

Laboratory Instruction Guide

EE 391 Electrical Engineering Laboratory II

Department of Electrical Engineering
University of Saskatchewan
57 Campus Drive
Saskatoon, Saskatchewan
S7N 5A9



© University of Saskatchewan
2001

Compiled and Revised 2001 by Dr. Mark Nerdoly
Revised 2002 by D. Lynch, J. Kowalski
Revised 2003 by D. Lynch, J. Kowalski
Revised 2004 by D. Lynch, J. Kowalski, B Fogal, A. El Damaty
Revised 2005 by D. Lynch, J. Kowalski, S. B. Ko
Revised 2006 by D. Lynch, J. Kowalski, H. Majid
Revised 2007 by D. Lynch
Revised 2008 by D. Lynch
Revised 2010 by D. Lynch
Revised 2011 by D. Lynch – added James Kowalski's safety instruction in Power Factor Correction Lab

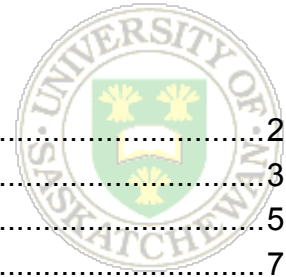


Table of Contents

Table of Contents	2
Marking Scheme and Notebook Guidelines	3
Privileges and Responsibilities of Electrical Engineering Students	5
Electrical Safety and Hazardous Materials	7
Electrical Engineering Laboratory Regulations	8
Release	9
1 Design Lab I: A Switch Mode Power Supply	10
1.1 Safety Advisory:	10
1.2 Introduction:	10
1.3 Part 1: SMPS Investigation and Analysis	10
1.4 Part 2: Design and testing of a regulated SMPS.....	12
1.5 Design Considerations:	17
1.6 Design Example I:	23
1.7 Design Example II:	25
1.8 Appendix A – Derivations Background.....	27
1.9 References:.....	31
2 Design Lab II: An Automatic Power Factor Correction System	32
2.1 Safety Advisory:	32
2.2 Background: Reactive Power and Power Factor Correction in Power Circuits ..	34
2.3 Part 1- Power Factor Calculation and Measurement	35
2.4 Part 2 – Voltage Drop and Control	36
2.5 Part 3 – Power Factor Correction.....	37
2.6 Part 4 – Design of a PFC System	38
3 Design Lab III: An Electronic Governor for Electric Motor Speed Control	39
3.1 Safety Advisory:	39
3.2 Introduction:	39
3.3 Part 1.....	40
3.4 Part 2: Design and testing of an electronic governor	42
3.5 Background Theory:.....	44
3.6 Design Examples:	48
3.7 References:.....	64
4 Appendix A: Data Sheets.....	65

Marking Scheme and Notebook Guidelines



Your grade in EE 391 is *tentatively* based on the following:

- Lab notebook keeping, Lab Reports ($\cong 40\%$)*
- Midterm exam ($\cong 20\%$)
- Final exam ($\cong 30\%$)
- Formal interview (final oral exam) ($\cong 10\%$)

**Also includes observation of your day-to-day performance in the lab; e.g. observing laboratory rules and procedures, proper set-up, handling and storage of equipment, attendance and punctuality, professional demeanor, division of responsibility with partner/teamwork.*

Check the Class Web Site for a final version of the criteria and weighting for grade determination. This will be posted prior to the start of laboratories in September.

Your lab notebooks and reports will be evaluated from time to time to help determine a grade and to provide feedback on recording techniques. You will normally have a one week notice to submit your lab book; your lab book will be returned one week after that. All labs **must** be performed, including the appropriate documentation, in order to pass the class. If one or more labs have not been performed, then a grade of "INC" (incomplete) will be submitted.

Lab notebooks can be considered to be fulfilling the same functions as logbooks in industry. Logbooks are used to record the results of all tests performed on systems, subsystems and equipment during the various phases of a project including R & D, design, systems integration, etc. Logbooks are official, permanent documents, and can be used in court to prove ownership of a design!

The following points must be followed when recording information in your lab notebooks:

- The first page must contain a table of contents.
- All pages in the notebook must be numbered.
- Formal structure is not critical; logical order is important.
- Try to avoid using pencil.
- Legibility and neatness are not inherently important, but in this setting others will definitely have to be able to read and interpret your notes, so please give due consideration to these aspects. In a non-academic setting, your notes will be primarily for your own use, but there will be circumstances when others will have to read and interpret what you've done, so get in the habit of keeping legible, orderly notes.

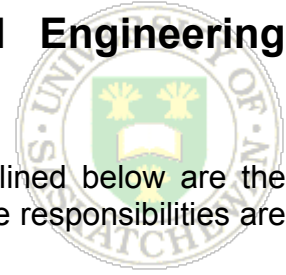
- The lab book is standalone. There generally should be no references to any outside documents. (There are, of course, exceptions such as common data sheets, although you may want to include important data such as pin outs or certain specifications for convenience.) Remember that typically you are only allowed to bring in your lab notebooks for exams. They are your reference notes—make sure they're complete!
- Theory and background information **must** be completed prior to the lab.
- Cross out unwanted or erroneous material with a single large X. Do not remove any pages from your lab book.
- It will be a very useful and time-saving skill to be able to record any relevant data in your notebooks directly as you do your work. If you do need to “re-write” your notes, use the left-hand page for rough calculations, notes, measurements, etc., and use the right-hand side for the revised version. Explanatory, directive notes will help both you and other readers find relevant information quickly.
- It is acceptable to paste material into your lab book; such as graphs, plots, experimental waveforms, and schematics. Use glue wherever possible; tape is acceptable, please don't use staples!
- One lab partner must have the original of any experimental waveform; his/her partner may have a photocopy of that waveform.
- Fully label all diagrams and schematics; include information about any equipment used!
- Interpretation and discussion of results and/or conclusions resulting from each portion of the lab should be found with that portion if it is reasonably independent of other parts. The most reasonable format will depend on the specifics in each lab exercise.

If in doubt about what to include (and how), remember that it should be clear, concise and *complete*. A good rule of thumb is “if you're not sure – include it”.

Procedures for reports and handing in and returning notebooks may be revised from year to year. Check postings on the EE391 class web site for the latest procedures and guidelines.

The examinations are intended to test your knowledge of experimental procedures including how to take adequate and complete notes. Questions about the related theory may also be included, but the primary focus will be on the practical aspects of technology as encountered in the laboratory. It will also be beneficial to become familiar with the test equipment available in the undergraduate laboratories and how to use it properly.

Privileges and Responsibilities of Electrical Engineering Students



You are responsible. Most of the laboratory policies outlined below are the result of student requests. The privileges will be available only if the responsibilities are accepted.

Electrical Engineering students may use the laboratories during regular school hours and after hours, provided they (a) assume responsibility for all activities in the lab and (b) relinquish equipment to other lab groups if they are performing a scheduled lab. If you intend to use the lab while other groups are scheduled to perform a lab, ask the instructor in charge if it is okay, and if there is a place where he/she would like you to work. **The labs must not be left unlocked and unoccupied even for quick washroom or coffee breaks, and the last student leaving the lab(s) must ensure that they are locked.** Students **must not work alone** with circuit potentials exceeding 25 V.

Electrical Engineering students wishing to use the labs after hours when the lab is locked but the building is open may obtain permission and a key to the labs from Mr. Dave Karaloff in room 2C94. This permission will not be granted for times when the building is locked. The building is normally open from 7:30 am – 11:00 pm Monday to Saturday and from noon – 6 pm on Sunday. It is locked and “out of bounds” to students on University holidays. It must be emphasized that **all** students working in the labs after hours are responsible for **all** activities in the labs. For weekend access to the labs, fill out the signup sheet outside 2C94. The student with the earliest Saturday time of arrival on the signup sheet should pick up the key for the labs.

You may leave a circuit set up, provided that you leave a piece of paper with your name and the date on the breadboard. Storage space for loaded breadboards is provided in 2C82. **All test equipment (scopes, power supplies, signal generators, DMMs, etc.) must be returned to their proper locations.** Any boards more than one week old may be dismantled or the student may be asked to do so. The extended setup privilege may be withdrawn if it is consistently misused.

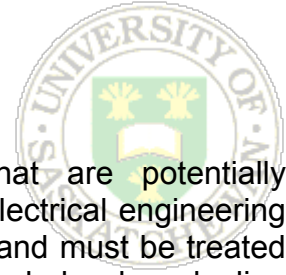
The department technicians, Mr. Dave Karaloff, Mr. Serge Nazarenko, Mr. James Kowalski and Mr. RLee Prokopishyn are located in 2C94. Report any equipment problems directly to them.

All equipment in the labs has been marked with a department of electrical engineering identifier code (the EE xxxx) number, as well as a location code. The location code tells you where a piece of equipment should be returned. For example, if a HP 54600A scope has the code “**82 – E7**”, then that scope should be returned to room **2C82**, the cabinet marked **E**, and the **7th** shelf down from the top. At that shelf, there

should be another tag corresponding to the EE number on the scope itself. **Always return the equipment to its proper location.**



Electrical Safety and Hazardous Materials



General

Any laboratory environment may contain conditions that are potentially hazardous to a person's health if not handled appropriately. The electrical engineering laboratories obviously have electrical potentials that may be lethal and must be treated with respect. In addition, there are also mechanical hazards, particularly when dealing with rotating machines, and some chemical hazards because of the materials used in various components. Our objective is to educate all laboratory users to be able to handle laboratory materials and situations safely and thereby ensure a safe and healthy experience for all.

Safety Procedures

A summary of safety and materials handling procedures is included in this section of the Laboratory Manual. In addition, regulations and procedures may be posted in and around the laboratories. Please familiarize yourself with these procedures and follow them at all times.

There are certain categories of hazard that are present in the Electrical Laboratories, depending on the particular equipment being used and the procedures involved. A summary of the hazards specific to each laboratory procedure is included at the beginning of the procedural write up for that lab. Be sure you are familiar with the appropriate safety procedures referred to in these summaries. Ask your instructor or teaching assistant if you are not clear.

You may be required to successfully answer (verbal or written) questions about, or physically demonstrate proper safety procedures.

Electrical Engineering Laboratory Regulations



General

Rules and regulations for related to use of the laboratories are established by the Manager, Electrical Engineering Undergraduate Laboratories with the approval of the Department. Check the class web site or postings in 2C94 for the current regulations.

Remember, students working the electrical laboratories do so at their own risk. Electric potentials used are dangerous to life. Students must observe all safety precautions and must not work alone with circuit potentials exceeding 25 V.

Smoking, food and drinks are not permitted in the laboratories at any time, with the exception of drinking water in spill-proof containers.

Equipment

All equipment must be returned to its proper storage location at the end of the lab period. Circuits may be left set up for short periods of time only by permission from the department technicians (room 2C94).

