

1 Assignment Deliverable

A well-formatted and laid out report containing details of your power supply design, with justifications for decisions made, and validated test results including analysis of output reliability and impacts on power quality.

Maximum of 10 pages, including design calculations, figures, tables, key schematics and waveform images. Additional waveform images may be included in an appendix and commented on in the main body of the report.

More details on project report expectations can be found in Section 6.

2 Overview

You work for a consumer electronics design company that is developing a *Smart-device Docking Station* capable of providing power to a range of smart devices (phones, tablets, music players, etc), audio amplifiers, as well as providing an auxiliary power supply. The docking station is required to be able to plug into any single-phase power outlet anywhere in the world, and should be compliant to applicable power quality standards. Your role at the company is to design and test a power supply system that is capable of supplying the various parts of the docking station with the necessary quantities of power at the required voltages. The requirements are detailed in Section 3.

There are four main parts to the project which you should complete: Converter Design, Stage 1 Converter Build and Test, Stage 2 Converter Build and Test, and then finally combining the two converter stages together into one Complete Power Supply Circuit and testing.

3 Power Supply Specifications

The basic power supply system and requirements are represented by the block diagram shown in Figure 1. You are required to use a standard diode bridge rectifier, and then design two separate DC-DC Converters (Stage 1 shown in green, Stage 2 in red in the block diagram). Your company requires that the second (red) DC-DC Converter that you design be flexible enough such that a single design can be manufactured which satisfies both separate end-use voltage and power requirements. The Stage 2 converter should be able to be powered via the mains/Stage 1 Converter. In order to help reduce manufacturing costs and to keep the product as lightweight as possible, you should

design all converters to operate at a switching frequency of at least f_{sw} kHz where $f_{sw} = (30 + \sum(\text{your numeric UNI} - \text{ID numbers}))$ kHz ¹, and avoid using excessively large components. It makes sure that there isn't a single solution for the design. So it is not like doing an assignment with a fixed solution (like early year AQF Level 7 work). It is closer to a real-world design requirement. Everyone will end up with a different design.

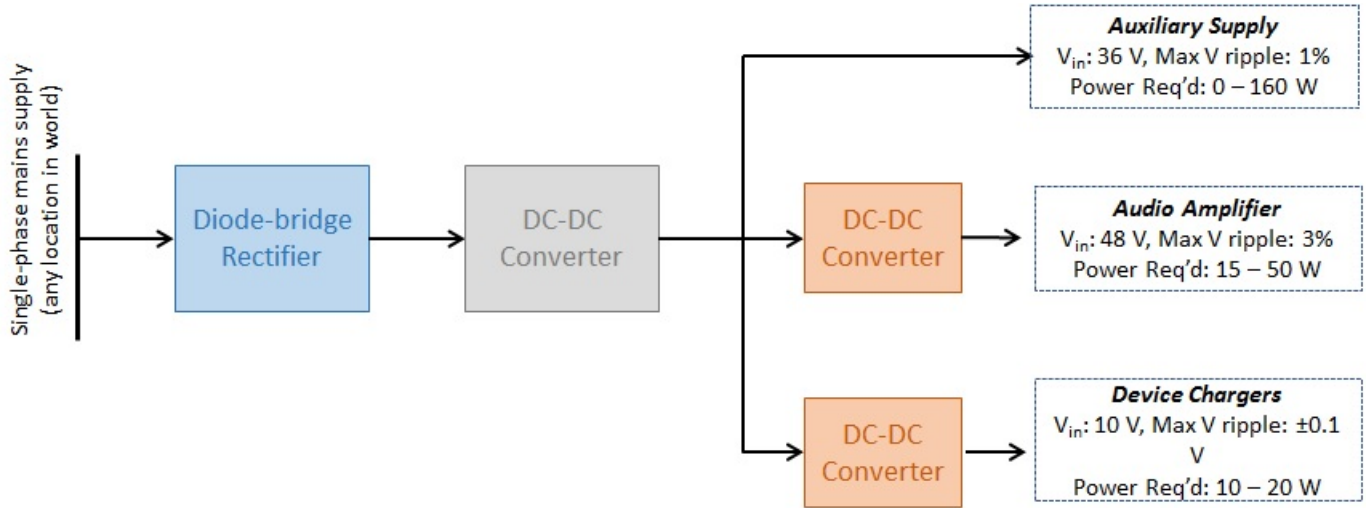


Figure 1: Power supply block diagram.

4 Project Requirements

Part 1 - Converter Design

Rectifier/Stage 1 DC-DC Converter: Start by designing the Rectifier and Stage 1 DC-DC converter. Unlike the rectifiers mostly discussed in the notes, you won't have a large enough load to enable inductive filtering on its own to smooth the output (without using a very large inductor). And like the hardware labs, if you use too large a capacitive filter it will result in large amount of harmonic distortion on the input side. You should instead allow the rectifier output to have some capacitive filtering but still fluctuate over a reasonable range, and rely upon the Stage 1 DC-DC converter to provide suitable regulation and a constant 36 V output.

