ECE 469 — Power Electronics Laboratory

LABORATORY INFORMATION AND GUIDE

Prof. P. T. Krein, Prof. R. C. N. Pilawa-Podgurski, Prof. A. Banerjee
Department of Electrical and Computer Engineering
University of Illinois
Urbana, Illinois

Version 3.4 — August 2019

Contents

Introductory Material

Preface and Acknowledgments

Power electronics studies the application of semiconductor devices to the conversion and control of electrical energy. The field is driving an era of rapid change in all aspects of electrical energy. The Power Electronics Laboratory course — one of only a few offered at the undergraduate level in the United States — seeks to enhance general material with practice and hands-on experience. The laboratory course provides instruction in general lab practices, measurement methods, and the design and operation of several common circuits relevant to the field of power electronics. It also provides experience with common components such as motors, batteries, magnetic devices, and power semiconductors. The course has a significant design component. The final weeks of the term are devoted to a power converter design project.

The equipment and instrumentation for ECE 469 were updated substantially in 2011, and our complete new laboratory was commissioned in 2014. Many people have helped in a wide variety of ways in the past, and their efforts are appreciated. Past work by Z. Sorchini, J. Kimball, R. Balog, and K. Colravy is acknowledged. The generous support of The Grainger Foundation has been instrumental in developing and improving the laboratory. The efforts of the ECE Electronics Shop and the ECE Machine Shop in preparing the benches and equipment are gratefully acknowledged.

Student feedback is encouraged throughout the semester. Your input will help make the course more interesting and enjoyable, and will increase its value over time. Comments are always appreciated. Experiments and other work can and will be modified quickly if the need arises. The course is designed as an advanced laboratory, primarily for seniors and graduate students. You will find that procedural details are up to the student teams. The requirements for lab reports and procedures reflect the standards of a productive industrial research and development lab more than the relatively routine work in beginning courses.

Introduction

Power electronics is a broad area. Experts in the field find a need for knowledge in advanced circuit theory, electric power equipment, electromagnetic design, radiation, semiconductor physics and processing, analog and digital circuit design, control systems, and a tremendous range of sub-areas. Major applications addressed by power electronics include:

- Energy conversion for solar, wind, fuel cell, and other alternative resources.
- Advanced high-power low-voltage power supplies for computers and integrated electronics.
- Efficient low-power supplies for networks and portable products.
- Hardware to implement intelligent electricity grids, at all levels.
- Power conversion needs and power controllers for aircraft, spacecraft, and marine use.
- Electronic controllers for motor drives and other industrial equipment.
- Drives and chargers for electric and hybrid vehicles.
- Uninterruptible power supplies for backup power or critical needs.
- High-voltage direct-current transmission equipment and other power processing in utility systems.
- Small, highly efficient, switching power supplies for general use.

Such a broad range of topics requires many years of training and experience in electrical engineering. The objectives of the Power Electronics Laboratory course are to provide working experience with the power electronics concepts presented in the power electronics lecture course, while giving students knowledge of the special measurement and design techniques of this subject. The goal is to give students a "running start," that can lead to a useful understanding of the field in one semester. The material allows students to design complete switching power supplies by the end of the semester, and prepares students to interact with power supply builders, designers, and customers in industry. Many of you will be surprised at how pervasive power electronics has become — and at how few people have a deep understanding of the field.

Power electronics can be defined as *the area that deals with application of electronic devices for control and conversion of electric power*. In particular, a power electronic circuit is intended to control or convert power at levels far above the device ratings. With this in mind, the situations encountered in the power electronics laboratory course will often be unusual in an electronics setting. Safety rules are important, both for the people involved and for the equipment. Semiconductor devices react very quickly to conditions — and thus make excellent, expensive, "fuses." Please study and observe the safety rules below.

Safety

The Power Electronics Laboratory deals with power levels much higher than those in most electronics settings. In ECE 469, the voltages will usually be kept low to minimize hazards. Be careful when working with spinning motors, and parts that can become hot. Most of our equipment is rugged, but some delicate instruments are required for our experiments. Even rugged instruments can be damaged when mishandled or driven beyond ratings. Please follow the safety precautions to avoid injury, discomfort, lost lab time, and expensive repairs.

- **GROUND! Be aware of which connections are grounded**, and which are not. The most common cause of equipment damage is unintended shorts to ground. Remember that oscilloscopes are designed to measure voltage relative to ground, not between two arbitrary points.
- **RATINGS!** Before applying power, check that the voltage, current, and power levels you expect to see **do not violate any ratings**. What is the power you expect in a given resistor?
- **HEAT! Small parts can become hot** enough to cause burns with as little as one watt applied to them. Even large resistors will become hot if five watts or so are applied.
- **CAREFUL WORKMANSHIP!** Check and recheck all connections before applying power. Plan ahead: consider the effects of a circuit change before trying it. Use the right wires and connectors for the job, and keep your bench neat.
- **WHEN IN DOUBT, SHUT IT OFF!** *Do not manipulate circuits or make changes with power applied.*
- **LIVE PARTS!** Most semiconductor devices have an electrical connection to the case. Assume that **anything touching the case is part of the circuit** and is connected. Avoid tools and other metallic objects around live circuits. Keep beverage containers away from your bench.
- **Neckties and loose clothing should not be worn when working with motors.** Be sure motors are not free to move about or come in contact with circuitry.
- Remember the effects of inductive circuits — high voltages can occur if you attempt to disconnect an inductor when current is flowing.
- **EMERGENCY PHONE NUMBER: 9-911**

The laboratory is equipped with an emergency electrical shutoff system. When *any* red button (located throughout the room) is pushed, power is disconnected from *all* room panels. *Room lights and the wall duplex outlets used for instrument power and low-power experiments are not affected.* If the emergency system operates, and you are without power, inform your instructor. It is your instructor's task to restore power when it is safe to do so. Each workbench is connected to power through a set of line cords. The large line cords are connected to two front panel switches labeled “3φ mains” and “dc mains.” The standard ac line cord is connected to the switch on the bench outlet column. Your bench can be de-energized by shutting off these three switches.

Equipment and Lab Orientation

Introduction

The Grainger Electric Machinery Laboratory was funded through a grant from The Grainger Foundation. The equipment, support, and even the entire facility have been renovated. The laboratory rivals many modern industrial research counterparts in terms of safety and instrumentation. The room includes a set of workstation panels to distribute power throughout the room and special lab benches that are the primary tool for all work. The benches hold rotating machines, dedicated power meters, an instrument rack, a cable rack, and connection panels. Extra instrumentation and equipment are stored in cabinets at the bottom of each bench.

Map of the Facility and Electrical Panels

The laboratory is located in room 4024 in the Electrical and Computer Engineering Building, as shown in Fig. 1. A storage area is located just east of the laboratory. Motors and extra instruments are kept in that area. Down the hall to the east is the Advanced Power Applications Laboratory, a research facility which shares many of the same features. The main laboratory is supplied by 60 Hz ac power at 208 V three-phase. A separate dc power supply system delivers ± 120 V at up to 24 kW. Power from the regular building supply is used for instruments and low-power experiments. The room includes an interconnect set for experiments that involve multiple benches. Up to 30 A can be imposed on any of these wires.

The master circuit breakers in the room have what is called a “shunt trip” mechanism. They can be turned off with a short pulse of ac power. When any of the large red “panic buttons” throughout the room is pushed, all master breakers feeding the workstation panels are forced to shut off. When this occurs, power is cut off at all lab station panels throughout the room. This provides an emergency disconnect capability. It does not affect lights or regular wall outlets in the lab.

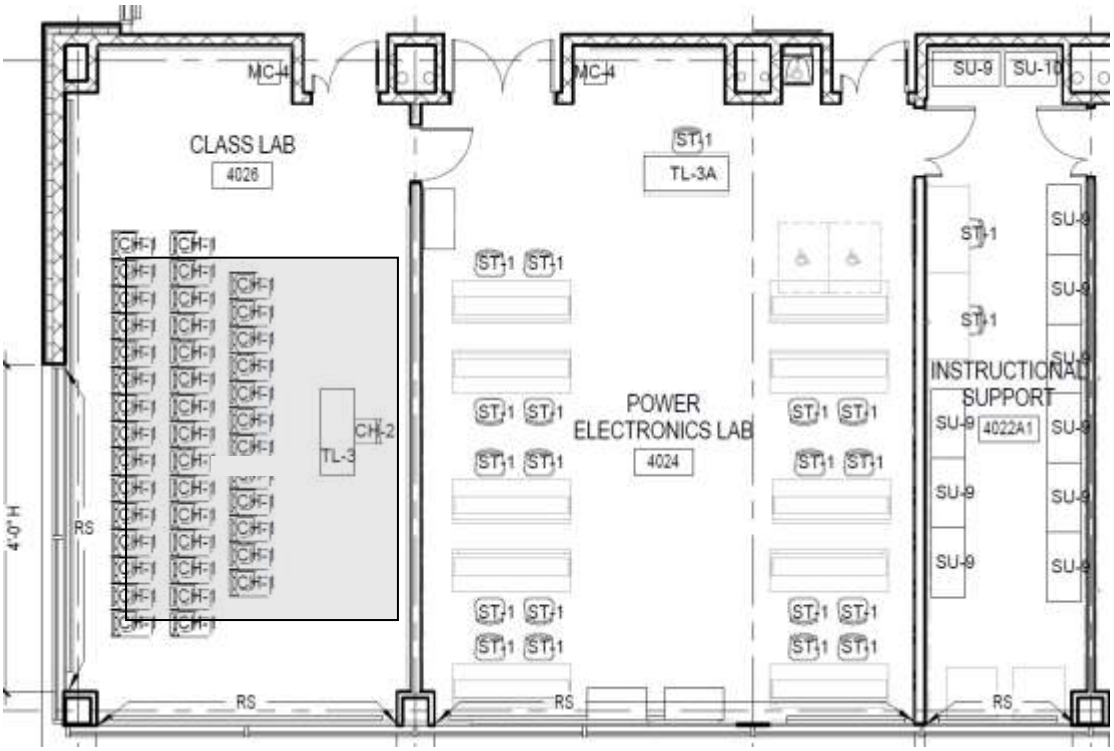


Figure 1. The Grainger Electric Machinery Laboratory and surroundings

Figure 2. Front view of workstation panel

A view of one of the workstation panels is provided in Fig. 2. The top portion contains two power outlets for convenient access to the high-power supplies. One of these is a 120/208 V three-phase source, which is also connected to an adjacent set of duplex outlets. The bottom of the panel holds eight “transfer jacks,” wired to the interconnect panel. There is a ground jack for access to a solid earth ground.

The Lab Workbenches

Overview

Each power lab bench is designed as a complete test station, with its own safety features and protective mechanisms. The benches have space for instrument operation and storage, rotating machines, and power connections. Photographs are shown in Figs. 3 and 4. There are two functionally identical bench versions — a right-hand unit and a left-hand unit.

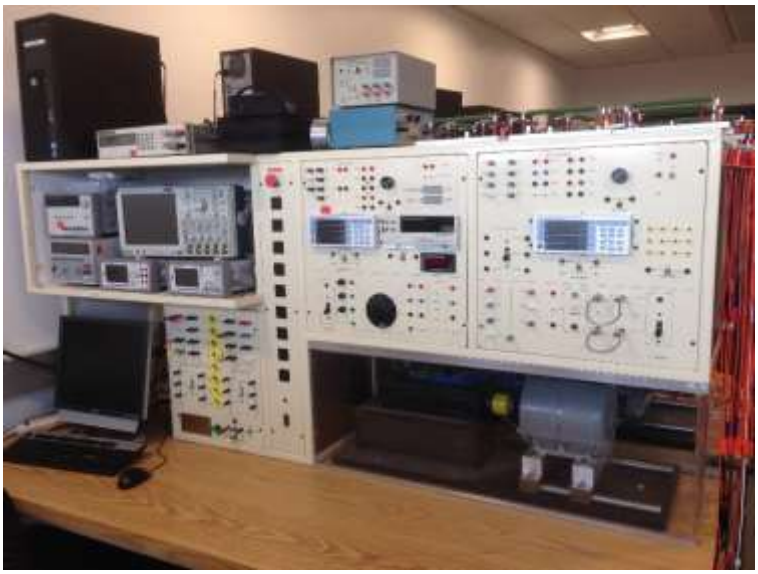


Figure 3. Laboratory bench, "left-hand" version

The benches plug into the workstation panel outlets with power-line cords accessed through the bench "window" behind the computer monitor. Many panel jacks on your bench have been pre-wired internally for your convenience. These jacks have identical labels. They allow short, organized connections. *Please be aware of these labels, and respect them.* The benches are divided into four major sections: input power handling and distribution, rotating machine connection panels, the instrument rack, and the load patch area.

The power-line cords have incompatible plugs to prevent errors in power access. They are of the twist-lock style to prevent accidental removal. Three-phase ac power to the bench is from the 120/208 V source. A double-throw center-off switch located beneath the bench must be set to select the proper source. In any case, three-phase ac power is wired to the "3φ mains" switch on the bench front panel. When this switch is off, no three-phase power will appear at the bench panels. Dc power to the bench is routed from the line cord, through a fuse box, and then to the "dc mains" switch on the front panel. As with ac power, turning this switch off will remove all panel access to the dc source.

Figure 4. The laboratory workbench

The single-phase ac instrument power is routed from the familiar 1φ line cord to the outlet column near the center of the bench, to outlets in the instrument rack, and to internal instrument power through a front-panel circuit breaker. The single-phase line cord should be plugged into the wall duplex outlet near the floor so that computers and instruments will not be affected by use of the room panic buttons. The other cords should be plugged in only as necessary for power access. Each bench can be shut off by turning off the 3φ mains switch, the dc mains switch, and the instrument power switch.

Inventory

Each bench is permanently equipped with the following:

- Variable three-phase ac transformer, 0-230 V, 0-10 A
- Yokogawa WT310 Power Meter (four display) RMS V A Watts, PF, dc to 100 kHz 20 A
- Fluke Model 45 dual multimeter
- Speed and torque meters connected to the machine set
- Westinghouse Power Miser ac motor start box
- Three-phase transformer set 120 V/25.2 V
- Variable power resistors 0-300 Ω, 150 W
- Power resistor, 100 Ω, 150 W
- Three three-pole 30 A switches
- Two one-pole switches, one rated 30 A, one rated 6 A for meter shunting
- Kollmorgen *Goldline* brushless servo motor and drive (B-206-C21 + SR20200), 0-4900 RPM and 0-19 N-m torque

In addition, each bench is supplied with the following equipment:

- Tektronix Model MS04304B scope, 350 MHz 2.5 GS/s, 4 CH Analog and 16 CH Digital
- Tektronix current and isolated voltage probes
- Kenwood PD56-10AD dc Power Supply 56 Vdc 10A
- Agilent 34461A 6½ Digit Display, Multimeter
- Agilent 3350B Waveform Generator Series Trueform
- Agilent E 3631A Triple Output dc Power Supply 6 Vdc 5 A, ±25 Vdc 1A
- Hewlett-Packard 6060B electronic load
- General Electric Model CD186AT 1.5 HP dc machine
- TECO/Westinghouse 2 HP 1.5 kW three-phase induction motor
- Advanced Motor Tech 1.5 kW wound-rotor synchronous machine
- Additional dc and ac machines
- Three power resistor boxes, each with ten 500 Ω resistors
- Three capacitor boxes, each with eight 6 μF capacitors
- Three 1 kVA, 240/120 V transformer boxes
- Computer with LabVIEW Dynamometer Control, GPIB, USB, Network, NI 6014 cards
- Lead rack with banana leads of various lengths

Additional instruments available in the laboratory for shared use include:

- Hewlett-Packard Model 4195A network/spectrum analyzer
- Tektronix Model 371 power semiconductor curve tracer
- Hewlett-Packard plotter and printer
- Philips Model PM6303 automatic RCL meter
- Laser printer and copier

The laboratory also has a tool set and selections of electronic parts.
Please make an effort to keep track of the equipment at your bench, especially portable items such as probes. It is important to take measurements carefully and in an organized fashion. Equipment damage is expensive and can cause time delays or inconvenience for you. Look over your station at the beginning of each lab, and return equipment to their proper places at the end of lab.

Course Organization and Requirements

The course consists of about fourteen lab sessions and a weekly lecture/discussion session. The hour of lecture/discussion each week will provide specific lab preparation, opportunity for general questions, time for elaboration on practical power electronics topics, and demonstrations. Required work is as follows: A short

pre-lab assignment accompanies each experiment. The purpose of this assignment is to help you prepare for the experiment. The problems apply directly to the procedure or report. *Pre-labs must be completed and turned in before performing the given experiment.* **Late pre-lab assignments will not be accepted**.

- The experiments and reports are semi-formal in nature. Proper lab notebooks must be maintained by all students. Reports are written independently by each individual, and follow the format given below. Correct spelling, grammar, and punctuation are expected. Most reports will cover a group of experiments.
- The final class session involves a brief oral presentation. Here, the design project is described and demonstrated.

Care and neatness in the maintenance of lab notebooks and in the preparation of reports is important. Your instructors will be pleased to assist you in generating quality work.

As you know, it is difficult to make up missed laboratory work. Please notify your instructor as soon as possible if illness or similar emergency prevents your attendance. In other cases, arrangements can sometimes be made, given enough advance warning; however, time demands on your instructors are such that **make-up sessions will not be held without acceptable excuse**.

Lab sessions will be divided into two major categories:

Demonstrations are conducted by your instructor and usually involve complicated laboratory work. They allow experiments, which require extensive setup time, unusual equipment, or intricate measurements. In the case of demonstrations, the pre-lab assignments serve to highlight major points. In general, you will be expected to take notes and record data during demonstrations for use in preparing reports.

Experiments are conducted by students in small teams. For each experiment, one team member serves as leader, another as recorder, and any others as helpers. Teams will be assigned early in the semester, and will generally stay the same throughout the course. Team duties rotate for each experiment.

The Lab Notebook

The laboratory notebook is a crucial tool for work in any experimental environment. A notebook used in a research lab, a development area, or even on the factory floor is probably the most valuable piece of gear in the engineer's arsenal. The purpose of the notebook is to provide a complete *permanent* record of your practical work. Why a notebook? It allows you to reproduce your own work, or to refer to it without having to duplicate the effort. It provides a single place that tracks your work in a consistent way. It provides a permanent physical record for legal purposes. Often, it permits us to “reverse engineer” and find errors of record or procedure.

The notebook is your record, but in most industry practice it is the property of your employer. For this reason, many companies have specific rules about notebook format, content, and usage. In the ECE 469 lab, your notebook will eventually become your property (although for the moment you should act as if it belongs to the State of Illinois). It should include:

- Diagrams of all circuits used in the lab. If the circuit is identical or almost identical to one in your procedure or book, you may reference (not copy) it. The important factor is to be able to reproduce your setup in case of errors.
- Procedures and actions. (But do not repeat steps in the lab manual.) The idea is to provide enough information so that you could repeat the experiment.
- Equipment used. (List only your bench number if you used only the standard bench equipment.) The model and serial numbers of special instruments and equipment should be recorded in your notebook. This is mainly for your protection in case a scale is misread or equipment is defective. Also include the values of all components you use.
- All data generated in the experiment. Be sure to include units and scale settings. For example, oscilloscope data might read "data in display divisions, 50 mA/div," and then list the numbers read. Use data in its most primitive form. Do not perform scaling or calculations when data is first recorded. The objective is to minimize errors.
- If hard copy plots or prints are generated, write the date on them and tape them into the notebook at the appropriate location.
- Names of the experiment team, with a summary of duties. Each team member should maintain a notebook in each session, although the recorder performs the bulk of this task each week. The recorder should provide copies of the original pages to all team members before leaving each week. Even though the recorder keeps notes for a given week, other team members should summarize their efforts in their own notebooks.
- Dated initials of the recorder on each page used for a given day's work.
- Your instructor's signature and signatures of all team members on the last page of the day's work.

It is entirely permissible to include calculations, observations, and even speculations in your notebook, provided these are clearly marked and kept apart from experimental data and actual bench work.

The notebook must be a bound book with permanent, pre-printed page numbers. Within these requirements, any type is acceptable. Do not use loose sheets for data or other information. It is absolutely not acceptable to recopy information into the notebook at a later time. Notebook errors should be crossed out (not obliterated) and initialed and dated by the recorder. Be sure to initial and date each page of your notebook as it is filled. Remember that paper is cheap: start a new page rather than cramming extra information onto one sheet. *The notebook must be kept in ink!*

Keeping a complete lab notebook sometimes seems inconvenient but in the long run saves a tremendous amount of time and effort. Some of the uses of an official notebook are:

- A record of your personal efforts for use with your manager or instructor.
- A history of work on a particular project or circuit. This avoids the need for duplicated effort.

- An official record for patent applications. If a patent is challenged in court, the notebook is the key document to be used.
- A complete technical record for use in reports, articles, specification documents, and drawings.
- Identification of points at which errors were made.

The notebook is the "who, what, where, when, how" of the technical world. Billions of dollars are wasted each year duplicating efforts which were not carefully documented or defending patents based on sketchy lab data.

The Lab Report

An experiment is not considered complete until the results have been properly reported. One of the primary tasks of an engineer is to *interpret* results of work, rather than just to gather data. A good report helps you understand the concepts in the experiment, and also helps you when you wish to discuss and communicate those results with others. A high-quality report allows a reader to understand your results and gain the benefits of your insights. Working engineers often mention technical writing as an area in which they could have used better preparation — because of the need for good engineering reports. To give you some additional practice along these lines, lab reports for ECE 469 are semi-formal in style. They should be prepared with a word processor and laser-quality printer. The computers in room 4024 can be used if necessary. Be sure to take advantage of spell-checking and similar features.

The report has six elements:

1. **The Title Area**. This must show the report title, author, dates, and names and duties of group members.

1a. **Table of Contents**. Required only on the Design Project report. This should show the locations of all headings and major subheadings.

2. **Abstract**. A one paragraph summary of the report, including:
   → A brief, clear summary of the objectives and results.
   → An indication of the system studied, loads used, and the basic work performed.

3. **Discussion**. This is the body of the report and may contain subheadings as needed. It should report on the laboratory effort, summarize the data and any calculated results, and briefly describe the important theory and concepts. It should compare measured results with those expected and contrast the various cases studied. It should discuss important sources of error and their relevance to the results. Finally, it should discuss any difficulties encountered and suggest what might have been done differently. Study questions assigned in class or in this manual should be addressed in the Discussion. Figures, tables, and circuit diagrams are encouraged. Laboratory reports in which the discussion merely paraphrases the lab manual are not acceptable. Suggested subheadings include:
   → Theory

Brief overview of theory and the methods used for the experiment and its analysis. This should provide sufficient background for the reader to understand what you did and why. It should help the reader follow along with the rest of your discussion. *Detailed or basic theory should not be repeated from the lab manual or textbook.*

→ Results

This portion provides an organized summary of your data and calculated results, in forms that help you interpret them. Graphs are a powerful tool for this subsection. When you include graphs, be sure to label them properly and talk about them in your discussion. A good sample graph from a student report appears in Fig. 5. In most reports, this subsection also will include tables of numerical results. When calculations are involved, you should show one example of each type of calculation (please do not provide extensive numerical calculations). A sample table from a student report appears in Fig. 6. Perhaps the most important aspect of your report is the task of analyzing the results, such as comparing expected and actual results. This discussion should be quantitative whenever possible. Be sure to include percent errors or other indication of deviations from expectations. Be aware of significant digits in your data and calculations. Keep in mind that engineering is about *interpretation* of results much more than generation of results. The Results subsection is the usual place to address study questions given in the lab manual.

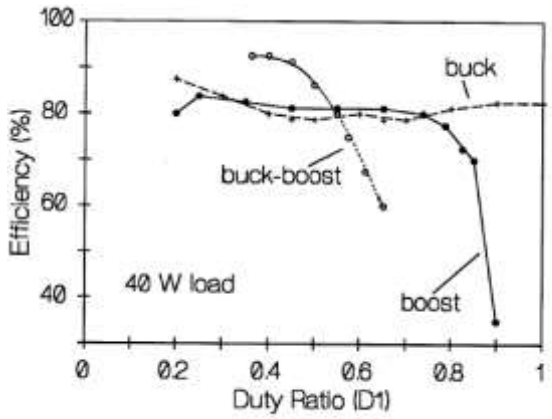


Figure 3: Efficiency vs. Duty Ratio for Dc-Dc Converters

Figure 5. Sample graph, including figure caption, from student report

→ Error Discussion

In most reports, it is important to point out sensitive places in the data. The issue is to determine the level of confidence in the results. Error issues should be discussed in quantitative terms. Consider the following example, from a student report:

TABLE 1.3 Third Test Setup

L = 370 uH
C = 50 uF/ 50 v
R = 16 ohm/55 watts
Period = 30.5 usec
Switching Freq. = 32.79K Hz

Vin	Iin	Pin	Vload	Iload	Pload	Vfet	D2	Ripple	Po/Pin
Vdc	Adc	W	Vdc	Adc	W	Vdc		Vpp	%
15.00	2.83	42.5	24.10	1.572	39.9	14.6	0.367	1.44	94.0
12.00	3.83	46.0	24.00	1.577	39.9	5.8	0.482	1.72	86.8
9.00	7.45	67.1	24.00	1.573	39.1	-10.0	0.649	2.48	58.3
7.20 (Unable to achieve 24 volt output at this level)									

Figure 6. Sample table from student report

"The calculated results depend on the phase measurements in Part I. These were hard to make and may not be very exact. Since the frequency was 50,000 Hz, 1° is only 56 ns. The scope gives up to 3% time error. On the 10 μs scale, this is 300 ns. So the measured phases could be 6° off. Our results are a lot better than this, so maybe the scope has less error."

4. **Conclusion**. A brief summary of the results and significant problems uncovered by the work. These should represent the actual results as opposed to any expectations you might have had. This is a good place to suggest how you would do it differently if you were to repeat the experiment.
5. **References**. A list of any references used. Please be aware of University regulations involving written work. Quotations or paraphrases from other works, **including the lab manual**, must be properly referenced. If the lab manual is the only source you used, you can just list "ECE 469 lab manual" as the reference. When other references are involved, list them in the order used. Examples of the format (IEEE style):

   [1] I. Rotit, *The Basics of PWM Inverters*. New York: Energy Printers, 2026, p. 142.

   [2] E. Zeedusset, "Phase error effect in bridge converters," *IEEE Transactions on Industrial. Electronics*, vol 66, pp. 4231-4236, October 2019.

   In the text, you should use the reference numbers. For example, "... methods for PWM control are described in depth in [1]...," or "... are discussed in detail in the ECE469 lab manual."
6. **Appendix**. *This must include copies of the original data sheets.* Number the sheets if you refer to them in the discussion. The Appendix should also include any auxiliary information such as semiconductor manufacturer's data sheets, a summary of the procedures actually used, and an equipment list if it differs from that in the lab handout. It is not necessary to include copies of material from the lab manual.

Lab reports should not be lengthy. Except for the Design Project report (which covers extra information), the total length of a report, except for the Appendix, *should not exceed 7 single-spaced, double-column pages*. Lab report grading will address format as well as each of the six major sections. The discussion is most important. More details about grading will be provided by your instructor. To help you in writing the report, there are several study questions given at the end of each experiment write-up. These questions do not substitute for a complete discussion of results but provide a starting point. They are not to be taken

as homework problems to be answered one by one in the lab report, rather they are important points that should be addressed in the body of the report. The study questions are of two types:

- Specific questions about results. These might request certain plots or calculations. You are expected to provide the expected information completely in your reports.
- Thought questions. These are intended to guide your thinking when evaluating the results. They should be covered in your discussion, but do not answer them one at a time as if they were test problems.

Instrumentation Notes

HP 6060B Electronic Programmable DC Load

The HP Programmable load is basically a variable load that accepts dc power and dissipates it as heat. It can be extremely useful since it is a versatile, programmable load.

The ratings on the load appear on the front panel and are as follows:

Voltage: 3-60 V, Current: 0-60 A,

Max Power: 300 W (notice that this is not the max current at the max voltage)

Always obey these ratings!

How to use the HP Load:

1. Turn the power on.
2. Select what mode you want. The choices are:

   Current – it behaves as an ideal current sink;

   Voltage – it behaves as an ideal voltage source with negative output current

   Resistance – you can set a specific resistance

   a. Press [**MODE**]
   b. Press either [**CURR**], [**VOLT**], or [**RES**]
   c. Press [**ENTER**]
3. Enter the value:
   a. Press the button of the mode you are in, for example: [**CURR**]
   b. Enter on the numeric keypad the value, for example: [5] (for 5 amps)
   c. Press [**ENTER**]
4. [**METER**] button toggles the display between volt / current and watts
5. [**INPUT ON/OFF**] can be used to disconnect the load

Advice: When you are first trying to get your circuit to work, use a power resistor from the lab stock. This removes the complication of the programmable load. It makes troubleshooting easier since we all know how a plain old power resistor should work. Once the results make sense, you are encouraged to replace the resistor with the active load.

ECE 469 POWER ELECTRONICS LABORATORY

EXPERIMENT #0 — Introduction to the Laboratory

Objective — This lab is intended to introduce some of the special equipment and methods of the Power Electronics Laboratory. Basic laboratory concepts and safety issues will be reviewed.

Pre-lab Assignment — Take a few minutes to read the safety information in the lab manual introductory pages. Be prepared with any questions. Also, be prepared to be quizzed about safety rules.

Discussion —

Introduction: In this lab, the unusual equipment associated with the Power Electronics Laboratory will be described and operated. For each lab station, basic electronic measurement gear is provided. In addition, three custom circuits have been constructed for your use. These are:

- A low-voltage ac supply, from a transformer bank, built into each bench. This supply provides polyphase output, with ratings of 117 V input to 25.2 V output. The switch allows the bank to draw power from either the bench single-phase source or three-phase source. A circuit diagram is shown in Fig. 1.

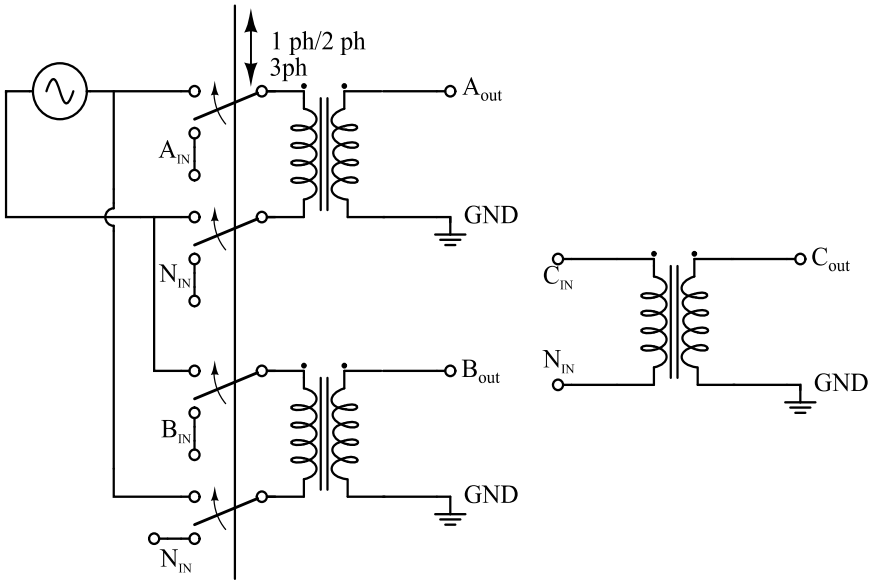


Figure 1. Polyphase transformer bank

With the panel switch in the "one-phase/two-phase" position, A and B outputs are available, with a 180° phase shift. The "C" output is indeterminate. With the panel switch in the "three-phase" position, three

outputs shifted 120° are available (if 3φ power is supplied to the bench). In both cases, the common neutral point is connected to bench frame ground.

