test points shown in blue

A description of each of the measurement points built into the board is given in the table below. In this exercise we are going to use the oscilloscope to measure the voltages at some of these points.

| Measurement Point | Measurement Type | Description of Measurement Point |
|-------------------|------------------|--|
| VCC | Power Circuit | Input Voltage |
| Track/SS | Control Circuit | This voltage sets the output voltage reference for the LTC7800 control chip |
| BG | Power Circuit | The gate of the lower device in MOSFET half-bridge within the buck converter. |
| TG | Power Circuit | The gate of the upper device in the MOSFET half-bridge within the buck converter. |
| SW | Power Circuit | The mid-point of the MOSFET half-bridge within the buck converter |
| ITH | Control Circuit | The feedback control pin in the LTC7800 control chip |
| Vout+ | Power Circuit | The positive output rail of the buck converter |
| Vout- | Power Circuit | The negative output rail of the buck converter. This rail also gives a voltage measurement |

Connect the channel 1 probe of the oscilloscope to the **SW** measurement point – you can do this while the board is operating. Examine the circuit diagram shown in Figure 1 so that you are sure of what point in the circuit you are actually measuring.

You will need to adjust the trigger level and settings on the oscilloscope so that you can get a steady measurement. You can then adjust the time-base and vertical scaling setting so that you can see the

voltage at the mid-point of the half-bridge within the buck converter. If you are familiar with using oscilloscope you may wish to do this yourself, otherwise you can ask a T&D to help with this.

Adding Additional Measurements

You should then add the **BG** measurement to channel 2. Adjust the vertical settings on the oscilloscope so that both signals can be easily seen at the same time (again ask a T&D if you are unsure). What relationship do you see between BG and SW? Does this relationship make sense to you – please discuss with a T&D and myself if you are unsure.

Adjusting the Trigger Channel

You should also try changing the trigger settings so that the scope triggers using the **BG** signal rather than **SW**. You can do this by firstly opening the trigger menu on the oscilloscope by pressing the 'Trigger' button in the top right of the oscilloscope. This will open the trigger menu for the scope. You should see an option to select the Trigger Source. By changing this you will change what channel the oscilloscope uses to trigger. If you scroll down in the trigger menu you will also see options for 'HF Reject' and 'Noise Reject'. These options may be useful latter on if you run into scenarios where it is difficult to trigger the oscilloscope due to noisy signals.

Measuring the Output Current

The next measurement you should add is **Vout-**, which you should add to channel 3 of the oscilloscope. This measurement is the negative output rail of the buck converter, but if you examine the electrical schematic of the converter you should notice something called a 'current sense resistor'. This is a low-inductance and low-resistance resistor that is used for taking current measurements. The LTC7800 control chip that runs the board uses this current sense resistor for several functions that we will explore in later lab sessions, but we can also use the voltage across this resistor to get a measurement of the current flowing through the buck converters output inductor – Please consult a T&D or myself if this is not clear.

To get an improved current measurement you can adjust the gain of the oscilloscope probe to 1:1 (there is a yellow slide switch at the end of the probe that controls the probe attenuation). You can also limit the bandwidth of the measurement for this channel to 20 MHz. You can change this option by opening a Channel menu for a channel on the oscilloscope. The Bandwidth option should be towards the top of this menu. If you scroll down you will see there is a 'Probe' sub-menu, in which you can adjust the probe attenuation for this channel on the oscilloscope.

Adding Labels

After adjusting this you should also add a label for this measurement. This can be done by opening the channel menu for a channel and then scrolling down to the 'Label' option at the very bottom of the channel menu. This will be important later when you are saving results and need to check which measurements have been saved to which channels.

Adding Maths Channels

Add the measurement point **TG** to channel 3 of the oscilloscope. If you examine the electrical schematic in Figure 1 you should see that this point gives a measurement of the gate of the top MOSFET in the half-bridge with respect to the GND of the circuit. You should remember that MOSFETS are turned on once a sufficient positive voltage is applied from their gate with respect to the source of the MOSFET. Examining the circuit identify which point in the circuit the source of this

MOSFET is connected to (Hint: we have already adding the measurement for this point to one of the oscilloscope channels).

Press the 'Math' button at the bottom of the oscilloscopes front panel. This will bring up the Equation Set Editor menu on the oscilloscope, which allows you to add up to four different math waveforms to the oscilloscope. Using this Equation Set Editor add a math waveform that subtracts the voltage at the mid-point of the MOSFET half-bridge from the measured voltage at the gate of the top MOSFET. This will result in the MOSFETs gate to source voltage being displayed.

Once this Maths channel has been added to the oscilloscope display, which should now show the gate-source voltage measurement of the top MOSFET, is added you can disable the display of channel 3 by pressing the channel button for this channel, while retaining the maths channel that shows the MOSFET gate to source voltage. Examine the relationship between this gate-source voltage measurement, the gate-source measurement of the bottom MOSFET device (**BG**) and the voltage at the mid-point of the MOSFET bridge (**SW**). Is this what you expect? – Discuss with a T&D if required.