Instruments and equipment are delicate and expensive. Improper use of equipment can result in instantaneous damage; check with you instructor if you are in doubt. The manuals for each piece of equipment can be obtained from the technicians in 2C94.

Damaged or faulty equipment should be given to the technicians along with a detailed description of the problem. **Do not return damaged or faulty equipment to the shelves.**

Release

Sign and return one copy of this page on the first day of your lab.

I, _____ acknowledge that certain laboratory processes may involve dangerous substances, procedures or materials (such as, but not limited to, electricity). I accept this risk and agree to abide by all safety precautions and laboratory regulations. I release the University and its employees from any damage or loss, of any nature or kind, which I may suffer as a result of my participation in this laboratory class.

DATED this _____ day of _____, 20____.

Witness

Student's Signature

1 Design Lab I: A Switch Mode Power Supply

(Denard Lynch, September 2010)



1.1 Safety Advisory:

The activity prescribed in this laboratory will be conducted in an environment where hazardous electrical potentials exist. The student should be aware of normally expected electrical laboratory hazards and follow procedures to minimize risk. Please refer to general safety precautions in the Laboratory Manual and those posted in the labs.

This laboratory exercise presents the following specific hazards:

- Shock: collapsing fields in inductors and transformers can produce extremely high potentials even with low input source voltages; allow for appropriate discharge paths and insulate yourself appropriately when contacting leads or components.
- Burn: circuit elements under load, especially abnormal load caused by circuit or design errors, can reach temperatures which will cause burns to skin if contacted. Do not attempt to determine the temperature of elements with your fingers! Approach potentially overheated elements with caution and allow sufficient time for cooling when necessary. In some cases, elements may burst into flames. This is usually very isolated and does not normally create a fire hazard. If overheating to flames do occur, immediately remove power and *determine and correct the cause before re-energizing!*
- Explosion: Capacitors that are subjected to potentials exceeding their rating can burst their cases. This especially applies to electrolytic capacitors commonly used as power supply filter elements. Electrolytic capacitors are typically polarized, and if they are connected with the polarity reversed, even below rated voltage, can explode on occasion. Make certain that all electrolytic capacitors are connected *with the proper polarity* before energizing the circuit.

1.2 Introduction:

Switch mode power supplies (SMPS) have become the architecture of choice for power conversion because of their economy, higher efficiency and lighter weight. In this lab you will investigate the basic operating principles of a switch mode power supply, and then design, build and test a simple supply given a few basic components.

A considerable amount of background theory and design advice is included following the Procedure description below. Please study this material to prepare for this lab.

1.3 Part 1: SMPS Investigation and Analysis

The first part of this lab involves investigating the principles of operation of a SMPS and the parameters that you will employ in the design.

1.3.1 Procedure:

Design and breadboard a simple inductor based flyback converter similar to that in the sample circuit given in Figure 1.1 below. To allow some flexibility and range for testing, design for an output voltage, V_o of approximately $\frac{1}{2}$ of V_{in} (approximately 10 - 12V) Utilize a bench DC power supply (25VDC) as an input source for the time being. You may also need to design a variable duty cycle driving circuit to drive the transistor switch, or simply use a suitable function generator (if necessary, check with your instructor about which function generators can provide an adjustable duty cycle). Construct your circuit on a prototype breadboard (available from the Tech office). Although each partner-group need construct only one circuit, each member of the team should be familiar with and participate in any circuit construction. (Practical exam questions will require each *individual* to demonstrate circuit construction and troubleshooting skills.)

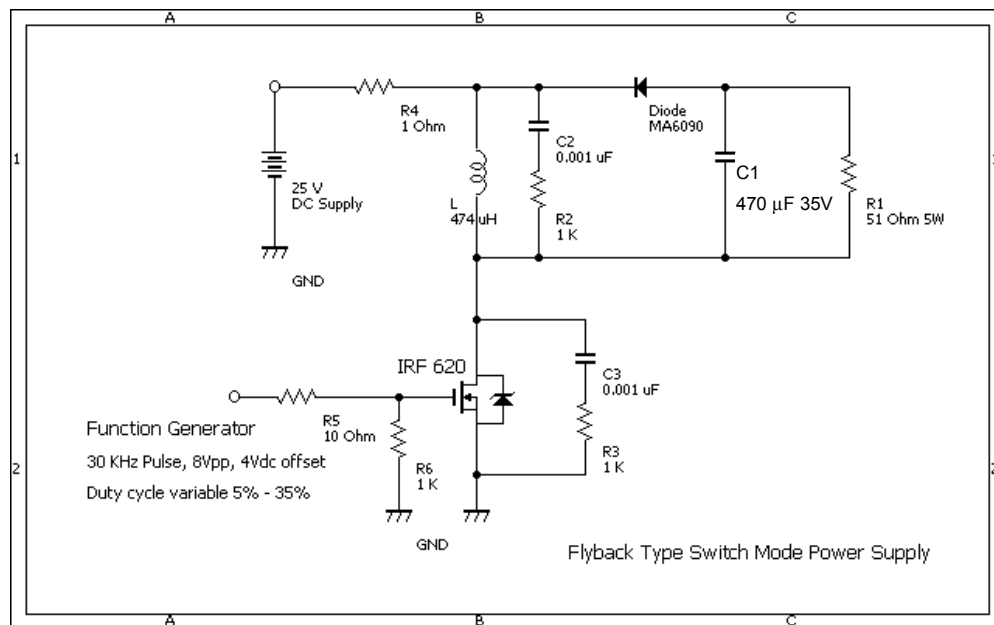


Figure 1.1

There are two additions in the sample circuit that are not included in the basic configuration described in Figure 1.2: a 1Ω current sense resistor in the supply line to monitor input current, and some R - C snubber networks to dampen ringing and protect the devices from excess voltage and current transients. If your driving circuit is capable of sufficient voltage to keep the FET turned on with a small lift in the source voltage (e.g. 8 or 9V), you may be able to put the current sense resistor between the source and ground. Use snubber networks to dampen the ringing, and thus excessive voltages, across the FET switch to avoid component damage.

Use a multi-channel oscilloscope and monitor: input current through your sense resistor; the output voltage, V_{DS} (which will indirectly give you V_L); and the gate drive voltage. (If a four-channel 'scope is not available, two two-channel scopes will provide the same information.) Be sure to monitor parameters like V_{DS} and I_L closely when you turn the power on. If you don't see the expected traces, reduce (or remove) power and re-check your circuit wiring.

(Note: Read the entire procedure, including background theory, before you start, and plan your investigation to make efficient use of your time and circuit setup.)

- 1) For this first step, maintain a constant period (frequency) and vary the duty cycle. Start with a load resistance that will only lightly load the supply and a very short duty cycle and then adjust the duty cycle slowly until the target output voltage is reached. (There are some high-current wire-wound adjustable load resistors available in the lab.) **Carefully** increase the load current (by decreasing the load resistance) and note the duty cycle adjustments necessary to maintain the output voltage at the target level. Graph the load current versus the duty cycle for a constant output voltage over a partial range of output current (say $.25$ to $.75$ of $I_{O\ Max}$, where $I_{O\ Max}$ is the output current at peak power output, or $P_{O\ Max}/V_{O}$). Discuss how this can be used to regulate the output under varying load conditions.
- 2) Attempt to regulate the output by keeping a constant *pulse width* and varying the operating frequency (period). Again, start with a low load current. Re-adjust your circuit by lowering the frequency first (probably something in the range $15 - 20\text{kHz}$), and then increasing the duty cycle to bring V_o back up to your target voltage. Now increase the load current and then while keeping the pulse width (i.e. on time) constant, increase/decrease the frequency (period) to maintain V_o . Note: Be sure you have the signal generator set up to provide a constant *width*, not *duty cycle*. Graph load current versus frequency and compare the potential for regulation with the varying pulse width technique examined in 1).
- 3) Also observe and measure the output voltage ripple at various load currents (e.g. $.25$, $.5$, $.75I_{O\ Max}$). (Remember, the output cannot be connected to the same ground as the input!) Compare the ripple with your calculations. Briefly discuss the advantages and disadvantages of pulse-width versus frequency-based regulation with respect to output ripple.
- 4) Set your circuit to maintain the target output voltage with a load current of approximately $\frac{1}{2}$ the maximum. Now adjust your input supply voltage (up and down $10 - 20\%$) and observe and record the change in output voltage. Adjust your drive circuit to maintain your target output voltage at the low and high input voltage. Discuss the “worst case” scenario from a design perspective; what conditions will define the limits of operation?

1.4 Part 2: Design and testing of a regulated SMPS

For the remainder of this lab, you will be asked to design, build and test a regulated Switch Mode Power Supply. The restrictions you will be required to work within are as follows:

- the input must be from the $120/24\text{VAC}$ step-down transformers available in the laboratory
- the integrated regulator circuit available to you is from STMicroelectronics’s UC384xB family: specifically, the UC3845B

- a switch mode transformer is available: TS223-Y01MS (or a TS227-Y05MS, TS202-Y03MS, or TS222-Y01MS) from Coil Winding Specialists. While these vary slightly in specifications, they are all designed for approximately the same output power and voltages.
- remaining components must be from laboratory stock (A reasonable selection of suitable components is available. Some parameters may have to be adjusted slightly to conform to component spec's.)

Additional information on these components will be available from the manufacturer, on the EE391 web site, or from your instructors.

1.4.1 Design Objective:

Your objective will be to build either a 5V or 12V, 5W regulated supply with a maximum 1% ripple at full load, maintained over a $\pm 10\%$ input voltage variation. If the available components are such that you must design for a slightly different output voltage, discuss this change in objective with your supervisor. It is more important that you understand the design process and characterize the supply you build than to have a specific output voltage. Note that you will also have to test your supply to verify the specifications, and to characterize other parameters such as its response to step changes in load current and how long it can maintain the output if the input supply is interrupted.

1.4.2 Report

You may be required to submit a report of your results (semi-formal report format). This report should cover your initial investigation into the theory of operation and regulation, as well as your design objectives, restrictions, decisions and results. Each student must write their own report, even though much of the data may be in common with your lab partner. A sample Table of Contents will be posted on the class website to guide your report development.

The due date for this report will be announced and posted on the EE391 web site. Note: Material covered in this laboratory will be examinable.

1.4.3 Background:

There are a number of different names used to describe different architectures and variations of switch mode power supplies, but they may be categorized in one of two basic types: the flyback or buck-boost converter, and the forward or buck converter. A third common designation is the push-pull converter, which is a variation of the forward converter. They all operate on the same basic principle of temporarily storing energy from a source in the magnetic field of an inductor (or the inductance of a transformer), and “extracting” this energy at the desired voltage or current. They also typically use a small, light inductor by exploiting the relationship between frequency, flux and voltage represented by: $E_{p-\max} = 2\pi f N_p \Phi_{\max}$. They typically regulate the output by varying the amount of energy “loaded” into the magnetic field in order to accommodate variations in the input supply or the output load. During the investigative part of this lab, you will look at either a flyback or a forward configuration.