- One-two-three phase SCR control unit. This box contains three silicon-controlled rectifiers (SCRs). Each SCR is controlled by a pulse transformer and is floating with respect to ground. The SCR is similar to a standard diode, except that it does not turn on until a pulse or a switching function is applied to a gate terminal. The three SCRs in the box are operated so that the switching functions are spaced a precise time interval apart, controlled from the front panel.

  The "phase" delay value sets the time shift among the three phases. For one-phase or two-phase circuits, it should be set to half the input period — $8\frac{1}{3}$ ms for a 60 Hz input. For three-phase circuits, it should be set to $\frac{1}{3}$ of a cycle, or $5\frac{5}{9}$ ms for a 60 Hz input.

  The "master" front panel delay control sets the time delay of the SCR "A" signal relative to the input zero crossing. When the value is set at zero, there is no delay. It can be adjusted in milliseconds up to about 40 ms. A front view of the box appears in Fig. 2.

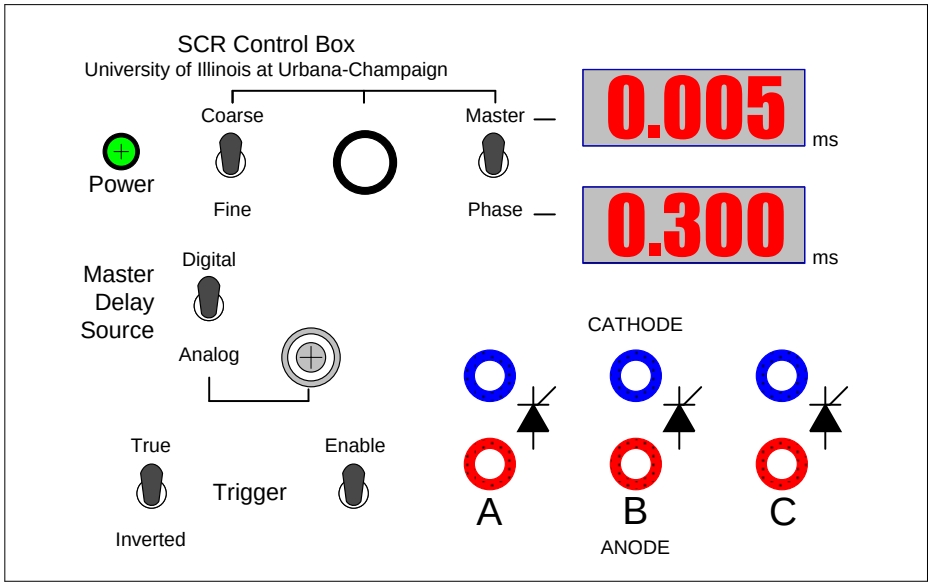
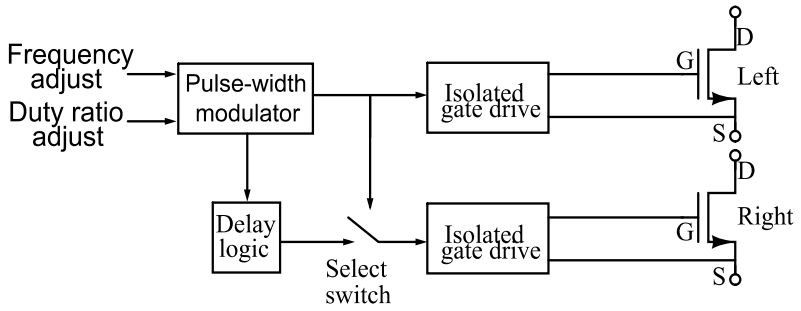


Figure 2. Polyphase SCR control unit

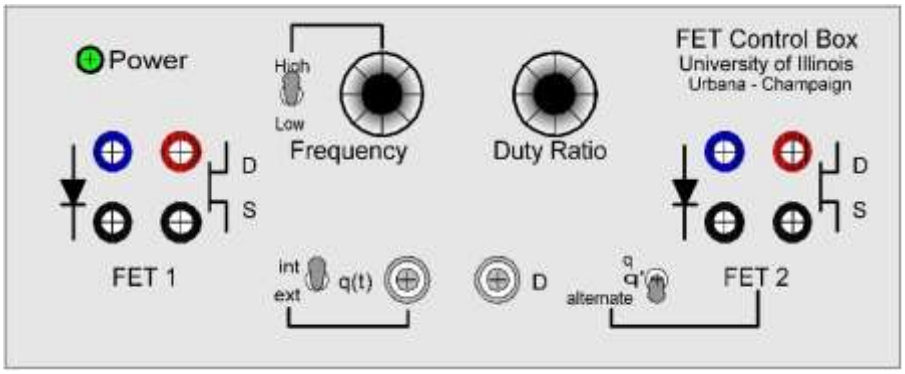
- Isolated dual power FET control box. This box contains two field-effect transistors (FETs), each rated for at least 300 V and 15 A. The FET is used as a switch, with either a small resistance (on state) or a very large resistance (off state). The drain and source terminals are floating and can be connected to any voltage which does not violate the ratings.

  Also contained in the box is a circuit, which operates the FET gates to turn the devices on and off. In essence, a square wave is applied to the FET, turning it on when the square wave is high and off when it is

low. Panel controls adjust the frequency of this square wave, the fraction of the time during which the wave is high, and the action of the square wave on the second FET. The operation is useful for dc-dc and dc-ac conversion applications. For convenience, unconnected power diodes are provided inside the box. A block diagram and a view of the front panel appear in Fig. 3.



a) Block diagram



b) Front panel

Figure 3. Isolated FET control unit

Basic Theory: Power electronics studies electronic circuits for the conversion of electric power. Examples are units which change dc voltage levels, convert ac to dc or dc to ac, or change the frequency of an ac waveform. Since a converter appears between a power source and a load, and because high power levels might be involved, efficiency is critical. Therefore, such circuits are built up from lossless devices or from low-power control electronics. The possible lossless devices include storage elements (capacitors and inductors), and transformers, but the most important lossless device in power electronics is the switch. A perfect switch has no voltage drop when on and no current flow when off; the power is always zero.

Probably the most familiar example of an electronic switch for power conversion is the rectifier diode. A rectifier circuit converts ac waveforms into dc power, with minimal loss. The drawback of simple diode-rectifier circuits is the inability to control them. A diode is on whenever current attempts to flow through it in a forward direction, and off otherwise. To improve on this, a family of semiconductor devices known as *thyristors* was invented. The SCR is the most basic thyristor. This device is similar to a diode, except that it

need not be on whenever a forward voltage is applied. Instead, turn-on can be delayed until a pulse is applied to a third terminal — the gate. Once the device is on, it functions like a diode. This ability to delay turn-on means that output can be adjusted. Outputs can range from zero (gate always off) to a full waveform equal to that of a diode (gate always on). The SCR is useful in applications which require ac to dc conversion, and power levels beyond 100 MW can be supported with commercial devices. Other power devices that can be turned on or off on command also exist. Both field-effect and insulated-gate transistors are used in power electronics for this purpose, along with various, more complicated types of thyristors.

Circuits: In this lab, both the SCR and the power FET will be used in converter circuits. The SCR set will be used in a controlled, full-wave rectifier circuit, while the FET unit will be used to form a basic dc-dc converter. These circuits are typical power electronics applications, and many commercial power units are based on them.

There are two major ways to form a full-wave rectifier, as shown in Fig. 4. One is with a rectifier bridge, which converts a single ac source into a full-wave rectified waveform. The second uses two diodes with a center-tapped transformer as a "two-phase" ac source. As in the figure, SCRs can substitute for diodes in a full-wave rectifier. The SCRs are operated half a cycle apart, with an adjustable phase angle delay.

The full-wave output is applied to a resistor in this lab. When the ac voltage at the top node of the transformer is positive, the top rectifier is forward biased. In the case of the diode, the device will turn on. In the case of the SCR, the device will turn on only if commanded to do so. When the bottom node of the transformer is positive, the bottom diode will be forward biased. Again, the SCR will turn on only when commanded to do so. Part 1 will explore this action.

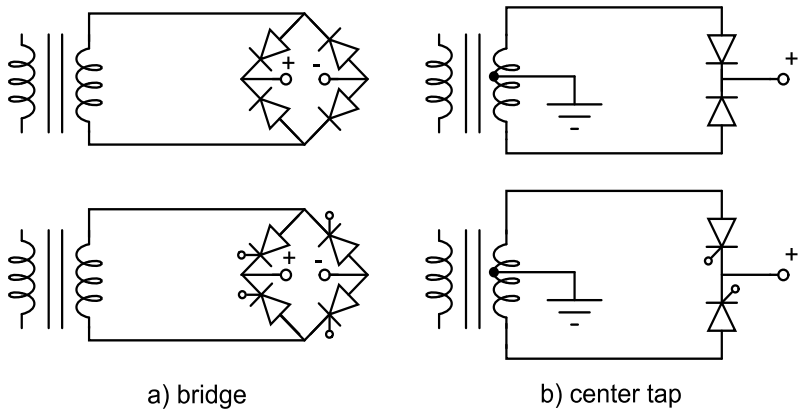


Figure 4. Basic diode and SCR full-wave rectifier circuits

Pulse-width modulation (PWM) is the most popular control tool for dc-dc and dc-ac conversion. When pulse width is adjusted, the average value of a waveform grows larger or smaller, following the width. If a circuit is set up

with unipolar dc input, the result is adjustable dc output. If both positive and negative inputs are provided, an ac output is possible when pulse width is adjusted gradually. Part 2 provides an introduction to PWM.

Procedure —

Part 1: Rectifiers and the SCR box

1. Connect the 25 V ac supply for two-phase operation. Connect two 1N4004 diodes for full-wave output (anodes to phase A and B output, cathodes in common to load). See Fig. 5.
2. Connect a load of approximately 50 Ω from the common cathode to ground. What should the resistor power rating be?
3. Observe the resistor voltage waveform. Observe diode currents and comment. Measure the resistor RMS voltage, RMS current, power, and average voltage. What is the relevance of each?
4. Rewire the circuit with SCRs in place of the diodes. Connect the 25 V ac supply for two-phase operation as in Fig. 5. Phases A and B should be wired to anodes A and B of the SCR box.

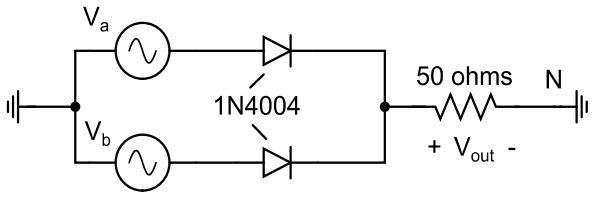


Figure 5. Two-phase diode rectifier test circuit

5. Connect the SCR cathodes in common, and to a power resistor of approximately 25 Ω. The resistor is then connected to supply ground. What power rating must the resistor have?
6. With the output enable of the SCR box off, set the phase delay to 8.33 ms for two-phase operation. Set the master delay to 0, then enable the box. Double check all connections, then turn the power on.
7. Observe the voltage waveform across the resistor. Notice how the waveform changes as the master delay is altered. Again measure RMS voltage, RMS current, power, and average voltage, and consider the relevance of each.

Part 2: Dc-dc conversion and the FET box

The FET unit will be used in a simple circuit which converts a dc voltage to a lower level with minimal power loss. The output is presented with a rapid switching of the input. The average output level (the dc portion) is lower than the input since the switch is on less than 100% of the time.

1. Set up the FET control box as shown in the circuit diagram in Fig. 6. Use the left FET in the box. You may use any resistive load you like so long as all power ratings are satisfied.

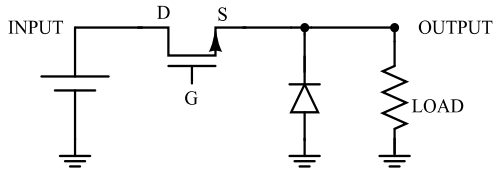


Figure 6. Simple dc-dc converter circuit

2. Connect a voltage source of 20 to 30 V to the input. Turn the unit on. Observe the drain-to-source voltage. Adjust the output for 50% duty ratio (50% on-time) and about 50 kHz.
3. Observe the output voltage waveform and average voltage. Adjust the duty ratio and notice the change. Adjust the frequency and notice the change. Explain the results.

Study Questions — We have performed some basic experiments with power electronic converters. This introductory lab is designed to help you work in the laboratory. The following group of study questions will be useful to consider in future reports:

1. In dc output converters, what is the relevance of RMS output voltage? What about average voltage? (Hint: Consider the effects on loads which are purely resistive, and on loads that include low-pass filtering.)
2. Why do we operate the A and B SCRs one-half cycle apart for the full-wave rectifier?
3. For the simple dc-dc converter in Part 2, explain how to predict the average value of output voltage from the input voltage, the switching frequency, and the switch duty ratio (the fraction of the time during which the switch is on).

Grainger Electric Machinery Laboratory Workstation

ECE 469 — POWER ELECTRONICS LABORATORY

EXPERIMENT #1 — Ac-Dc Conversion, Part I: Basic Rectifier Circuits

Objective — Measurement techniques of power electronics will be studied in the context of half-wave and full-wave rectifier circuits. Load effects in diode circuits will be explored. The silicon controlled rectifier (SCR) will be introduced, with an R-C delay circuit for gate control.

Pre-Lab Assignment — Read the discussion below. Study the procedure, and bring any questions to class. This experiment is not a long one, *provided that you are familiar with the procedure before beginning.* Solve the following, on a separate sheet, for submission as you enter the lab. Please note that your instructor may elect to assign another problem.

1. A four-diode bridge is used with an ac voltage source with a RMS value of 480 V and a frequency of 60 Hz to produce a full-wave rectifier. The rectifier can be attached to any of several loads. **Sketch** the load voltage and diode current vs. time for the following cases. **In these assignments and in the lab procedures, "sketch" means to draw the waveshape and its important features, without much regard for numerical values**. A sketch represents shape information, as opposed to detailed numerical data. You are encouraged to use tools such as *MATLAB, Mathematica* or *Mathcad* to generate graphs and solutions, but be sure to submit the commands used when you turn in the assignment. The cases are:
   a. A purely resistive load of about 40 Ω.
   b. A constant current source with a dc value of 10 A in series with a 25 Ω resistor.
   c. A 120 mH inductor in series with 30 Ω.
   d. An 800 μF capacitor in parallel with 40 Ω.

Discussion —

Introduction: A bridge rectifier circuit provides an ac-dc conversion function (rectification). But the waveforms and operation of such a circuit depend on the output load. Furthermore, diodes do not permit any control. This ties the dc output level to the ac input voltage. You have studied the properties of simple R-C, R-L, and R-L-C circuits in previous courses. Properties of "D-C" circuits (diode-capacitor circuits), as well as D-L, D-L-C, and various D-R-*x* circuits are nonlinear and cannot be studied with familiar linear methods. The behavior of these circuits provides a practical look at power electronic converters, both from the standpoint of energy conversion applications and from the standpoint of laboratory measurements. This is the focus of Experiment #1. Also, the SCR will be introduced, with an R-C circuit applied to provide a time delay in the action of its gate signal.

Basic Theory: *Diode* is a general term for an electronic part with two terminals. The most common type of diode is the rectifier diode (or forward-conducting, reverse-blocking switch). Silicon P-N junction devices and metal-semiconductor junction devices known as *Schottky* diodes are used for this function. Modern silicon diodes have

impressive ratings — currents of more than 5000 A can be carried by units which can block reverse voltages of more than 6000 V. Actually, the analysis of diode-based circuits is direct given a single additional consideration. A rectifier diode acts as a switch: It is either on or off. Once this "switch state" is determined, circuit analysis can proceed along conventional lines. The state of a diode — whether it is on or off — is determined uniquely and immediately by the terminal conditions. If forward current flow is attempted, the diode will turn on, and will exhibit only a small residual voltage drop. If reverse current flow is attempted, the diode will turn off and only a minuscule residual current will flow. No third "gate" terminal is needed.

In the half-wave rectifier in Fig. 1, the state of the diode depends on the input voltage polarity — and also on the load. With no information about the load, it is *not possible* to predict either the load current or voltage (convince yourself of this; how would one assign the on or off state?). Let us begin with a resistive load, shown in Fig. 2. In a resistor, the voltage and current are related by a constant ratio, and the load voltage is zero when the diode is off.

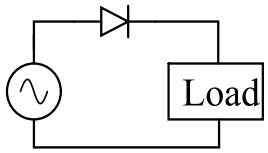


Figure 1. Basic half-wave rectifier diode circuit

One way to find the circuit action (even though we already know what the circuit does), is to take a "trial" approach. For example, consider the case when V_{in} is positive. We might guess that the diode is off. Then the voltage across the diode is just V_{in}, which is positive. But an off diode cannot block forward voltage, so the guess was wrong — the diode must be on. Similarly, consider the case when V_{in} is negative, and guess that the diode is on. Then diode current must be negative. But the diode can only support forward current, and so again the guess is wrong. In reality, the correct guess can be made most of the time. The important principle is that all currents and voltages in a diode circuit must be consistent with the restrictions imposed by the diodes. Even a complicated diode circuit combination can be understood quickly with the trial method. The essence of the method is this: Once the diode switch state is determined, the circuits are easy to analyze. If we do not know the switch state, we can just assign it in some assumed manner, then proceed with circuit analysis and check for consistency.

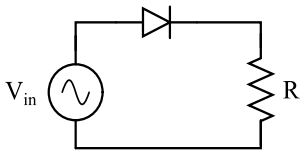


Figure 2. Half-wave rectifier circuit with resistive load

Now, look at the inductive load of Fig. 3. Assume that the inductor is large. If current is initially flowing in the inductor, the diode is on. Inductor voltage V_L will be positive or negative, depending on the input voltage and the inductor current. Since there is current flow, the diode will stay on for some time regardless of the V_{in} value. If $v_L = L\frac{di_L}{dt}$ is negative, the inductor current will fall, possibly even to zero. The diode must stay on until the current reaches zero. This time will be delayed relative to the voltage zero-crossing.

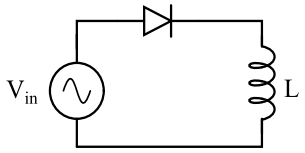


Figure 3. Half-wave circuit with inductive load

Capacitive loads bring about a different problem. Imagine the capacitor of Fig. 4, fully charged and supplying energy to the resistor. As long as $V_{in} < V_{out}$, the diode will be off. When V_{in} becomes larger than V_{out}, the diode must turn on. But then a large forward current $i_C = C\frac{dv_c}{dt}$ will flow until V_{in} becomes less than V_{out}. Brief, large current spikes are characteristic of diode-capacitor circuits. Such waveforms are typical of the power supplies in cheap electronic equipment. This is not the best situation, since the large current spikes generate noise.

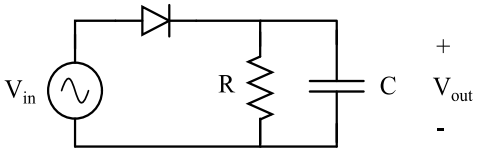


Figure 4. Half-wave circuit with capacitive load

So far, these circuits have no control. The switch necessary to add control still needs to carry current in only one direction, but must be capable of blocking forward voltage when required. The SCR provides this function. The SCR is a triode (three-terminal electronic device), built as a four-layer P-N-P-N configuration. It is called a "latching" device, because the on-state is self-sustaining once it is established. For our purposes, this means that the device is off until commanded to turn on and exactly equivalent to a diode once on. As in Demonstration #1, a set of SCRs can replace the diodes in a simple rectifier to bring about control.

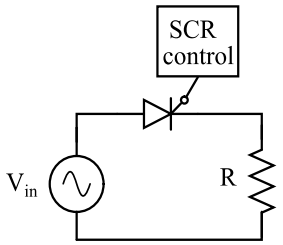


Figure 5. Basic SCR half-wave circuit with resistive load

The basis of rectifier control when SCRs are involved is turn-on delay. Consider the half-wave resistive circuit in Fig. 5. Turn-on of the SCR can be delayed to alter the wave shape. The turn-on delay is traditionally measured in degrees relative to a full diode waveform. The action of this control is not hard to determine for a known load, since the input waveform is simply being switched on and off.

Measurement Issues: In the Power Electronics Laboratory, measurement interpretation is an important ingredient. Many of the waveforms are not sinusoidal, so we will need to supplement conventional measurements with current waveforms, etc.

The input voltages in a rectifier circuit are sinusoidal ac waveforms, while the intended output is dc. Consider the ac input, which has an average value of zero. A root-mean-square (RMS) measurement would be appropriate for magnitude. Consider the rectifier output. The dc portion (the average value) is of interest. Typical laboratory meters provide the necessary capabilities. For example, the Fluke 45 multimeter displays average value whenever it is set for dc. When set to ac, this meter directs the input through a capacitor, and computes the RMS value of the filter output.

Often, a waveform we measure will contain both dc and ac. The "true RMS" value might be useful in this case. Power is also an important quantity. The Yokogawa model WT310 power meters will be useful for these kinds of measurements. These instruments perform the arithmetic necessary to find actual RMS and P_{ave} values. They have a voltage input and a series resistor for sensing current. A front view of the WT310 with its panel connections is shown in Fig. 6. The voltage to be sensed is applied in parallel with the input. The output connection forces the current to flow through the sensing resistor. The shunt switch should be set to *read* in operation. The purpose of the shunt is to direct current around the meter if needed to avoid short high-current exposure. An input signal up to 100 kHz will give a true RMS display, as long as the value is within the meter range. Current up to 20 A and voltage up to 600 V can be measured.

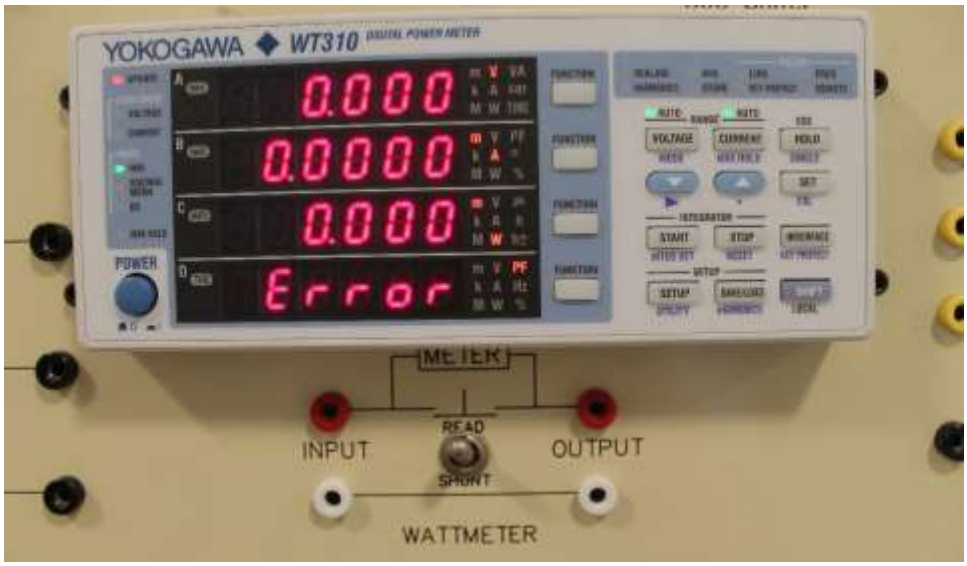


Figure 6. Front panel view of Yokogawa model WT310 meter, mount, and connections

Like current magnitudes, current waveforms can be observed by using a low-impedance series-sensing resistor. An alternative that does not introduce grounding problems is to sense the magnetic field produced by flowing charges. The laboratory is equipped with Hall-effect current probes for this function. These probes can measure current from 0 to 30 A (peak) at frequencies between 0 and 120 MHz. One drawback is that the probes may have an internal dc offset, which may drift for about twenty minutes during warm-up. A second is that they are rather delicate. You will probably find these drawbacks minor compared to the information that can be gained.

Timing is critical in the operation of most power converters. Since switches are the only means of control, the exact moment when a switch operates is a key piece of information. Consider the waveforms in the Fig. 7. A sinusoidal waveform (perhaps the input voltage to a rectifier) serves as a timing reference. It is straightforward to measure the time shift between this wave's zero crossing and the turn-on rise of the switched signal below it. The example in the figure shows two 60 Hz waveforms. The time delay τ_d can be measured directly from the graph as about 1.25 ms. Since a full 360° degree period lasts 16.667 ms, an angle φ_d can be calculated from τ_d as

$$\phi_d = \frac{1.25ms}{16.667ms / cycle} \bullet \frac{360°}{cycle} = 27°$$

One important detail in making this measurement is to make sure both oscilloscope traces are being triggered simultaneously. Make sure the scope is not in an "alternating" waveform mode, and that a single trigger signal (such as the power line) is used.

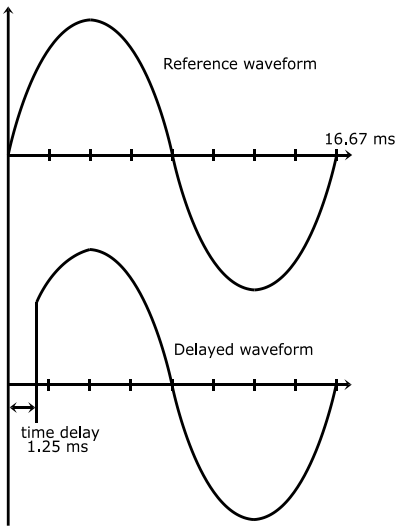


Figure 7. Phase-delay measurement example

Procedure —

Part 1: The Diode

To permit testing with small inductors and capacitors, we will use a waveform generator for the early parts of today's experiments. Please use care in connecting and operating this instrument. It is not designed to supply high current.

1. Obtain a toroidal or audio power transformer from the lab selection.
2. Construct the circuit shown in Fig. 8. Diodes should be standard 1N4001 or 1N4004 rectifiers, or similar.

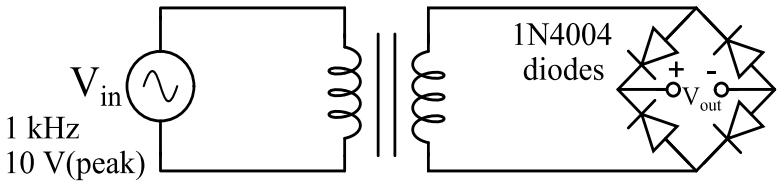


Figure 8. Rectifier test circuit

3. Be careful with ground connections. Set the waveform generator for approximately 1 kHz output at about 10 V peak. Observe V_{out} with the scope.
4. Quickly sketch the output waveform under “no load” conditions (the scope probe gives a slight load).
5. Load the bridge with a 500 Ω resistor (**compute the necessary power rating first!**). Observe and sketch the output voltage waveform. Use the current probe to observe and sketch the current out of the transformer.

6. Connect an inductor, specified by your instructor, in series with the resistor. Observe and sketch the resistor voltage waveform and the current into the bridge. Measure the average resistor voltage with a multimeter.

7. Remove the R-L load. Instead, attach a 47 Ω resistor in series with a 1 μF capacitor, as shown in Fig. 9. Place a 1 kΩ resistor across the capacitor terminals. Observe and sketch the capacitor voltage waveform and the bridge input current waveform. Also, measure the average capacitor voltage.

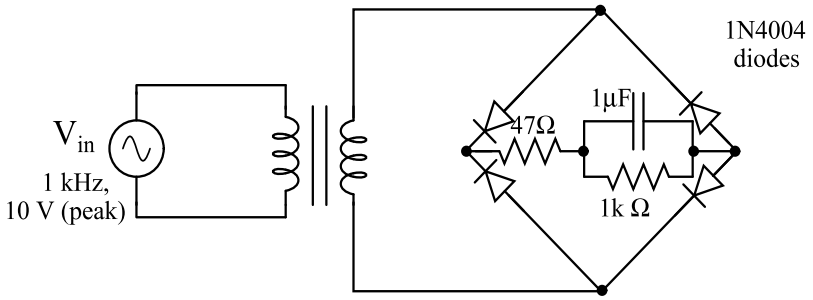


Figure 9. Rectifier test circuit with R-C load

Part 2: The SCR

In Demonstration #1, we saw a simple controlled rectifier example and the waveforms, which result. The gate control was described as a pulse, but it was not observed or controlled. Today, we will actually operate an SCR with external control, in order to gain some feeling for the gate signals expected.

1. Construct the circuit shown in Fig. 10. The input source should be the 25 V 60 Hz supply.

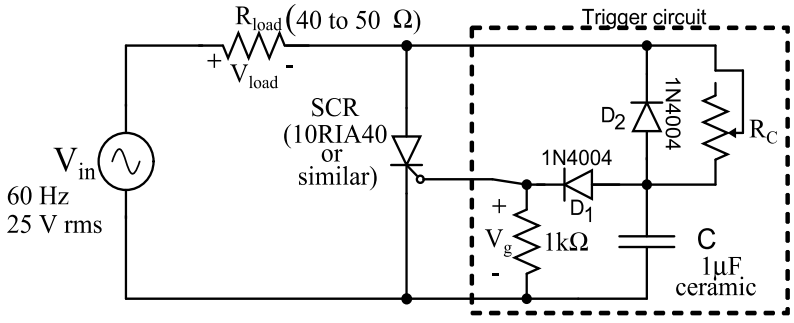


Figure 10. SCR rectifier test circuit

2. Use a 10 kΩ potentiometer or a resistor substitution box for R_c. Observe the V_g and V_{load} waveforms for R_c values of 0, 200, 400, 600, 800, and 1000 Ω. Sketch the two waveforms for two or three values of R_c.

Measure the turn-on delay by comparing V_g or the gate current with V_{load} with the oscilloscope set for line triggering. Also, use the meters to measure the average and RMS values of V_{load} in all cases.

Study Questions —

1. For Part 1, what waveforms would you expect for R, R-L, and R-C loads? Hint: Think in terms of low-pass or high-pass filtering.
2. Compare the actual waveforms with those expected. Compute the actual circuit time constants, and discuss how they might affect the waveforms.
3. The R-L and R-C cases of Part 1 represent different output filter arrangements for a rectifier circuit. Discuss the circumstances under which each of these will be effective for generation of dc output.
4. Comment on how the diode forward voltage drop (about 1V) affects the waveforms.
5. From the Part 2 data, tabulate and plot the average load voltage vs. the value of R_c.
6. Compute the SCR gate circuit R-C time constants. Is the turn-on delay governed by the time constant?
7. The circuit shown in Fig. 10 is similar to that used in incandescent light dimmers. Can you suggest other applications?
8. Could a battery be substituted for the load resistor in Fig. 10 to form a controlled charger?