Adding Measurements

During this lab, you will need to take numerous measurements of different waveforms. One of the first tools you can use for doing this is the cursors. The 'Cursor' button that enables these is in the 'Analysis' section of the oscilloscopes front panel. By pressing this button, you can control what channel the cursors are applied to and if you have horizontal or vertical (or both) cursors. Using the cursors and the **SW** measurement point you should measure:

- The switching frequency of the converter
- The duty cycle of the converter
- The peak-to-peak voltage of voltage at **SW**

An alternative to using the cursors to take measurements is the oscilloscopes automated measurement features. Press the 'Meas' button just to the right of the 'Cursor' button on the oscilloscopes front panel. This allows you to add up to four different automated measurements of different types to the oscilloscope. Try adding automated measurements that mirror those you made manually using the cursors. Compare the values you got using both methods. You should also try enabling the 'statistics' option for each measurement. This uses up more screen space but can be useful for measuring un-steady values that are changing.

Warning: Some of the measurements you take will involve noisy power electronic circuits, particularly around the switching edges of the MOSFET devices. There can be a risk when using the automated measurement features that this noise will spuriously throw off the automated measurements. You should use the cursor system to verify at least some of the automated measurements before trusting them.

Saving Results

During lab sessions 2 & 3 you will have to save waveform results as you go along onto a USB stick. You can do this quickly by pressing the button with the camera logo on the front panel of the oscilloscope. This will quick save a png image to the USB stick with an automatically named file. If you click the 'Save Load' just to the right of the camera button this will open a menu on the oscilloscope. If you select the 'Screenshots' option, you will be able to adjust the filename of the screenshot.

Exercise 3: Reading Datasheet and Following PCB Schematics (target Time 45 Minutes)

An important part of this lab is developing familiarity with reading device datasheets and linking the descriptions of how different components work to the physical implementation on the board.

High Efficiency 3.3V 2.1MHz Step-Down Regulator

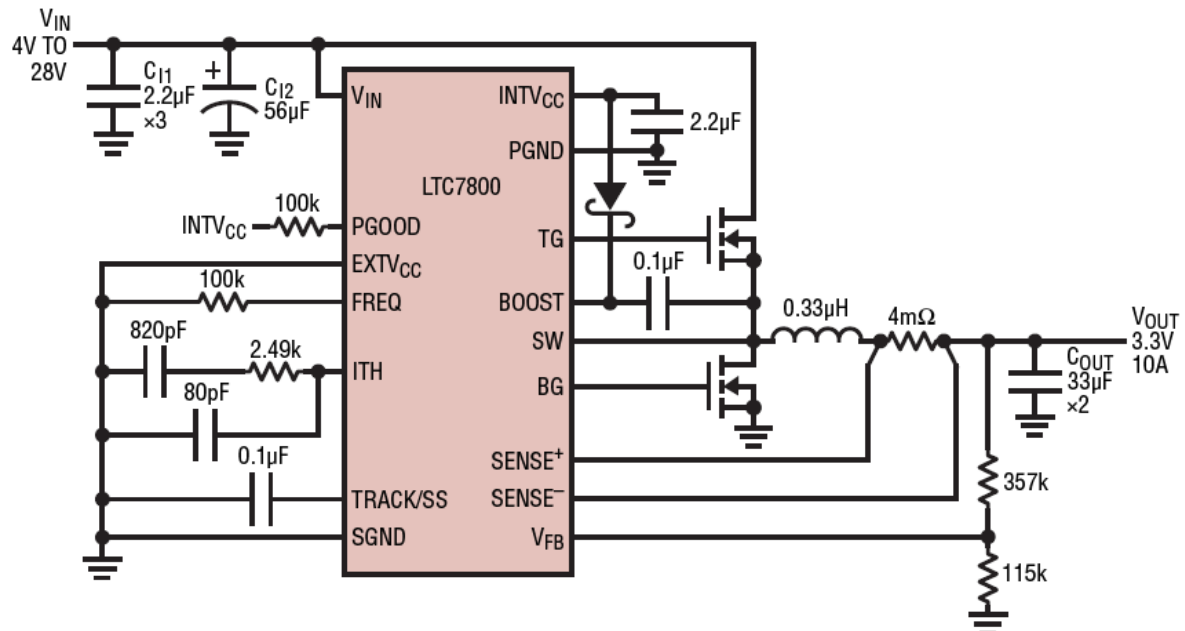


Figure 2: Typical application of the LTC7800 control chip.

During weeks 2-3 you are going to explore various advanced features that the LTC7800 control IC has. Doing so is going to require you to become comfortable with reading different device datasheets, and then reading the schematic files of the PCB to reconcile what you see on the oscilloscope with the functions of the LTC7800 control IC.

To help teach you how to do this, this exercise will guide you through the common steps required to understand a function of the LTC7800 IC, and how this function has been physically implemented on the PCB. The Buck converter board has a nominal adjustable switching frequency of 330 KHz – 1200 KHz. In this exercise you will learn how to adjust this frequency.