1.4.4 Flyback Converters:

A flyback converter is a fairly simple architecture that is easy to construct and manipulate. To understand the basic design, considering the inductor storage element is a good starting point. You may recall that the energy stored in an inductor is:

$$W = \frac{LI^2}{2}, \quad (1)$$

and that the voltage across an inductor is:

$$v_L = L \frac{di}{dt}. \quad (2)$$

Using these basic relationships from inductor theory and simple circuit analysis techniques such as Kirchhoff’s Voltage Law, you can understand how a flyback converter operates and calculate the requirements for the required components. The basic flyback configuration is shown in Figure 1.2. When the switch is closed (i.e. $V_{sw} = 0V$), V_{in} is present across the inductor (i.e. $v_L = v_{in}$), L , and the diode, $D1$, is reverse biased (i.e. no current flows to the output). The current builds up in the inductor

according to the expression given in Equation (2) (re-arranged): $di = \frac{v_{in}}{L} dt$. If V_{in} and L are at least reasonably constant, and the current starts at $0A$, it will rise linearly as long as the switch is closed (Δt) and the *inductor does not saturate*, up to some peak current, I_{peak} , letting us express this change as:

$$I_{peak} = \frac{v_{in}}{L} \Delta t. \quad (3)$$

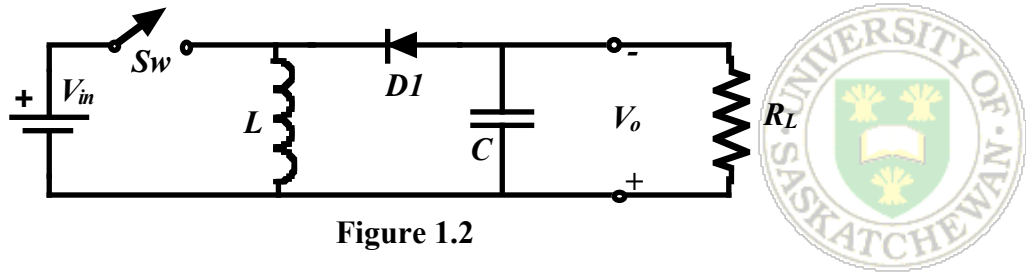


Figure 1.2

When the switch is opened, the magnetic field that was built up in the inductor core material will collapse, inducing a voltage across the inductor of a polarity *opposite* to V_{in} which will forward bias $D1$ and allow current to flow, delivering energy to the load, R_L , and charging the output filter capacitor, C . The current generally decreases linearly again according to Equation (2), and using KVL we find that V_L must be equal to $(V_o + V_{D1})$, where V_{D1} is the forward voltage drop across $D1$ while it is conducting. We can rearrange this equation to find the time required for the current, and thus the magnetic field, to fall from I_{peak} to zero:

$$\Delta t = \frac{L}{V_o + V_{D1}} I_{peak} \quad (4)$$

It is important that we eliminate any residual build-up of magnetic flux in the inductor core from one charge cycle to the next so the inductor does not saturate. To do this, we must either allow sufficient time for the current to fall to zero each cycle or employ other techniques to dissipate any remaining energy in the core.

The advantage of the flyback configuration is that the full input voltage, V_{in} , can be used to “charge” the inductor, regardless of the output voltage, and the inductor will generate what ever voltage is require to discharge this energy when the field collapses. Thus, the output voltage, V_o , may be either lower (buck, or step-down) or higher (boost, or step-up) than V_{in} . The amount of charge put into C each cycle will depend on the energy that was stored in the inductor (given in Equation (1)), which is proportional to I_{peak} which in turn is dependent on the length of time the switch was closed, Δt . By adjusting the *length of time* or *how often* the switch is closed, we can provide just enough energy to the output capacitor to make up for the current that was drawn by whatever load, R_L , we might attach to the output (ignoring the initial transient stage where the output voltage builds up to its target). Of course, between charges the voltage across the capacitor will decrease, giving V_o a “ripple”. The amount of ripple will depend on the size of C , the load current, and how often the energy is “replenished”.

There are two other considerations with the simple configuration shown. First, there is no electrical isolation between the input and output. When dealing with off-line power supplies (i.e. those connected to a commercially supplied source such as the power available from standard wall receptacles), V_{in} could easily be several hundred volts, which presents a safety hazard. The second issue is that V_o is inverted with respect to V_{in} , which will make it much more complicated to monitor V_o and adjust the parameters of the “charge” cycle and regulate the output accordingly. Figure 1.3 shows a simple version of the flyback configuration that uses a transformer as the inductance which

addresses both of these problems. The operation is identical except that the energy built up by current in the primary when the switch is turned on is recovered through current flowing in the secondary when the switch is turned off. The other “improvement” made in Figure 1.3 is that we’ve used a FET (field effect transistor) as a switch and put it in the ‘ground’ side of the circuit. This facilitates turning the “switch” on and off very quickly with an electrical signal. You can also see that, because the secondary is electrically isolated, we are no longer faced with the same safety hazard, and the negative side can potentially be connected to the same electrical ground as the switch control circuitry making it easier to use feedback from the output to control the switch (In practice, an opto-isolator is often used to provide additional isolation between the output and input, especially when the control circuitry is powered from V_{in} .)

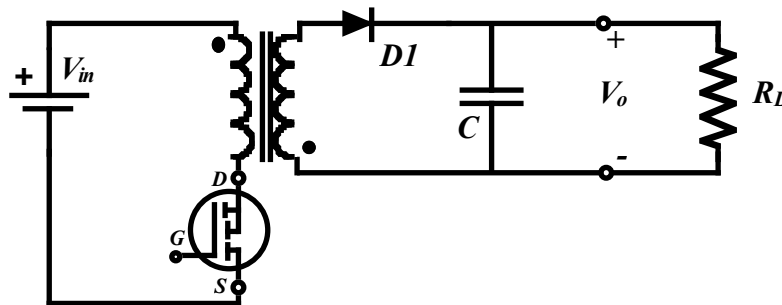


Figure 1.3

One more advantage of this modification is that we can use the turns ratio, n , of the transformer to limit or adjust the available voltage present on the load side of this circuit.

1.4.5 Forward Converters:

The forward converter also works on the same basic principles as the flyback converter. A slightly different component arrangement allows current to flow to the output during both the charging and discharging cycles of the inductor. A basic forward configuration is shown in Figure 1.4.

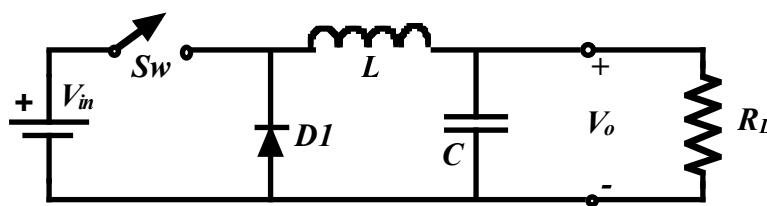


Figure 1.4

When the switch is closed, diode $D1$ is reverse biased and current will again increase through the inductor according to Equation (2). This current will also flow through the load, R_L , and potentially increase the charge on the output filter capacitor, C . When the switch is opened and the field collapses, the voltage across the inductor will be the opposite polarity to that in the charging part of the cycle, and again contribute energy to the load and filter capacitor. One important difference in this configuration is that the voltage impressed across the inductor is the *difference* between V_{in} and V_o . Thus the relationship shown in Equation (3) for the “charging” phase in the flyback converter must be modified for the forward converter and becomes:

$$\Delta i = \frac{V_{in} - V_o}{L} \Delta t \quad (5)$$

This relationship shows that the output voltage can clearly never be greater than the input voltage, and thus this specific forward converter configuration is limited to step-down (or buck) applications. Once again the current will decrease linearly (described generally again by Equation (2)) once the switch is opened over a time described by Equation (4).

While the polarity of the output is the same as the input in the forward converter's case, we still have the potential safety issue of no input-output electrical isolation. Again, the use of a transformer as the inductor can provide the required isolation as well as, by the appropriate design of the turns ratio, potentially allow a step-up (boost) converter. An isolated version of the forward converter is shown in Figure 1.5.

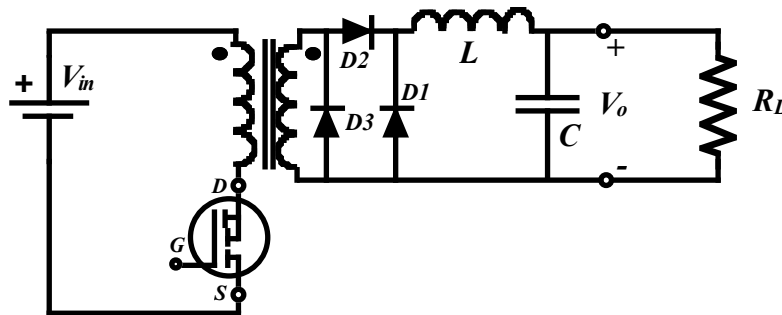


Figure 1.5

Unfortunately, we pay a price for this isolation: the forward inductor is still required along with $D2$ and $D3$. We have no simple way of recovering the energy stored in the transformer core, and it must be dissipated each cycle through $D3$ (which could also be placed on the primary side) to prevent eventual saturation, and $D2$ is required to isolate the output from the reverse polarity secondary during the discharge cycle.

There are numerous variations of each of these basic types and a variety of modifications developed to improve performance. The reader is directed to the references which represent a small sample of the literature available on the subject of SMPSs. For this laboratory exercise, the basic flyback configuration provides a simple, flexible architecture that illustrates the basic design and performance considerations involved in this popular type of power supply.

1.5 Design Considerations:

This section describes one procedure for determining component specifications for a flyback converter. Other procedures are given in the references, but they are all based on the same basic theory, and generally give very similar results for component values. Some procedures devote considerable effort to designing the required transformers and inductors, while the procedure given here determines only the basic parameters of these components and assumes a stock part will be available.