ECE 469 — POWER ELECTRONICS LABORATORY

EXPERIMENT #2 — Ac-Dc Conversion, Part II: Single-Phase and Poly-Phase Conversion

Objective — This series of two experiments will examine the properties of controlled rectifiers, or ac-dc converters. The first experiment will concentrate on the single-phase (half-wave) circuit, while the second will examine two- and three-phase converters. Converter concepts such as source conversion and switch types will be studied. Popular applications such as battery chargers and dc motor drives will be introduced.

Pre-Lab Assignment — Read this experiment. Study the procedure, and bring any questions to class. Solve the following on a separate sheet, for submission as you enter the lab. The dc motor can be modeled as a constant voltage proportional to speed. For this particular motor, the constant is 50 V per 1000 RPM. Please neglect any series inductance. Your instructor may elect to assign another problem.

1) A dc motor control circuit is shown in Fig. 1. It is drawing an average current of 1 A. Study it, and answer the following.

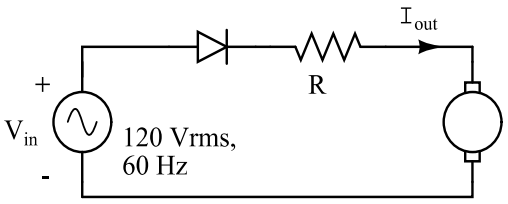


Figure 1. Motor supply circuit for pre-lab assignment

a) Sketch the waveform for the motor current.

b) Find the motor speed.

c) Now, given the motor speed, refine your current waveform to make it an accurate plot.

d) Repeat a), b) and c) using a four-diode bridge instead of a single diode.

Discussion —

Introduction: In Experiment #1, you had a chance to become familiar with the basic action of the SCR, and also studied a number of non-resistive rectifier load circuits. The SCR was seen to support control, since it can block forward voltage until commanded to turn on. In today's experiment, the lab SCR boxes will be used to study controlled-rectifier action in more depth.

The 120 V ac waveform available to us in the lab (or the 25 V ac waveform at our transformer output) is a good voltage source. It provides a consistent sinusoidal ac voltage at current levels from 0 up to many Amperes. In a switched converter network, KVL will not allow us to connect this ac voltage source directly to a dc voltage

source. Thus, if the objective is to obtain a controllable dc voltage, the ac source must be converted into current or buffered with some other part before it can be otherwise processed. Many types of loads act to provide current conversion. The source conversion concept is one subject of today's work.

Basic Theory - Single-phase ac-dc conversion: Switching networks allow efficient conversion of electrical energy among various forms, but one disadvantage is that the KVL restriction does not allow direct energy conversion between different voltage sources. Similarly, energy cannot be transferred directly among different current sources by a switch network. To deal with these constraints, it is necessary to provide intermediate conversions. For example, energy from a voltage source can be transferred to a current source, and then on to another voltage source.

A resistive load is a trivial example of current conversion: the incoming voltage produces a proportional resistor current. This is a far cry from a current source, but it is not a voltage source, and it does result in a conversion function. An inductive load is a better conversion example: the inductor voltage $v_L = L\frac{di}{dt}$ resists any change in current. If *L* is very large, any reasonable voltage will not alter the inductor current, and a current source is realized. A capacitive load has the opposite behavior. The capacitor current $i_C = C\frac{dv}{dt}$ responds whenever an attempt is made to change the capacitor voltage. If *C* is very large, no amount of current will change the voltage, and a voltage source is realized.

In Experiment #1, you saw three types of rectifier behaviors. With a resistive load, the rectified waveform has a dc component with substantial ac components as well. With an inductive load, the current is closer to a fixed dc value. With a capacitive load, current is transferred in large spikes during a fraction of each cycle — a behavior which intensifies noise as well as wear and tear on the electrical parts. The inductive circuit came closest to the desired current source conversion. Many electrical loads, especially motors, are inductive. As a result, most circuits behave in such a way that current does not change much over very short periods of time. In power electronics practice, this behavior, along with the desired source conversion function, means that most loads are treated as “short-term” current sources.

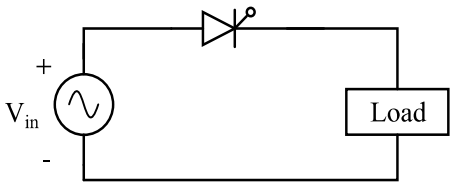


Figure 2. Basic single-phase controlled rectifier

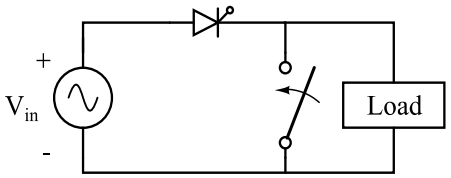


Figure 3. Single-phase converter with second switch for KCL

The basic single-phase controlled rectifier is shown in Fig. 2. If the load is inductive, there is no KVL problem. But there is a KCL problem: the switch cannot be turned off if the load is inductive. To see that this is so, observe what happens to $L\frac{di}{dt}$ when an attempt is made to turn the switch off. The current must approach zero almost instantaneously. The value of $L\frac{di}{dt}$ is a huge negative voltage. In practice, this voltage will likely exceed the blocking capabilities of the switch, which will be damaged. A solution is to provide a second switch across the load, as shown in Fig. 3. In simple ac-dc converters, this switch can be a diode, or it can be a second bidirectional-blocking forward-conducting device. Any load which resembles a current source can be taken into account by addition of this second switch.

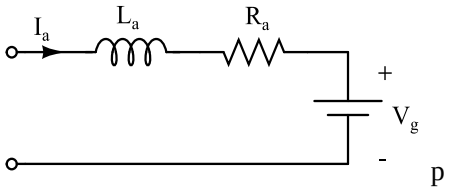


Figure 4. Basic dc motor circuit equivalent

Two popular loads for ac-dc converters are the dc motor and the rechargeable battery. The dc motor has an inductive model, as shown in Fig. 4. The motor looks rather like a voltage source, but its windings provide series inductance. This combination has many current-source properties.

A battery provides a good dc voltage source. To support the desired energy conversion function, something must be placed in series with the battery for current conversion. A resistor can be used, as in the Pre-Lab, with a loss in efficiency. Alternatively, an inductor can be used, as in the circuit in Fig. 5. Analysis of this circuit is not trivial. The controlled switch is on whenever V_{in} is positive. When it is on, the output current is given by

$$\frac{di_L(t)}{dt} = \frac{V_{in} - Ri_L - V_{battery}}{L}$$

When the controlled switch is off, inductor current is

$$\frac{di_L(t)}{dt} = \frac{-Ri_L + V_{battery}}{L}$$

If L is large, the current is approximately constant. These differential equations can be solved to give actual values of current. Charging current is controlled by the resistor value and also by the phase delay used to operate the SCR.

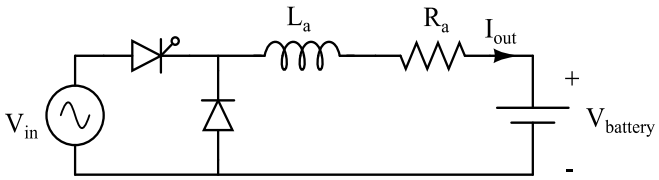


Figure 5. Single-phase battery charger with inductive filter

The form of the circuit now expected for single-phase conversion appears in Fig. 6. The switches are implemented with an SCR and a diode. In this case, the output voltage is V_{in} whenever the SCR is on, and zero when it is off. The SCR turns off automatically when $V_{in} \le 0$. The voltage waveform is a familiar one: a half-wave rectified waveform, possibly with a delayed turn-on. The voltage has an average value given by

$$V_{out(ave)} = \frac{1}{2\pi}\int_{\alpha}^{\pi} \sqrt{2}V_{ac(RMS)} \sin\theta \, d\theta$$

where α is the angle of delay before the SCR is turned on. This integral can be computed to give

$$V_{out(ave)} = \frac{\sqrt{2}V_{ac(RMS)}}{2\pi}(1 + \cos\ \alpha)$$

Since the output is a dc current source, average power exists only at dc, and is simply $I_{out} \cdot V_{out(ave)}$.

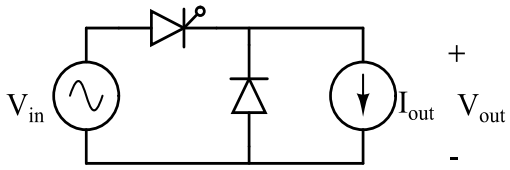


Figure 6. A single-phase ac voltage to dc current converter

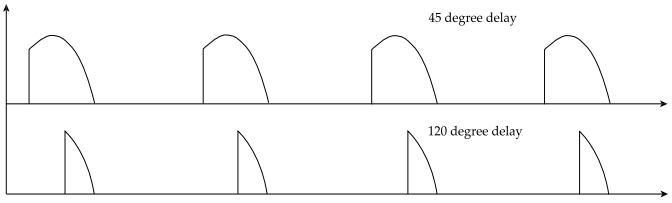


Figure 7. Typical output voltage waveforms

Basic Theory - Poly-phase ac-dc conversion: When multiple input sources are available for ac-dc converters, it is natural to use all of them. The circuit in Fig. 2 shows the most general of such converters — an *m*-phase to dc converter. Today, we will examine a slightly simplified version of this converter — one in which there is a common neutral connection between input and output. This is called *a midpoint* converter, and appears in Fig. 8.

In the midpoint converter, the KVL and KCL restrictions are reasonably clear: no more than one switch may be on at any time, and one switch must be on if the load current is not zero. This implies that $\Sigma\, q_i \le 1$. If the load is a current source, $\Sigma\, q_i = 1$, and the switching functions are said to form a "complete set." We want to operate the switches so that the dc value is maximized and the unwanted ac components are minimized. It can be *proved* that if the switching frequency is chosen to equal ω_{in}, the best choice of switching functions is to follow the polyphase input: each switch is on 1/*m* of the time, and switching functions are spaced 360°/*m* apart. The dc output component can be controlled by adjusting the phase of the switching functions. The output voltage is given by $V_{out} = \sum_{i=1}^{m} V_i h_i$.

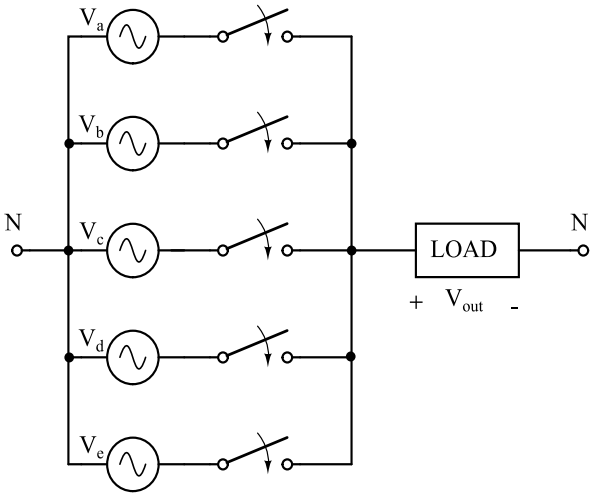


Figure 8. Polyphase midpoint ac-dc converter

This gives a complicated series, in the form

$$V_{out} = \left(\frac{1}{m} + \frac{2}{\pi} \sum_{n=1}^{\infty} \frac{\sin\left(n\pi/m\right)}{n} \cos\left(n\omega_{in}t + n\phi_0\right) \right) V_0 \cos(\omega_{in}t)$$

$$+ \left(\frac{1}{m} + \frac{2}{\pi} \sum_{n=1}^{\infty} \frac{\sin\left(n\pi/m\right)}{n} \cos\left(n\omega_{in}t + n\phi_0 - \frac{2\pi n}{m}\right) \right) V_0 \cos\left(\omega_{in}t - \frac{2\pi}{m}\right)$$

$$+ \left(\frac{1}{m} + \frac{2}{\pi} \sum_{n=1}^{\infty} \frac{\sin\left(n\pi/m\right)}{n} \cos\left(n\omega_{in}t + n\phi_0 - \frac{4\pi n}{m}\right) \right) V_0 \cos\left(\omega_{in}t - \frac{4\pi}{m}\right)$$

+...

The dc component appears in the *n*=1 term, and is given by

$$V_{out(ave)} = \frac{2}{\pi}\sin\left(\frac{\pi}{m}\right)\frac{V_0}{2}\cos(\phi_0) + \frac{2}{\pi}\sin\left(\frac{\pi}{m}\right)\frac{V_0}{2}\cos(\phi_0) + \ldots$$

This, in turn, can be simplified to give

$$V_{out(ave)} = \frac{m}{\pi}\sin\left(\frac{\pi}{m}\right)V_0\cos(\phi_0),\ m \geq 2$$

Notice that $V_{out(ave)}$ depends on ϕ_0, where ϕ_0 is defined as the delay angle between any voltage and the switching function associated with it. The result is a controlled rectifier. Unwanted components appear at frequencies that are multiples of $m\omega_{in}$. This is a better situation than the single-phase case, which had a lower dc component and large unwanted components at multiples of ω_{in}. The unwanted components can be reduced by increasing *m*. In large power system applications such as high-voltage dc transmission, as many as 48 phases are used at the converter input, to reduce unwanted components.

In this experiment, the two-phase source will be obtained by means of a center-tapped transformer (the one-phase/two-phase switch setting on the bench front panel). The standard 120 V ac line is supplied to the transformer, and 25 V direct and inverted outputs are provided. The SCR switching functions must be spaced 180° apart ($\frac{1}{2}$ cycle) in order to best effect conversion. The three-phase source is stepped down from laboratory power. In this case, the SCR control box will need to be adjusted to provide the necessary 120° delays, based on phase A for a reference.

Procedure —

Part 1: Ideal R-L load

1. Connect phase "A" of the nominal 25 V 60 Hz supply, the SCR labeled "A" in the SCR box, and a resistive load, as in Fig. 9. Estimate the required resistor power rating, and abide by it. Remember that the source neutral is grounded. Record the actual RMS value of the source voltage.

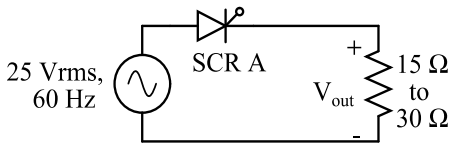


Figure 9. SCR test circuit, resistive load

2. Observe the output voltage waveform for SCR master delays corresponding to approximately 0°, 45°, 90°, and 135°. *Sketch* just one or two typical waveforms, rather than the whole series. Measure $V_{out(ave)}$ and $V_{out(rms)}$ for all SCR delays. Note that the phase delay setting is not relevant for this experiment.
3. Place an inductor specified by your instructor in series with the resistor, as shown in Fig. 10. Record input

and output power so you can compute efficiency. Repeat #2 above for this new circuit, but include both the output and load voltage waveforms in your sketches and also measure $V_{load(rms)}$.

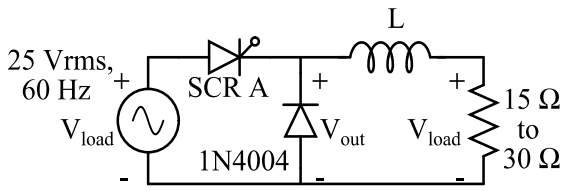


Figure 10. SCR test circuit, series R-L load

Part 2: Battery load

1. Connect a rechargeable battery to the output of the circuit in Fig. 11.

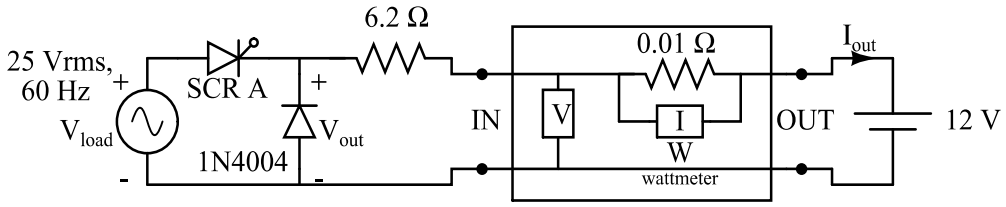


Figure 8. SCR test circuit with battery charger load

2. Measure the average battery current $I_{out(ave)}$ with the Fluke multimeter and the RMS current $I_{out(rms)}$ with the wattmeter for at least four different SCR delay angles. Record the master delay times. Observe and sketch the resistor current waveform at one intermediate SCR delay angle. Record information to permit computation of efficiency. What data will be needed?
3. Add an inductor in series with the load and then repeat step 2.

Part 3: Three-phase converter

1. Connect the 25 V 60 Hz supply and SCR box for three-phase operation as shown in Fig. 12. Plug the SCR box into the outlet labeled Phase A on the power panel at your bench. Adjust the phase delay to reflect $\frac{1}{3}$ cycle delay at 60 Hz. Turn on 3φ power.

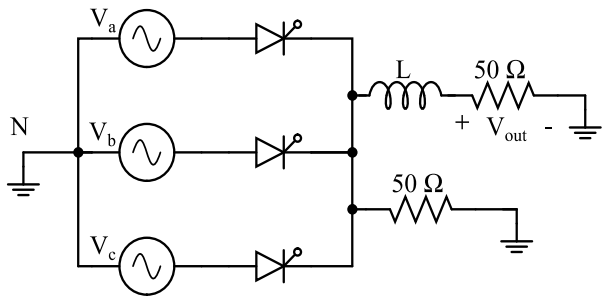


Figure 12. Three-phase SCR converter test circuit

3. Set the SCR master delay to zero (i.e. set it to act as a diode). Observe the output voltage waveform. Measure $V_{out(ave)}$ and $V_{out(rms)}$, and sketch the waveform.
4. Repeat #3 for master time delays corresponding to approximately 45°, 90°, and 135°. Sketch just one typical waveform, rather than the whole series.
5. If there is time, repeat #3 with a dc motor as the converter load, except set the master delay at about 120° initially. Observe and sketch the motor current and voltage waveforms with no motor shaft load.
6. Add some shaft load, and observe current and voltage waveform changes. Record your observations.

Study Questions —

1. For the loads of Part 1, tabulate and plot $V_{out(ave)}$ and $V_{out(rms)}$ vs. the SCR delay, computed in terms of angle. Compute a theoretical result, and compare it to the data. Do these agree?
2. For the battery load in Part 2, tabulate and plot $I_{out(ave)}$ and $I_{out(RMS)}$ vs. the SCR delay angle. Again consider whether your results are consistent with theoretical expectations.
3. Why is the “flyback” diode included in today's circuits?
4. Compute the efficiencies of the SCR circuit with R-L and battery charger loads.
5. Comment on how the diode and SCR forward voltage drops affect your results.

Optional Study Questions —

1. For each of the load and source combinations, tabulate and plot $V_{out(ave)}$ and $V_{out(rms)}$ vs. the SCR delay angle. What would you expect in theory? How well do your results agree with the theory?
2. For one delay angle (pick 45°, for instance), plot $V_{out(ave)}$ and $V_{out(rms)}$ vs. the number of input phases (you have data for one, two, and three). What do you expect to happen when more phases are used?

3. Why is the flyback diode *not* included in today's circuits?

4. A bridge converter, as in the pre-lab (Fig. 1), implements the full switch matrix in Fig. 2. How would $V_{out(ave)}$ be affected by the use of a bridge rather than a midpoint converter?

ECE 469 — POWER ELECTRONICS LABORATORY

EXPERIMENT #3 — Dc-Dc Conversion, Part I: One-Quadrant Converters

Objective — This series of two experiments will examine dc-dc conversion circuits and their major applications. Pulse width control is the basic technique for operating such converters, and this method will be used. The first experiment will examine most of the simple one-quadrant converters (buck, boost, buck-boost, boost-buck). The second will examine dc-dc converters for motor drives.

Pre-Lab Assignment — Please look at Figure 4 (Page 33 below). You will be building this buck converter. Please use the inductor and capacitors as given in the figure to calculate and sketch (a)–(d). Use the same input voltage as you will be using in this experiment: 100 kHz switching frequency and a duty ratio of the MOSFET of 0.3.10 Ohm load.

a) Please sketch the inductor current. Please calculate the expected ripple current through the inductor.

b) Please calculate the expected ripple voltage across the output capacitor (1 uF).

c) Please sketch the expected voltage across the diode (marked by Vout).

d) Please calculate the expected ripple voltage across the input capacitor (47 uF tantalum).

Discussion —

Introduction: The general two-input two-output switch network has four switches. It will provide dc-dc conversion if a dc source is used at the input and pulse width control is used to operate the switches. In this experiment, some common one-quadrant converters will be implemented and tested. Most commercial dc switching power supplies use one of these one-quadrant methods as their basis. The main tool of this experiment is our FET switch control unit. This box contains two sets of power switches — a pair of power metal-oxide-semiconductor field-effect transistors and a pair of diodes — along with switching function circuitry for the FETs. The circuitry provides direct front-panel control of the switching function frequency and duty ratio. The power devices in the FET control box are fully isolated to permit their use in any switching circuit.

The FET is a bidirectional-conducting forward-blocking switch, and so it can be used in place of the simple forward-conducting forward-blocking switch needed for one-quadrant dc-dc conversion. In this experiment, we will set up two one-quadrant circuits and characterize their operation. The converters we will build are fairly sophisticated, with fast switches and switching frequencies as high as 100 kHz.

Basic Theory: In theory discussions, dc-dc converters are depicted as providing energy transfer between two ideal sources. In practice, one of the sources is almost always implemented as an electrical energy storage element. For

example, the buck converter stores energy in an inductor when the controlling switch is on, and discharges that energy through the load when the controlling switch is off. The objective is to keep energy flow into the load nearly constant as the switches operate. Many converters have nearly constant voltages or currents for the storage elements.

The general dc-dc converter is shown in Fig. 1. Several one-quadrant dc-dc converters can be built from simplified versions of this matrix, and the four major ones are shown in Fig. 2. Notice that the combination converters (buck-boost and boost-buck) are really two matrices connected together. As long as the inductors and capacitors maintain non-zero voltage and current levels, KVL and KCL requirements must be accounted for in switch operation. When $V_C > 0$ and $I_L > 0$, the two switches must have $q_1 + q_2 = 1$, and $D_1 + D_2 = 1$.

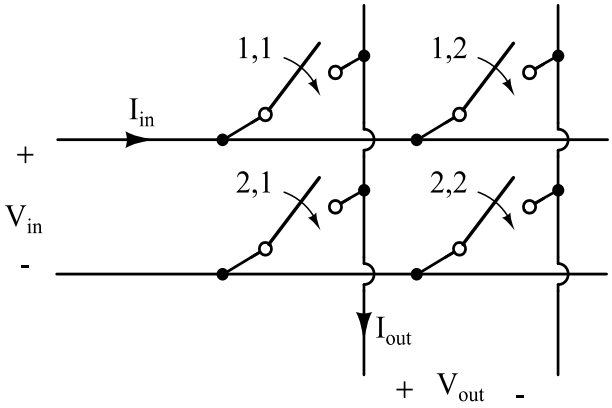


Figure 1. General dc-dc converter matrix

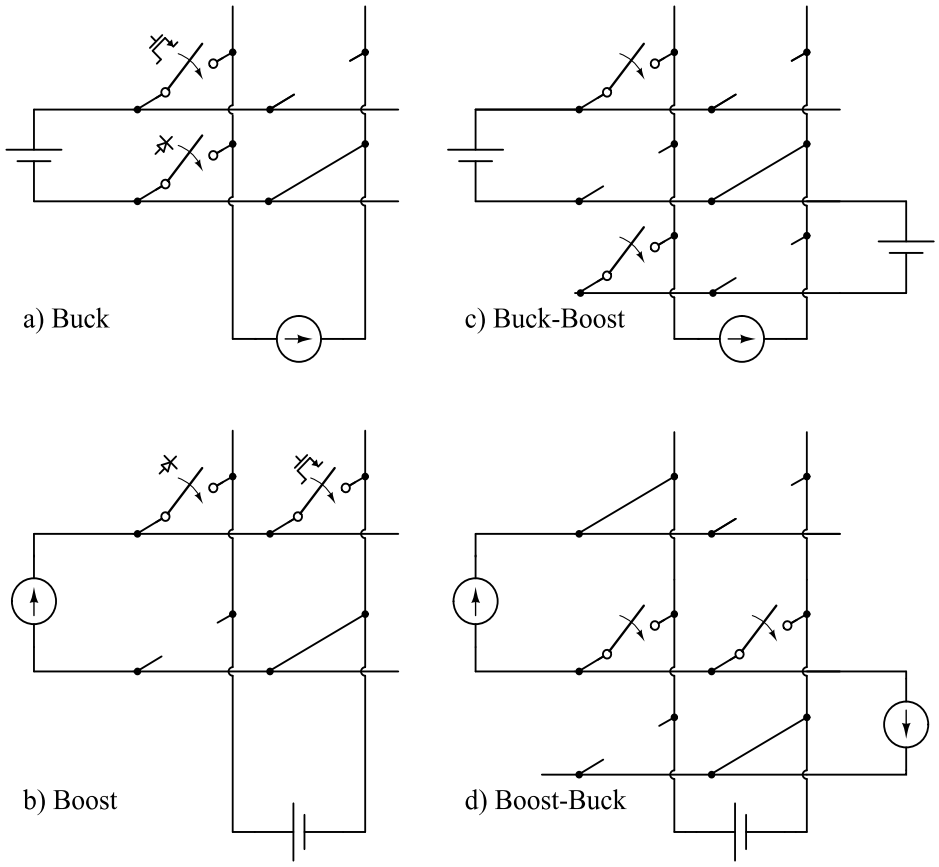


Figure 2. Major one-quadrant dc-dc converters

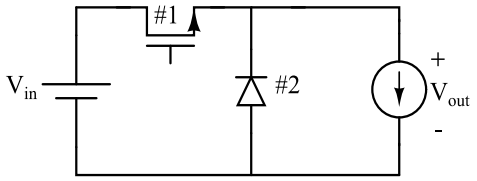


Figure 3. The buck converter

In every dc-dc converter, the output waveform is a scaled version of one or more switching functions. Average values are therefore determined by the switching-function duty ratios. For example, the buck converter shown in Fig. 4 has $V_{out} = q_1 V_{in}$, which in Fourier form is: $V_{out} = V_{in}\left[D_1 + \frac{2}{\pi}\sum_{n=1}^{\infty}\frac{\sin(n\pi D_1)}{n}\cos(n\omega_{switch}t + n\phi_0)\right]$.

The average value of this series is $D_1 V_{in}$. Unwanted Fourier components appear at the switching frequency f_{switch} and its multiples. The largest unwanted component is

$$\frac{2}{\pi}\sin(\pi D_1)\cos(\omega_{switch}t + \phi_0)$$

A special advantage of dc-dc converters is the arbitrary value of f_{switch}. This frequency can be chosen in any manner, as long as the desired duty ratio can be provided. Some common constraints used to select the switching frequency include:

— The speed of switch transition from on to off or off to on.

— Allowed size and weight of inductors and capacitors.

— Sensitivity of the load or source to ripple voltages and currents.

— Radio-frequency interference caused by high-frequency components.

Modern dc-dc converters are implemented with GTOs (Gate Turn-Off thyristors) and FETs. Each of these has a preferred switching frequency:

— GTO-based dc-dc converters (used for dc motor drives and very high power conversion): up to several kilohertz

— FET-based dc-dc converters: up to 2 MHz or more.

— FET-based "resonant" dc-dc converters: up to 20 MHz or more.

In general, as switching frequencies increase, the values of *dv/dt* and *di/dt* values also increase, and smaller inductors and capacitors can be used without sacrifice in performance.

Procedure —

Part 1: Buck converter

1. Set up the FET control box as a buck dc-dc converter as shown in Fig. 4. Use the left devices. The red jack is the FET drain lead (the "forward" switch terminal). Be sure to provide the capacitor shown across the input power supply. The capacitor should be placed as close to the FET box as possible so that the inductance of the wires to the FET will be very low. The HP 6060 electronic load can be used as appropriate.

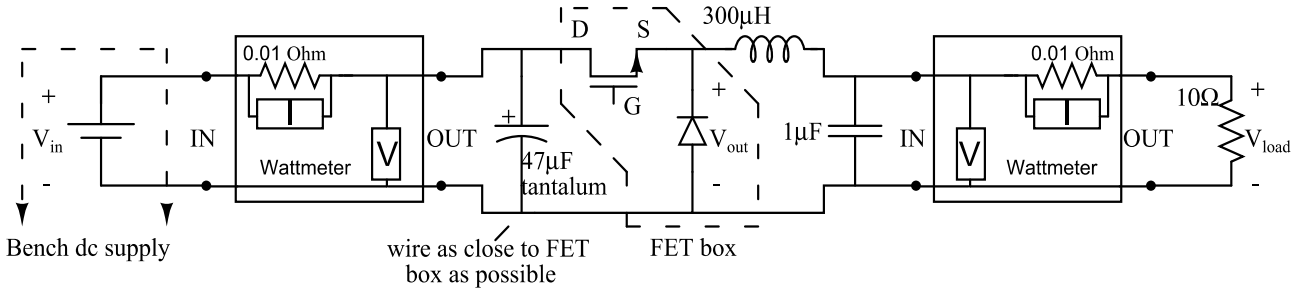


Figure 4. Buck converter test circuit

2. Set the power supply current limit for about 3 A, and the voltage to 12 V. Turn on the FET box, monitor q(t), and set the duty ratio to 50%. Set f_{switch} to about 100 kHz. Connect oscilloscope probes to measure V_{load} and V_{out}. The oscilloscope input coupling should be set to "dc."
3. Turn on the input power supply. Notice that the V_{out} waveform allows easy measurement of the switching function. Confirm that the duty ratio is close to 50%. Sketch the V_{out} and V_{load} waveforms.
4. Use the oscilloscope to measure the peak-to-peak ripple on V_{load} at duty ratios of 10%, 50%, and 90%. Measuring directly at the capacitor leads may provide cleaner measurements. The ripple measurement can be performed by setting the oscilloscope input coupling to "ac," and expanding the voltage scale. Do not forget to return the coupling to "dc" when you finish the ripple measurement.
5. Measure average values of V_{load} and I_{in} at duty ratios of close to 10%, 30%, 50%, 70%, and 90%. Use your multimeter for $V_{load(ave)}$. The meters on the Kenwood power supply monitor average values. Record the duty ratio, as well as the input and load RMS voltages, currents, and powers from the wattmeters.
6. Change f_{switch} to 5 kHz. Observe the inductor current rise and fall. Note the current value at a few points in the waveform. The objective is to determine di/dt as the basis for estimating the inductance in your report.

Part 2: Buck-boost converter

1. Set up the FET control box as a buck-boost dc-dc converter, as shown in Fig. 5. Again, remember that the red jack is the FET drain lead.
2. Set the power supply current limit for about 3 A, and the voltage to 12 V. Set the FET box duty ratio dial to the zero position. Connect oscilloscope probes across the load resistor and across the inductor.