On page 1 of the LTC7800 datasheet you should read the **Description** section to get a broad overview of the LTC7800 chips function. You should also note how many pins this IC has. For this exercise we are interested in the $FREQ$ pin. The datasheet gives an overview of each of the IC's pins starting on pg. 8. Locate the entry about the $FREQ$ pin. The datasheet then gives further extensive information about the function of each pin in the **Applications Information Section**, which starts on pg. 14 and comprises a total of 16 pages. This is a lot of information. For now, you should just read the section on **Frequency Synchronization and Selection** (pg. 21). Some of this information will be non-relevant as the Buck Converter designed for this lab is not synchronised to an external signal – The last paragraph of this section is the most relevant and gives details of how the frequency of the converter can be programmed by setting a resistor value between the $FREQ$ pin and the Signal Ground ($SGND$) of the board.

To next step in adjusting the frequency of the converter is to consult the PCB schematic files ('/PCB Schematic and Design Documents/SchematicAndBoardFiles.pdf) to see what components on the

Buck Converter PCB have been connected to the FREQ pin. The first page of the shows the top-level schematic for the overall PCB. You will note that this contains multiple components as well as several sub-sheets, which are shown in green. The schematics for these sub-sheets are given in the later pages of the 'SchematicAndBoardFiles' PDF.

Locate the LTC7800 IC in these PCB schematics. You should be able to find the **Freq** pin and then trace backwards to the schematic to find what components are connected to this pin.

By doing this you should have located a resistors, a potentiometer and a capacitor. The resistor and potentiometer are used to adjust the resistance connected to the **Freq** pin, while the capacitor has been added to filter out noise that might couple into this circuit.

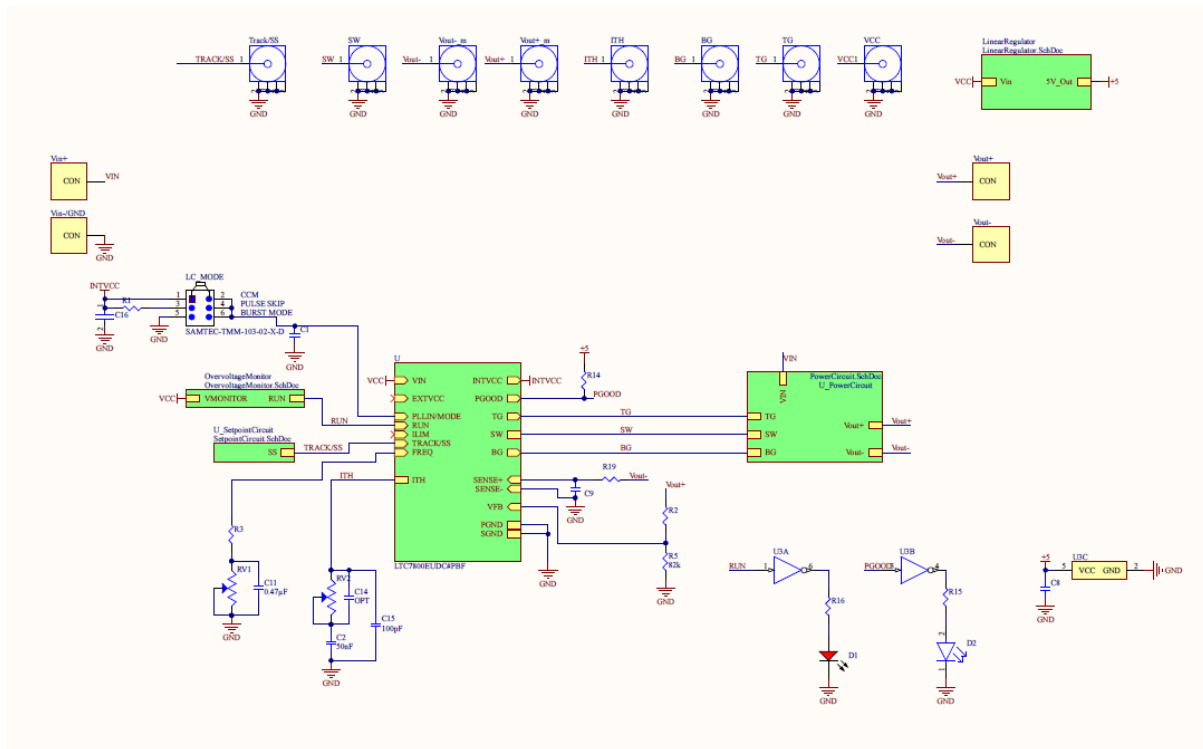


Figure 3: PCB Top Level Schematic

You should finally try and find these physical components on the PCB board itself. Finally locate the potentiometer you have identified as controlling the switching frequency of the converter. Using this potentiometer and the potentiometer trimmer tool that should be on the lab bench adjust the frequency of the converter to 1 MHz.

As a summary of steps:

- Read the datasheets to identify the pin(s) on the LTC7800 that control a specific feature (we will give hints in the later labs)
- Read the relevant sections of the datasheet that
- Find that pin in the PCB schematic and identify the components that connect to it.
- Check the component values of the components connected to that pin, and any available datasheets.

During the later lab sessions, you will need to go through this process yourself several times for different board functions.

relied upon an LTSpice model to verify that the circuit could be expected to work correctly, and to investigate the amount of power loss that could be expected to ensure that the thermal design of the PCB was sufficient.