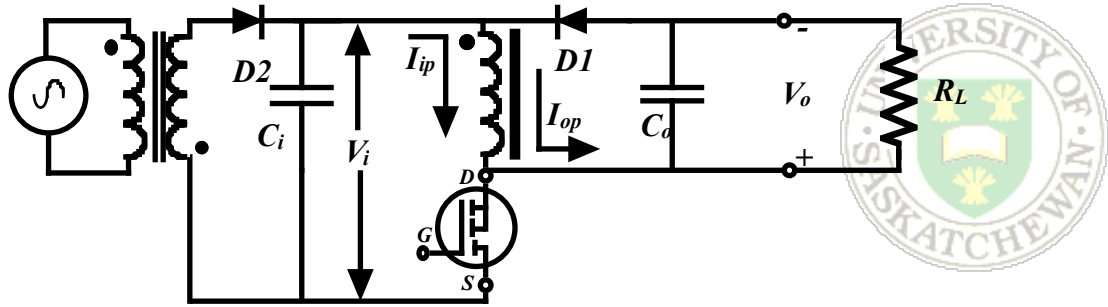


Figure 1.6

As a starting point, consider the non-isolated flyback converter given in Figure 1.6. Based on the discussion of basic operation given above, we can graph the input and output voltages and currents, which are shown in Figure 1.7.

To recap, we see that when the FET is turned on, the Drain-Source voltage of the FET, V_{DS} , is approximately $0V$, and the voltage across the inductor, V_L is approximately V_{in} . During this time the input current, I_i increases linearly from $0A$ to the input peak current, I_{ip} . following Equation (3), where Δt is the duty cycle, γ , times the period, T ($\Delta t = \gamma T$). Once the FET is turned off, the voltage across the inductor inverts, the current through the inductor, which is now the output current, I_o , decreases linearly according to Equation (4), and the drain-source voltage across the FET is at its maximum. We can also observe from Equation (4) and Figure 1.7 that the time required to reduce the inductor current to zero *increases* as V_o *decreases*. In fact it is only the same time if the V_o is approximately equal to V_i ! As discussed before, this can be problematic if $(1 - \gamma)T$ is not enough time to reduce the current, and thus the

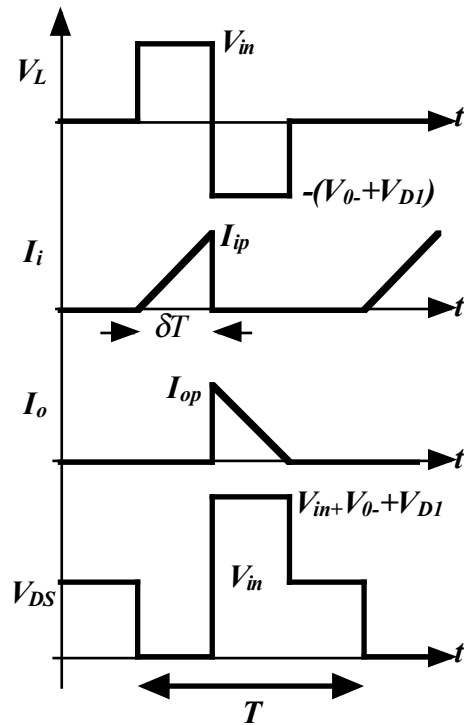


Figure 1.7

flux, in the inductor to zero. Finally, we also can determine the V_{DS} requirement for the FET if we know V_i and V_o . We can use the relationships depicted graphically in Figure 1.7 to help determine the parameters of a flyback converter design

First, we can use the target output parameters, I_o and V_o . Their product gives us the desired output power: $P_o = I_o V_o$. Next we could either assume an operating frequency and determine the required inductance, or start with an inductance and determine the required operating frequency. The amount of energy that must be stored in the inductor during each cycle is simply:

$$\frac{P_{Out}}{f} \quad (6),$$

which must be stored temporarily in the inductor. The energy stored in the inductor, and available for transfer to the output, occurs when $I = I_{ip}$ (the input peak current) and therefore:

$$\frac{P_{Out}}{f} = \frac{I_o V_o}{f} = \frac{LI_{ip}^2}{2}. \quad (7)$$

If we assume an ideal situation (i.e. no losses, negligible diode voltage drops), and also assume that under the worst extremes, the time to build up the energy in the inductor and the time to transfer it all to the output takes up the whole period, T , then using the conservation of energy and Equations (3) and (4) the maximum duty cycle, γ is:

$$\gamma = \frac{V_o}{V_o + V_i} \quad (8)$$

Of course, under less than extreme conditions, (higher input voltage, less load current), the sum of the charge and discharge times will be less than the period, T . In addition, a little manipulation of Equation (7) with some input from Equation (4) yields a relationship between the operating frequency, f , and the inductance, L :

$$L = \frac{V_i^2 \gamma^2}{2P_{out} f} \quad (9).$$

It is an simple rearrangement to find the required operating frequency for a given inductance, L .

Next, we need some way to determine I_{peak} , the maximum inductor current required to provide the target output power. Again equating the input and output energy in terms of the peak inductor current and the average output current will lead to an expressions for the target peak current:

$$I_{L-peak} = \frac{2I_o}{(1-\gamma)} \quad (10).$$

If we utilize a transformer to store and transfer the energy to the output, we can modify the basic equations to reflect the influence of the transformer's turns ratio, n . A basic transformer flyback configuration is shown in Figure 1.8.

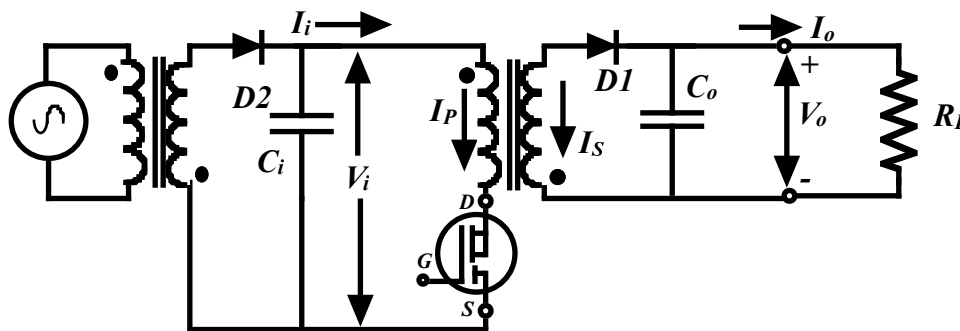


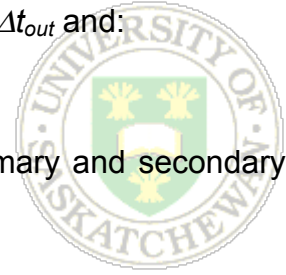
Figure 1.8

Recalling that $n = \frac{N_s}{N_p}$, $I_s = \frac{I_p}{n}$ and $V_s = nV_p$, we find that $\Delta t_{in} = \Delta t_{out}$ and:

Maximum duty ratio: $\gamma = 0.5$ (8.1),

Relationship between f and L_{primary} : $L_p = \frac{V_i^2 \gamma^2}{2P_{out} f}$ (9.1), and the primary and secondary

peak currents are: $I_{S-peak} = \frac{2I_o}{(1-\gamma)}$, and $I_{P-peak} = nI_{S-peak}$ (10.1).



1.5.1 Dampening Oscillations (Snubber Networks)

The stray capacitances and inductances in the circuit along with that of the components can create resonant circuits that respond to step changes in current or voltage by “ringing”. The overshoots and transients caused by this ringing can create stress for the switching elements and other circuit components. A variety of techniques have been developed to address this issue in switch mode power supplies. These “snubber networks” can be: rate-of-rise voltage or current snubbers, voltages clamps, or voltage or current damping snubbers. The first two varieties can be either dissipative (consume power) or non-dissipative. In the simple flyback configuration we are studying here, the output capacitance of the switching transistor and the leakage inductance from the inductor (transformer) are likely the main contributors to circuit ringing and excess dv/dt or di/dt values. The only snubber type discussed here in relation to the SMPS under investigation in this lab will be the “simple R-C voltage snubber”.

Philip Todd describes a procedure for determining the component values for a dissipative R-C snubber to provide damping across output inductors and transformers as well as diodes and switches. A paraphrase of the method follows:

- 1) with no snubbing in place, observe the frequency, f_s , of the ringing to be damped (at the drain of the FET switch)
- 2) place a small capacitor (alone) in the snubber position (D-S of the FET); adjust its value (by substituting other values), if necessary, until the observed frequency is approximately $\frac{1}{2}$ the original frequency. Recalling that the resonant frequency, $f_s = \frac{1}{2\pi\sqrt{LC}}$, this indicates the total capacitance is now *four* times the original, meaning the added snubbing capacitor, C_S , is three times the original circuit capacitance (near optimum for critical damping), or $C_S = 3C_{\text{Circuit}}$.
- 3) determine C_{Circuit} ($=C_S/3$), and use this to calculate the inductance: $L = \frac{1}{\omega_s^2 C_{\text{Circuit}}}$
- 4) calculate the characteristic impedance, $Z_o = \sqrt{L/C_{\text{Circuit}}}$
- 5) use a value of $R_S = Z_o$ in series with C_S as the snubber network.
- 6) in-circuit “tuning” of these values may improve performance.

Example: Observed ringing frequency = 10MHz.

Leakage inductance = 12 μ H, $C_{Sw} = 330$ pF.

Placing a .001 μ F capacitor across the circuit component reduces the ringing to approximately 5MHz, thus $C_{\text{Circuit}} = C_S / 3 = 333\text{pF}$.

$$L_{\text{Circuit}} = \frac{1}{(2\pi f)^2 C_{\text{Circuit}}} = \frac{1}{(2\pi 10\text{MHz})^2 (333\text{pF})} = .76\mu\text{H}$$

Now calculate R_S :

$$R_S = Z_O = \sqrt{L_{\text{Circuit}} / C_{\text{Circuit}}} = \sqrt{.76\mu\text{H} / 333\text{pF}} = 47.8\Omega$$

So a 50 Ω resistor in series with a .001 μ F capacitor should be tried in the circuit. Small variations could be tried to see if they will improve the damping. Snubbing may also have to be applied in other areas of the circuit. Snubbers in one part of the circuit will affect the observed ringing in other areas. Snubbing in those areas is usually based on the originally observed ringing frequency.

Prof. Chowdhury has also written a detailed explanation of the requirement for, and the application of snubber networks in circuit applications involving switching elements. Please refer to his notes (see 'References' at the end of this document) for additional understanding of the theory behind these necessary additions.

1.5.2 Thermal Considerations:

A necessary part of design is to determine the thermal effects of the circuit operation on the various components. The end objective is to provide an adequate path to dissipate any excess heat in order to prevent destruction of components and thus improve reliability of the device.

In a design similar to the one developed in this lab, the main component categories to consider are:

- the transformer and other inductors
- rectifier diodes
- resistors, including current sense elements and resistors in snubber networks.