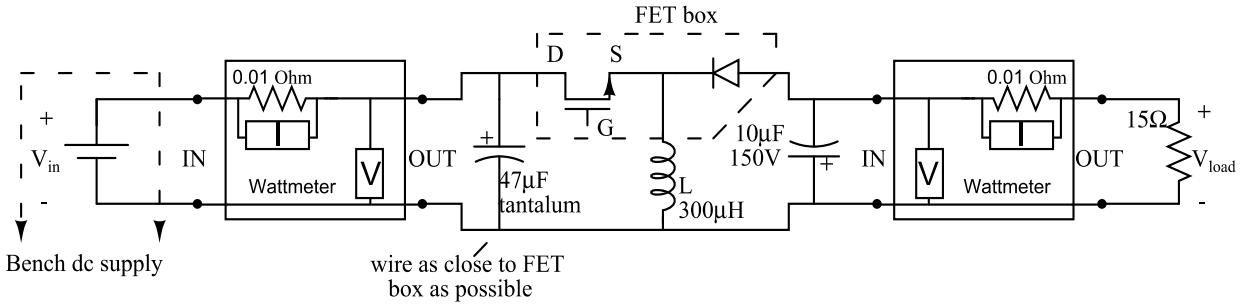


Figure 5. Buck-boost test circuit

3. Turn on the FET box and the input power supply. Set f_{switch} to about 100 kHz.
4. Set the duty ratio to about 50%, and confirm this. Sketch the inductor voltage waveform. Use the current probe to sketch the inductor current waveform. Observe and sketch the load voltage waveform, as well as its ripple component using ac coupling.
5. Measure average and RMS values of V_{load} and I_{in} at duty ratios of close to 20%, 40%, 50%, 60%, and 70%. Confirm whether or not the output polarity is reversed. Record power readings as well. **The output voltage becomes high quickly with D > 60%, so be careful.**

Study Questions —

1. Tabulate your data in an organized fashion. Compare the RMS readings from the wattmeters with the various average readings. Do they agree?
2. Compute and tabulate ratios of $V_{load(ave)}/V_{in}$ for these converters. Are the results consistent with the duty ratio settings?
3. Estimate the *average* input and output power from the *average* readings of V_{load} and I_{in} for each operating condition. Compare these results to the wattmeter readings. Calculate efficiency, P_{out}/P_{in}, from the wattmeter readings.
4. Estimate the inductor value from the measurements of the current waveform in the buck converter. Use this value to compute an expected load voltage ripple at 100 kHz. How do your results compare with the data? If the inductor value were to double, how would this affect the behavior of these circuits?
5. What is the impact if a student tries to test a boost or buck-boost circuit under no-load conditions?

ECE 469 — POWER ELECTRONICS LABORATORY

EXPERIMENT #4 — Dc-Dc Conversion, Part II: Converters for Motor Drives

Objective — The second in a series of two experiments will examine the operation of dc-dc converters in the context of motor drives. Motor drives are typical applications of multi-quadrant dc-dc converters. Motors are not true current sources. Some types of switching motor drives take advantage of this fact in their operation. Others are directly equivalent to the dc-dc converters studied so far.

Pre-Lab Assignment — Read this experiment. Study the procedure, and bring any questions to class. Solve the following on a separate sheet, for submission as you enter the lab. Your instructor may elect to assign another problem.

1) A dc-dc buck converter has a 48 V input. This converter is connected to a dc motor that spins at 2000 RPM given 24 V at its terminals, 1000 RPM for 12 V, and follows a linear speed relation at other voltages. The motor is characterized by a voltage source in series with an inductance and resistance. The resistance is 0.5 Ω. The inductance is 10 mH. The converter switches at 20 kHz. At 2000 RPM with no mechanical load, the motor draws 0.5 A.
   a) What is the no-load motor speed at 20% duty ratio? At 65% duty ratio?
   b) The buck converter has been set to operate the motor at 2500 RPM, when the FET abruptly turns off. Sketch the motor current after this turn-off. Where in the circuit does the current flow? When does it equal zero? (You may assume the motor EMF remains constant after the FET turns off.)

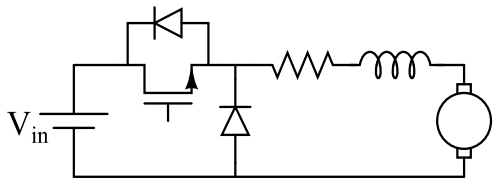


Figure 1. Dc-dc converter for pre-lab assignment

Discussion —

Introduction: One common example of the applications of dc-dc converters is the switching power supply, in which a single rectified waveform can be converted with minimal loss into a variety of dc voltages. This makes switching dc supplies common in computer systems, where three or four different dc voltage levels are often needed. Dc power supplies are usually always based on one-quadrant circuits: the output power and current are positive at all times. A second major application of switching dc supplies is the dc motor drive. In this case, dc power from some source (perhaps a rectifier) is to be controlled in order to achieve a mechanical function. Converters or other electric controllers used for such applications are called *motor drives*, and dc-dc converters

are often called *chopper drives* in this context. Motor drives range from units rated at a few watts (for motors in printers, computer disk drives, and CD players) to units rated at thousands of horsepower (used in locomotives, ships, and large pumps).

In large dc motor drives, the rotating kinetic energy of a large motor is considerable, and in a braking situation, this energy must be removed. The energy can be converted to heat and lost, as it is in gasoline engine vehicles, or it can be directed back to the energy source for recovery. This "regeneration" energy can be controlled only with multi-quadrant circuits. Multi-quadrant dc-dc converters are the main topic of this experiment.

Basic Theory: The equivalent circuit of a dc motor is repeated in Fig. 2. Rotation of the motor produces a "back EMF," equal to $k\omega i_f$, where k is some constant, ω is the motor shaft speed in rad/s, and i_f is the *field exciting current* or some other variable representing the magnetic field strength inside the motor. Note that this is not the same as the armature current, i_a, which flows through the motor terminals. The back EMF, symbol V_g, appears in series with the inductance and resistance of the motor windings. If a voltage V_t is applied to the motor terminals, a current $(V_t - V_g)/R_a$ will flow in the steady state. This current sends power into the motor, which is converted to mechanical energy. The input power accelerates the motor until the electrical input power exactly balances any mechanical energy delivered to a shaft load.

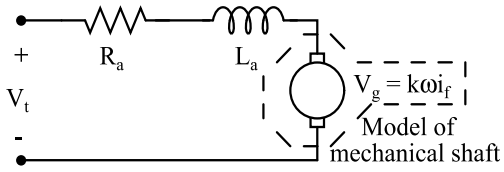


Figure 2. The dc motor equivalent circuit with mechanical model

The power input, and hence the operating speed, of a dc motor can be controlled by altering the value of V_t. Since the actual output power is into some mechanical load, with its associated inertia, ω will not change much over short time periods, and V_g will be nearly constant. This is a good match for a dc-dc converter, since the average electrical power into the motor will be determined by the average value of V_t, even when the inductance is low. For example, a simple buck converter could be used to control V_t, as in Fig. 3. The output could vary from 0 to 100% of some input dc source voltage, and thus speed could be altered from near 0 to near rated speed.

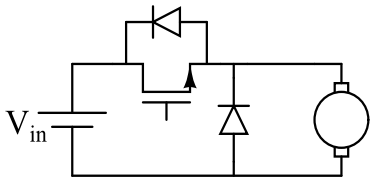


Figure 3. Buck converter in a dc motor drive application

If i_f is held approximately constant, as it is in "separately excited" motors or motors with permanent magnets, the input power is a very direct function of converter duty ratio, and speed control is made easy. In fact, many converters have feedback systems built in for this purpose. Consider the unit of Fig. 4, in which the transistor duty ratio depends on a low-power input voltage. This voltage could be generated, as shown in the figure, with some reference signal being compared to information from a tachometer. In this unit, the duty ratio automatically increases if the motor speed drops. This results in additional input power and is intended to keep running speed nearly constant. The operating speed can be changed easily by adjusting the reference voltage, and will be held constant at the desired value.

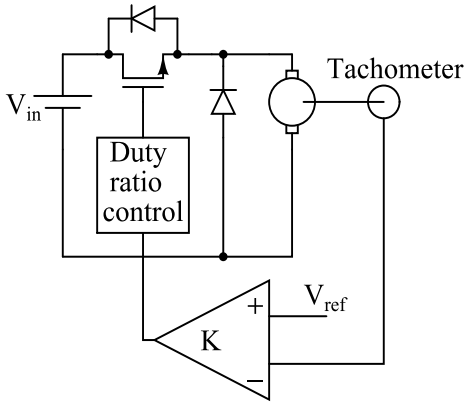


Figure 4. Dc-dc converter set up for speed control

In high power and traction motor drives, a considerable amount of energy is represented during a speed decrease. An example is the braking of an electric vehicle. The energy removed from a motor during deceleration is equal to the energy needed to accelerate the motor, at least in principle. In mechanical systems, this braking energy is converted to heat, dissipated, and lost. This is not an acceptable situation in battery-powered electrical systems. Multi-quadrant dc-dc converters can be used to convert the braking energy back into electrical form. The energy can be dissipated just as it would be by mechanical brakes, or recovered. The dissipation version is called *dynamic braking* and is often used for very rapid deceleration of electric motors. The recovery version is called *regenerative braking*, or just *regeneration*, since it uses the motor as a generator during deceleration.

The buck converter is sometimes called a *class-A chopper* when used as a motor drive [1]. This converter makes a useful, simple motor controller. When braking, the controlling switch turns off, and only the diode conducts. If inductance is low, the output current quickly goes to zero, and the motor merely coasts to a stop. If inductance is high, the braking energy is dissipated in R_a. Fig. 5 shows some typical waveforms. A boost converter can be used to return generated energy from a motor to a source, provided that the source voltage V_t is greater than V_g. Such a converter, shown in Fig. 6, can provide regenerative or dissipative braking. It is sometimes referred to as a *class-B chopper* [1].

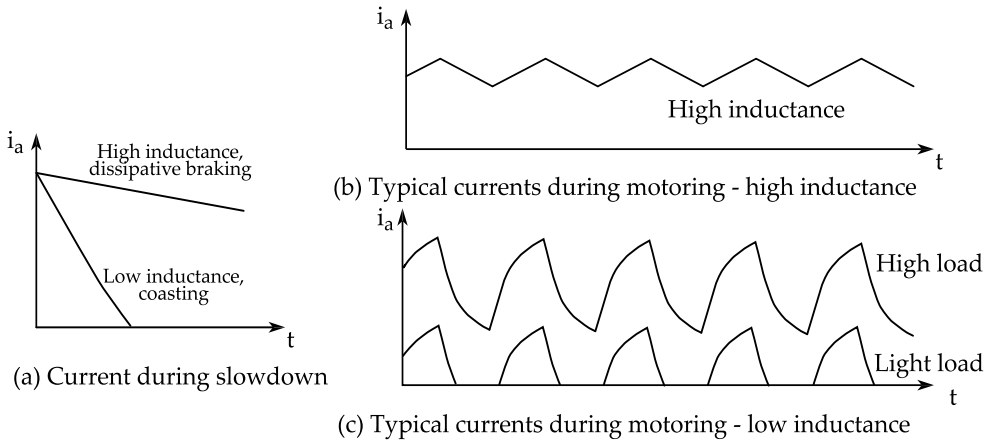


Figure 5. Buck converter current waveforms

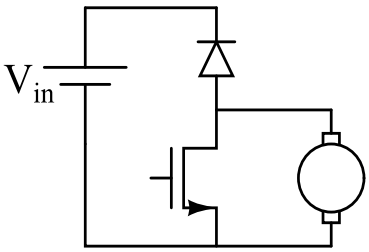


Figure 6. Boost converter for braking energy recovery

A motor could be attached to both a buck and a boost converter, with the buck converter operating (with $I_a > 0$) during "motoring," and the boost converter operating (with $I_a < 0$) during regeneration. Such a unit is called a *class-C chopper*, and is widely used for dc motor drives. In a class-C chopper, care must be taken to avoid turning on both active switches simultaneously, since this would short the input source. To operate a class-C chopper, the boost converter active switch is shut off, and the buck converter is used to accelerate and provide running power to the motor. When regeneration is desired, the buck converter is shut off until I_a becomes

negative. At that point, the boost converter can begin operating, and will transfer energy from the motor back to the source. In effect, the two converters operate independently.

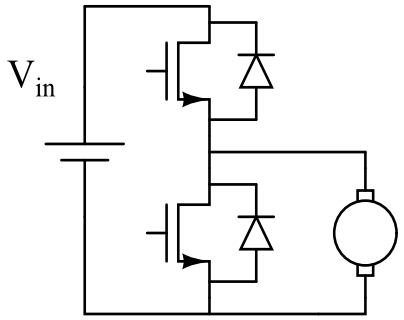


Figure 7. Class-C chopper circuit

The general dc voltage to dc current converter shown in Fig. 8 is not especially useful for dc motor control, since the inductance value is usually too low for useful regeneration. If a large inductor is placed in series with the motor, the circuit is capable of regeneration, since V_t can be negative with I_a positive. The converter cannot provide negative steady-state voltage to the motor, and so is no more useful than the class-C chopper above. It is called a *class-D chopper*.

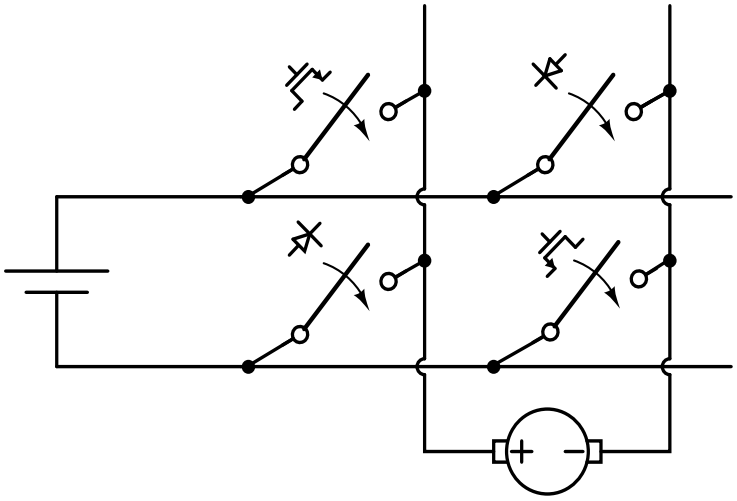


Figure 8. Two-quadrant dc-dc converter

In many applications, it is necessary to reverse the motor direction, while maintaining both motoring and regeneration. Two class-C choppers, one for each motor V_t polarity, can be assembled for this purpose. The resulting *class-E chopper* consists of four converters, each of which operates in only one quadrant. It is shown below.

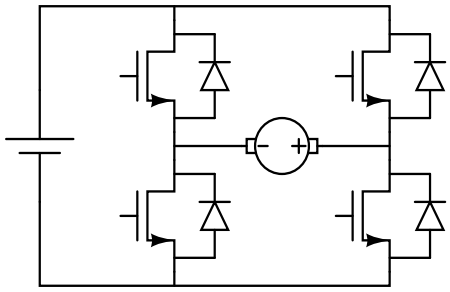


Figure 9. Class-E chopper circuit for bidirectional dc motor drive

The class-E chopper is more often termed an *H-bridge*, based on its shape. Low-power IC H-bridge devices are the basis of many computer motor drives. In practice, the most common multi-quadrant converters operate as if they are built up from multiple one-quadrant converters. It is possible to view ac-dc, dc-ac, and ac-ac converters as if they were multi-quadrant dc-dc units. We will carry this idea further in later experiments.

Procedure —

Part 1: Class-A (one quadrant) motor drive

1. Choose a dc motor from the lab selection. Use your multimeter to measure its R_a.
2. Set up the FET control box as a buck dc-dc converter, with a dc motor as the output load. Remember that the red jack is the FET drain lead (the "forward" switch terminal). Be sure to provide the capacitor and resistor shown across the input power supply. The resistor is intended to take up any regenerated energy.
3. Set the power supply current limit for about 5.0 A, and the voltage to 24 V (depending on the motor ratings). Set the duty ratio dial to zero. Connect an oscilloscope probe across the motor. Set up the current probe to measure the motor armature current.
4. Turn on the FET box and the input power supply. The motor voltage waveform gives you a way to observe switching frequency (how?). Set f_{switch} to about 20 kHz (a period of 50 μs). Monitor and record the power supply current and voltage; shut it down if anything unexpected appears.
5. Observe and sketch the motor voltage and current waveforms, as well as measure the average motor voltages and currents at: the lowest duty ratio for which the motor runs steadily, a duty ratio of about 50%, and a duty ratio of about 90%. In each case, measure the peak-to-peak ripple of I_a. Qualitatively observe the effects of shaft load on the current waveform.

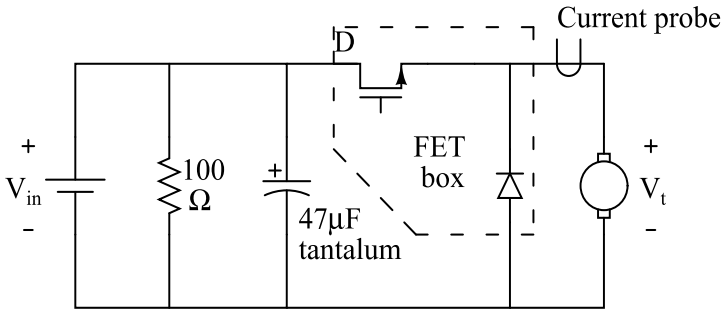


Figure 10. Class-A chopper circuit for test circuit

6. Change f_{switch} to 2 kHz. Take special care to record the value of voltage during intervals where $I_a = 0$. Take current waveform data that will allow you to determine the time constant of I_a during its fall. This will allow you to estimate L_a later.

Part 2: Class-C (regenerative) chopper circuit

1. The two active switches of the class-C chopper operate separately. Be aware that either one can be operated only when the other is off. The dual FET box can be configured to provide a "dead time" to avoid any possible overlap. This is done by setting the switching function of the second FET to the q' (note that there are **three** possible settings). Use both devices in your FET control box to set up the circuit in Fig. 11.
2. Use an input voltage of 12 V or 24 V (depending on the motor rating) and a switching frequency of 20 kHz. Observe operation at approximately 50% duty ratio. Sketch the motor voltage and current waveforms, and record the average motor voltage and current. Measure the peak-to-peak ripple on I_a.

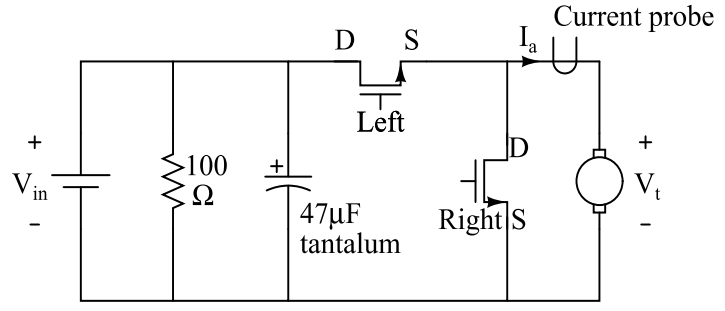


Figure 11. Class-C chopper test circuit

3. Increase the duty ratio to near the maximum amount; then drop the duty ratio abruptly in an attempt to observe converter action during regeneration. You may need to repeat this step, or spin the shaft externally

by hand or with another motor to observe regeneration behavior. Make note of your observations. Try to obtain a situation in which I_a reverses, in which case the machine is a generator.

Study Questions —

1. During intervals when $I_a = 0$, the motor voltage is not necessarily zero. Interpret its value during such intervals. (Hint: Since $I_a = 0$, this is an open-circuit voltage.)
2. Compute average power into the motor for each operating condition.
3. Estimate the motor series inductance from the buck converter data. Use this to estimate the current ripple at 20 kHz, 50% duty ratio. Does the calculation match your measurements?
4. Is the average motor voltage determined by the duty ratio?
5. Why is the class-A chopper incapable of regeneration? What will happen in such a converter if V_t is suddenly lowered?
6. What would happen if no dead time were present (i.e. what if the controls for the two switching devices in Part 2 were to overlap for a short time, such as 100 ns)?

References —

[1] S. B. Dewan, G. R. Slemon, A. Straughen, *Power Semiconductor Drives*. New York: John Wiley, 1984.

ECE 469 — POWER ELECTRONICS LABORATORY

EXPERIMENT #5 — Dc-Ac Conversion, Part I: Voltage-Sourced Inverters

Objective — This series of two experiments will examine inverter circuits. Inverters, or dc-ac converters, are used for generation of backup ac power, for ac motor drives, and for switching amplifiers. Commercial inverters are also used in alternative energy applications such as solar power. Uses of inverters are growing rapidly with expanded use of backup power, inverter-based amplifiers, and ac motor controllers.

Square-wave (voltage-sourced) inverters will be studied in this first experiment of the series. Emphasis will be placed on phase control techniques and resonant filtering.

Pre-Lab Assignment — Read this experiment. Study the procedure, and bring any questions to class. Solve the following on a separate sheet, for submission as you enter the lab. Your instructor may elect to assign another problem.

1. The half-bridge converter shown in Fig.1 has a square waveform for V_{out}. To avoid any dc component in V_{out}, the two switches are operated with a duty ratio of 50% at a frequency equal to the intended ac output. The capacitor bridge also prevents any dc component from flowing in the output. The intended output frequency is 50 Hz. Select L to make this load resonant at 50 Hz, so just a single harmonic flows.

   a) Sketch the voltage waveform across the load resistor.

   b) Sketch the voltage and current waveforms for either of the two large capacitors in the divider network.

   c) Can the average output power be varied by adjusting the phase or duty ratios of the switching functions? Why or why not?

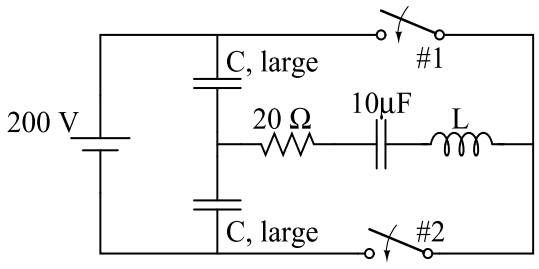


Figure 1. Dc-ac converter circuit for pre-lab assignment

Discussion —

Introduction: Dc-ac converters, or inverters, can be classified according to the properties of the dc source. Those with a dc current source are often used for sending energy into an ac power system. Dc power regeneration and HVDC transmission are two examples. Such inverters blur distinctions between rectification and inversion. In

practice, the only difference between rectifiers and inverters when dc current sources are involved is the direction of any unidirectional switches used in the circuits and the current flow direction.

Ac-dc/dc-ac converters which involve dc voltage sources are more common and operate differently from the current-sourced versions. These *voltage-sourced inverters* imply an ideal ac current load — rare in practice. Normally, the "ac current sink" is instead a model for an electrical load such as a motor. In an energy sense, these loads are *passive*: they consume energy delivered to them, and can be represented as an equivalent load impedance under steady-state conditions. A passive load just follows its input waveform, and do not establish a time of reference for phase purposes. The phase just leads or lags the input voltage by a certain amount. Without a phase reference, there is nothing to be gained by adjusting the phase of the switching function set, so that approach to control is not available. An alternative is to use one of the switching functions as a phase reference and then adjust others relative to it. This gives rise to *relative phase control*, which is used in some backup systems. Today's experiment will consider the basic square-wave inverter, widely used for backup power and general ac power needs.

Inversion is important in three general contexts:

1. Transfer of energy from some dc source into an ac power system. Applications include dc motor regeneration and alternate energy source conversion. Typical examples are inverters for solar power or fuel cell power.
2. Backup ac power. Since most electrical equipment is intended for ac power from a utility, switching power converters which produce ac from storage batteries are of growing importance.
3. Ac motor drives. Induction motors, synchronous motors, and so-called brushless dc motors require ac power at variable frequency. An inverter with adjustment capability is essential for ac motor drive applications.

The first of these is usually addressed with current-sourced inverters or with voltage-sourced inverters that apply relative phase control. In the current-sourced case, series inductance is added on the dc side, and the ac utility line provides an absolute phase reference. Phase advance or delay is used, and the control is just like that in a rectifier. In the second application, square-wave inverters (perhaps with added filters) often serve the purpose. There are somewhat more sophisticated backup inverters that create better waveforms, but square-wave circuits remain the most common. In the third case, we must be able to create a good sinusoid with full control over amplitude and frequency. Next week's experiment will consider PWM methods for this purpose.

Basic Theory: The most general dc-ac converter (based on single sources) is shown in Fig. 2. It has at most four switches. The switching functions are constrained by KVL and KCL as usual, but they must also have a Fourier component at the output frequency. An important practical consideration is that most ac utility sources involve transformers and cannot tolerate a dc component. This further constrains the functions. An obvious choice is to operate the switches in pairs, in a manner similar to the two-quadrant buck or boost dc-dc converters. The switching functions are given a duty ratio of 50% for symmetry, to avoid any dc component in the ac output.

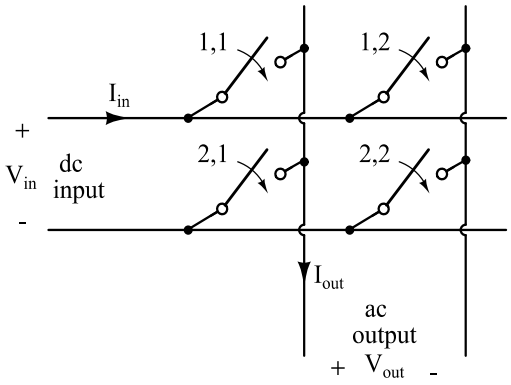


Figure 2. Basic dc-ac converter

The dc voltage to ac current case is shown in Fig. 3. As in a two-quadrant buck converter, V_{out} can be either V_{in}, $-V_{in}$, or 0, depending on the switch combination. Notice the switch types to be used in this converter. The devices must support current in either direction, although there is only one voltage polarity.

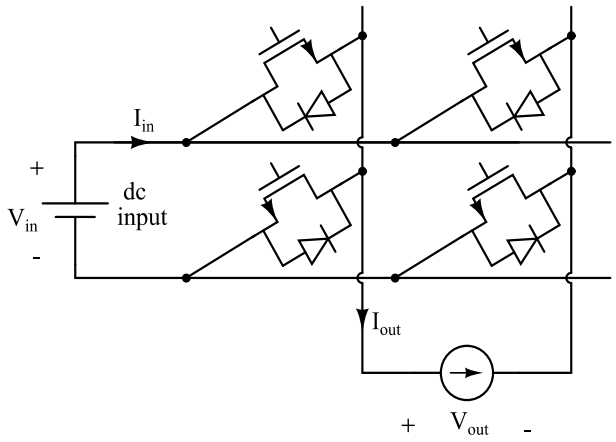


Figure 3. General voltage-sourced dc-ac converter

An induction motor equivalent circuit is shown in Fig. 4. This circuit acts as an inductive load. Since the current cannot change instantaneously, a current source is an appropriate model over short times. Nonetheless, the phase of this current source is linked to the phase of the input voltage. Changing the phase of the input voltage will change the current phase but not alter power. This is a reminder that an absolute phase control approach does not apply to this type of load.

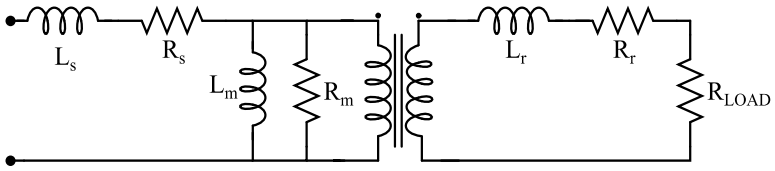


Figure 4. Induction motor equivalent circuit for each phase

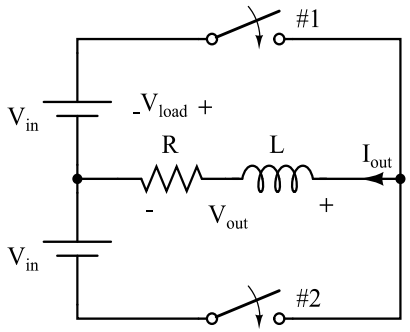


Figure 5. Half-bridge inverter with R-L load

Because of the ac current source phase behavior, properties of the load are important to converter operation. A half-bridge converter with an R-L load is shown in Fig. 5. KVL and KCL require $q_1 + q_2 = 1$. The practical requirement of no dc component in V_{out} can be met if $D_1 = D_2 = ½$. V_{out} is a square wave at the desired frequency, and has a value of $V_{out} = (2q_1 - 1)V_{in}$. This generates the Fourier series: $V_{out} = \frac{4V_{in}}{\pi}\sum_{n=1}^{\infty}\frac{\sin\left(n\pi/2\right)}{n}\cos\left(n\omega_{out}t - n\phi_0\right)$

Output current and power depend on the load, in a manner that can be computed term-by-term based on $V_{out} = RI_{out} + L\frac{dI_{out}}{dt}$. At steady state, I_{out} is periodic at the same frequency as V_{out}, and has a Fourier series: $I_{out} = c_o + \sum_{n=1}^{\infty} c_n \cos\left(n\omega_{out}t + \theta_n\right)$ This series can be differentiated. Let $X_n = n\omega L$. Then we can solve for I_{out}, with the final result:

$$I_{out} = \frac{4V_{in}}{\pi}\sum_{n=1}^{\infty}\frac{\sin\left(n\pi/2\right)}{n\sqrt{R^2 + X_n^2}}\cos\left(n\omega_{out}t - n\phi_0 - \tan^{-1}\left(\frac{X_n}{R}\right)\right)$$

This process can be automated, and the current at any point in time can be found by adding several terms in the series. A sample current waveform is given in Fig. 6. Some loads of this type will be tested in this experiment.

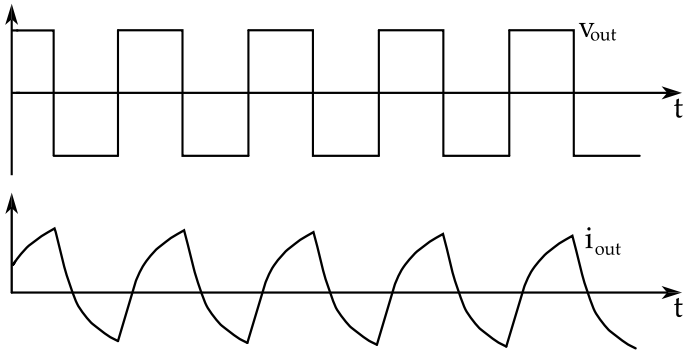


Figure 6. Typical output waveforms of voltage-sourced inverter

Procedure —

Part 1: Voltage-sourced inverter, R-L and R-L-C loads

1. Set up a FET control box with a short dead time between the devices to form a half-bridge inverter. Use a single supply with a capacitor bridge as depicted in Fig. 7. Do not confuse the FET drain and source leads.