LTSpice can be downloaded from here: <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html> and is available for both Windows and Mac computers.

The zip folder for this lab will contain the LTSpice files required for the lab. You will modify this file multiple times during the following weeks, and so it is suggested that you copy and rename the master file for each different thing you investigate during lab sessions 2 & 3.

Open the main LTSpice file (it will end with a '.asc' file extension)

The LTSpice model has three main parts. The control chip implementation shown below

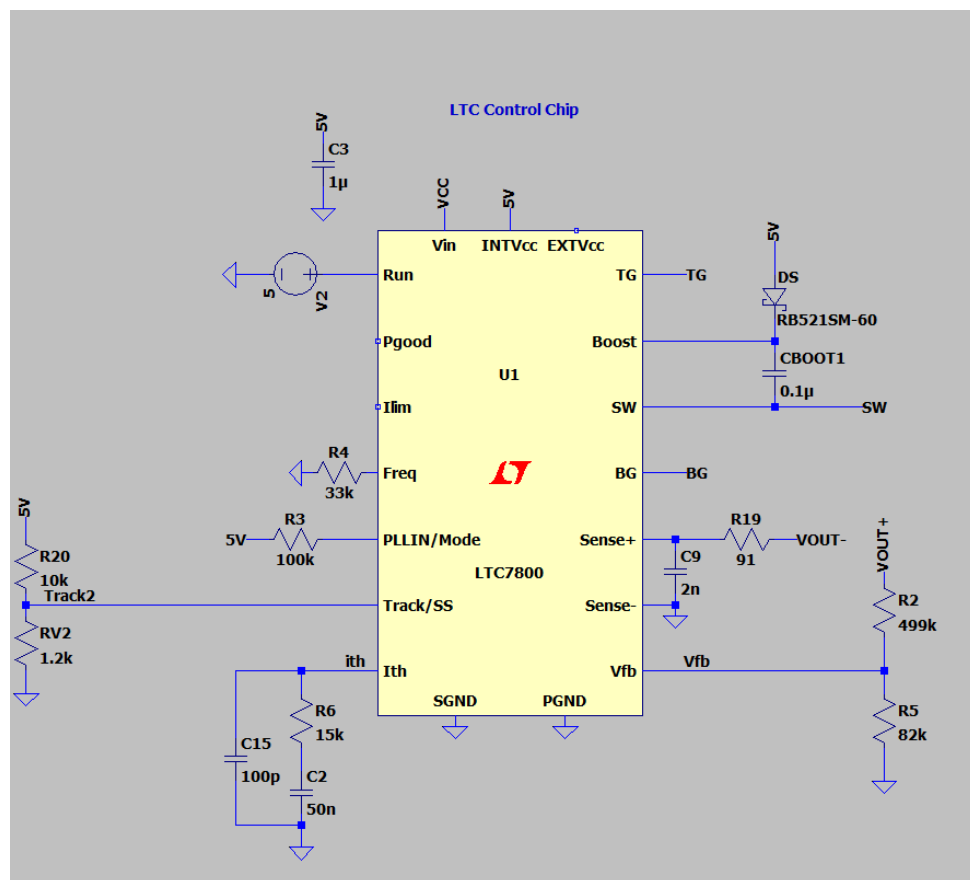


Figure 5: LTSpice LTC7800 Circuit

And the electrical circuit of the buck converter shown below.

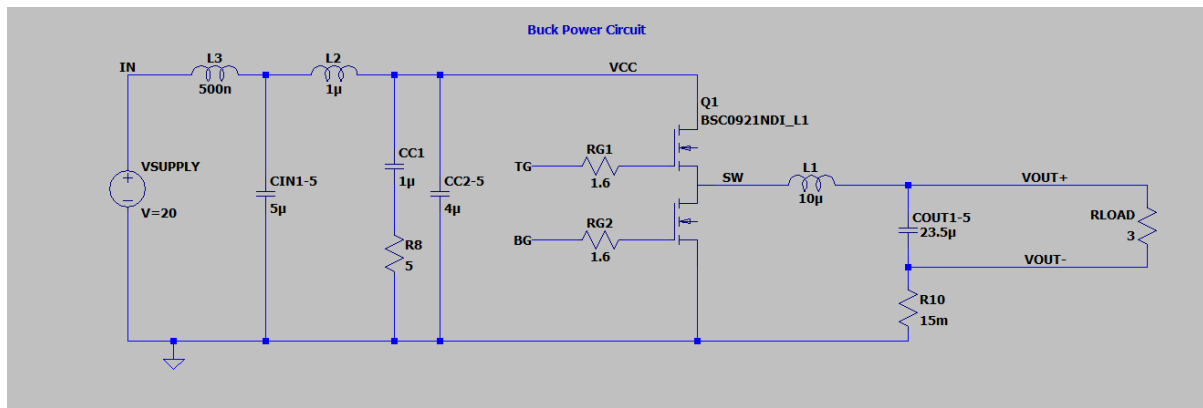


Figure 6: LTSpice Buck Converter Circuit

You should note how there are no physical electrical connections shown between the two circuits. This has been done for neatness. Instead, electrical connection have been made using netlabels. These netlabels electrically connect different parts of the overall circuit together.