Transformer and inductor data is usually provided by the manufacturer, including the DC resistance of the coils and the design operating frequency range. If operated within the specifications given by the manufacturer (including environmental conditions such as humidity and temperature), magnetic elements will typically dissipate the heat they generate without compromising their operation. Core losses from eddy currents or hysteresis losses cannot be ignored but are typically small compared to I^2R heating losses in the coils. With some reasonable assumptions, one can perform a simple calculation to determine the expected winding dissipation. For example, consider the coil used in Example I below. If the manufacturer's data (or a direct measurement) gives the DC resistance of the winding as .2 Ω , and we assume a triangular current waveform that ramps from 0 – 2.7A in a maximum duty cycle, $\gamma_{\text{Max}} = .25$, we can calculate:

$$P = \left(\frac{2.7\text{A}}{\sqrt{3}} \right)^2 (.2\Omega)(.25) = 121.5\text{mW}$$

Here we've used the RMS value of a triangular waveform and recognized that the I^2R heating only occurs during the conduction cycle, or $\frac{1}{4}$ of the time to give us the average power. Note, that if the circuit contains a transformer, there is heating from both the primary and secondary to consider.

The rectifier diode dissipation can be calculated simply using their average current and forward voltage drop, or their average conduction interval current, forward voltage drop, and conduction ratio (both give the same result). If we again use the values from Example I below, D_1 (output) maximum dissipation will be $(0.375A)(0.7V)=0.263W$ and the input rectifier, D_2 dissipates $(0.25A)(0.7V)=0.175W$.

Finally, dissipation requirements for any resistors can be calculated using $P = I^2R$ or IV and the appropriate RMS or average currents. Dissipation for resistors in snubber networks is a bit more difficult to estimate, and they often have to dissipate much more power than one might think. Take the values determined in the snubber example above for instance: a 50Ω resistor in series with a $.001\mu F$ capacitor, applied across the FET drain-source operating at 30kHz. One way to simplify the situation to estimate the dissipation is to assume that the R-C network is subjected to a 30kHz square wave of amplitude 36V (worst case). The time constant, τ is $.05\mu s$, and the peak current through the network would be $36V/50\Omega = .72A$. To simplify the calculation, we can assume that the current through this network, and thus the resistor, decays linearly over 5τ , or $.25s$. This would occur twice each cycle, or $16.67\mu s$. Using a similar method to else where, we can estimate the dissipation as follows:

$$P = \left[\left(\frac{36V/50\Omega}{\sqrt{3}} \right)^2 (50\Omega) \right] \left(\frac{.25\mu s}{16.67\mu s} \right) = 0.130mW$$

where we have again used the RMS value for a triangular wave and pro-rated the power based on the conduction ratio. A $\frac{1}{4} W$ resistor would do in this case, but note that if the value had been even 22Ω , the dissipation would have been $.7W$; enough to destroy even a $\frac{1}{2} W$ resistor before too long!

This discussion has focused a few of the main dissipating components to serve as an example. Of course, a good design will check all components to make sure that they are specified properly and will result in a safe, reliable device. In addition to calculations for individual components, one should also consider the enclosure and restrictions on air movement. In many cases, components specifications must be de-rated because of the mounting environment in small, enclosed spaces, or additional cooling ventilation may have to be provided.

1.6 Design Example I:

Let us assume an inductor-based arrangement like that shown in Figure 1.6, with an input voltage of 30V minimum and 36V maximum, and a desired output of 10V at a maximum current of 1A (10W). With the formulas given earlier, Kirchhoff's Voltage Law and reference to Figure 1.6, we can determine the requirements for the components.

We have a minimum input voltage and a target output voltage, so we can determine the maximum "on" time for the FET that will control the charging of the inductor. Using Equation (8):

$$\gamma_{Max} = \frac{V_o}{V_o + V_i} = \frac{10V}{10V + 30V} = 0.25$$

We can next determine either an inductor value or an operating frequency, or an iteration between the two. We would like the unit to operate above the audible frequency range to avoid noise pollution, so let's start by assuming a 20kHz operating frequency and determine an inductor value using Equation (9):

$$L = \frac{V_i^2 \gamma^2}{2P_{out} f} = \frac{30V^2 (.25)^2}{2(10W)20,000} = 0.141mH$$

Since we know the output power and voltage, we can use the average output current to determine I_{L-peak} using Equation (10):

$$I_{L-peak} = \frac{2I_o}{(1-\gamma)} = \frac{2\left(\frac{10W}{10V}\right)}{(1-.25)} = 2.7A$$

Unfortunately, all we can find in this range is an inductor that is 0.25mH and is rated at only 1.25A before it starts to saturate. Since this is an experiment, and an example, we can simply modify our specification to suit what is available. If we decide to maintain the 10V output voltage, we can use Equation (1) to determine our maximum output current:

$$I_o = \frac{I_{L-peak}(1-\gamma)}{2} = \frac{1A(1-.25)}{2} = 0.375A$$

Note we've used a peak current of only 1A to allow a little margin for unforeseen circumstances. This new current gives us 10V X .375A = 3.75W output power. Using this with the available inductor:

$$f = \frac{V_i^2 \gamma^2}{2P_{out} L} = \frac{30V^2 (.25)^2}{2(3.75W).00025H} = 30,000kHz$$

Our period, T , is 33.33μs, and of this, we can leave the FET turned on for a maximum of .25(33.33μs) = 8.33μs. In practice, we may again want to leave ourselves a bit of margin to make sure there is sufficient time to discharge the inductor fully even at maximum load, so we would limit the on time to something less than 8μs.

The rectifying diode in the output, $D1$, must handle a maximum current of I_{L-peak} , or 1A, and withstand a reverse voltage of $V_{in} + V_{out}$ during the charging phase. In this case, a minimum of 40V would be required, but we would likely specify the next highest

standard value of 50 or 100VDC. It must also be “fast” enough to operate at least 30kHz. This peak current also specifies the current handling requirements for the FET, 1A. The maximum drain-source voltage, from Figure 1.7, is $V_{in} + V_o + V_{D1} \approx 36V + 10V + 1V = 47V$. Again, a conservative selection would be in order. Any driver circuit could presumably be designed to not exceed the Gate-Source limit, or a suitable FET would need to be found to withstand the drive voltage.

The output filter capacitor value can be calculated using Equation (11) rearranged. Assuming we’d like to keep the ripple to no more than 0.1V:

$$C = \frac{I_o}{fV_{Ripple}} = \frac{.375A}{(30000Hz)(0.1V)} = 125\mu F$$

A working voltage of twice the output voltage should provide enough margin (e.g. 20VDC). Choosing a low ESR (Equivalent Series Resistance) capacitor will also improve the circuit’s performance.

The input capacitor and diode specifications can be determined in a similar fashion except that we must recognize that the input rectifier diode must supply the energy require to replenish the energy in the filter capacitor that feeds the inductor (at a relatively constant voltage) in a fraction of the cycle if the ripple is to be acceptable.

We should easily be able to tolerate about a 10% ripple in the input, and assuming a full-wave rectified input ($f = 2 \times 60Hz$):

$$C_i = \frac{\overline{I_{i-Avg}}}{fV_{Ripple}} = \frac{.25A}{2(60Hz)3V} = 694\mu F ,$$

where I_{i-Avg} is approximately $.25I_{i-peak}$ for a ramp current that occupies about $\frac{1}{2}$ the period. We may specify a *slightly* higher value for the filter capacitors, but at the expense of increased space and cost, but *too much* higher may put added stress on the rectifier diodes.

The rectifier diode, D2, must also supply the same ‘average’ current of .25A that is flowing out of the filter capacitor, but it must do it during its conduction period. With a 60Hz input source, the conduction time, t can be found from:

$$t = \frac{T}{2(4 \text{ for half-wave})} - \frac{\sin\left(\frac{V_M - V_{Ripple}}{V_M}\right)}{21600}, \text{ (for full-wave rectifier and argument in degrees)}$$

The conduction ratio is simply $\frac{t}{T}$, where T is the period (8.33ms for a full-wave rectifier and 16.67 for a half wave rectifier on a 60Hz source). In this case, if we assume a 10% ripple voltage from the input source, the ratio is $\frac{1.197ms}{16.667ms} = 0.0713$. With an average current out of the capacitor of 0.25A as determined above, the diode will have to handle $\frac{.25A}{.0713} = 3.501A$. It must also withstand a reverse bias voltage of V_{in-Max} or 36V. Once again a 50 or 100VDC would be specified.

1.7 Design Example II:

Let us keep the original specifications for our power supply assumed in Example I, except that we will use a transformer-based configuration like that shown in Figure 1.8. Armed with the same tools, we can determine the specifications for the basic components we need.

A reasonable place to start this time is with the transformer turns ratio, n . When V_{in} is impressed across the primary, the primary current, I_P , builds up to its maximum while the FET is turned on (Equation (3)). When the FET is turned off, the current in the secondary (I_P/n) will start to flow inducing an appropriate voltage in the secondary. That voltage must be a minimum of $V_o + V_{D1}$ in order for the inductor to discharge current as the field collapses. The turns ratio must be such that the applied voltage on the primary will build up the required current so the secondary current will induce the minimum voltage required there. We need to insure that we have 10V at the output when we have the minimum voltage of 30V present at the primary, so that turns ratio is:

$$\frac{V_o + V_{D1}}{V_i} = \frac{10V + 1V}{30V} = 0.37$$

Again, for a bit of margin, we would probably specify about 10% more voltage, or pick the next winding ratio available (that would provide a slightly higher output for a given input voltage).

Now starting as in Example I, we can see that the maximum duty ratio is 0.5 (Equation (8.1)), but we should allow some margin for error, so let's assume a maximum of 0.45.

We can now calculate a required primary inductance from Equation (8.2) assuming we want at least a 20kHz operating frequency to keep any noise out of the audible range:

$$L_p = \frac{V_i^2 \gamma^2}{2P_{out} f} = \frac{30V^2 (.45)^2}{2(10W)20,000} = 0.455mH$$

Fortunately, we happen to have a SMPS (i.e. high frequency) transformer with a primary inductance of 600 μ H. It is designed for an output current of about .4A at 12V or approximately 5W, and has several secondary windings, one of which has a turns ratio of 0.32. Given these restrictions, we end up with an operating frequency of:

$$f = \frac{V_i^2 \gamma^2}{2P_{out} L} = \frac{30V^2 (.45)^2}{2(5W).0006H} = 30,375kHz,$$

which should easily be attainable and well out of the audible range.

Our peak primary and secondary currents can be calculated using Equation (10.1):

$$I_{S-peak} = \frac{2I_o}{(1-\gamma)} = \frac{2(.4A)}{(1-.45)} = 1.45A, \text{ and } I_{P-peak} = nI_{S-peak} = .32(1.45A) = 0.47A.$$

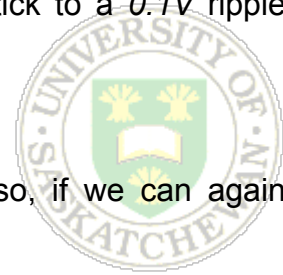
This will set the specifications for current handling for the output rectifier, $D1$, at about 1.5A, the FET at about 0.5A, and the input rectifier, (using the same methods as in Example I) must handle 1.655A during its conduction interval. The associated voltage limits, using KVL, are $(nV_{in-peak} + V_o) \approx 22V$ for $D1$, $V_{in-max} = 36V$ for $D2$, and $(V_o + V_{D1})/n + V_{in-max} \approx 2V_{in-Max} = 72V$ for V_{DS} for the FET.