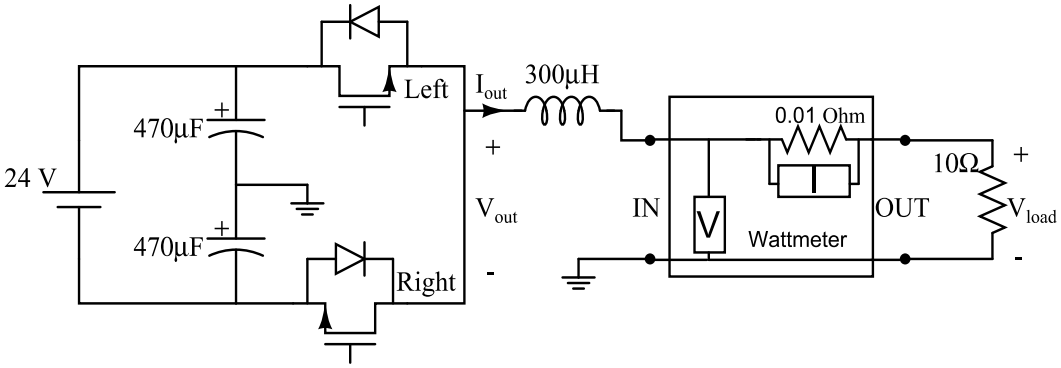


Figure 7. Voltage-sourced inverter test circuit

2. Set the power supply current limit for about 2.0 A. Set the duty ratio dial to 50%. Connect oscilloscope probes across the load resistor and converter output, as shown in Fig. 7.
3. Turn on the FET boxes and the input power supply. Set f_{switch} to about 10 kHz.

4. Measure the output average voltage. Is it zero? Adjust the duty ratio to get zero average output. *Small average offsets can swing the current to the power supply current limits.* Some supplies must be shut off to reset if current limits are reached.
5. Sketch the load resistor voltage and the output voltage waveforms. Record the RMS voltage, current, and power from the wattmeter. Measure the average input current from the supply.
6. Compute the series capacitance needed for resonance with the inductor at the switching frequency. Add this capacitor in series with the inductor. Adjust the frequency as needed to obtain near-resonant operation.
7. Sketch the resistor voltage waveform with the R-L-C load. Record the RMS value of the load resistor voltage, the output power, and the average input current from the supply.

Part 2: Isolated converter with ac link (half-bridge forward converter)

1. Set f_{switch} to 50 kHz. Wire a toroidal transformer into the circuit shown in Fig. 8.

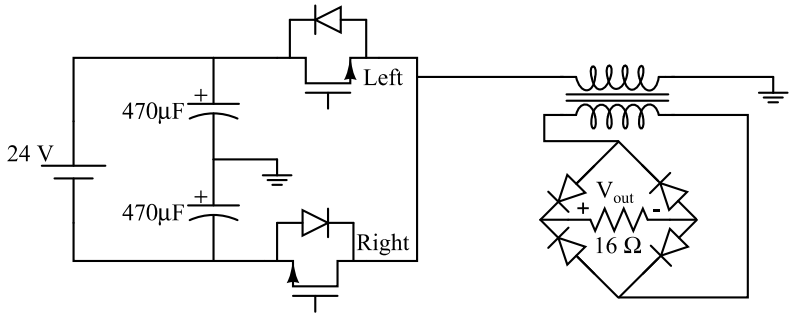


Figure 8. Isolated ac-link converter

2. Measure the average output voltage of this circuit. Sketch the waveform.
3. Place a 1 μF capacitor across the output terminals. Again measure the average output voltage. Also measure the average input current.

Study Questions —

1. How do you expect waveforms for an R-L load to change with frequency?
2. An alternate method for control of $V_{out(wanted)}$ is to add a third switch across the output combination. This allows zero as a possible output value. The duty ratios D of the first two switches can then be adjusted so that V_{out} changes. Assuming that the dc component remains at zero, find $V_{out(wanted)}$ as a function of D for this control scheme.

3. What is the effect of a resonant load, such as that in Part I?
4. Why is dead time required in these applications?
5. What is the efficiency of the converter circuits tested above?

ECE 469 — POWER ELECTRONICS LABORATORY

EXPERIMENT #6 — Dc-Ac Conversion, Part II: Pulse Width Modulation Inverters

Objective — The second in the series of inverter experiments involves pulse-width modulation circuits. The intent is to gain familiarity with the waveforms and concepts of pulse-width modulation as it applies to power electronics. PWM inverters are quickly becoming the method of choice; their use is likely to expand. The PWM inverter will be examined in the context of an ac motor drive. Basic ac drive concepts will be one feature of this experiment.

Pre-Lab Assignment — Read this experiment. Study the procedure, and bring any questions to class. Solve the following on a separate sheet, for submission as you enter the lab. Your instructor may elect to assign another problem.

The PWM waveform shown in Fig. 1 has a switching frequency of 5000 Hz and a modulating frequency of 200 Hz. It is applied to a series R-L circuit with $R = 8\ \Omega$ and $L = 10$ mH. Sketch the expected steady-state current waveform. You are not expected to compute it in detail.

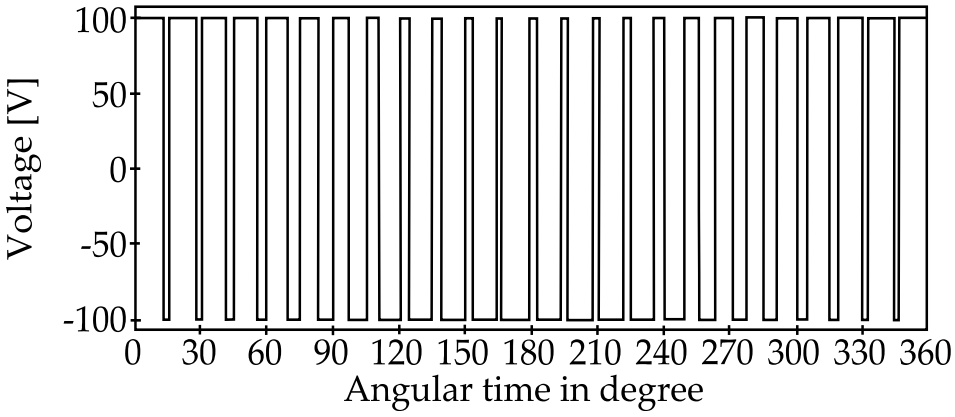


Figure 1. Sample PWM waveform for pre-lab assignment

Discussion —

Introduction: Control of output voltage in dc-ac converters is critically important in ac motor drives, and is very useful in all inverter applications. With output control, the converter can compensate for input changes or allow the user to set a desired output.

There are two methods in common use for controlling the output voltage (and power) of an inverter. The first is to use "relative phase control," in which the two half-bridges in a full-bridge inverter are adjusted in phase relative to each other. This approach is common in high-power inverters and is also used in low-cost low-power inverters. The drawback is that there are significant unwanted harmonics in the output. In fixed-frequency applications, these can be addressed with resonant filters, as in Experiment 6. Relative phase control permits

slow switches, such as SCRs, to be used in inverters. The switching frequency is the same as the intended output frequency.

The second method, pulse-width modulation (PWM), decouples the switching function frequency and the intended output. This method can provide the desired results if the switches operate much faster than f_{out}. Modern FETs, IGBTs, and GTO thyristors have made PWM possible in nearly all contexts. To understand PWM, consider a very fast square wave. The duty ratio can be varied slowly between 0 and 100%, in a manner similar to pulse width control in dc-dc converters. In a buck converter, for example, this would have the effect of slowly changing $V_{out(ave)}$. This slow change can be sinusoidal. V_{out} can change at the rate of 60 Hz if, for example, a switching frequency of 10 kHz is used. To allow V_{out} to appear in two quadrants, a full-bridge inverter can be used. This circuit operating under PWM is identical to a two-quadrant buck converter. The pulse width is now intentionally varied, rather than held constant.

The drawbacks of PWM include the relatively high switching rates and power losses which increase with switching rate, the wide range and high magnitude of unwanted components, and radio interference which results from the unwanted components. Advantages include the ease with which $V_{out(wanted)}$ can be varied, the fact that adjusting f_{out} is just as easy as adjusting $|V_{out(wanted)}|$, the simplicity of PWM systems, and the possible wide frequency separation between wanted and unwanted frequency components. This wide frequency separation simplifies filtering — a low-pass filter is generally all that is used.

We will examine PWM in the context of ac motor control and R-L loads. The ac induction motor, for example, is mechanically simple and rugged, requires no electrical connections between stationary and rotating parts, and is inexpensive. Ac motor speed control in general requires adjustment of the motor input frequency. Unfortunately, ac motors also have frequency-dependent impedance. As the frequency is decreased, input current and internal magnetic flux rise. To counteract these effects, the voltage must change along with frequency. If the voltage is altered in the correct manner as a function of frequency, an ac motor can be made to act almost like a dc motor in terms of speed and torque control. Today, ac motor drives are displacing dc motors in most applications. These drive units are nearly always based on PWM.

Basic Theory: The half-bridge inverter in Fig. 2 has output $V_{out} = (2q_1 - 1)V_{in}$. If the duty ratio is varied with time as some *modulating function M(t)*, V_{out} becomes:

$$V_{out} = V_{in}\left[2M(t) + \frac{4}{\pi}\sum_{n=1}^{\infty}\frac{\sin(n\pi M(t))}{n}\cos(n\,\omega_{switch}t - n\phi_0)\right] - V_{in}$$

Remember that D must be between 0 and 1. Substitute ½[k·cos(ω_{out}t)+1] for M(t). Then V_{out} becomes:

$$V_{out} = V_{in} k \cos\left(n\omega_{out} t\right) + \frac{4V_{in}}{\pi} \sum_{n=1}^{\infty} \frac{\sin\left(\frac{n\pi}{2}\left[k\cos\left(n\omega_{out} t\right)+1\right]\right)}{n} \cos\left(n\omega_{switch} t - n\phi_0\right)$$

Notice that this is *not* a Fourier series — time appears in several places, as does sin[$nk\ cos(\omega_{out}t)$]. It can be decomposed into a Fourier series (by using properties of Bessel functions), and the Fourier components that result appear at frequencies of $n\omega_{switch} \pm m\omega_{out}$. The components drop in amplitude quickly for increasing *m*, but slowly for increasing n. If the ratio f_{switch}/f_{out} is large, very little of the energy appears between ω_{out} and ω_{switch}.

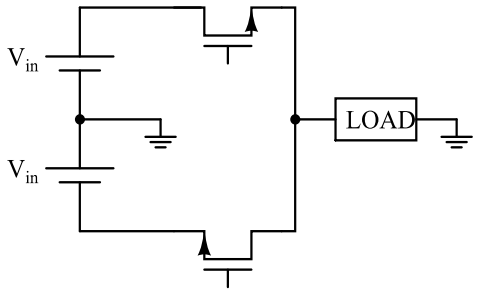


Figure 2. Half-bridge inverter for PWM control

Since the output is a voltage source, a current-sourced interface such as an inductive load is appropriate. A series R-L load would act as a low-pass filter (just as in the dc-dc buck converter), separating f_{out} from the unwanted components. This function can be performed by a deliberate R-L load, a motor winding, or a transformer. This experiment will illustrate both PWM and the filtering process.

Procedure —

Part 1: PWM inverter, R-L load

1. Set up the dual FET control box to form a half-bridge inverter, as in Fig. 3. Use a fixed 24 V supply as the input energy source. Set the box for dead-time between the two switching functions.
2. Set the power supply current limit to about 1.5 A, and the voltage to zero.
3. Set up the waveform generator at your bench to produce a 50 Hz sinusoidal voltage such that the lowest voltage is 1 V and the highest is 3 V. Remember to use high-Z mode. Connect this voltage to the duty ratio input BNC jack on the FET box.
4. Observe both the half-bridge output voltage and the (filtered) resistor voltage. Turn on the FET box and the input power supply. Set f_{switch} to about 20 kHz.

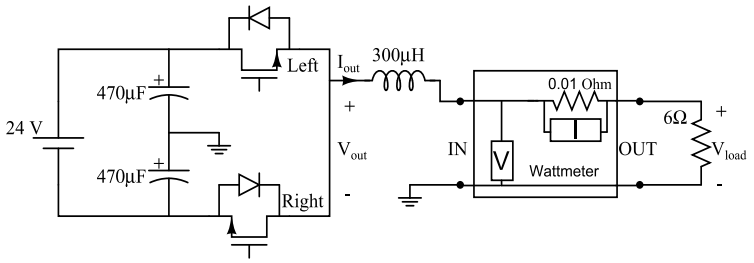


Figure 3. Voltage-sourced inverter test circuit

5. Monitor the average value of V_{out} and the power supply current. Adjust the function generator to get near zero average output. Remember that some supplies must be turned off if their current limit becomes active.
6. Sketch the V_{out} waveform. Can you see the modulation process?
7. Increase f_{switch} to about 50 kHz. Record the RMS voltage, current, and power into the load resistor. Measure the average input current.
8. Decrease the amplitude of the function generator modulating waveform by about 50%. Sketch the V_{load} waveform. Record meter data as in steps 6 and 7.

Part 2: Induction motor drive

1. Turn off the power supply. Replace the R-L load with a series combination of a 5 Ω resistor and a transformer winding with a 12.6 V rating.
2. Place a 1 kΩ load across the transformer secondary terminals. Turn on the power supply, and measure the RMS and average voltage across this resistor, and observe and sketch the waveform. **Be aware of the relatively high voltage level.**
3. With the supply off, wire the small quadrature motor into the circuit, as shown in Fig. 4. Apply power. Sketch the converter output voltage and resistor voltage waveforms (refer to the figure).
4. Change the sine wave frequency and amplitude to observe an ac motor control function.

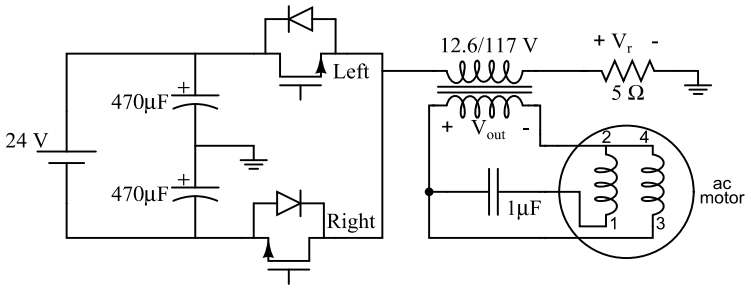


Figure 4. Ac motor drive test circuit

Part 3: Audio amplifier application

1. Set up the audio amplifier circuit provided in the laboratory. Use a power supply of 9 V, and prepare a 1 kHz sinusoidal modulating signal for input. The load can be a loudspeaker or a 6 to 10 Ω resistor.
2. Based on the amplifier data sheets, observe the modulating function, carrier waveform, and switching function for at least two different amplitudes of modulating signal. Please keep the signal below the saturation limit.
3. Record your observations.

Part 4: Commercial drive demonstration

Your instructor will demonstrate a commercial PWM ac motor drive.

1. Connect a PWM drive to a motor of proper ratings, then to 208 V three-phase power.
2. Observe current and voltage traces. Turn on the drive.
3. Operate the drive at frequencies of approximately 20 Hz, 40 Hz, and 60 Hz. Record your observations.

Study Questions —

1. For this type of converter, how do you expect waveforms for an R-L load to change with switching frequency? With modulating frequency?
2. Why is PWM advantageous in ac motor control?
3. Draw the full-bridge inverter for PWM. What relationships would you expect among the various switching functions?
4. The three-phase inverter uses six switches. What waveforms do you expect for line-to-line and line-to-neutral voltages?
5. Compute and tabulate the converter's efficiency from your Part 1 data.
6. Compare PWM with the simpler inverter scheme of Experiment #6.
7. Compare the PWM inverter from Part 2 to the commercial unit of Part 4.

ECE 469 — POWER ELECTRONICS LABORATORY

EXPERIMENT #7 — Real Components, Part I: Models for Real Capacitors and Inductors

Objective — This series of experiments will study the behavior of real components. In the first experiment, basic characteristics of realistic capacitors and inductors will be examined, with emphasis on impedance effects. The second experiment explores the topic of high-side gate driving, a key implementation detail pertinent to many power converters.

Pre-Lab Assignment — Read this experiment. Study the procedure. *There is a considerable amount of data to be taken for this experiment. The recorder should prepare appropriate tables prior to the lab session.* Solve the following on a separate sheet, for submission as you enter the lab. Your instructor may elect to assign another problem.

1) A sinusoidal voltage of amplitude V_0 and frequency f_s is applied to the series R-C circuit shown.
   a) Find the amplitude and phase of V_c as a function of frequency. Tabulate the values of amplitude and phase for frequencies of 200 Hz, 20 kHz, and 2 MHz.
   b) If the capacitor is really a series-resonant L-C pair (as in Fig. 5), what is V_c at resonance?

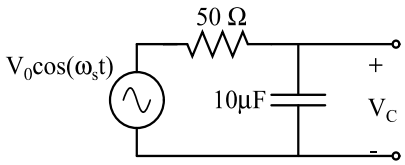


Figure 1. R-C voltage divider for pre-lab assignment

2) A sinusoidal voltage of amplitude V_0 and frequency f_s is applied to the series R-L circuit shown.
   a) Find the amplitude and phase of V_L as a function of frequency. Tabulate values at 200 Hz, 20 kHz, and 2 MHz.
   b) If the inductor has significant series resistance, can the value be determined knowing V_L and f_s? If so, what is the value in terms of known and measured quantities?

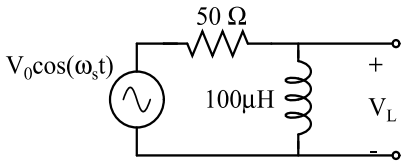


Figure 2. R-L voltage divider for pre-lab assignment

Discussion —

Introduction: The capacitors, inductors, and resistors used in circuit analysis have ideal properties: $i_c = C\frac{dv_c}{dt}$, $v_L = L\frac{di_L}{dt}$, $v_R = Ri_R$. As in the case of voltage and current sources, real devices are not ideal. In the context of power electronics, these effects are often important. After all, we often seek large values of *L* and *C* at high voltage, current, power, and frequency. We often use resistors in high-speed circuits.

Some of the nonideal effects are straightforward: wires have resistance and inductance, coils of wire have capacitance between the turns, and so on. In order to use passive components in power electronic circuits, it is important to understand what the effects are, how they are characterized and measured, and the implications for design. In these experiments, we will examine practical effects in passive devices.

Basic Theory, Capacitors: When two conductors of any shape are placed in an electric field, a charge develops on each one. The amount of charge changes with the voltage difference between the conductors. In a system which consists only of perfect conductors and perfect insulators, the charge *Q* depends on voltage in a linear fashion, so that $Q = CV$. The constant of proportionality is defined as *capacitance*. Since voltages perform work in separating charge, the charge and capacitance represent stored energy, in the well-known form $W = \frac{1}{2}CV^2$. Current is, of course, the time rate of change of charge. If capacitance is constant (e.g., the conductors are stationary), this leads to the simple relationship $i = C\frac{dv}{dt}$.

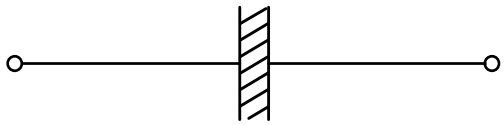


Figure 3. Parallel-plate capacitor

A real capacitor must provide conductors to hold the charge, wires to allow application of voltage, insulation which physically supports and separates the conductors, and a protective package to prevent damage. There are two major classes of commercial capacitors. The first consists of two flat metal conductors, separated by a dielectric layer. The second, known as the *electrolytic capacitor*, consists of an oxidized metal conductor and a nonmetallic conductor. Either of the two types is capable of providing storage of electric charge. An excellent overview of capacitor types can be found in [1].

Both types can be modeled with a parallel-plate geometry, shown in Fig. 3. The capacitance value depends on the plate spacing d, the plate area A, and the dielectric constant of the insulator, ε, according to the relation $C = \varepsilon\frac{A}{d}$. A large value of capacitance requires large plates, small spacing, and high dielectric constant.

Capacitors possess both current and voltage ratings. The dielectric must not break down. To avoid this, the electric field magnitude E = V/d must be kept sufficiently low. The voltage rating of a capacitor, then, depends on *d*. When the applied voltage changes rapidly, significant currents will flow. The wires and plates must be large enough to handle the applied current without overheating. In general, one wants to place very large plates in a small package, along with a dielectric of high ε. The choices mainly involve plate thickness, wire size, dielectric thickness, and dielectric type. The nonideal effects are also relatively clear: the wires and plates introduce series inductance and resistance, and an imperfect dielectric could allow some current flow between the plates. A candidate circuit model emerges:

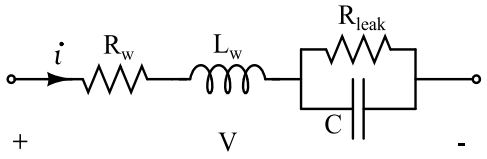


Figure 4. Proposed circuit model for a simple dielectric capacitor

This circuit model can be simplified at a specific frequency. Then the parallel portion can be transformed into a series equivalent. This gives the **standard model**:

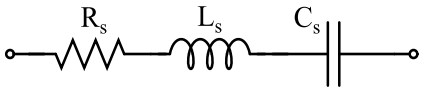


Figure 5. Standard series equivalent circuit of a capacitor

Many manufacturers use this standard model as a basis for describing their capacitors. The **equivalent series resistance** (ESR, another name for the R_s of the circuit) is often given at a frequency such as 120 Hz. The **equivalent series inductance** (ESL) is often given in the form of a "self-resonant frequency," f_r, The properties of the standard circuit include:

- The voltage drop across the ESR reduces the stored charge relative to expected values.
- Any current flow produces loss in the ESR: the higher the current, the higher the losses.
- The resonant effect of the series R-L-C circuit means that a plot of impedance vs. frequency will fall at first (capacitive impedance), reach a minimum, and then rise (inductive impedance).
- Above f_r, the device is an inductor!

- The circuit does not model real behavior well at very low frequencies. For instance, the original circuit in Fig. 4 shows that some leakage current will flow when a dc voltage is applied. Real capacitors show this effect. The standard model does not predict it.

The ESR mainly represents the dielectric properties. As you might guess, dielectrics are strange materials, and do not act in accordance with Ohm's law. Because of this, the ESR is generally some kind of nonlinear resistance; for example, it varies with frequency. A traditional way of characterizing such materials is with the so-called **loss tangent**. The loss tangent, also called tan δ or **dissipation factor** (*df*), is defined as *the ratio of resistance to reactance for a series R-X circuit.* For the standard model of the capacitor (at low frequencies), $\tan \delta \approx \omega RC$. The loss tangent has a characteristic value for a given capacitor dielectric material, regardless of the plate geometry. It also happens that many good dielectric materials are characterized by a loss tangent, which is roughly constant over a wide frequency range. For these reasons, the loss tangent or *df* is often given in capacitor specification sheets. The ESL is more closely related to packaging and lead structure, since wire inductance is the most important factor.

In electrolytic capacitors, the insulating layer is formed through an electrochemical reaction between the two conductors. If voltage of the wrong polarity is applied, the reaction reverses, and the insulating layer is destroyed. The device becomes a resistor, and usually overheats and fails catastrophically. When the correct polarity is applied, the electrolytic capacitor shows the same general properties as the simple dielectric version, with two changes: the leakage current levels are much higher, which means ESR is significant; and the effective plate surface area is very high, which allows high values of capacitance per unit volume. Electrolytic capacitors are more nearly characterized by constant ESR than by constant *df* values.

Basic Theory, Inductors: Inductors are formed simply by wrapping a coil around a magnetic material. Current in the coil creates magnetic flux in the material. If the material is linear, the flux is proportional to current, so that $\lambda = Li$. The constant of proportionality in this case defines inductance. By Faraday's law, if this flux varies with time, it gives rise to a voltage v_L. Thus $v_L = L \cdot di/dt$. Magnetic materials are generally *not* linear; these effects will be studied in more detail in lecture.

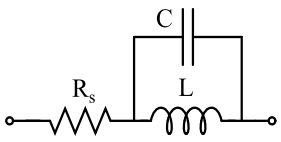


Figure 6. Proposed circuit model for a basic inductor

Even without considering the effects of nonlinearities, some aspects of real inductors are clear. The wire coil has resistance. Incoming and outgoing wires act a bit like parallel plates. A simple circuit model might be the one in Fig. 6. Below resonance, the L-C combination can be treated as just an inductor, so an R-L series combination is sufficient as a model. Since no computation or "equivalencing" is required to find R, it is usually just called the *series resistance*. In reality, the magnetic material also shows loss, so other resistances appear. As in the capacitive case, the loss tangent is defined *as the ratio of resistance to reactance*. It is hard to build inductors, which handle high power levels and still have very low loss tangents.

Inductor types are defined mainly by the magnetic material. This can be a linear material, such as air, or any magnetic material. Good linearity is a desirable feature, so most practical inductors have an air gap. In this experiment, inductors will be tested for their ESR values and resonant frequencies.

Procedure —

Part 1: Reference measurements. *Do these as time permits rather than first, so teams can trade off.*

1. Your instructor will assign a set of capacitors, inductors, and resistors to each team. Use the automatic RLC meter in the lab to obtain 1 kHz values for C_s/L_s, C_p/L_p, R_s, R_p, and $D = df$ for each of your assigned parts. Special emphasis should be placed on the values relevant to the various models.

Part 2: Frequency sweeps

1. Insert one capacitor in the circuit shown below. Use the highest available V_{in} value. If the capacitor is electrolytic, be sure to observe the polarity marks, and add a dc offset to V_{in} so that $V_c > 0$ at all times. If it is not electrolytic, use an offset of zero. If the capacitor is less than 0.5 μF, substitute a 1 kΩ resistor for R_s.

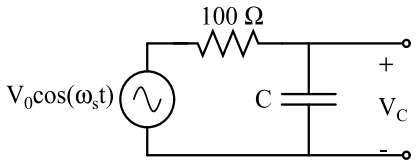


Figure 7. Capacitive impedance test circuit

2. Set f_s to 1 kHz, and observe V_{in} and V_c on your oscilloscope. Measure the peak-to peak amplitudes and the phase shift between them in degrees. (Hint: Phase measurements can be made more easily by using the oscilloscope cursors to measure time differences.)
3. Look for the resonant frequency of the part by adjusting f_s until the phase shift is close to zero. How do you expect the amplitude to behave? Record amplitudes of V_{in} and V_c at this frequency.

4. Choose at least three frequencies below resonance and three above it. The frequencies you choose should cover roughly a factor of 100 in frequency. Record amplitudes of V_{in} and V_c, and the phase shift between them, at each of these frequencies.

5. Repeat parts 1–4 for your other assigned components. In some cases, you may want to change the resistor value to allow better measurements of V_c. If you do this, be sure to use $R \geq 50\ \Omega$ and record the actual R value used. A value $R = 1\ k\ \Omega$ is suggested for inductors.

Study Questions —

1) Use data for V_{in}, V_c, and phase to compute the impedance for each tested component.

2) Plot impedance magnitude and phase vs. frequency for each component.

3) Calculate ESL for each capacitor based on the resonant frequencies you measured.

4) Using your frequency sweep data, calculate ESR at 1 kHz and at a frequency near resonance for each capacitor and inductor. Compare the 1 kHz results to the RLC meter data.

5) Discuss how your data conform to the proposed simple models.

References —

[1] Donald M. Trotter, Jr., "Capacitors," *Scientific American*, vol. 259, no. 1, July, 1988, pp. 86-90B.

ECE 469 — POWER ELECTRONICS LABORATORY

EXPERIMENT #8 — Real Components, Part II: Switching Devices and Gate Drive Circuits

Objective — The purpose of this week's lab is to develop an understanding of the non-idealities of common switching devices and how they can be controlled to minimize power loss and optimize circuit performance. The operation of high-side gate drivers will be introduced which will allow the implementation of a buck converter using discrete components.

Pre-Lab Assignment — Read this experiment. Study the procedure, and bring any questions to class. Solve the following on a separate sheet, for submission as you enter the lab. Your instructor may elect to assign another problem.

1. The MOSFET in the buck converter shown in Fig. 1 has a threshold voltage of 5 V. For this analysis, assume that it is an ideal switch, meaning that the switch will be fully off (infinite resistance) if the gate-source voltage is below 5 V, and fully on (zero resistance) if the gate-source voltage is above 5 V. The MOSFET is driven by a square-wave signal that has a high of 10 V with respect to circuit ground, and a low value of 0 V with respect to circuit ground. The slope of the square wave is infinite.
    a. Calculate the maximum voltage that can be observed at the switching node (labeled N_{SW}) while the switch is still on.
    b. What is the maximum source voltage for the circuit?

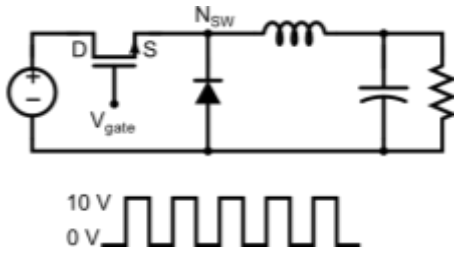


Figure 1. Buck converter

    c. For the buck converter of Fig. 4, sketch the gate voltage of the transistor with respect to ground, for an input voltage of 20 V, a switching frequency of 100 kHz, and a 50% duty signal applied to SW1 and SW2. Note that SW1 and SW2 are operated in a complimentary fashion, so that only one switch is on at a given time.

Discussion —

Basic theory of gate drives: Up to this point in the class, we have assumed the existence of ideal switching devices. In reality, the switching must be performed by a practical device such as a BJT, MOSFET or IGBT.

A BJT can be modeled as a current-controlled current source, and requires current into the base when maintaining the device in the on state. This current is supplied at a low voltage (so that power is low), but can be as high as a tenth or more of the switch on-state current. Base drives for power BJTs can be a challenge, since they usually must supply several amps in a precisely-controlled square wave. In the case of the FET and other field-controlled devices, the steady-state gate current is essentially zero, and the gate voltage determines switch operation. However, there is still a need for high short-term currents to charge gate regions rapidly and force the fastest possible switching. These voltage-controlled gate-drive circuits are much easier to design than BJT base drives, which helps explain the declining use of power BJTs in switching converters. Because FETS have arguably the simplest gate-drive requirements and have come to dominance in many power electronics applications, they will be used for this lab.

The MOSFET consists of a channel of doped semiconductor (most often p-type) spread between the drain and source terminals. A small region of heavily doped material of opposite type is implanted at the two terminals. A layer of insulation is mounted on top of the channel. When voltage is applied to the gate electrode, charge carriers are attracted into the region near the gate. More carriers come in as the gate voltage rises, until the channel is "inverted," and matches the polarity at the source and drain. Current can flow between the source and drain if adequate numbers of charge carriers are present.

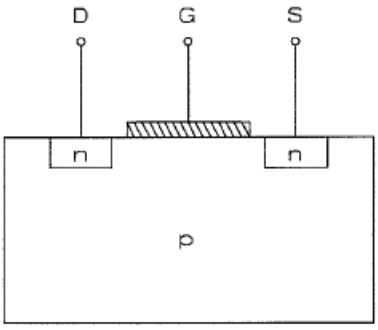


Figure 2. Simplified MOSFET cross section

To turn the FET on, the gate region must be charged. The region represents a capacitance, and the gate drive must be designed to charge this capacitance sufficiently. The channel will invert when the gate voltage exceeds a threshold level, V_{th}, and the gate voltage must be maintained at a level considerably greater than V_{th} (usually above $2V_{th}$) so that enough charge will be present in the channel for low effective channel resistance. The gate drive must therefore apply an *overdrive* gate voltage on the order of more than 100%.

To turn the FET off, the gate region must be discharged, and the gate voltage must be decreased and maintained below V_{th}. Real devices have wide error tolerances on V_{th}. For example, the IRFP360 FET used in the FET boxes can have any V_{th} between 2.0 V and 4.0 V, depending on the device and operating temperature. For good performance, a gate drive would apply about 10 V between gate and source for such a device.