The last part of the circuit is shown below and contains multiple spice directives. These are code snippets that control multiple options

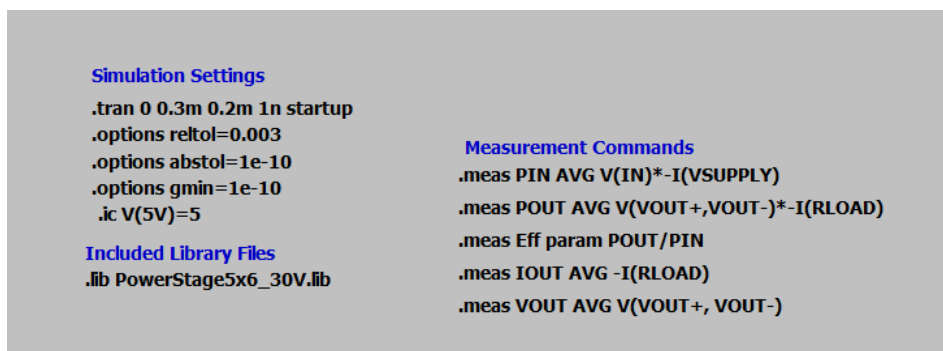


Figure 7: Spice Directives

If you right click the '.tran 0 0.3m 0 1n' directive, which controls the length of the simulation this should open a menu in LTSpice. You can use this menu to adjust the length of the simulation and several different options. We will come back to this menu later in this exercise.

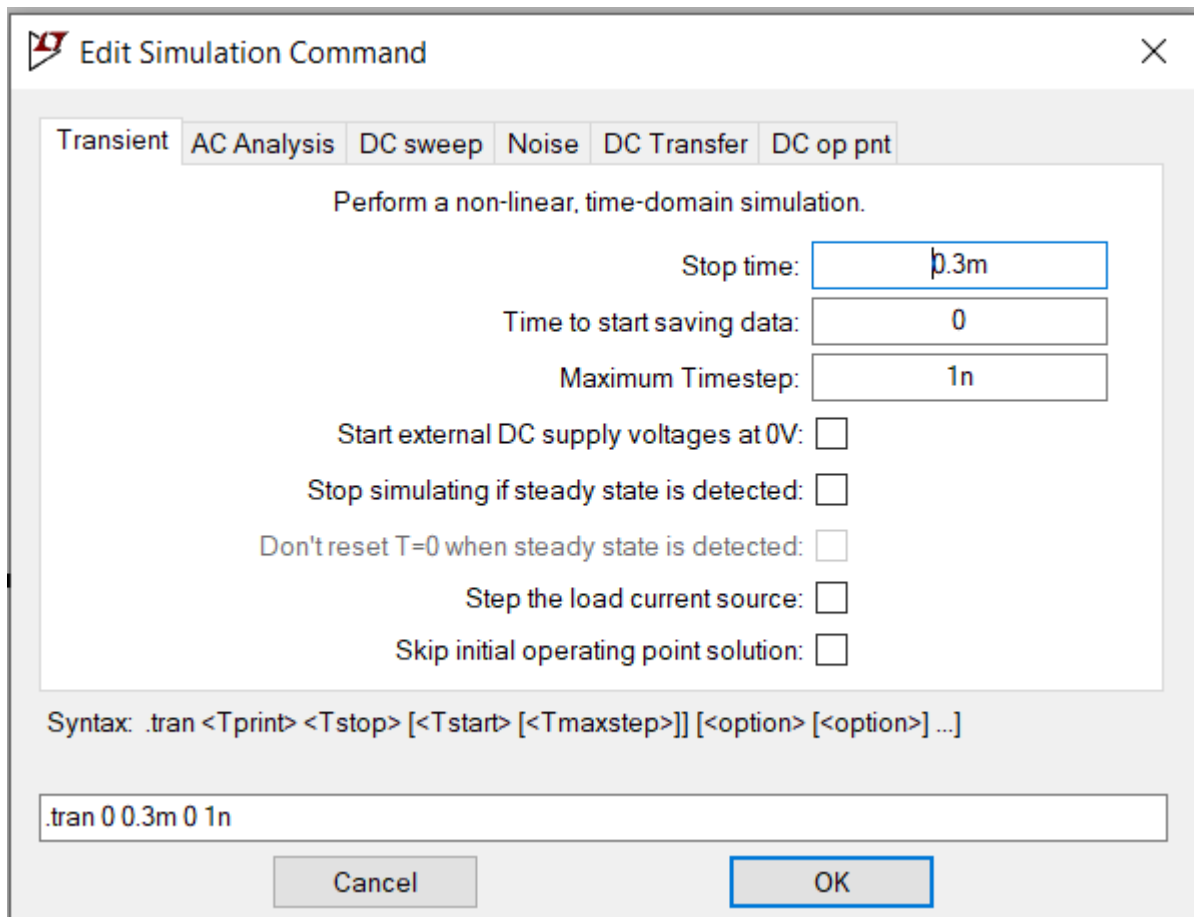
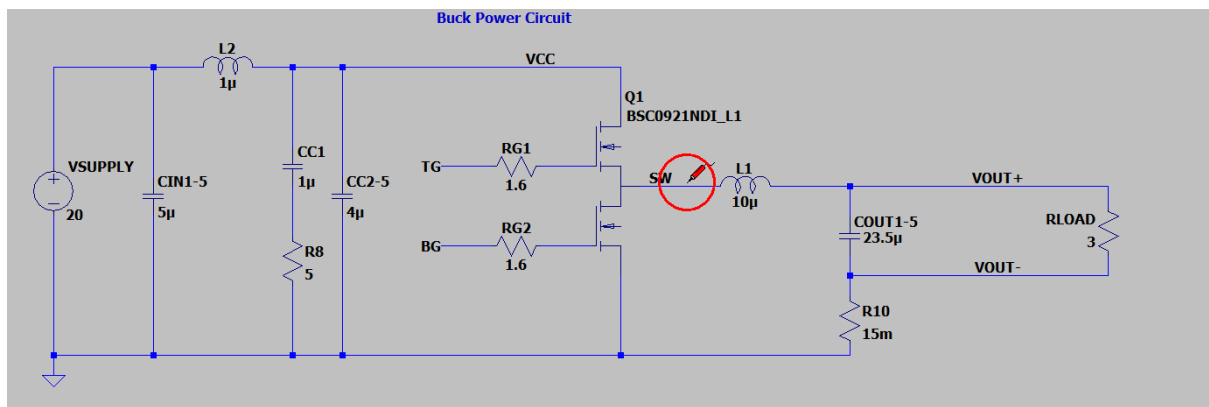