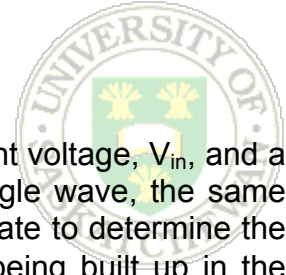
Finally, the value of filter capacitors can be calculated. If we stick to a $0.1V$ ripple target for the output:

$$C_o = \frac{I_o}{fV_{Ripple}} = \frac{.36A}{(30375Hz)0.1V} = 119\mu F ,$$

where I_o is approximately $.25I_{S_peak}$ at a duty cycle of 50%. Also, if we can again tolerate about a 10% ripple in the input:

$$C_i = \frac{I_i}{2fV_{Ripple}} = \frac{.118A}{2(60Hz)3V} = 326\mu F .$$





1.8 Appendix A – Derivations Background

1.8.1 Derivation of duty cycle:

The instantaneous power per cycle into the inductor given a constant voltage, V_{in} , and a linear rise in current up to I_{ip} , (which also results in a simple triangle wave, the same shape as the current shown in Figure 1.7) lets us graphically integrate to determine the average power over this interval, γT , while the magnetic field is being built up in the inductor. This energy, ignoring losses, is also equal to the power delivered from the inductor to the output during the discharge phase, which is the peak current, I_{op} , linearly decreasing over a maximum time $(1-\gamma)T$ times the relatively constant output voltage, V_o . This lets us express the input and output powers in terms of these parameters:

$$P_{in} = \frac{(\gamma T)(V_{in} I_{ip})}{2T} = \frac{\gamma V_{in} I_{ip}}{2} \quad (i)$$

$$P_{out} = \frac{((1-\gamma)T)(V_{out} I_{op})}{2T} = \frac{(1-\gamma) V_{out} I_{op}}{2} \quad (ii)$$

Equating these two expressions and recalling that $I_{op} = I_{ip}$ and cancelling some terms leaves us with:

$$\gamma V_{in} = (1-\gamma) V_{out} \quad (iii)$$

Rearranging into two ratios we get:

$$\frac{\gamma}{1-\gamma} = \frac{V_{out}}{V_{in}} \quad (iv)$$

and with a bit more manipulation:

$$\gamma = \frac{V_o}{V_o + V_i} \quad (\text{Equation 8})$$

which gives us the maximum fraction of the period that can be used to charge the inductor given the input and output voltages.

In the case where we use a transformer, as shown in Figure 1.3, with a turns ratio, $n = \frac{N_s}{N_p}$, then $V_S = nV_P$ and $I_S = \frac{I_P}{n}$. Equating the right sides of (i) and (ii) above yields:

$$\frac{\gamma V_{in} I_{ip}}{2} = \frac{(1-\gamma) V_{out} I_{op}}{2} \quad (v),$$

and recalling that, if we ignore rectifier forward voltage drops, $V_{in} = V_P$, and $V_{out} = V_S$, this can be re-written:

$$\frac{\gamma V_P I_P}{2} = \frac{(1-\gamma)(V_P n) \left(\frac{I_P}{n} \right)}{2} = \frac{(1-\gamma) V_P I_P}{2} \quad (vi).$$

Cancelling terms leaves us with $\gamma = (1 - \gamma)$, which means that we can have a maximum duty ratio, γ , of 0.5 regardless of the ratio of V_{in}/V_{out} if we set an appropriate turns ratio, n .

1.8.2 Relationship between frequency of operation and inductance:

Recall the energy stored in an inductor is given by:

$$W = \frac{1}{2} L I_{peak}^2 \quad (1)$$

Since this is the source of energy to supply the output load, the amount of energy needed each cycle of operation is $P_{out} T = \frac{P_{out}}{f}$, which must equal the energy stored in the inductor at peak current:

$$\frac{P_{out}}{f} = \frac{1}{2} L I_{peak}^2 \quad (vii).$$

Also recall that the peak current after a time, Δt , with a constant input voltage is:

$$I_{peak} = \frac{V_{in}}{L} \Delta t \quad (3)$$

and since $\Delta t = \delta T$:

$$I_{peak}^2 = \left(\frac{V_{in}}{L} \gamma T \right)^2 \quad (viii).$$

Substituting into (iii):

$$\frac{P_{out}}{f} = P_{out} T = \frac{1}{2} L \left(\frac{V_{in}}{L} \gamma T \right)^2 = \frac{L V_{in}^2 \gamma^2 T^2}{2 L^2} \quad (ix).$$

Cancelling terms and isolating L leaves:

$$L = \frac{V_i^2 \gamma^2}{2 P_{out} f} \quad (9)$$

In the case where we use a transformer instead of an inductor, Equation (9) still holds, with the qualification that L is the primary inductance. Using V_o instead of V_i will yield the secondary inductance.

1.8.3 Relation between average output current, I_o , and the peak inductor current: $I_{peak} = I_{ip} = I_{op}$.

Again we simplify this basic derivation, for the time being, by assuming no losses and neglecting any forward voltage drop across the rectifiers. Another reasonable assumption

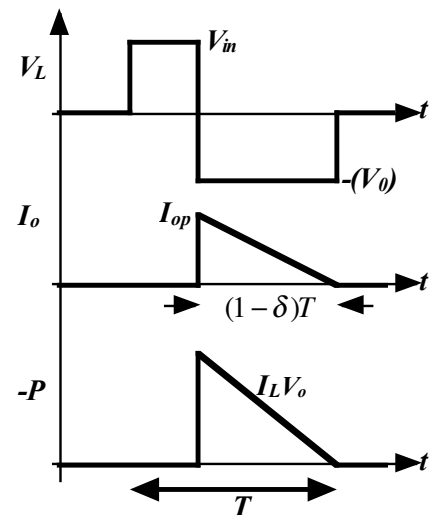


Figure 1.9

is that if the output voltage ripple is small, that $V_{o-RMS} \cong V_{o-AVG}$. We can then see that the output power = $I_o V_o$, and that this must equal the average power delivered from the inductor.

The instantaneous power delivered from the inductor is shown in Figure 1.9 (inverted for clarity) along with the related voltage and current for reference. We can see that the energy delivered during one cycle is:

$$\frac{W_L}{\text{cycle}} = \int_0^T i(t)v(t)dt = \frac{1}{2} I_{L-peak} V_o (1 - \gamma) T \quad (\text{x}),$$

and the energy delivered to the load is

$$\frac{W_o}{\text{cycle}} = I_o V_o T \quad (\text{xi}).$$

Equating the expressions:

$$\frac{1}{2} I_{L-peak} V_o (1 - \gamma) T = I_o V_o T \quad (\text{xii}),$$

which results in:

$$I_{L-peak} = \frac{2I_o}{(1 - \gamma)} \quad (10)$$

If a transformer is used instead of an inductor, the only modification required is that the peak currents on the primary and secondary sides are no longer identical, but related by the ratio n such that

$$I_{S-peak} = \frac{2I_o}{(1 - \gamma)} \quad (11), \text{ and}$$

$$I_{P-peak} = n I_{S-peak} \quad (12)$$

Finally, the primary and secondary inductances are different, and related by the ratio:

$$\frac{L_S}{L_P} = \frac{N_S^2}{N_P^2} \quad (13),$$

but by again using Equation (1) and the relationship between primary and secondary current given above, $I_S = I_P/n$, we can prove that the energy initially stored in the core through the primary current is exactly the same as, and can be retrieved through the secondary. In addition, Equations (3) and (4) can be modified to reflect the primary and secondary inductances:

$$\Delta t_{in} = \frac{L_P}{V_i} I_{P-peak}, \text{ and } \Delta t_{out} = \frac{L_S}{V_o} I_{S-peak} \quad (14.1, 14.2)$$

Using Equation (13) and $I_S = I_P/n$ and $V_S = nV_P$ in Equation (14.2), we get:

$$\Delta t_{out} = \frac{n^2 L_P}{n V_i} \frac{I_{P-peak}}{n} = \frac{L_P}{V_i} I_{P-peak} = \Delta t_{in} \quad (\text{xiii}),$$

which means that even if we use the maximum time, $0.5T$ to charge the inductor, we should need *no more than* the same amount of time to completely recover the energy from the secondary! In practice, we should allow some margin to ensure that we don't risk driving the transformer (inductor) core into saturation.

1.8.4 Voltage Ripple and Filter Capacitance:

The expression most commonly used to estimate the voltage ripple in relation to the value of the filter capacitor in a simple, capacitance filtered supply is:

$$V_{Ripple} = \frac{I}{fC} \quad (11).$$

If we examine the situation in our flyback switching power supply circuit for example, we can consider how the output voltage varies with respect to the current coming from the inductor or going out to the load. We note the time to “discharge” the inductor was $(1-\gamma)T$, and recall from Figure 1.7 and Figure A that the “charging current” as a function of

time may be described as $i_{inC}(t) = I_{peak} - \frac{I_{peak}}{(1-\gamma)T}t$ (for the worst case, when the charge

and discharge phases sum to T). Clearly the capacitor is being charged as long as i_{in} is greater than I_O . After i_{in} drops below I_O , the draw on the capacitor will be $I_O - i_{in}$. After i_{in} falls to 0 , and until the next capacitor charging phase (inductor discharge phase), the drain on the capacitor is I_O . The peak-to-peak ripple will be the drop in voltage across the capacitor from the time when $i_{in} = I_O$ until the next capacitor charging phase. This will obviously be something less than T , and will depend on the duty cycle, γ , and the ratio of I_{L-peak} to I_O .

This exact calculation will clearly lead to a smaller value for C , but for the time being, the estimate given by (xv) is sufficient and conservative.

1.9 References:

Chrysiss, George C., "High Frequency Switching Power supplies Theory and Design", 2nd Ed., McGraw-Hill, New York, 1989

Pressman, Abraham I., "Switching and Linear Power Supply, Power Converter Design", Hayden Book Co. Inc., New Jersey, 1977

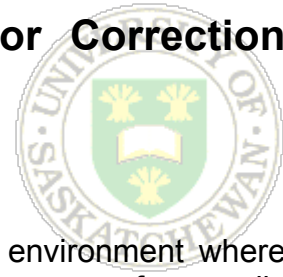
Todd, Philip C., "Snubber Networks: Theory and Design", Texas Instruments Inc., May 1993

Chowdhury, Nurul, "Snubber Circuits", downloaded from
http://www.engr.usask.ca/classes/EE/443/notes/snubber_circuits.pdf



2 Design Lab II: An Automatic Power Factor Correction System

(Dr. Rajesh Karki, Sept. 2009)



2.1 Safety Advisory:

The activity prescribed in this laboratory will be conducted in an environment where hazardous electrical potentials exist. The student should be aware of normally expected electrical laboratory hazards and follow procedures to minimize risk. Please refer to general safety precautions in the Laboratory Manual and those posted in the labs.