It would seem optimal to supply the highest current possible to the FET gate in order to minimize switch transition times and corresponding losses. Indeed, this would be true in a circuit with no parasitic inductance. However, remember that the inductor voltage is proportional to the rate of change of current flowing through it. By forcing the current flowing through the circuit's parasitic inductances to change too quickly, significant voltage ringing will be created that can destroy switching components. Provided that the circuit connections have been made neatly with minimal connection lengths, the FET should be switched at the highest speed possible without excessive ringing.

The basic objectives of a gate drive circuit can therefore be summarized as follows:

- For turn-on, the drive must rapidly charge the gate capacitance to a voltage much higher than V_{th}, Without inducing excessive ringing which might exceed the dielectric breakdown limit of the gate insulator.
- For turn-off, the drive must rapidly discharge the gate to a voltage lower than V_{th}, again without inducing excessive ringing.

One of the challenges of designing drivers for N-channel devices is the need to apply a substantial voltage between the gate and source as long as the device is to be on. In many converters, this is not a trivial matter. As an example, consider the familiar buck dc-dc converter as shown in Fig. 3. For the circuit in Fig. 3a to work, a voltage equal to V_{in} + 10 V must be applied to the FET gate whenever the switch is to be on. If the switch is relocated as in Fig. 3b, the problem of a high gate voltage is alleviated, but the output is no longer relative to ground potential.

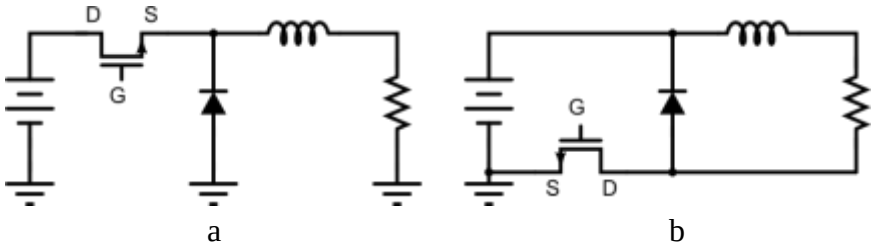


Figure 3. Buck converter with FET switch

High-side gate drivers: The gate potential problem is solved by means of a high-side gate drive circuit. Figure 4 depicts the functionality of the circuit. With switch 1 closed, the gate of the FET will be pulled to V_{DRV} higher than the source and the device will be fully on. When switch 2 is closed, the gate of the FET will be connected to the source and the device will be fully off. Although an isolated source such as that in Fig. 4 is sometimes used in high-power applications, there are more cost-effective options for typical converters.

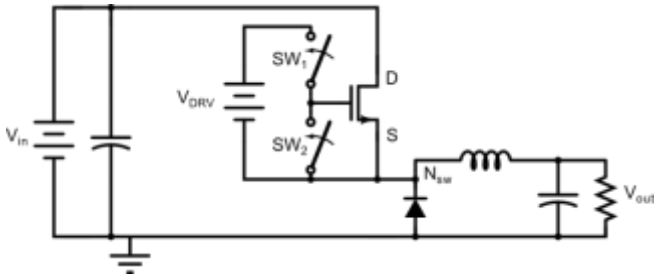


Figure 4. Buck converter with functional high-side gate driver

The circuit in Fig. 5 shows the partial implementation of a more practical gate driver. In order to understand the operation of this circuit, note that the voltage of N_{sw} is near ground when the diode is conducting and at V_{in} when the diode is off. Because the switching function of the diode and FET are opposite, $N_{sw} \approx 0$ V when the FET is off and $N_{sw} \approx V_{in}$ when the FET is on.

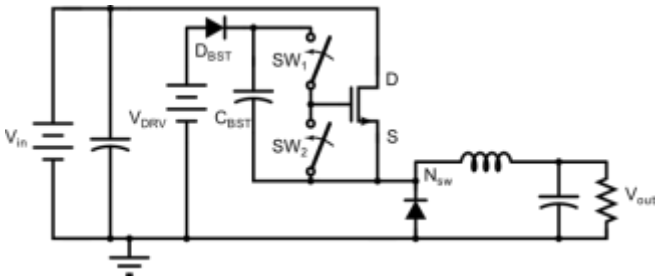


Figure 5. Buck converter with discrete high-side gate driver

During the off-time of the FET, when $N_{sw} \approx 0$ V, current is able to flow through D_{BST} to recharge C_{BST} to the value of V_{DRV}. When SW_1 is turned on, V_{gs} of the FET is charged to V_{DRV}, and therefore the gate of the FET is raised to $V_{DRV} + V_{in}$ above ground. Thus, C_{BST} serves the same purpose as the floating voltage source in Fig. 4. The source of V_{DRV} is typically a separate low voltage supply of 5–12 V, depending on the desired gate voltage.

The control of SW_1 and SW_2 are the only components of the circuit left to discuss. This control is frequently accomplished using BJT devices driven by a level shifter. Fortunately, integrated high-side drivers are available to handle this control, and therefore circuit design only consists of selecting the driver capacitor (C_{DRV}), C_{BST}, R_{gate}, and the boot-strap diode. Some devices, such as the LM5101 from TI, contain an internal boot-strap diode (D_{BST}), as shown in Fig. 6, which further simplifies implementation.

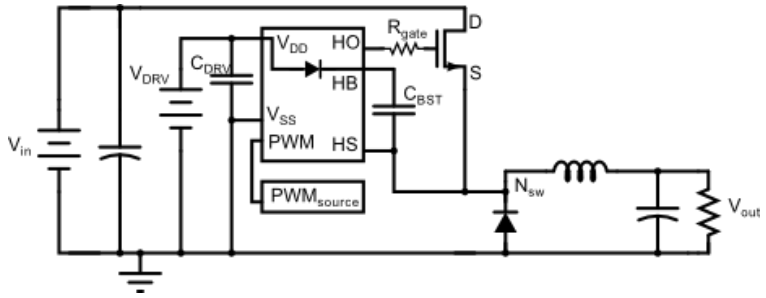


Figure 6. Buck converter with integrated high-side gate driver

The boot-strap capacitor is responsible for providing the gate charge of the FET (Q_G), the reverse recovery charge of the diode (Q_{RR}). the leakage current of the boot-strap diode ($I_{LK,\,D}$), the quiescent current of the level shifter ($I_{Q,\,LS}$), the high-side gate driver operating current ($I_{Q,\,DRV}$), and the gate-body leakage current or the leakage current between the gate and source terminals (I_{GS}). Based on this information, the required boot-strap capacitance can be calculated as:

$$C_{BST} = \frac{Q_G + Q_{RR} + (I_{LK,D} + I_{Q,LS} + I_{Q,DRV} + I_{GS})\frac{D_{MAX}}{f_{DRV}}}{\Delta V_{BST}}$$

The following information for the devices which will be used in this lab is available in the device data sheets.

Driver- IRS2183:	Diode- FES16DT:	MOSFET- IRF100B202:
$I_{Q,\,LS}$: 50 µA	Q_{RR}: 1.2 nC	I_{GS}: 100 nA
$I_{Q,\,DRV}$: 150 µA	$I_{LK,\,D}$: 10 µA	Q_G: 116 nC

For a switching frequency of 100 kHz, a boot-strap voltage ripple of 0.5 V, and a maximum duty ratio of 90%, the required capacitor size is calculated to be 80 nF. For good measure, a 0.1 µF cap will be used for C_{BST}. C_{DRV} is customarily made at least an order of magnitude larger than C_{BST}, so a 1µF cap will be used. Note that sizing the boot-strap capacitor in circuits using special control techniques which allow the switch to remain on or off for multiple cycles will require additional considerations. There are many application notes available from component suppliers discussing such details.

A great deal of analysis can be invested in the calculation of the correct gate resistance, but this calculation does not take into account variable parasitics, such as the inductance of the wires used in the lab. In order to allow for optimization based on the actual circuit construction, the gate resistance will be first estimated and then tuned during operation. Calculations regarding gate charge and switching times have some merit in that they can provide a result within an order of magnitude or so. However, as you have already seen, parasitic elements can change the performance of power electronic circuits very significantly.

Figure 7 shows some of the locations where parasitic inductance affects FET switching. Inductance in the gate drive circuit (L_{gate}) can form a tank circuit with the FET gate capacitances and cause oscillations during switch transitions. Similar ringing can also result from the parasitic inductance in series with the FET. Noting L_{series} in Fig. 7, it can be seen that a sudden change in current flowing in the circuit will result in a large potential build-up across the inductor. In order to limit the ringing magnitude, the rate of change of current can be reduced by increasing the gate resistance and thereby increasing the switching time. Alternatively, L_{series} can be reduced by keeping interconnections small.

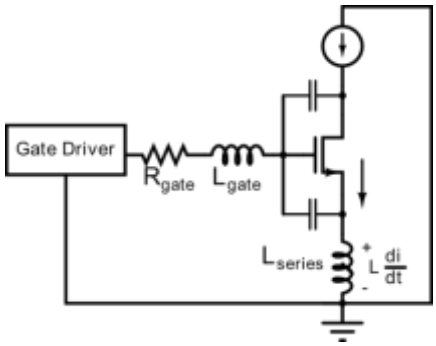


Figure 7. Schematic showing the effect of parasitic inductance on hard switching

Figures 8 through 11 demonstrate the effects of tight circuit wiring on converter performance. Figure 8 shows the gate switching waveform and FET drain to source voltage for the neatly wired buck converter shown in Fig. 12. Adding 1 meter of wire between the diode cathode and the FET source results in voltage spikes during turn-off, as depicted in Fig. 9. Adding an additional meter of wire between the input filter capacitor and FET drain results in even larger voltage ringing with the peak V_{DS} reaching 84 volts in a converter with only a 12 volt input. Based on this example, it is easy to understand why poor circuit layout can damage FETs. The IRF530N device used in this lab has a max V_{DS} of 100 V. The excessive ringing can be partially remedied by adding gate resistance as illustrated in Fig. 11. By increasing the gate turn-on time, the peak V_{DS} ringing is reduced to 48 volts.

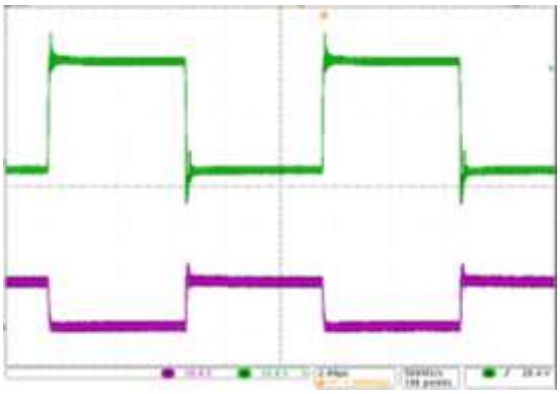


Figure 8. Tightly wired (Top: V_{gate}, Bottom: V_{DS})

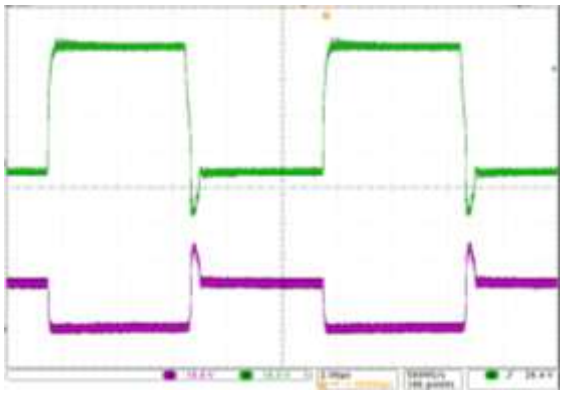


Figure 9. One meter of wire between diode cathode and FET source

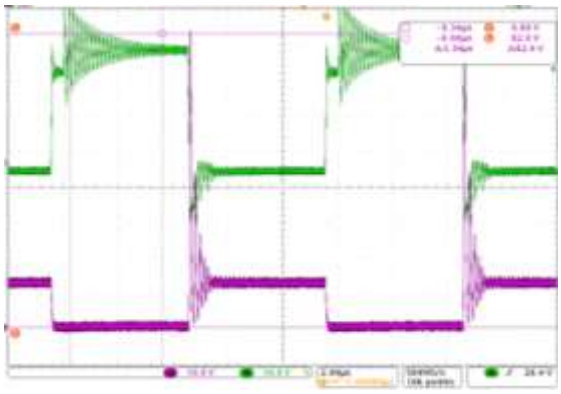


Figure 10. Same as Fig. 8 plus one meter of wire between input cap and FET drain. $V_{DS\ peak}$ = 84 V

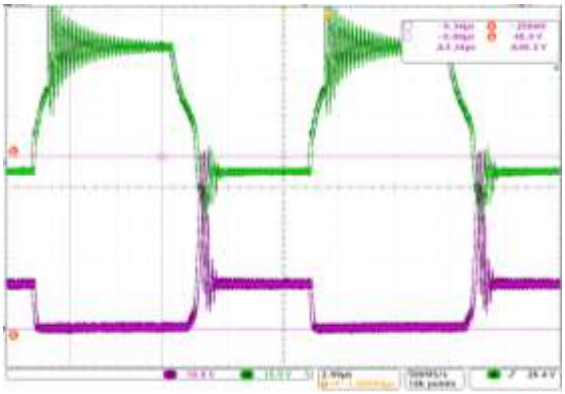


Figure 11. Same as Fig. 9 with added gate resistance. $V_{DS\ peak}$ = 48 V

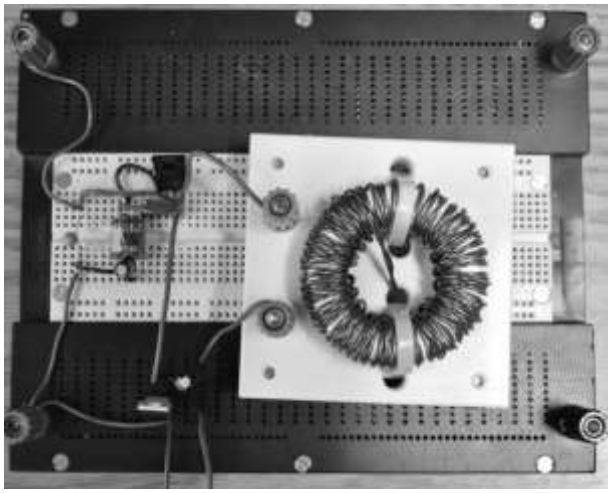


Figure 12. Tightly wired, low-power buck converter on a breadboard

Procedure —

1. Build the circuit depicted in Fig. 13. Use a 0.1 µF cap for C_{BST} and a 1 µF cap for C_{DRV}. Both should be ceramic and be placed close to the chip. A 500 Ω potentiometer set to 120 Ω will be used for R_{gate}, and the PWM signal can be obtained from the signal generator. Refer to the IRS2183 data sheet for the appropriate amplitude of your PWM signal.
2. Set the power supply current limit to 3 A and the voltage to 12 V. Turn on the electronic load and place it in resistance mode at 25 Ω, or use a 25 Ω power resistor.

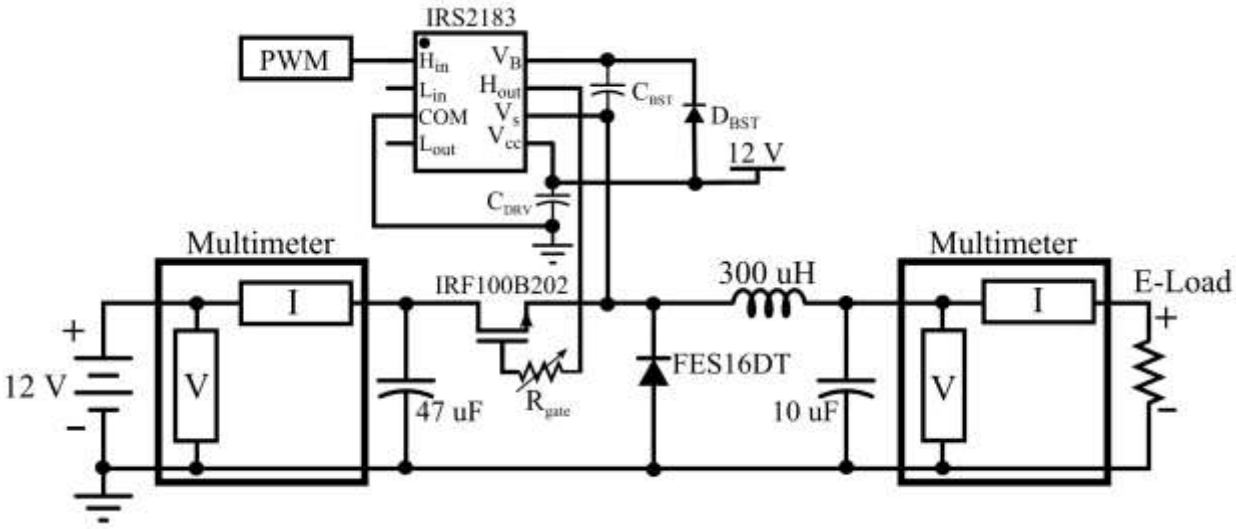


Figure 13. Buck converter test circuit

3. Set the switching frequency to 100 kHz and the duty ratio to 50%. Verify that the PWM signal is reaching PIN 7 of the gate driver.
4. Place the *differential* oscilloscope probe between the drain and source of the FET, and a regular probe between ground and the cathode of the power diode. Also set up the scope to observe the FET gate and the output voltage.
5. Power up the converter and capture a screenshot of the gate signal. Measure the time constant of the gate-charging circuit.
6. Adjust the resistance of the gate drive circuit until the turn-on and turn-off intervals of the FET are about equal, as long as the V_{DS} ringing does not exceed about 60 volts. Be aware that at higher voltage levels (as may be encountered in the final project), this ringing can easily destroy the FET if it exceeds switch ratings. Record T_{on} and T_{off}. Note whether T_{on}, T_{off} or both are influenced by driver circuit resistance.

7. Adjust R_{load} to 4 Ω. After waiting about 2 minutes for converter temperature to stabilize, take measurements of V_{in}, V_{out}, $I_{in,}$ and I_{out} with which to calculate converter efficiency. Measure FET case temperature as well.
8. Turn off the circuit and record the potentiometer resistance R_{gate}. Adjust the potentiometer to 100 Ω, restart the converter, and again measure T_{on} and $T_{off.}$ Take efficiency data and measure FET temperature after waiting about two minutes for converter temperature to stabilize. Repeat for 200 and 400 Ω, taking data at each point.
9. Add an extra length of wire (around 1 meter) between the source of the FET and the cathode of the diode, as well as between the input cap and the drain of the FET. Find the lowest gate resistance at which the circuit can be operated while keeping the V_{DS} ringing less than 50 volts with both inductances in place. What is the minimum peak ringing you can achieve? Save a screenshot of circuit operation with both inductances, with each of the added inductances separately, and without extra parasitic inductance.
10. Do not take apart your circuit, as you will be using it in future labs.

Study Questions —

1. Tabulate your data in an organized fashion.
2. Calculate an approximate value of the gate capacitance of the FET at a V_{DS} of 12V.
3. Calculate and plot the converter efficiency as a function of gate resistance. Comment on whether you were able to make T_{on} and T_{off} equal for your circuit, or whether excessive ringing prevented you from doing so.
4. List and comment on the changes in FET temperature as the gate resistance was increased. What is the reason for this additional power loss in the FET?
5. Comment on the effect of adding extra parasitic layout-inductance to the converter and include plots of normal operation and operation with each of the added inductances separately. Estimate the amount of parasitic inductance added.
6. Is there anything that prevents you from operating the high-side switch in the on-state indefinitely?

References —

[1] L. Balogh, “Design and Application Guide for High Speed MOSFET Gate Drive Circuits” Texas Instruments Inc. [Online]. [Accessed Aug. 16, 2013].

[2] Advanced Power Technology, Appl. Note APT0102. Rev Oct 29, 2001 [Online]. [Accessed Aug. 16, 2013].

ECE 469 — POWER ELECTRONICS LABORATORY

EXPERIMENT #9 — PWM Generation and Control

Objective — The purpose of this week's lab is to generate a pulse-width-modulated (PWM) waveform utilizing a triangle wave generator. The PWM waveform thus generated will be used in a dc-dc power converter.

Pre-Lab Assignment — Read this experiment. Study the procedure, and bring any questions to class. Solve the following on a separate sheet, for submission as you enter the lab. Your instructor may elect to assign another problem.

1. For the Schmitt trigger circuit in Fig. 4, derive an expression for the low and high threshold values (the values of V_{in} when the comparator will toggle). Note that the output voltage will alternate between 0 and V_{dd}. By taking the difference between the high and low threshold values, find the hysteresis voltage. Observe that the hysteresis voltage is the same as the peak-to-peak amplitude of the triangle wave.
2. For the integrator circuit in Fig. 2, derive an expression for the output voltage V_{tri} in terms of parameters R_{s1}, R_{s2}, R_{int}, C_{int}, V_{dd}, and V_{ref}.
3. For a triangle wave with amplitude of 2.5 V_{dd} , centered at 2.5 V, calculate the value of resistor R_{s1} if R_{s2} has value 10 kΩ and V_{dd} is 5 V. What should V_{ref} be?
4. Derive an expression for the frequency of the triangle waveform generated by the integrating circuit. Express your answer in terms of R_{s1}, R_{s2}, R_{int}, C_{int}, V_{dd}, and V_{ref}. Note that the triangle waveform amplitude will be as found in Problem 1.
5. Find the datasheets for the ICs (LM339 and LT1632) needed to build the PWM generator. These will prove helpful when constructing the circuit.

Discussion —

Basic theory of PWM: A key component of any power converter is the PWM generator. This component is responsible for generating pulse widths of suitable lengths to drive the gate of the active switch. Up until now, you have used the built-in PWM generator of the FET box. In this lab, you will build your own PWM generator, and in the process gain a better understanding of how they operate. Note that the pre-lab exercises of this experiment are more rigorous than what you have experienced so far. You are advised to start them early!

The basic task of a PWM generator is to convert an analog input (such as a voltage reference) to a time-domain series of pulses, where the width of the pulses are proportional to the input reference. Shown in Fig. 1 is an example of a method to accomplish this using a triangle wave (sometimes referred to as *the carrier wave*).

Whenever the reference signal is higher than the triangle wave, the PWM generator outputs a “high” signal, and whenever the reference signal is lower than the triangle wave, the PWM generator outputs a “low” signal. Note that in addition to triangular waveforms, other signals (such as sawtooth) can also be used as the carrier wave.

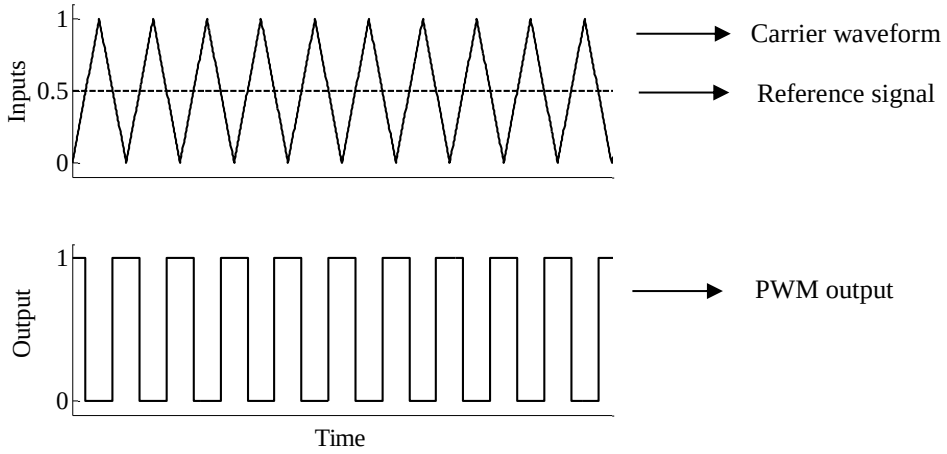


Fig. 1. Pulse width modulation for a constant reference signal using a triangular carrier

PWM generator circuit implementation: Shown in Fig. 2 is a circuit diagram of a simple PWM generator. It consists of a triangle-waveform generator and a comparator. Various components and IC part numbers are also indicated in the figure. Each of the parts is briefly described in the following paragraphs. The pin configurations of the ICs LM339 (used for comparators) and LT1632 (used for integrator) are given in Fig. 3. Note: the LM339 is an open-collector device and therefore requires a pull-up resistor. The advantage of this approach is that the driving strength of the logic signal can be adjusted as required to optimize power consumption or overcome system noise.

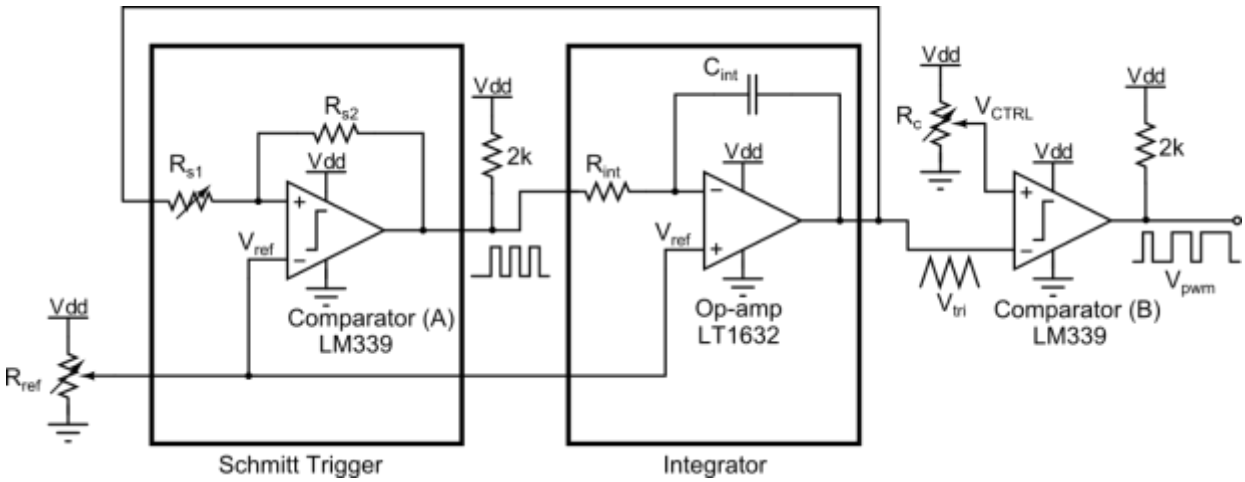


Fig. 2. PWM generator circuit

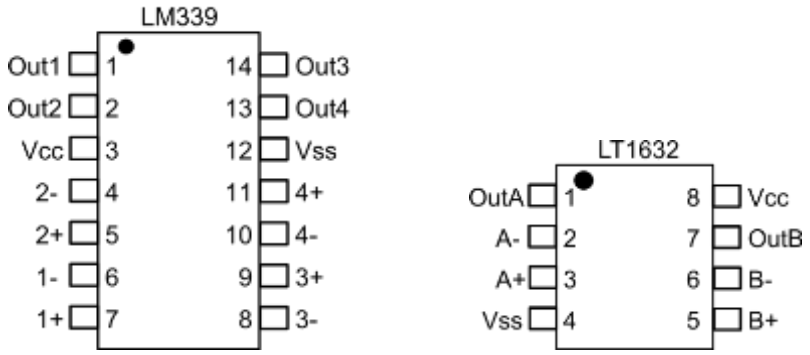


Fig. 3. Pin configurations of the LM339 and LT1632

Triangle wave generator: The key element of the triangle-wave generator is the operational amplifier configured as an integrator. As you should remember from introductory EE classes, the integral of a dc signal is a ramp, a fact that is used in this circuit. In case you do not recall the operation of an op-amp based integrator, you will have a chance to review this material as part of your pre-lab exercises. When the input to the integrating op amp is high, its output will decrease linearly (providing the down ramp of the triangle wave). Conversely, when the input is low, the op-amp output will *increase* linearly (providing the up ramp). By alternatively providing a high and a low input to the integrating op amp, a triangle wave can be produced. The choice of passive components (resistor and capacitors) will set the ramp slopes. These will control the waveform frequency.

The role of comparator (A) is to provide this alternating high and low input. We wish for the comparator to change from a "high" to a "low" output when the down ramp has reached its desired minimum value. Likewise, we want comparator (A) to change from "low" to "high" when the up ramp has reached peak value. To accomplish this, we need a comparator that has two *different* thresholds. A regular comparator cannot accomplish this, but through a technique known as **hysteresis**, we can make the comparator perform this functionality. The comparator in Fig. 2 is connected as a Schmitt trigger, a very useful circuit implementation that performs a comparison with hysteresis. The Schmitt trigger creates a switching band with an amplitude equal to

$$\Delta V = \left(\frac{V_{dd}}{2}\right)\left(\frac{R_{s1}}{R_{s2}}\right)$$

This amplitude will thus be the amplitude of your triangle waveform, labeled V_{tri} in Fig. 2. By analyzing the integrating circuit, the operating frequency can be determined to be:

$$F_{op} = \frac{R_{s2}}{R_{s1}}\frac{1}{R_{int}C_{int}}\frac{V_{ref}(V_{dd}-V_{ref})}{V_{dd}^2}$$

You will derive these relationships in the pre-lab. Ideally, we wish to operate with a triangle waveform spanning from 0 to V_{dd} for maximum resolution.

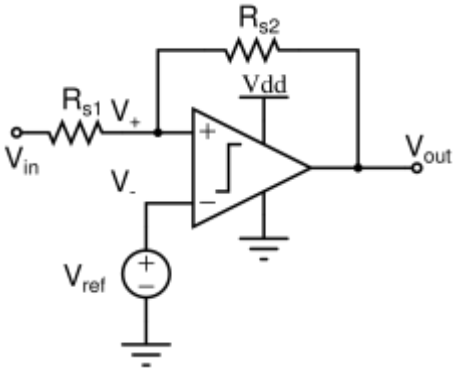


Figure 4. Schmitt trigger circuit

Comparator: The triangle waveform that we generated can now be compared to a reference voltage, which is produced by the potentiometer (R_c in Fig. 2). By adjusting the potentiometer, the input voltage of the comparator (V_{CTRL}) can be adjusted to change the duty ratio of the PWM waveform.

Printed Circuit Board (PCB)

You will be using a specially designed PCB for the rest of your labs to eliminate the challenges that you faced while building the buck converter on the breadboard. A PCB implementation will minimize noise and improve reliability and performance. As a part of this process, you will have to solder components for both experiments 8 and 9. Figure 5 shows the overall architecture of the PCB that you will be using.