Figure 8: Simulation Command Window

At the top menu click 'Simulate -> Run' – This will run the simulation of the circuit. The progress of the simulation will show in the bottom right of the window. A new window should also appear – This is the waveform viewer. Once the simulation finishes it is time to start adding waveforms to the viewer.

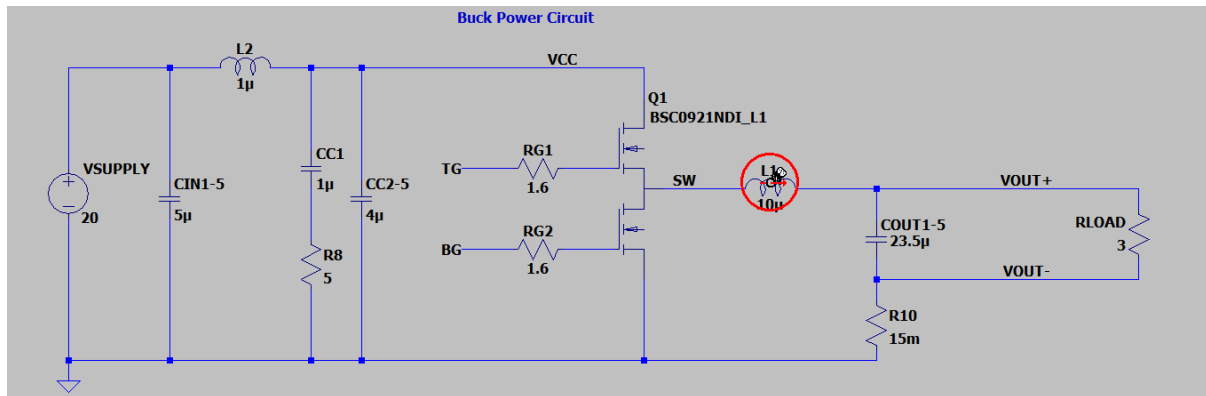
Taking Voltage and Current Measurements

You can add voltage waveforms by mousing over points in the circuit and clicking. Add the voltage at the switch node (**SW**)

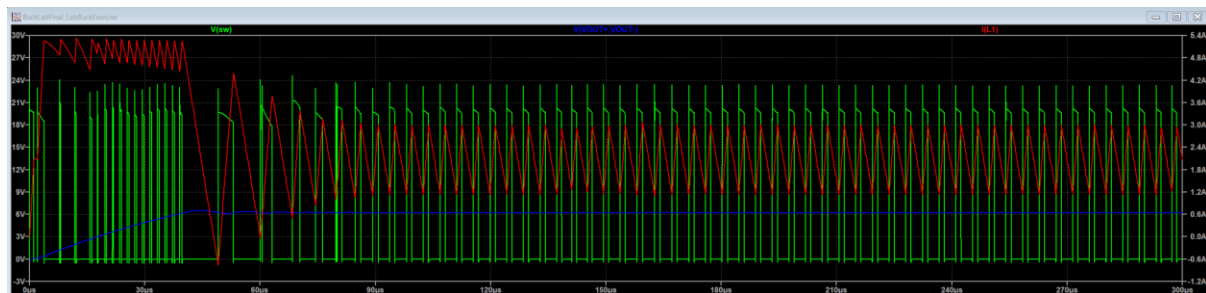


You can also add differential voltage measurements by clicking and holding on one point of the circuit and then dragging to another point. Try adding the voltage across the load resistor (RLOAD). In the waveform viewer this voltage should be labelled 'V(VOUT+,VOUT-)'.