This laboratory exercise presents the following specific hazards:

- Shock: Collapsing fields in inductors, transformers and motors can produce extremely high potentials even with low input source voltages; allow for appropriate discharge paths and insulate yourself appropriately when contacting leads or components.
- Burn: Circuit elements under load, especially abnormal load caused by circuit or design errors, can reach temperatures which will cause burns to skin if contacted. Do not attempt to determine the temperature of elements with your fingers! Approach potentially overheated elements with caution and allow sufficient time for cooling when necessary. In some cases, elements may burst into flames. This is usually very isolated and does not normally create a fire hazard. If overheating to flames do occur, immediately remove power and *determine and correct the cause before re-energizing!*
- Explosion: Modern capacitor designs using metal cases with pressure relief mechanisms have proven very safe and should be used in all new designs. However, capacitors that are subjected to potentials exceeding their rating can, on very rare occasion, rupture their cases. This especially applies to electrolytic capacitors commonly used as power supply filter elements. Electrolytic capacitors are typically polarized, and if they are connected with the polarity reversed, even below rated voltage, can overheat quickly. Make certain that all electrolytic capacitors are connected *with the proper polarity* before energizing the circuit. If in doubt, re-check the polarity of the voltage rails *without* the filter capacitor and then mount the element correctly.
- Rotating Shaft Hazard: There is an electric motor with a rotating shaft. Although small, the shaft and mounted sensor disk may pose a slight abrasive hazard if contacted. Caution should be exercised when the motor is running even though the risk of serious injury is negligible.

Some additional procedures for dealing with high potential equipment are included below. Please familiarize yourself with these and follow them when appropriate.

Turning Power ON, Taking Readings and Turning Power OFF

- Make sure everyone is clear of the circuit and equipment. Announce your attention to turn the power on. Turn on the **table breaker** first to avoid accidental human contact with the energized terminals on the table breaker. Turn on the 100-amp **breaker at your work station (bench panel breaker)** second to energize the circuit.
- **Take meter readings immediately. Do not delay.** Only one member of the group should take readings from the meters. Place both hands behind your back. Call out the readings. Be careful not to contact any part of the circuit or equipment. Another member of the group should record the readings called out. The recording person should be positioned a safe distance from the circuit. All other members of the group should stay clear of the circuit and the meter reading person. **Turn power off immediately after the readings are taken. Do not delay.**
- Turn the 100-amp **breaker at your work station (bench panel breaker)** off first, followed by the **table breaker**. Always turn both breakers off. Have the circuit rechecked if **Any** changes are made to the circuit.

Circuitry of High Power Ratings (> 100V):

- Work only with one hand; keep the other in a pocket or behind your back when working with powered equipment and circuitry. This will reduce the chance of touching two powered points at the same time and will greatly reduce the extent of injury if something should happen.
- ALWAYS disconnect power to the circuit when not in use and when changing connections
- Place all meters nearest to the edge of the circuitry nearest to you so that you do not have to lean over powered circuitry to make readings
- Keep wiring as neat as possible so that you can easily determine how the circuitry is connected without having to grab wires to determine where connections go.
- Use wires that are as short as possible to make connections to equipment, but not so short that they are stretched to their limit (use good judgment)
- In a learning lab environment ALWAYS have your circuit checked by an instructor before powering the circuit

2.2 Background: Reactive Power and Power Factor Correction in Power Circuits

The average power drawn by a load connected to an energized power circuit is also known as the active power or real power, and is calculated using Equation (1).

$$P = V \cdot I \cdot \cos \theta \quad (1)$$

where, V is the RMS voltage across the load, I is the RMS current drawn by the load, and θ is the phase angle between the voltage and the current waveforms. For given voltage and current levels, the average power is maximized when the value of $\cos \theta$ (which is known as the **power factor**) is maximized, i.e. equal to 1. It is, therefore, desired that electrical loads operate close to unity power factor in order to increase the efficiency of the power system.

$$\text{Power Factor (p.f.)} = \cos \theta \quad (2)$$

where, θ is known as the power factor angle, and is the phase angle between the voltage and the current waveforms. When the current waveform lags the voltage, the power factor is said to be lagging power factor. This occurs when the load has inductive reactance. A load with lagging power factor is considered to draw reactive power from the circuit. On the other hand, when the current waveform leads the voltage, the power factor is said to be leading. This occurs when the load has capacitive reactance. A load with leading power factor is considered to inject reactive power into the circuit. Equation (3) shows one method to calculate reactive power, Q .

$$Q = \text{SQRT}(A^2 - P^2) \quad (3)$$

where, A is the apparent power and P is the average (or real) power of the load. The apparent power is the product of the RMS voltage and current of the load. Power factor is also defined as the **ratio of the average power and the apparent power**.

Power factor can also be calculated from the impedance, Z of the load. If Z is expressed in the polar form, the angle of the impedance is the power factor angle, θ . If Z is purely resistive, θ is zero, and therefore, power factor is unity.

Most residential and industrial loads are of inductive nature, and therefore, will cause lagging power factor. Power factor correction (PFC) is usually done by connecting a capacitor of appropriate rating to inject reactive power and bring the power factor close to unity. When the reactive power injected is equal to the reactive power drawn by the inductive load, the power factor is corrected to unity. The capacitance required to inject reactive power Q can be calculated using Equation (4).

$$C = Q / (2\pi f \cdot V^2) \quad (4)$$

where, V is the RMS voltage across the capacitor and f is the frequency.

It is obvious that the capacitance required for power factor correction of a power utility customer will vary as the number and type loads vary throughout the day. It becomes important to avoid resonance type situations when capacitors are used to neutralize the lagging effect of inductors in a circuit, or else, some parts of the circuit may be subjected to very high voltages or currents. Care must be taken to ensure the circuit components are able to withstand the resulting currents or voltages.

It is important to maintain a steady RMS voltage at the load terminal, since residential as well as industrial loads are rated to operate at a specific voltage level. When a load is

connected, the voltage at the load terminals will however decrease due to a voltage drop in the circuit upstream to the load terminals. Corrective actions, such as transformer tap-changing or reactive power injection, must be taken if the voltage at the load terminal drops significantly. It should be noted that the corrective actions should be removed or altered when the load is changed or disconnected, or else the voltage at the load terminals can be excessive. These factors require careful attention when designing a PFC system for a typical power consumer. PFC system can be static as well as dynamic through automatic switching.

2.3 Part 1- Power Factor Calculation and Measurement

1(a):

- Connect a resistor in series with an inductor to form an inductive load.
- Calculate and measure the total impedance to obtain a power factor of 0.8.
- Use a variac to energize the load with a suitable voltage, so that the current through the circuit components (including the connecting cables) are within allowable limits.
- Measure the phase angle between the voltage and the current waveform using an oscilloscope and calculate the power factor.
- Connect a power analyzer and measure the power factor. It should be noted that a negative reading indicates leading power factor, and a positive reading indicates lagging power factor.

1(b):

- Connect the circuit as shown in the single-line diagram in Figure 2.1.
- This represents a power supply system to a residential/industrial customer. The inductor coil represents the upstream circuit impedance. Make sure that the components are able to withstand the operating voltages.
- Connect a 60 W incandescent lamp at Load 1 terminal in series with a switch. You should have an estimate of the current that will flow in different parts of the circuit, and make sure that the components and cables are able to carry the current.
- Measure the voltage at the load terminal, the current, active and reactive powers and the power factor.
- Replace the 60 W lamp with a 23 W compact fluorescent lamp (energy efficient lamp), and take the readings.

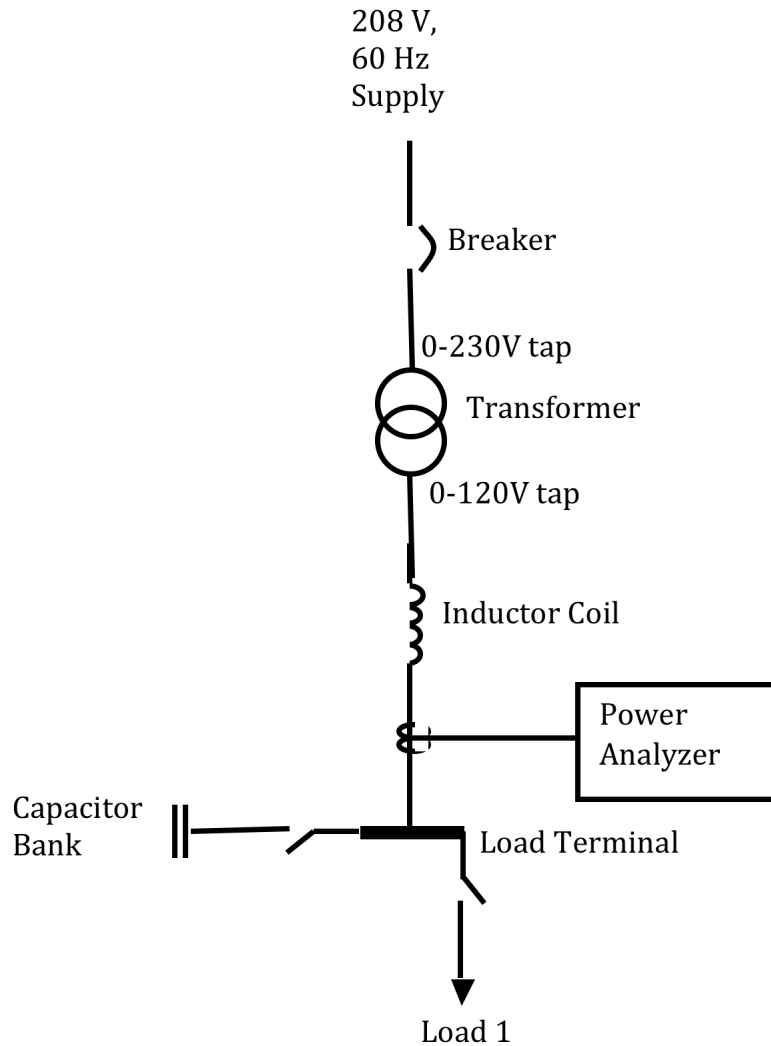


Figure 2.1

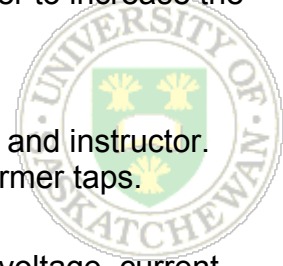
2.4 Part 2 – Voltage Drop and Control

2(a) – Voltage Drop Measurement:

- Energize the above circuit without connecting a load, and measure the no-load voltage at the Load terminal.
- Switch-on a 150 W incandescent lamp at the Load Terminal.
- Measure the voltage at the load terminal, the current, active and reactive powers and the power factor
- Calculate the voltage drop (reduction in voltage) at the load terminal from the measured voltages.
- Verify with calculations that $V_{\text{no-load}} - V_{\text{drop@coil}} = V_{\text{load}}$.