Stage 2 DC-DC Converter: You need to come up with a single converter design that will work for both loads. The converter should be operated at least f_{sw} kHz, although it does not have to be the same switching frequency as the stage 1 rectifier.

¹For example if your UNI-ID is u5664317, then switching frequency will be $30 + (5+6+6+4+3+1+7) = 62$ kHz.

Part 2 - Implement and Test Stage 2 Converters

Use LTSpice to build the Stage 2 converter with open loop control (unregulated), and then test the converter for all three load types and load sizes. Verify that the converter produces the required outputs, stays in continuous conduction. Measure the input power required in each load scenario and measure converter efficiency under a range of different load scenarios. *Optional:* Implement closed loop/feedback PWM control to automatically manage the duty cycle to achieve the required output. Note: it can be harder to get this right than for the Stage 1 converter - in fact it is recommended to do stage 1 converter feedback control first.

Part 3 - Implement and Test Rectifier and Stage 1 Converter

Use LTSpice to build the Rectifier and Stage 1 converter, complete with closed loop PWM control of the DC-DC converter section (regulated supply). Validate that the circuit works for both maximum and minimum output loads and for the maximum and minimum AC voltages that it can experience. Check that the converter produces the required 36V output, that it stays in continuous conduction and also that there are no high current spikes in the circuit. Measure the input power required in each load/input scenario, identifying where the losses are, and calculate the Rectifier/Converter efficiency. Observe and comment on the AC supply current waveform.

Part 4 - Join the Rectifier/Stage 1 converter and three Stage 2 converters and test

Connect the Rectifier/Stage 1 converter to two copies of the Stage 2 converter and test under all real conditions. Comment particularly on whether you are still able to achieve the desired output voltages and low ripple, and if required what you think might do to address that. Calculate the combined/overall efficiency of supply to the loads.

Comment on what you observe at start-up, that is when you first plug in the power supply and all output voltages are zero. Do some research and see what can be done in practice to address this.

5 Notes/hints on LTSpice Components

Diode: You may pick any available diode, as long as it can handle a large enough forward current and withstand a large enough reverse bias voltage. The MUR460 is recommended.

Inductor: Use the standard inductor, but to make it more realistic include in series with it (or define in the inductor specs) a resistance which you must scale with inductance value, using 20 m Ω per 100 μ H, Figure 2.

Switch: Start with a simple generic Switch model (SW), rather than using a FET or IGBT. You won't then have to worry about maximum current or reverse bias or about

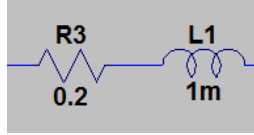


Figure 2: Realistic inductor model.

switching transients (or possible simulation hang-ups). To make it a little more realistic though you should accompany your switch with a 100 m Ω resistor in series with it (this represents the ON resistance of the switch), Figure 3.

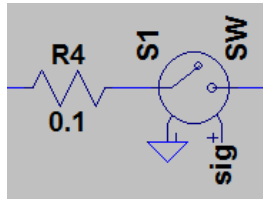


Figure 3: Realistic switch model.

PWM control: When it comes to implementing PWM control of your DC-DC converter you will need to compare the output voltage against a desired value and then against a triangle waveform. For similar reasons as for the switch, it is recommended to start with that you use a simple behavioural voltage source (bv) to generate the PWM pulse train, Figure 4. You can input into this any equation you like... you can consider this to be a legitimate software implementation of PWM feedback control.

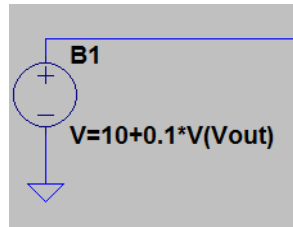


Figure 4: Behavioural switch.

Node labels: It will help, both for plotting and for defining feedback equations, if you label some nodes, Figure 5.

Transient simulation time: Make sure that when you measure average values and ripple, and FFT that you ignore the transient start-up component. For the case of the Rectifier and stage 1 converter simulating for at least 5 - 10 lots of 50 Hz cycles is recommended. For the Stage 2 converter on its own you will probably not need to simulate for as long.

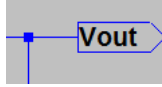


Figure 5: Node label.

6 What your project report should contain

Your project report should contain an abstract or summary which describes in summary form what you did: your power supply design, any special features you uses, your observations and results, comments on any difficulties that need addressing, and which extra bits you might have attempted and with what result. Your report should then ideally contain a section for each separate part of the project, detailing for example the design calculations and choices, schematics from LTspice of your final circuit, summary of validated results and measurements, include key waveforms and any unusual observations and what you did to fix them. You should include a short conclusion or discussion section which wraps it all up.