You can then add current measurements by mousing over a component as shown below. Add the current through the inductor to the waveform viewer



In the waveform viewer you should now have three measurements overlaid on top of each other

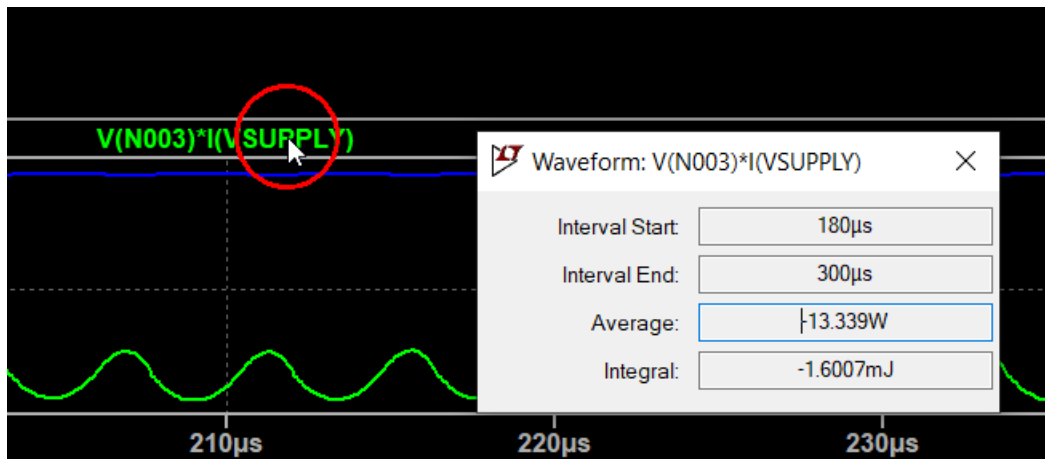


You can add additional plot panes by right clicking in the waveform viewer and then selecting 'Add Plot Pane'. You can click and drag measurements to these new plot panes by clicking their label at the top of the viewer. You can also zoom into the plot by clicking, holding and dragging the mouse over the area you wish to zoom.

Taking Power Measurements

In LTSpice you can take power measurements by holding the 'alt' key and mousing over a component. Add power measurements for both the supply voltage source (VSupply) and the load resistor (RLOAD).

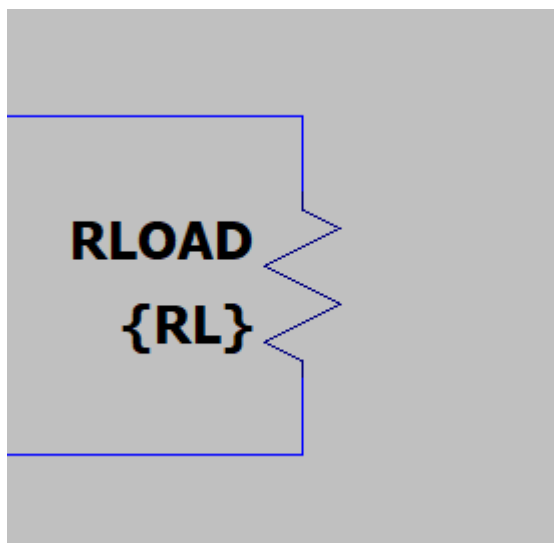
In the measurement viewer you can get average values of measured signals by mousing over them, holding the 'ctrl' key and clicking. You may need to zoom into the waveform so that the circuit has reached steady state as the large inrush currents during the start of the converters turn-on will result in inaccurate efficiency measurements being made. Try zooming in so that the time from 200 – 300 µs is displayed in the waveform viewer. You should now be able to get an estimate of the circuit's efficiency using LTSPICE.



Another way of doing the above is to instruct LTSpice to only take measurements after a certain time. This is also useful when setting up automatic measurements. Right-click the '.tran' statement we examined earlier to open the 'Edit Simulation Command' window. You should be able to adjust the time when LTSpice starts saving data here. Set it so that data is saved after 200 us and rerun the simulation.

Automating Multiple Runs

It is possible to setup multiple runs in LTSpice so that you can check how varying circuit parameters impact performance. Right click the load resistor and change its value to '{RL}'.



Click the 'spice directive' button in the top menu. This has a symbol of '.op'. This will allow you to add a new spice directive to the simulation.

Type '.step param RL list 3 4 5' into the spice directive and place this on the schematic. This translates to a command to run multiple simulations in which parameter RL is stepped through the list of values 3, 4, & 5. More info on the step command is here:

<https://www.analog.com/en/technical-articles/ltspace-using-the-step-command-to-perform-repeated-analysis.html>

Run a new simulation. LTSpice will now run three separate simulations and overlay the results. If you keep a single measurement in each plot pan, LTSpice will also colour code each run. This will not work if more than one measurement is in a Plot Pane

It is possible to automatically log measurements in LTSpice using measurement commands (these are spice directives that start with a '.meas' statement. The LTSpice model provided has five included measurement statements:

Measurement Commands

```
.meas PIN AVG V(IN)*-I(VSUPPLY)
.meas POUT AVG V(VOUT+,VOUT-)*-I(RLOAD)
.meas Eff param POUT/PIN
.meas IOOUT AVG -I(RLOAD)
.meas VOUT AVG V(VOUT+, VOUT-)
```