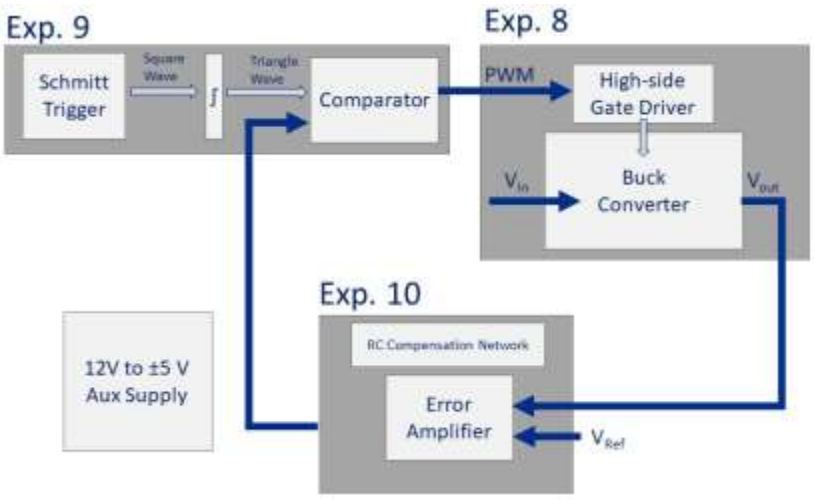


Figure 5. Architecture of the PCB

Figure 6. The actual populated PCB.

Figure 6 shows the populated PCB, including circuits from Experiments 8, 9, and 10. Exp. 8 is the recreation of the power stage of a buck converter built on the breadboard in Experiment 8. Exp. 9 is the PWM generation circuit which will be built in this lab and enlarged in Figure 7. The PWM signal is generated by comparing a triangular waveform with an input voltage. The input voltage is chosen by placing a jumper at "VREF for PWM Triangle Wave" header. Three possible positions correspond to different input voltage sources: (position 1) from potentiometer Rc1, (position 2) from external reference voltage applied to pin JPWM_VREF1, and (position 3) from feedback output signal generated by the Exp. 10 circuit.

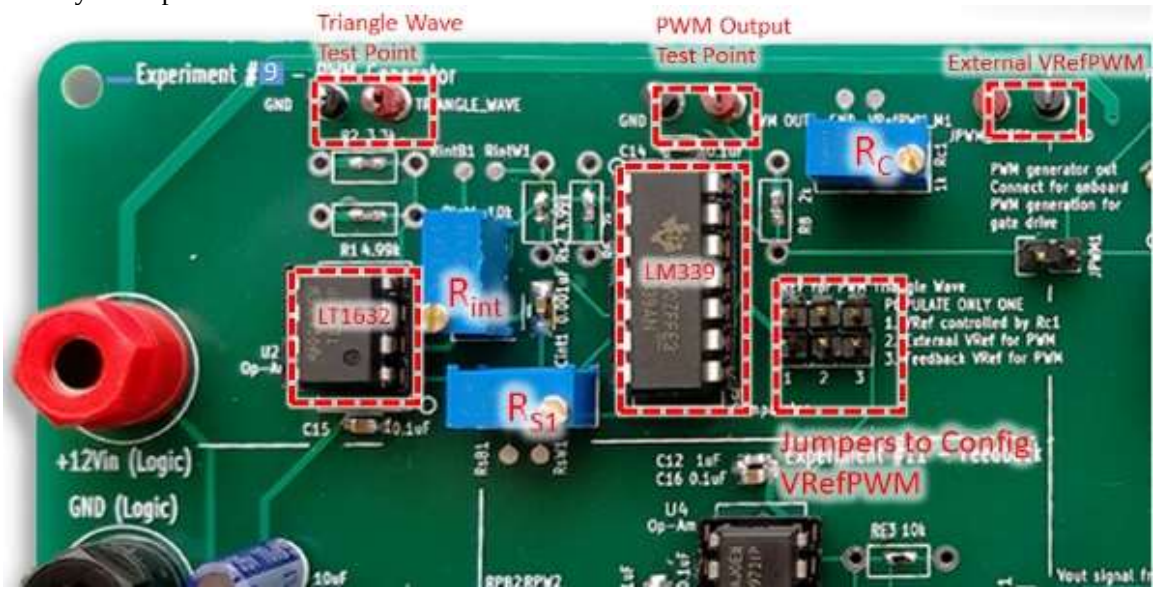


Figure 7. Key components on PCB

Figure 8 shows the schematic of the PCB for your reference.

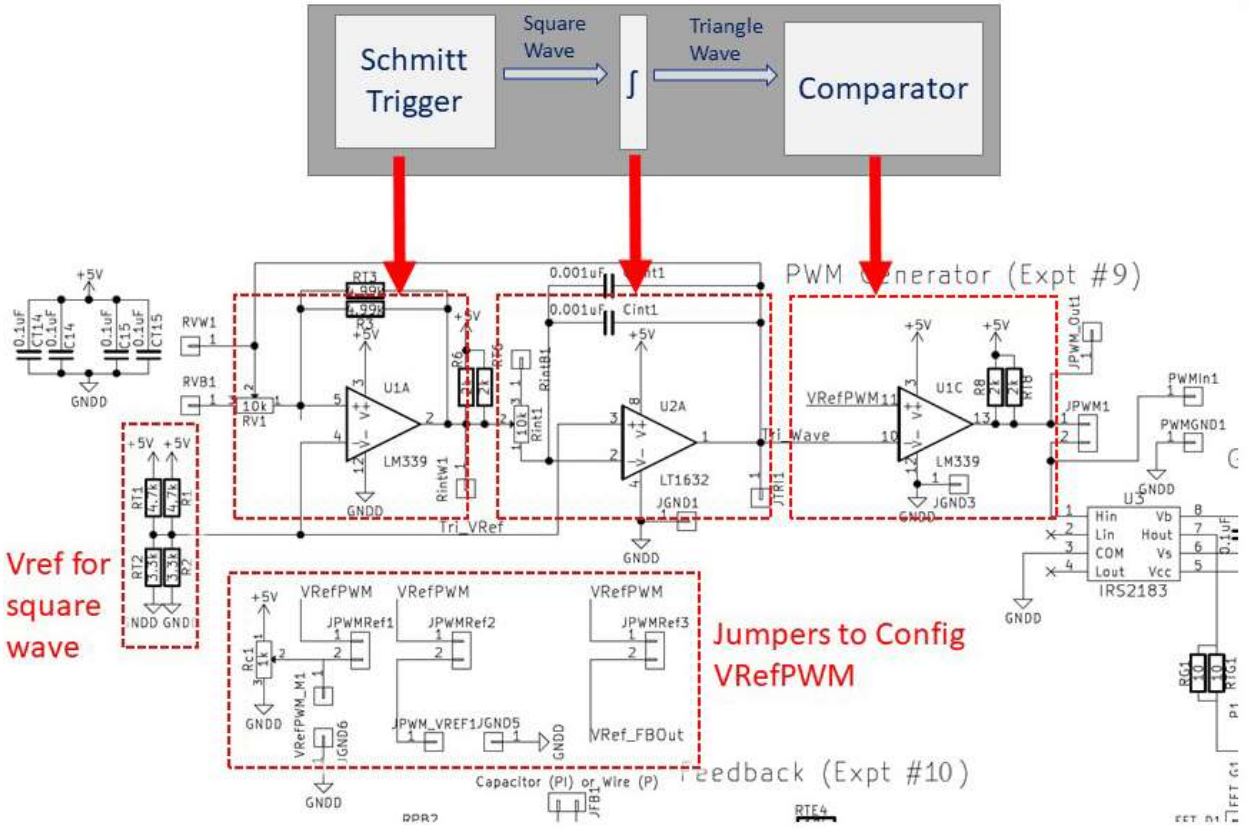


Figure 8: Schematic for the PCB used to generate PWM

Procedure —

Note: use two different power supplies, one for the logic circuit, and another one for the power circuit.

Part I: PWM Generation

1. Finish soldering "±5V Logic Power Supply" and "PWM Generator"/Experiment 9 sections as shown in Figure 6.
2. Provide a 12V supply for +12Vin (logic).
3. Generate 2.5V at VRefPWM using onboard voltage divider R_{C1}. Set jumper to the appropriate position (1).
4. Adjust width of hysteresis band by adjusting R_{S1}. Make the triangle waveform rising edge and falling edge close to linear and its minimum value close to zero.
5. Adjust the triangle waveform frequency to 50 kHz by adjusting R_{INT1}. Record the triangle waveform.
6. Adjust the voltage at VRefPWM to generate duty ratios of 10%, 50% and 90%. Save the PWM waveforms.

Generate a sinusoidal signal of frequency 60 Hz, amplitude 3 Vpp and centered at 2.5 V using function generator. Feed this signal to the VRefPWM by setting the jumper to appropriate position (2).

7. Capture a screen shot of at least one fundamental cycle of the PWM waveform, along with the triangle (carrier) and modulating (VRefPWM) waveforms on the same screen.

Part II: Buck Converter on the PCB

8. Finish soldering "Gate Driver and Buck Stage"/Experiment 9 section.
9. Connect 300 μH inductor and 25 Ω load to finish the buck converter.
10. Put the jumper at JPWM1 to connect PWM signal to the gate driver.
11. Provide 12V supply to Vin. Verify that you can generate an adjustable dc output voltage. Take screenshots of the PWM and Vout waveforms for duty cycles of 10%, 50%, and 90%. Note that the jumper for VRefPWM need to be set in the appropriate position (1).

Study Questions —

1. Comment on the harmonic performance of the PWM compared to the conventional square wave inverter.
2. Discuss what will happen if the modulation index/depth is chosen to be greater than 1. What effect will it have on the inverter output voltage?
3. Is there a disadvantage to making the triangle waveform amplitude too small?

References —

[1] J. Caldwell, "Analog Pulse Width Modulation" Texas Instruments Inc. [Online]. [Accessed Aug. 16, 2013]. (URL: http://www.ti.com/lit/ug/slau508/slau508.pdf)

ECE 469 — POWER ELECTRONICS LABORATORY

EXPERIMENT #10 — PWM Generation with PI Control

Objective — An analog PI controller will be constructed with the purpose of automatic output voltage regulation of the buck converter with high-side gate driver and PWM signal generator from experiments 8 and 9.

Pre-Lab Assignment — Read this experiment. Study the procedure, and bring any questions to class. No pre-lab assignment is due for Experiment 10. However, it is imperative that you come to lab prepared this week; make sure you already know how to answer study questions 1–4. The additional circuitry is just two op-amps and a few resistors; feel free to get a head start. Your instructor may elect to assign another problem.

Discussion —

Basic Theory of Converter Control: An essential function of a power converter is regulation. Regulation typically means good control of one or more of the converter's state parameters (e.g., output voltage, input current, etc.). The most common form of regulation is output voltage regulation. For instance, if your laptop's logic requires a steady 5 V, then the power converter must be equipped with a mechanism that can deal with variations in line voltage, load transients, component parametric drift, and several other converter disturbances and non-idealities while maintaining the desired converter state. Without regulation, any changes in load or line voltage could cause the output voltage to deviate from the desired 5 V output. This may exceed downstream device ratings and damage them.

Of the many ways to implement feedback control in a dc-dc converter, perhaps the simplest is called **output voltage control**. Here, the output voltage is sensed and subtracted from a given reference value, which generates the error signal, $e(t)$. This signal is amplified and applied to the duty ratio input of a PWM generation circuit to regulate the sensed output voltage to the desired reference voltage. Figure 1 illustrates this connection. Keep in mind that *low-power* voltages are easily generated for reference purposes with conventional analog IC design techniques, using band-gap references and linear regulators. However, these circuits provide only a limited amount of current and therefore are not suitable for high-power purposes.

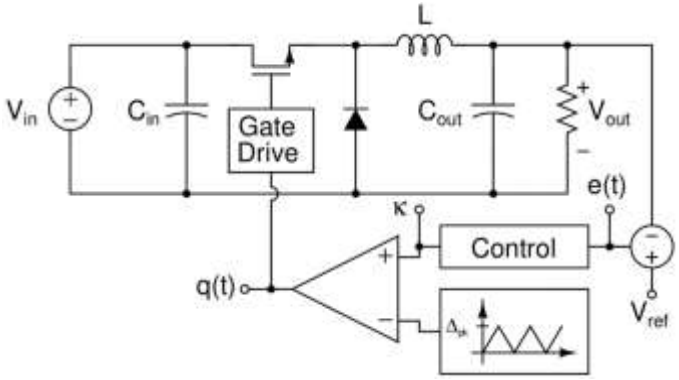


Figure 1. Buck converter with feedback

In a buck converter with **proportional control**, the control output κ is strictly proportional to the error signal:

$$e(t) = V_{ref} - V_{out}$$

$$\kappa = k_p \times e(t)$$

where k_p is the proportional gain of the control block in Fig. 1. **NOTE: V_{ref} is the set point, where you would like your output voltage V_{out} to be. This is not the same as VRefPWM you used in Experiment 10**. The output signal of the controller κ is used to generate the PWM signal in the comparator circuit above. In closed loop, VRefPWM is connected to κ by placing the jumper in position 3. Thus, the duty cycle of the resulting PWM signal is directly proportional to κ. With the addition of a normalization factor n, the resulting duty ratio is expressed as:

$$D = \begin{cases} 1, & \kappa \geq \Delta_{peak} \\ 0, & \kappa \leq 0 \\ n \times \kappa, & otherwise \end{cases}$$

The normalization factor is equal to the inverse of the peak voltage of the triangle waveform, denoted as Δ_{peak}, used in the comparator circuit. The equations above lead to the following input-output relationship for a buck converter with proportional control:

$$V_{out} = n \times k_p \times (V_{ref} - V_{out}) \times V_{in}$$

Rearranging:

$$V_{out} \times \left(1 + n \times k_p \times V_{in}\right) = n \times k_p \times V_{in} \times V_{ref}$$

If k_p is large such that $n \times k_p \times V_{in} \gg 1$, then both sides can be divided by $n \times k_p \times V_{in}$ to give:

$$V_{out} \cong V_{ref}$$

This result applies even when non-ideal voltage drops are included; a large gain will produce an output equal to the reference value, in spite of changes in supply, load, or other uncontrolled parameters, as long as the converter duty ratio is between 0 and 1.

While this style of proportional gain feedback is appealing and useful, it has two important drawbacks. The first is that the output is not exactly equal to the reference, even in steady state, because k_p cannot be infinite. This means that there will always be a non-zero tracking error. This is clear when you consider the relation for D above. Since we expect D to have some value greater than zero, the error result $V_{ref} - V_{out}$ must also be greater than zero. A second drawback is that if k_p is too large, then any slight disturbance will drive D to the limits 0 or 1, and the converter output will jump around almost at random.

A way to minimize the steady-state error is to combine the proportional feedback with **integral control** in the control block in Fig. 1**.** The signal $\int e(t)\,dt$ will rise when the error is positive, fall when it is negative, and remain constant when the error is zero. When the integrator drives the control parameter κ, the output voltage of the buck converter will vary in the previously described manner until the error is exactly zero. Notice that this effect is independent of gain. Even an integrator with very low gain will provide zero error in steady state. The combination of integral control and proportional control is termed **proportional-integral (PI) control**, and the control output can be expressed as:

$$\kappa = k_i \times \int e(t)\,dt + k_p \times e(t)$$

Printed Circuit Board

You will be continuing to build the PCB from where you left on Experiment 10. Figure 2 specifically shows the section on the PCB that you will have to solder this time. If Jumper JFBIN1 is shorted, the PI controller will sense the output from the buck converter in Experiment 9. If it is open, feedback is disabled. JFB1 is the connector for Ci for the RC compensation network for PI controller.

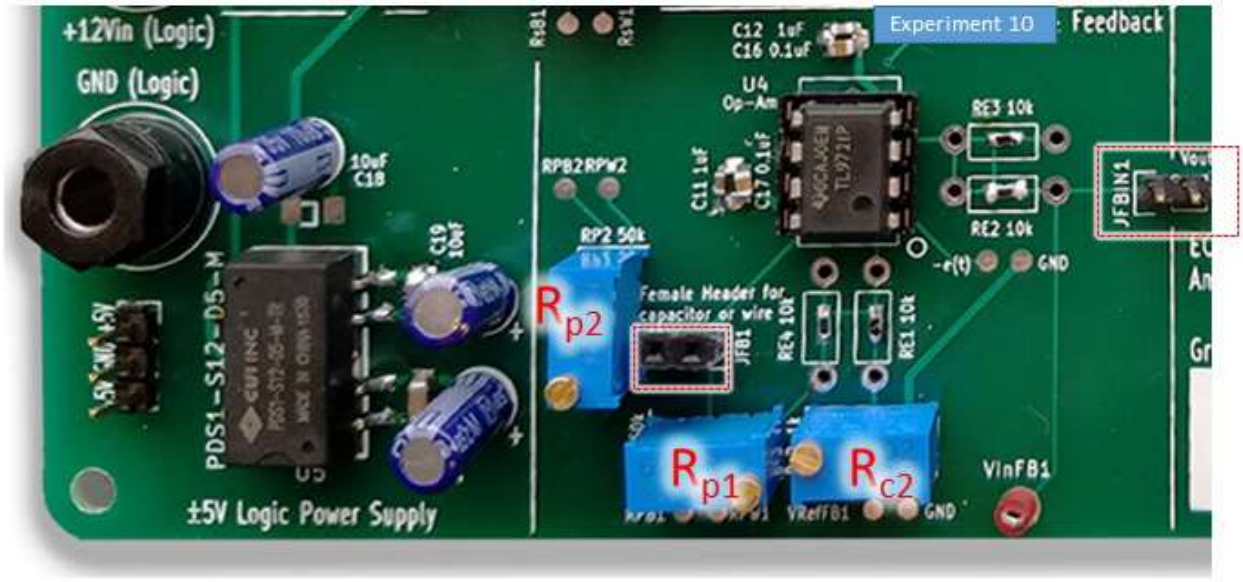


Figure 2. Key components on the PCB for the Feedback Control

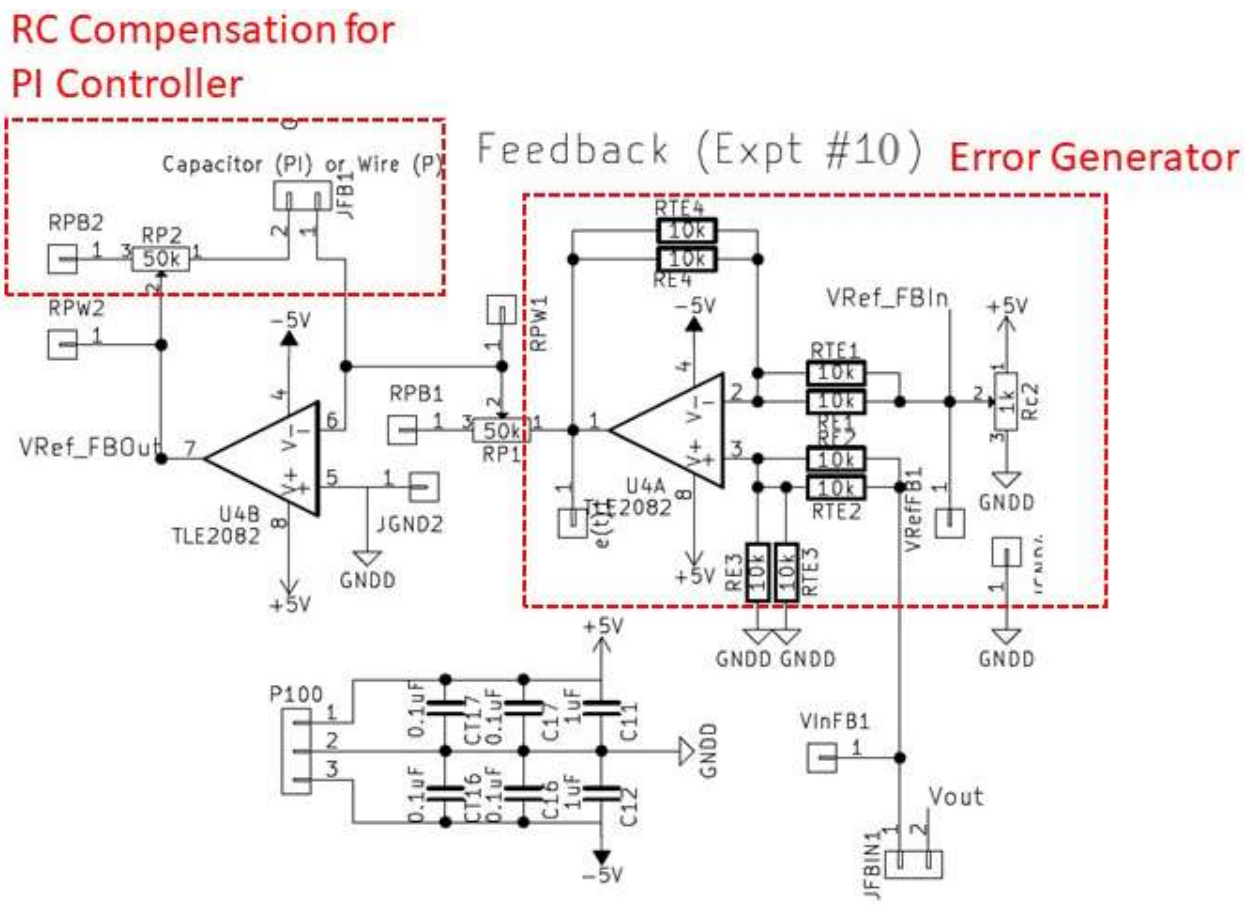


Figure 3. Schematic for the PCB used to create the PI controller using OpAmps

Figure 3 shows the schematic of the PCB and should help you to identify the potentiometers R_{P1}, R_{P2}, and R_{C2}, that will be used to change the reference output voltage through R_{C2}, proportional gain by adjusting the ratio R_{P2}/R_{P1} and integral gain by adjusting the ratio $1/CR_{P1}$.

Procedure —

Note: Use two different power supplies, one for the logic circuit, and another for the power circuit. If the controller does not work, try turning off and turning on the logic supply while keeping the power circuit supply on.

Part 1: Error Generation

1. Finish soldering the "Feedback" section. Place the jumper on VRefPWM to 1, such that you can use R_{C1} to adjust the open-loop duty ratio. We will close the loop after we have verified that your error generation is working.
2. Put the jumper at JFBIN1 to feed the Vout to the control circuit.

3. Provide 12V supply for +12Vin(logic). Generate 50 kHz PWM signal with 30% duty cycle using the onboard PWM generation module and feed this signal to the gate driver.
4. Set controller reference voltage at VRefB1 to 3.5 V.
5. Connect 300 µH inductor and electronic load to finish the buck converter.
6. Set the electronic load in controlled current mode and set the load current to be at 1.5A.
7. Provide 12 V supply for Vin with current limit of 2A.
8. Take screenshots of the error generator output –e(t) for Vout = 3 V, 3.5 V and 4 V. Note that Vout is changed by varying R_{C1}. Do the error measures agree with the expected values?

Part 2: Proportional (P) Control

9. With the power off, set RP1 and RP2 to 10 kΩ. Note that resistance measured between points RPB1 and RPW1 (or RPB2 and RPW2) is the portion of the potential meter that is not used in the circuit.
10. Short the two terminals of JFB1. We are not using integral control yet.
11. Set the jumper appropriately so that VRefPWM is connected to output of the controller (Position 3).
12. Supply 12V to +12Vin(logic). Set VrefFB1 to 3.5 V and observe the output of your proportional controller (at VRefPWM). Does this value agree with what is expected? Do not continue until you get the expected value.
13. Provide 12V to Vin. Note that power supply for logic circuit and power circuit are separate. Observe Vout and –e(t) for Vin = 10 V, 12 V and 14 V. Repeat for Iout = 1 A and 2 A. Measure average Vout and e(t). Take one screenshot for Vin = 12 V and Iout = 1.5 A.
14. With Vin = 12 V and Iout = 1.5 A, vary the proportional gain k_p of the P controller. This could be done by adjusting RP2. Comment qualitatively on the impact of k_p on the error. Take screenshot of Vout and –e(t). Measure the Vout and –e(t) average for your maximum k_p, record values of k_p, RP1 and RP2.
15. Keep your k_p between 1 and 2.5 before proceeding to the next part.

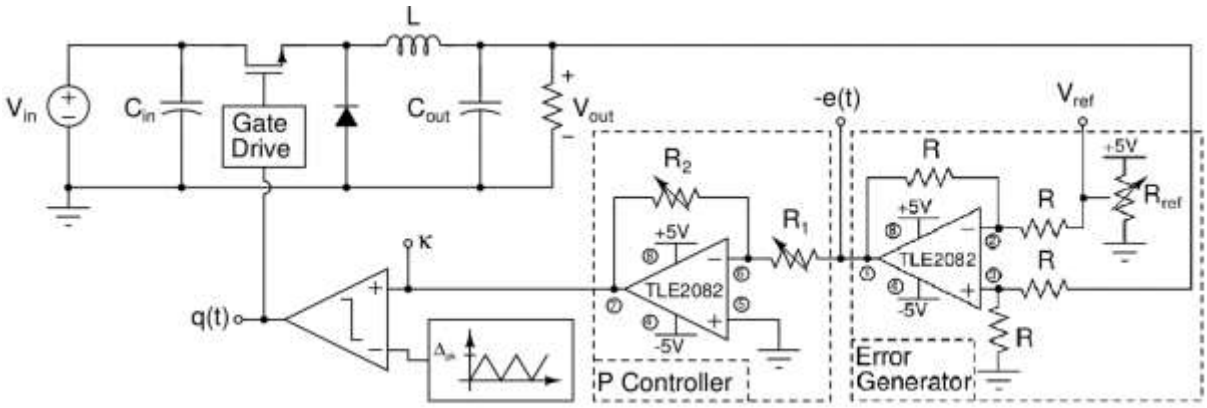


Figure 4. Proportional control circuit

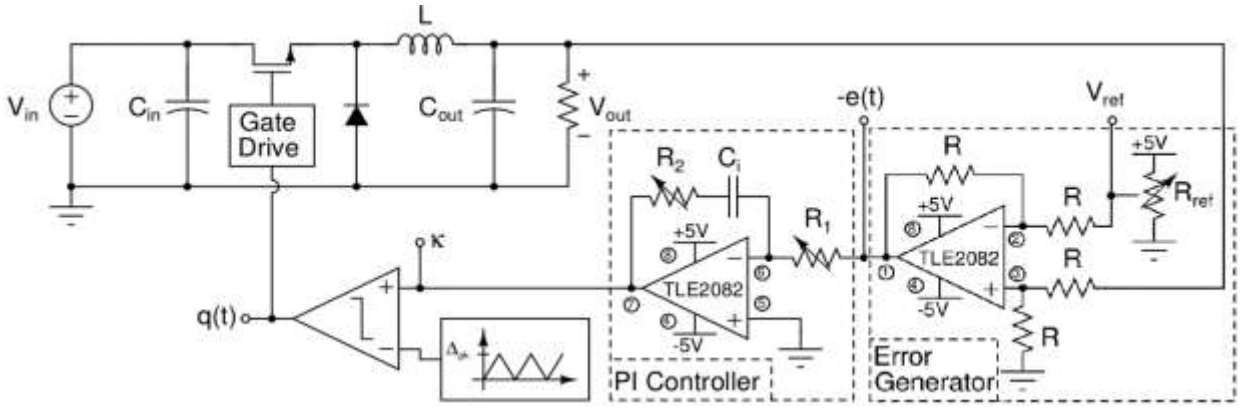


Figure 5. Proportional-integral control circuit

Part 3: Proportional-Integral Control

16. Power off the circuit. Put a 1 µF capacitor across JFB1 to create a PI controller. This capacitor is C_i.
17. Supply 12V to +12Vin(logic). Set VrefB1 = 3.5 V.
18. Repeat step 13 of Part 2. Pay attention to improvement on the average –e(t). Also, pay attention to the output of your controller i.e. the signal going into VRefPWM.
19. If time permits, tune your gains until you are satisfied with the tracking/dynamic performance. Record any relevant data.
20. If your controller does not work, try (1) turning off and turning on the logic supply while keeping the power circuit supply on, or (2) short the capacitor C_i momentarily. (Can you tell, why? Hint: Think about initial condition.)

Study Questions —

1. Derive an expression of the output of the error generator in Fig. 2 in terms of V_{ref}, V_{out}, and R. Keep in mind that this output is the *negative* of the desired error signal, $e(t)$. Why are we generating the negative of the error signal?
2. Find an expression for κ in Fig. 3 in terms of $e(t)$, R_1, and R_2. The R_1 and R_2 variables can be grouped as the proportional gain variable k_p. Write down this expression, and calculate k_p for the initial and final R_1 and R_2 values.
3. Describe how the proportional control works, starting from error generation to PWM signal ($q(t)$) generation.
4. Find an expression for κ in Fig. 4 in terms of $e(t)$, R_1, R_2, and C_i. The R_1 and C_i variables can be grouped as the integral gain variable k_i. Write down this expression, and calculate k_i for the initial and final R_1 and C_i values.
5. Compare your data and observations from the proportional control experiment with those of the proportional-integral control experiment. Analysis should be both qualitative and quantitative.

ECE 469 — POWER ELECTRONICS LABORATORY

EXPERIMENT #11 — Digital Control of a Power Converter

Laboratory Objective: To replace the analog controller of Experiment 10 with a digital one to control the output voltage of a buck converter

Learning Objective: First, be able to recognize the control framework in a digital controller. This framework is widely used—if you understand the framework, you should be able to apply it in many applications, including senior design projects. Second, compare the performance of a digital controller versus the analog controller in Experiment 10, while the system you are controlling (a buck converter) remains the same. Last, this lab will help you understand the debugging process of an embedded system.

Pre-lab assignment: We will use digital control platform from ECE 110—Arduino. You must have the code that works in a standalone mode before you come to the lab. Try it on an Arduino so that it can compile successfully. There won't be time to debug and do the experiment if you don't come with a working code. The PWM frequency should be 50 kHz.

Your code should have these three modules individually tested (rudimentary). The building blocks are detailed in the discussion section.

Module 1 (Error generator):

Input: Reference signal: r(t) at A1 (analog input) Feedback signal: u(t) at A0 (analog input)	**Output:** Error e(t): An integer variable that ranges between 0-1023
Building blocks: Analog Read, Serial command (needed for testing only)	
Standalone verification test: Try out with on-board 0 and 5 V. Connect ground to analog Pin A0 and connect 5V to analog Pin A1. Use analogRead() to read these values. Calculate the error with equation ($e(t) = 3r(t) - 3u(t)$) and output the error to the serial monitor. The error should be 3069 (multiplication of 3 will be explained later).	

Module 2 (PI Controller with an output limiter):

Input: Error e(t): An integer variable that ranges between 0-1023	**Output:** Duty ratio d(t): An integer variable that ranges from 0 to 1023. Combined with output limiter, the target output range is reduced to 0-1002
Building blocks: micros (measuring sample time), Bilinear Transformation, Serial command (needed for testing only), Anti-windup.	

Standalone verification test: In theory, the output duty ratio ranges from 0 to 100%. However, because the MOSFET has a turn-on and turn-off time, the ratio between output voltage and input voltage is no longer linear at an extremely low and high duty ratio. For Arduino Uno, the valid range of duty ratio that can successfully generate the desired PWM signal is slightly reduced. Test K_p first. Set the error e(t) to be 100, K_p to 0.05 and K_i to 0. You will find that the duty ratio is 5 in Serial Monitor as output. Test K_i. Set the error e(t) to be 1 and K_p to 0. Try K_i at 0.00005 and 0.005 and see the resulting D in Serial Plotter. You will see that higher K_i gives a greater slope. You might not see a linear curve. because the Serial Command. might interfere with the millis() command. As a result, sampling time may not be the same in each loop execution.