The measurement statements log:

- The average input power.
- The average output power.
- The efficiency
- The average output current
- The average output voltage

You can view the logged measurements in the 'spice error log' that is produced after the simulation stops running. You can view this by clicking View -> Spice Error Log in the top window menu of LTSpice. If you then scroll down you will find the measurements towards the end of the file

| Measurement: pin | | | | |
|-------------------|-------------------------------|---|--------|----|
| step | AVG(v(in)*-i(vsupply)) | | FROM | TO |
| 1 | 13.3479 | 0 | 0.0001 | |
| 2 | 10.1046 | 0 | 0.0001 | |
| 3 | 8.13272 | 0 | 0.0001 | |
| Measurement: pout | | | | |
| step | AVG(v(vout+,vout-)*-i(rload)) | | FROM | TO |
| 1 | 12.8768 | 0 | 0.0001 | |
| 2 | 9.68219 | 0 | 0.0001 | |
| 3 | 7.75747 | 0 | 0.0001 | |
| Measurement: eff | | | | |
| step | pout/pin | | | |
| 1 | 0.964708 | | | |
| 2 | 0.958195 | | | |
| 3 | 0.95386 | | | |
| Measurement: iout | | | | |
| step | AVG(-i(rload)) | | FROM | TO |
| 1 | 2.07177 | 0 | 0.0001 | |
| 2 | 1.55581 | 0 | 0.0001 | |
| 3 | 1.24559 | 0 | 0.0001 | |
| Measurement: vout | | | | |
| step | AVG(v(vout+, vout-)) | | FROM | TO |
| 1 | 6.21532 | 0 | 0.0001 | |
| 2 | 6.22322 | 0 | 0.0001 | |
| 3 | 6.22793 | 0 | 0.0001 | |

It is possible to plot these measurements by right clicking anywhere in the Spice Error Log and selecting the 'plot .step'ed .meas data'. This will open a new plot window. You can right click in this plot window and select 'Add Trace' to select measured data you would like to plot.

The y-axis in this plot window will, by default, use the parameter that the '.step' command changes during each run – In our case it will be the value of the resistor RL. It is possible to change this to another one of the measurements logged by LTSpice. You may for example wish to plot the efficiency against the output current. To do this right click the y-axis enter the name used in the '.meas' statement for the output current in the 'Quantity Plotted'.

Changing Values of Parameters During Runs

Another powerful feature of LTSpice is the ability to change the values or parameters of components during run-time. Delete the '.step' spice directive we added in the last section. You can now change the value of the resistor to the following conditional IF statement.

`R=IF(time>0.2m, 5, 10)`

This changes the value of the load resistor from 10 Ohms to 5 Ohms, with the change occurring when the simulation time exceeds 0.2 ms. Inspect the simulation waveforms and verify that you can see the transient occurring in the converter at this time (hint: you may need to change the time at which LTSpice starts logging data, remember that we changed this in the previous section).

You can also use the same syntax to change the input voltage source. Try setup the simulation so that the input voltage changes from 20 V to 10 V at 0.25 ms.

Saving Waveforms

You can save waveforms from LTSpice for insertion into your PowerPoint presentation. To do this you can right click in the waveform viewer and then select View -> Copy Bitmap to Clipboard. This option will copy the waveform as it is displayed to your clipboard. From there you

For both options you may wish to resize the waveform viewer so that the text and the size of the plot is of a suitable size to include in your PowerPoint.

End of Lab Session 1 Checklist

Before finishing this lab, you should go down through this checklist and check off each line to ensure that you have successfully developed the skills this lab was designed to give you. All these skills will be vital for the week 2 & 3 labs, which will involve a more explorative approach to learning about this practical implementation of a Buck Converter.

PCB Understanding

I can connect the PCB to the power supply, power analyser and load resistor correctly.

I understand what each physical measurement point on the PCB measures, and how it physically relates the circuit diagram.

I understand how to read the datasheet and link the information in the datasheet to the components on the PCB.

Oscilloscope Understanding

I can connect oscilloscope probes to the board.

I understand how to get the oscilloscope to trigger off signals and have identified which measurement points are the most useful for triggering off.

I can save measurements to a USB stick using the oscilloscope.

I can add labels to channels using the oscilloscope.

I can adjust the time-base on the scope.

I can adjust the y-axis scale for individual channels on the oscilloscope.

I can add cursor measurements (both horizontal and vertical) to individual channels on the scope.

I can add measurements to individual channels using the oscilloscope.

I can use math features on the scope to take differential measurements between channels.

I can use the single mode of the oscilloscope to capture transient events.

LTSpice Understanding

I can run the LTSpice simulation, and display measurements for voltage and currents.

I can measure instantaneous and average power of the overall circuit and individual components in LTSpice.

I can adjust the values of components within the LTSpice model of the buck converter.

I can setup multiple automatic simulations with a varying parameter using the .step command and can plot measurements from these simulations using the .meas command and the results stored in the Spice Error Log

I can setup transient events using conditional IF statements within LTSpice.

I can copy and past waveforms from LTSpice to PowerPoint.