Module 3 (PWM Block):

Input:	**Output:**
Duty ratio d(t): An integer variable that ranges between 0-1002	Switching signal q(t): A digital output at Pin 5 of the Arduino-Uno. Takes value either 0 or 5 V.
Building blocks: Timer 0	
Standalone verification test: Try this with an LED: Use open-loop control and connect the LED across digital Pin 5 and Ground. Set the duty ratio d(t) to 10, 100, 300 to see the difference in the LED brightness (green LED preferred). Note that for low duty ratio d(t), (for example, in the range of 0-6 in the 0-1023 scale), the output PWM waveform is 0%. The actual value may vary from board to board.	

Discussion: For analog control as we did in previous labs, PWM frequencies, duty ratio and k_p, k_i are adjusted by potentiometer. Physical tuning is required if any of these values need to be changed. Some of you might need to short the capacitor to initiate the control. In modern power electronics, PWM is usually generated digitally. For Experiment 11, let's look at how the modern controller is implemented. We will use Arduino.

Figure 1 is an implementation of an Arduino-controlled buck converter. We will use only the power section of the PCB board from Experiment 8, which consists of a gate driver, MOSFET, diodes, and filter capacitors. Since the output ranges from 0 to 12V, which is beyond the measurement range of the Arduino Board, a resistor divider is needed. In this case, we will choose resistor R1 and R2 such that the voltage division is 3. Similarly, the reference voltage is also scaled down by a factor of 3.

The Arduino takes REF and FDB signals from an external source and the output voltage, respectively, and it outputs the PWM signal to the gate driver. When you use Analogread() function to read voltages from Pin A0 and A1, don't forget to multiply the result by 3!

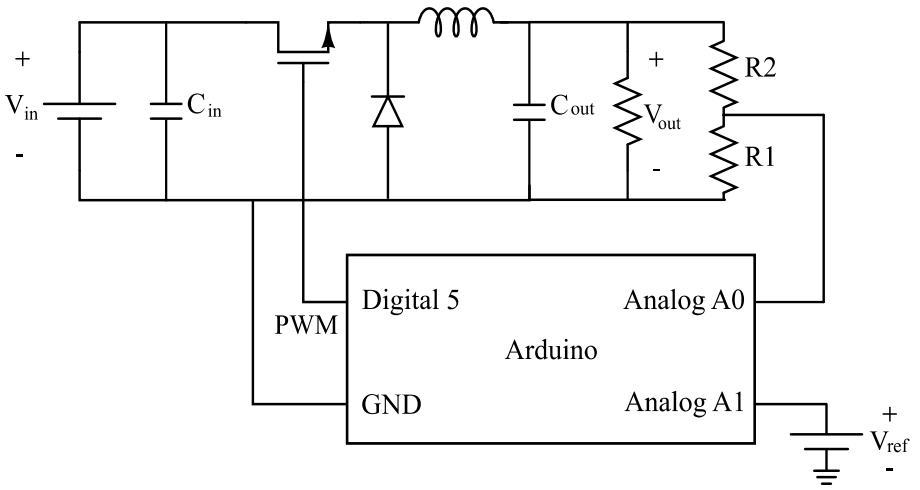


Fig. 1 High Level Description of the Digital Controlled Buck Converter.

In Experiment 10, the output PWM is generated through a comparator. As a comparator must have a reference signal, triangular wave is implemented. Digital control via Arduino, on the other hand, need no reference. In addition, the feedback circuit in experiment 10 is incorporated into the Arduino. Using digital control has the additional advantage of implementing algorithms too difficult or complex to do in analog, such as anti-windup PI control. For example, additional programming can be done to achieve DCM operation as well, which adds significant flexibility. In addition, applying digital control can reduce hardware usage. As a result, reliability is improved (fewer components means a smaller possibility of hardware failure).

There are also challenges in digital control: limited input voltage range and onboard storage (for example the maximum reading voltage is 5V for Arduino Uno), sampling time is not exact (either in this lab's method of using built in function millis() or a more advanced interrupt method).

Arduino Building Blocks: These functions are: micros(), Serial, analogRead(). A detailed description and example code can be found in Arduino Reference library [1].

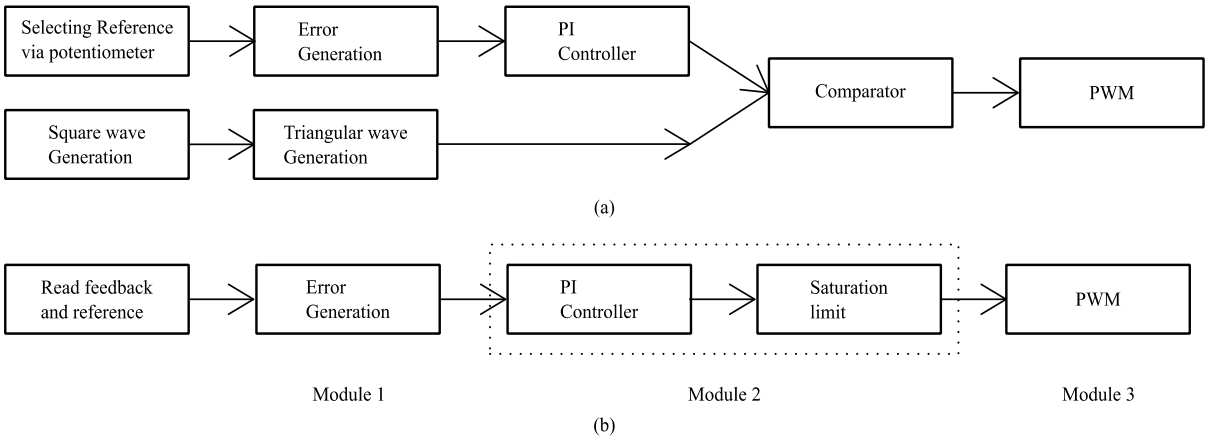


Fig. 2 Block Diagram of the (a) Analog Controller and (b) Digital Controller

A. micros(): No input is needed. This function simply returns the number of microseconds since the Arduino board began running the current program. We use two micros() functions to calculate the time gap of each loop, namely the sampling time.
B. Serial Command: Communicating with the Arduino in real time. We will use the Serial.print() and Serial.println() functions to upload the data we want to observe to Tools/Serial Monitor and Tools/Serial Plotter to test if the program is running as expected.You should be able to understand the usage of these two functions after reading Arduino reference library [2].
C. analogRead(): Reads voltage from the specified analog pin and returns an integer ranging from 0 to 1023, mapping voltage from 0 to 5V in linear scale. We are using analogRead() to measure the reference and feedback voltage.
D. millis(): Returns the number of milliseconds passed since the Arduino board started to run the current program. Arduino can run approximately 50 days before overflow.
E. Timer: Arduino has three timers (timer0, timer1 and timer2). These can be configured to do either PWM or interrupt. We will use an example from Gammon using timer0 to generate PWM at phase correct mode. The original link can be found at [3].

```
#include <TimerHelpers.h>
void setup() {
  pinMode (timer0OutputA, OUTPUT);
  pinMode (timer0OutputB, OUTPUT);
 TIMSK0 = 0;  // no interrupts
 Timer0::setMode (5, Timer0::PRESCALE_1, Timer0::CLEAR_A_ON_COMPARE |
Timer0::CLEAR_B_ON_COMPARE);
  OCR0A = 200;    // number of counts for a cycle
  OCR0B = 150;    // duty cycle within OCR0A
}  // end of setup
void loop() {}
```

Fig. 3 Sample code for Timer0 from Gammon [3].

Output Compare Register (OCR0A) determines the PWM frequency. The ratio between Output Compare Registers (OCR0A and OCR0B) determines the PWM duty cycle.

$$f_{pwm} = \frac{f_{clock}}{2 * OCR0A}$$

$$Duty\ Ratio\ = \frac{OCR0B}{OCR0A}$$

With a system clock of 16MHz, the PWM frequency and duty ratio can be calculated as:

$$f_{pwm} = \frac{f_{clock}}{2 * OCR0A} = \frac{16MHz}{400} = 40kHz$$

$$Duty\ Ratio\ = \frac{OCR0B}{OCR0A} = \frac{150}{200} = 75\%$$

F. Digital PI Controller & Bilinear Transformation:

Since we cannot use components like a capacitor to perform a simple integral in the digital domain, you will need to dig into the control theory a bit. We will use Bilinear transformation to perform integration and an anti-windup method to prevent saturation.

Bilinear transformation allows conversion of continuous-time transfer functions to discrete-time domains thereby enabling digital signal processing. In a digital controller, we often use bilinear transformation to perform the integral from sampled data points (z-domain). In the Laplace domain, the integral is performed by $\frac{1}{s}$, and the bilinear transformation is $s = \frac{2}{T}\frac{z-1}{z+1}$, where T is the sample time. For our case, the integral control can be expressed as: $D_I(s) = \frac{K_I \times e(s)}{s}$.

Performing bilinear transformation:

$$D_I(z) = \frac{\mathrm{T}}{2}\mathrm{K_I}[\mathrm{e(z)} + \mathrm{e(z-1)}] + D_I(\mathrm{z-1})$$

If you are interested, here is the link to learn more: [4]

The proportional part is more straightforward. The proportional gain K_p will produce $D_p(z) = K_p \times e(z)$.

Adding the integral and proportional gain, our overall duty ratio input to the PWM block becomes:

$$D(z) = D_I(z) + D_p(z)$$

The above equation is only true if the integrator does not hit the saturation limit and falls within its allowable range. Since the duty ratio can only range from 0 to 100%, $D(z)$ must be within 0 to 1023.

G. Anti-windup Method:

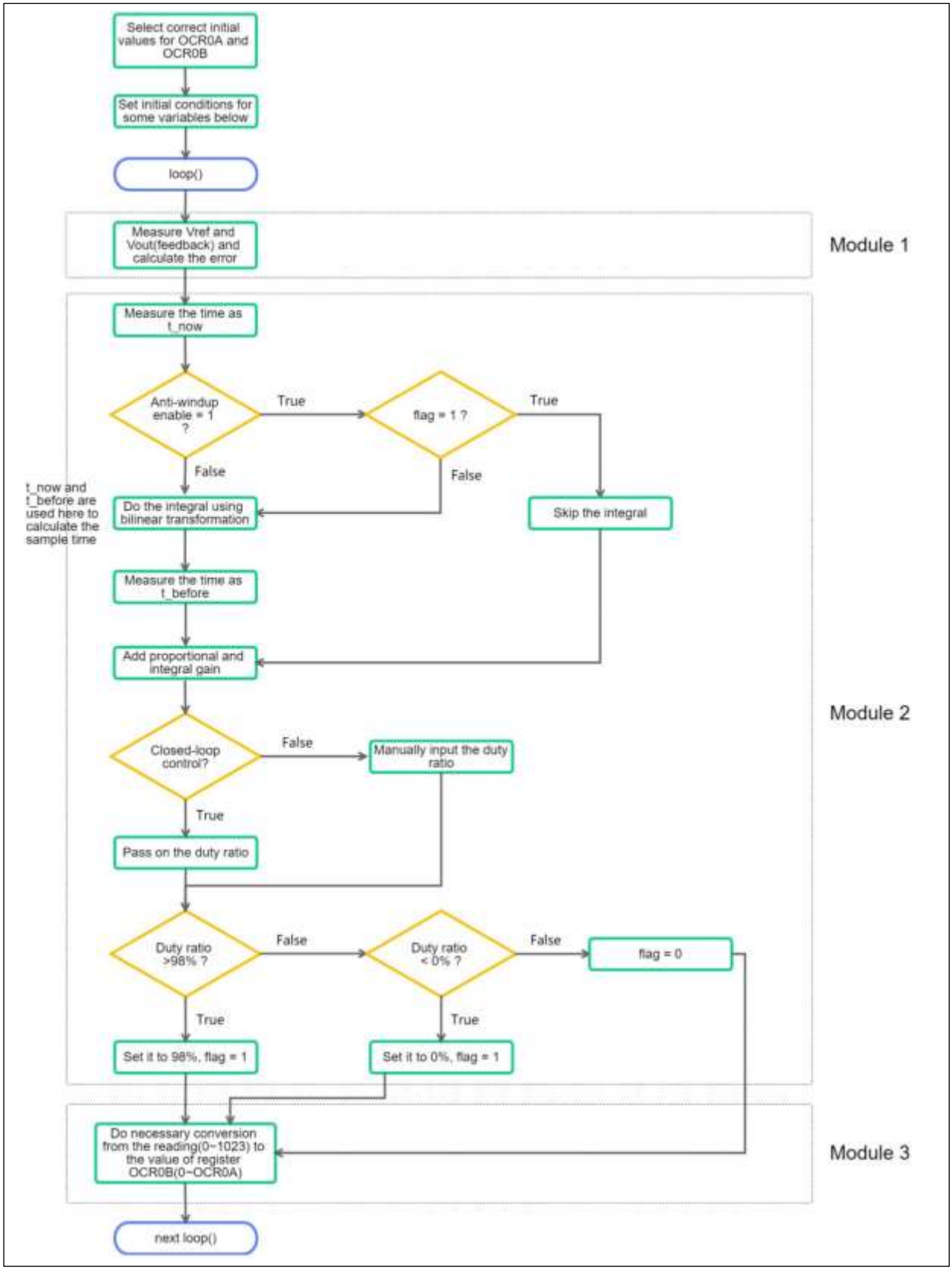
As the PI controller is working, the integrator may get saturated. A typical way to avoid this is by using the anti-windup method. When the system has accumulated more error than it can process, the integral will be paused so that the system will be able to recover a normal error level.

$$D(z) = D_p(z)$$

Please follow reference [5] to learn and implement the simplest anti-windup method.

If you are struggling with writing the code or want to dig out more about using an Arduino, there is a paper [6] that might be useful. Flowchart and Pseudo code are provided below:

Flowchart


Select correct initial values for OCR0A and OCR0B
Set initial conditions for some variables below
loop()
Measure Vref and Vout(feedback) and calculate the error
Module 1
Measure the time as t_now
Anti-windup enable = 1 ?
True
flag = 1 ?
True
False
False
t_now and t_before are used here to calculate the sample time
Do the integral using bilinear transformation
Skip the integral
Measure the time as t_before
Add proportional and integral gain
Module 2
Closed-loop control?
False
Manually input the duty ratio
True
Pass on the duty ratio
Duty ratio >98% ?
False
Duty ratio < 0% ?
False
flag = 0
True
True
Set it to 98%, flag = 1
Set it to 0%, flag = 1
Do necessary conversion from the reading(0~1023) to the value of register OCR0B(0~OCR0A)
Module 3
next loop()

Pseudo Code:

```
In the loop() function:

    ref = Read(A1);         //Record the reference input

    fdb = Read(A0);         //Record the feedback input

    err = K * (ref - fdb);  //Here the error is K times the difference of reference and feedback, you need to think about what
    K is

    if close-loop:

        t_now = Readtime();

        P_out = KP * err;       // P_out is the proportional output

        if (AW_enable)          // AW_enable is an boolean to trigger anti-windup

            if (!flag)

                I_out += bilinear_integral * conversion_1;      //Anti Wind-up is tested here. Use t_now and t_before here.
            And conversion_1 is 1e-6, to convert the us scale to s scale.

        else

            I_out += bilinear_integral * conversion_1;

        t_before = t_now;     //Remember to set an initial value for t_before

        last_err = err;

        D = P_out + I_out;

    else:  //open loop

        D =          ; //Enter your duty ratio here

    if (D > 98%) //Mind that 98% should be a level of the duty ratio in a certain scale. You probably won't directly use 0.98
    as an upper limit here.

        D = 98%;       flag = 1;

    else if (D < 0%)

        D = 0%;         flag = 1;

    else

        flag = 0;

    PWM = conversion2(D); //Convert the D from 0-1023 scale to 0-OCR0A scale, depending on the magnitude of OCR0A

    OCR0B = PWM +1;    //Compensate for the rounddown in these conversion
```

Procedure:

Part 1: Arduino Sensing Verification and Open Loop Control

1. Verify the open-loop duty ratio of the output generated by Arduino Digital PWM Pin 5. Connect Pin 5 to Oscilloscope and capture the PWM waveform with 30% duty ratio. Your switching frequency should be 50kHz.
2. Connect the circuit as instructed in Figure 5 and 6. Set the electronic load in controlled current mode and set the load current to 1.5A. Provide 12V supply for V_{in} with current limit of 2A.
3. Since the range of V_{out} is from 0-12V, which is greater than the range of the analog input of 0 to 5V, use the resistor divider to divide V_{out} by the scale of $1/3$. The range of the new V_{out}' is from 0 to 4V. Then connect $V_{out}'$to Analog A0.
4. Set the controller reference voltage to 3V using dc power supply (+6V output).
5. Calculate and print the digital error (with the necessary conversion from the readings) on the Serial monitor for D = 30%, 40% and 50%. Also measure the output voltage on the oscilloscope. Note that V_{out} is changed by varying D. Do the error measures agree with the expected values?

Part 2: Proportional (P) Control

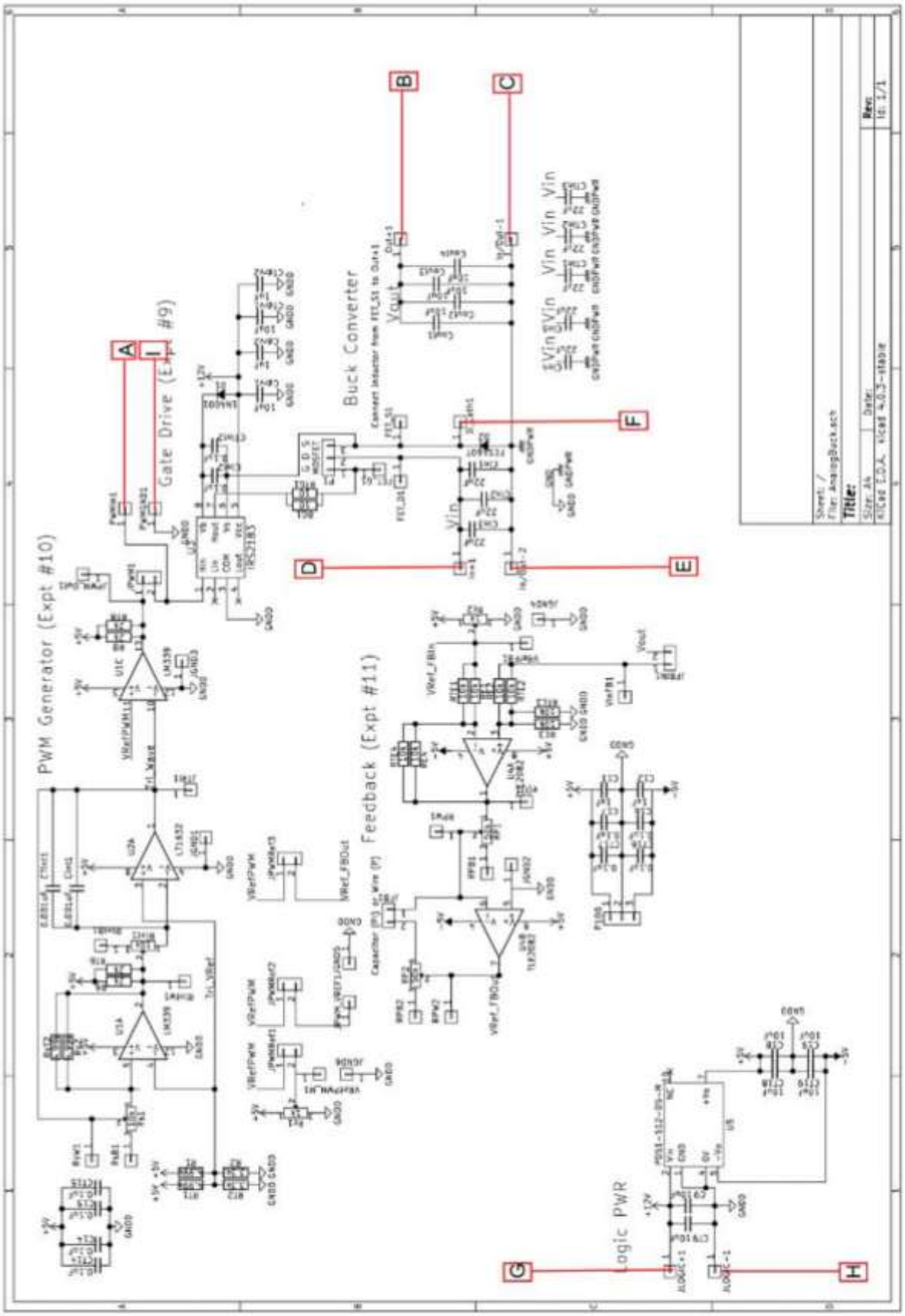
6. Turn Keithley DC supply off, upload Arduino using the closed-loop control method with k_p=0.5 and k_i=0.
7. Supply 12V to +12Vin. Observe the output of your proportional controller. Does this value agree with what is expected? You may need to adjust the value of KP slightly from 0.5 to make it work.
   If the output is not stable, change your KP to stabilize it.
8. Observe V_{out}, error average and duty ratio for V_{ref} = 3V, 6V, 9V and 11V. Repeat for I_{out} = 1A and 1.5A and tabulate these results. Take one screenshot for V_{ref} = 9V and I_{out} = 1.5A.
9. Set V_{ref} = 6V and I_{out} = 1.5A, varying the proportional gain k_p of the P controller (KP in the code). Comment qualitatively on the impact of k_p on the error. Take a screenshot of V_{out} and error average for your maximum k_p that can stabilize the output; record values at that k_p.

Part 3: Proportional-Integral Control

10. Power off the circuit. Change the value of k_i to roughly 5×10^{-5} and a stable k_p in Part 2. Supply 12V to +12Vin.
11. Repeat step 8 of Part 2. Record error average improvement.

12. Find the execution time of each loop. You may want to send a low signal at the beginning and a high one at the end of the loop and output the signal to the oscilloscope. Be careful of the Serial commands. Record the average execution time.
13. If time permits, tune your gains until you are satisfied with the tracking/dynamic performance. To find an optimal k_i, do a single trigger when the i_{out} goes from 0.5A to 1.5A. Take a screenshot of i_{out} and V_{out}.

See Fig.5: "Ports on the PCB Board with External Connections" on page 102.


Buck Converter
Gate Drive (Expt #9)
PWM Generator (Expt #10)
Feedback (Expt #11)
Logic PWR
Vin
Vout
A
B
C
D
E
F
G
H

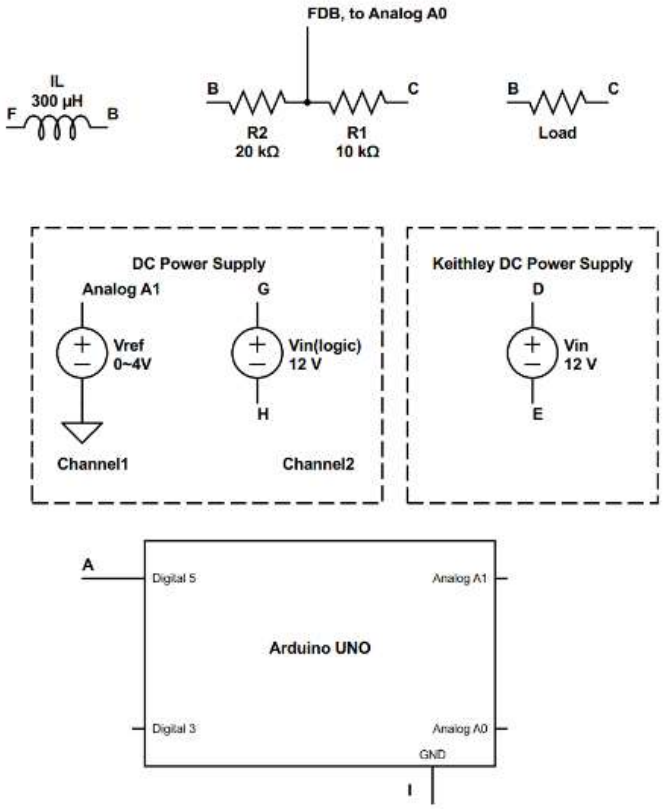


Fig. 6 Ports of External Devices Connected to the PCB Board

Study Questions:

1. Compare the digital and analog errors in Part 1. Is there a relationship between these values?

2. In Part 2, with i_{out} and k_p fixed, explain why changing the reference voltage will result in a different error. What is your maximum k_p?

3. Why is the error is reduced if you switch from P control to PI control?

4. (Optional) How long does it take for V_{out} to reach steady state (±5%) when you change the load from 0.5A to 1.5A? What is the fluctuation of V_{out} that deviates from the reference value? Does V_{out} ever become greater than the reference value? Explain why.

References:

[1] Arduino Reference: https://www.arduino.cc/reference/en/

[2] Arduino Reference: Serial.print()
https://www.arduino.cc/reference/en/language/functions/communication/serial/print/

[3] Gammon Forum : Electronics :Microprocessors :Timers and counters
http://www.gammon.com.au/forum/?id=11504&reply=3#reply3

[4] Bilinear Transformation:
https://en.wikibooks.org/wiki/Digital_Signal_Processing/Bilinear_Transform

[5] Understanding PID Control, Part 2: Expanding Beyond a Simple Integral
https://www.mathworks.com/videos/understanding-pid-control-part-2-expanding-beyond-a-simple-integral-1528310418260.html

[6] *L. Müller, M. Mohammed and J. W. Kimball, "Using the Arduino Uno to teach digital control of power electronics," 2015 IEEE 16th Workshop on Control and Modeling for Power Electronics (COMPEL), Vancouver, BC, 2015, pp. 1-8.*

URL: http://ieeexplore.ieee.org/stamp/stamp.jsp?tp=&arnumber=7236487&isnumber=7236433

APPENDIX

Standard EIA Decade Resistor Values —

5% tolerance:

1.0	1.1	1.2	1.3	1.5	1.6	1.8		
2.0	2.2	2.4	2.7	3.0	3.3	3.6	3.9	
4.3	4.7	5.1	5.6	6.2	6.8	7.5	8.2	9.1

1% tolerance:

1.00	1.02	1.05	1.07	1.10	1.13	1.15	1.18	1.21	1.24	1.27	1.30
1.33	1.37	1.40	1.43	1.47	1.50	1.54	1.58	1.62	1.65	1.69	1.74
1.78	1.82	1.87	1.91	1.96							
2.00	2.05	2.10	2.15	2.21	2.26	2.32	2.37	2.43	2.49	2.55	2.61
2.67	2.74	2.80	2.87	2.94							
3.01	3.09	3.16	3.24	3.32	3.40	3.48	3.57	3.65	3.74	3.83	3.92
4.02	4.12	4.22	4.32	4.42	4.53	4.64	4.75	4.87	4.99		
5.11	5.23	5.36	5.49	5.62	5.76	5.90					
6.04	6.19	6.34	6.49	6.65	6.81	6.98					
7.15	7.32	7.50	7.68	7.87							
8.06	8.25	8.45	8.66	8.87							
9.09	9.31	9.53	9.76								

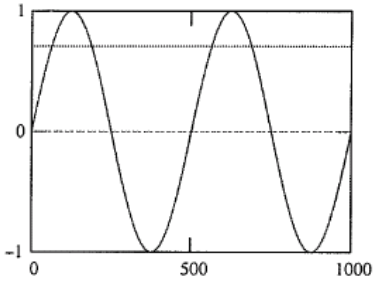
Color codes —

Black: 0	Brown: 1	Red: 2	Orange: 3	Yellow: 4
Green: 5	Blue: 6	Violet: 7	Gray: 8	White: 9

As multipliers, all these are 10^n. Gold indicates 10^{-1} and silver 10^{-2}.

Common Waveforms —

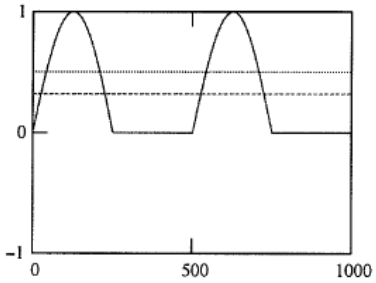
Sine Wave



AVE: $\frac{1}{T}\int_0^T \text{vsin}(i)\,di = 0$

RMS: $\sqrt{\frac{1}{T}\int_0^T \text{vsin}(i)^2\,di} = \frac{1}{\sqrt{2}} = 0.707$

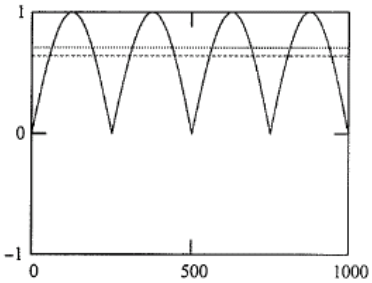
Half Wave Rectified Sine



$\frac{1}{T}\int_0^T \text{HWR}(i)\,di = 0.318$

$\sqrt{\frac{1}{T}\int_0^T \text{HWR}(i)^2\,di} = 0.5$

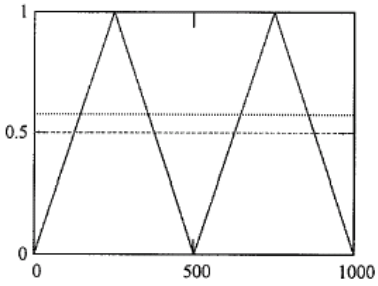
Full Wave Rectified Sine



$\frac{1}{T}\int_0^T \text{FWR}(i)\,di = 0.637$

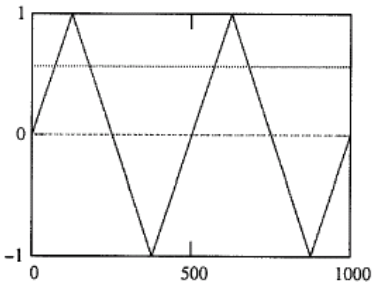
$\sqrt{\frac{1}{T}\int_0^T \text{FWR}(i)^2\,di} = \frac{1}{\sqrt{2}} = 0.707$

Triangle wave



AVE: $\frac{1}{T}\int_0^T \text{tri1}(i)\,di = 0.5$

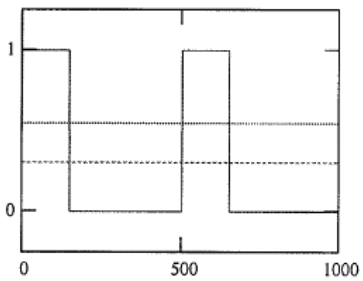
RMS: $\sqrt{\frac{1}{T}\int_0^T \text{tri1}(i)^2\,di} = \frac{1}{\sqrt{3}} = 0.577$



$\frac{1}{T}\int_0^T \text{tri2}(i)\,di = 0$

$\sqrt{\frac{1}{T}\int_0^T \text{tri2}(i)^2\,di} = \frac{1}{\sqrt{3}} = 0.577$

Switching function



$\frac{1}{T}\int_0^T q(i)\,di = D$

$\sqrt{\frac{1}{T}\int_0^T q(i)^2\,di} = \sqrt{D}$

(example: $D = \frac{1}{3}$)