2(b) – Voltage Control Using Transformer Tap-changing:

- Calculate the level of voltage increase that can be obtained through tap changing of the transformer
- De-energize the circuit by opening the breakers.



- Perform tap-changing at the primary winding of the transformer to increase the load voltage across the 150 W load.
- Measure the voltage at the load terminal.
- Disconnect the load. What is the voltage at no load?
- Discuss on-line and off-line tap changing with team members and instructor.
- De-energize the circuit and switch back to the original transformer taps.

2(c) – Voltage Control with Reactive Power Injection:

- Connect a 150 W lamp at the load terminal and measure the voltage, current, real and reactive power.
- Connect a capacitor bank in series with an ammeter at the load terminal.
- Gradually increase the reactive power injection, and monitor the current through the capacitor, the reactive power injected, and the voltage at the load terminal.
- Obtain a load voltage close to the original no-load voltage, and measure the connected capacitance.
- Also take the readings from the power analyzer.
- Switch off the 150 W load and measure the voltage at the load terminal.
- De-energize the circuit and change the switch position, so that both the load and the capacitor bank are controlled by the same switch.
- Measure the no-load and load voltage.
- De-energize the circuit and disconnect the capacitor bank.

2.5 Part 3 – Power Factor Correction

3(a):

- Connect an inductor in series with a variable resistor at the Load Terminal in Fig. 1. Also connect an ammeter to measure the current drawn by the load. You should have an estimate of the current that will flow in different parts of the circuit, and make sure that the components and cables are able to carry the current.
- Vary the resistance and/or inductance to obtain a lagging power factor of 0.7.
- Energize the circuit and measure the voltage at the load terminal, the current, active and reactive powers and the power factor.
- Calculate the capacitance required to correct the power factor close to unity.
- Connect the desired value of the capacitor bank to the Load Terminal through an ammeter. Estimate the current that will flow through the capacitor bank and make sure the circuit can handle that current.
- Measure the Voltage at the load terminal, the current, active and reactive powers and the power factor
- Compare the currents drawn by the capacitor, the load and the total current shown by the power analyzer.
- Verify that the total current drawn from the source is the summation of the current through the capacitor and the load.
- Disconnect the resistor/inductor load and measure the voltage at the Load Terminal.
- Discuss appropriate switch positions for the load and capacitor bank with your group and your instructor. Also discuss the applications of static and automatic power factor correction systems.

3(b) – Automatic Power Factor Correction

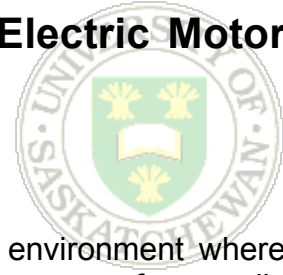
- Obtain 3 different power factor levels for resistor/inductor load by changing the value of the resistance and/or inductance.
- Calculate the capacitance required to correct the power factor close to unity.
- Connect the desired value of the capacitor bank to the Load Terminal through an ammeter.
- Measure the Voltage at the load terminal, the current, active and reactive powers and the power factor
- Disconnect the resistor/inductor load and measure the voltage at the Load Terminal for each case.

2.6 Part 4 – Design of a PFC System

- Consider an industrial power customer that is supplied through a 5 kV distribution feeder. The customer has different types of loads (such as ovens, air-conditioners, motors, etc.) that have different power factors.
- Draw a single line diagram of the system.
- Design a power factor correction system for the customer, taking into consideration the power factor correction range, maintaining acceptable voltage at the customer terminal at all times, and using components with proper ratings to withstand operating voltages and currents

3 Design Lab III: An Electronic Governor for Electric Motor Speed Control

(Denard Lynch, September 2008, revised Sept. 2009)



3.1 Safety Advisory:

The activity prescribed in this laboratory will be conducted in an environment where hazardous electrical potentials exist. The student should be aware of normally expected electrical laboratory hazards and follow procedures to minimize risk. Please refer to general safety precautions in the Laboratory Manual and those posted in the labs.

This laboratory exercise presents the following specific hazards:

- Shock: Collapsing fields in inductors and motors can produce extremely high potentials even with low input source voltages; allow for appropriate discharge paths and insulate yourself appropriately when contacting leads or components.
- Burn: Circuit elements under load, especially abnormal load caused by circuit or design errors, can reach temperatures which will cause burns to skin if contacted. Do not attempt to determine the temperature of elements with your fingers! Approach potentially overheated elements with caution and allow sufficient time for cooling when necessary. In some cases, elements may burst into flames. This is usually very isolated and does not normally create a fire hazard. If overheating to flames do occur, immediately remove power and *determine and correct the cause before re-energizing!*
- Explosion: Modern capacitor designs using metal cases with pressure relief mechanisms have proven very safe and should be used in all new designs. However, capacitors that are subjected to potentials exceeding their rating can, on very rare occasion, rupture their cases. This especially applies to electrolytic capacitors commonly used as power supply filter elements. Electrolytic capacitors are typically polarized, and if they are connected with the polarity reversed, even below rated voltage, can overheat quickly. Make certain that all electrolytic capacitors are connected *with the proper polarity* before energizing the circuit. If in doubt, re-check the polarity of the voltage rails *without* the filter capacitor and then mount the element correctly.
- Rotating Shaft Hazard: There is an electric motor with a rotating shaft. Although small, the shaft and mounted sensor disk may pose a slight abrasive hazard if contacted. Caution should be exercised when the motor is running even though the risk of serious injury is negligible.

3.2 Introduction:

There are many applications in engineering that require control of motor speed, and various methods of achieving that control. While there is evidence of feedback control being used centuries ago, it was the application of this principle in the early 18th century on steam engines that laid the groundwork for modern control theory and engine speed control. These early designs were entirely mechanical, and although simple, were fairly

efficient and effective. As our understanding of feedback and control improved, and with the dawn of the electronic age in the mid-20th century, the benefits of, and demand for, speed control were applied to increasingly sophisticated systems and thousands of new applications.

Controlling engine speed has two general benefits. First, it allows machinery to operate at a safe and constant speed even if subjected to varying loads. This could benefit workers on a factory floor or improve the operation of internal combustion engines, which are designed for optimal performance at a certain rpm or at best over a narrow range. It also enables accurate reproduction of recorded material like that on tapes, CDs, DVDs, BDs etc. The second main benefit is protection of the engine or motor itself. Without feedback-based control, removal of a load without reducing the energy input somehow can easily lead to self destruction by “over revving” or at least a waste of energy.

The original benefits and theory for mechanical systems are now applied regularly to electric motors and drives. How “tightly” the speed has to be controlled depends on the application and will help determine the design requirements. The design challenge in this laboratory is to design and build an electronic system that will monitor and control the speed of a small motor or engine. The target for your prototype will be a small, brushed DC electric motor, but you should consider scalability in your design.

The microcontroller unit (MCU) suggested as the base for this design, Microchip’s PIC16F886, has a number of features that are potentially useful for monitoring and electric motor control. Study the data sheet and sample application notes to become familiar with its more popular features.

3.3 Part 1

The first part of this lab involves characterizing the target motor and familiarizing yourself with the PWM, comparator and programming features of the PIC16F886 MCU.

3.3.1 Procedure (Motor characterization and MCU setup):

Although it is desirable that your design be able to “scale” to handle different types and sizes of electric motors (e.g. brushed or brushless DC, 1-phase or 3-phase AC etc), at least to some extent, your initial design and prototype will use a specific target motor, and it is useful to gather some data about its characteristics such as voltage, current and speed at certain loads. Some small, 6VDC electric motors are already conveniently mounted so they will plug on to your prototype boards. They also have a reflective object sensor mounted near the strobe wheel as one possible way of monitoring output speed, although there are certainly other techniques available.

- 1) Take a few basic measurements of voltage, current and speed under varying load conditions (note: you can manually apply a slight load by carefully using friction on the strobe wheel). Note the no-load rpm and current draw at various terminal voltages (caution: do not allow the motor to be “over driven”). Map the terminal voltage versus rpm for possible use later. Also note the change in current draw if the wheel is loaded slightly; this will help determine power handling requirements in your design. You might sign-out a hand-held optical tachometer to measure the shaft speed (available from the technicians’ office).

- 2) The shaft speed of DC motors typically responds to the magnitude of the voltage at their terminals. There are different ways of controlling the voltage and thus achieving some control over the shaft speed. Two popular methods are: linear regulation, and pulse-width (duty-cycle) control. Consider the pros and cons of each keeping in mind the capabilities of the MCU available, and design and build the motor drive electronics. Test the capabilities of your design by driving the current switch (e.g. FET) from an external source like a signal generator, and verify that you can achieve the required control by repeating a couple of the tests you did above. For example if you decide to use PWM control and a FET switch, add a slight load and verify that you can maintain the rpm by adjusting the pulse duty cycle. Note: to avoid excess mechanical heating or damage to components, do load testing at a low power point in the operating range, or apply the load intermittently while you observe the data.
- 3) As you noted by studying the MCU datasheet, the PIC16F886 (and other members of this family) are capable of producing a variable frequency, variable duty cycle output. In addition, it has a comparator and A/D converters that may prove useful in your design. Do enough preliminary design to decide on a strategy for driving the motor and monitoring its speed. Set up and verify the operation of the features you will need in your final system. Some popular possibilities are outlined below.
- 4) Your MCU will require basic programming to operate at all. There are some code snippets in the Appendix and posted on the class web site that will help you generate a basic software load for this MCU. Launch "MPLAB IDE" under All Programs/ee/Microchip/MPLAB IDE. When the development environment opens, select "Project Wizard" from the "Project" pull-down menu. Create a new project selecting the 'PIC16F886' device, the 'HI-TECH Universal ToolSuite' as the active toolsuite (HI-TECH Ansi-C Compiler). Select a project name and suitable location to store your files; click 'Finish'. Compose a C-file, using segments from snippets if necessary, and compile the program.
- 5) Your program should contain segments that allow you to verify the operation and necessary control over the features you might require for your final design. For instance, again using the PWM example, you should initialize the appropriate registers to set up the ECC Module for PWM operation (registers PR2, CCPR1L, CCP1CON, T2CON) and perhaps an input interrupt routine to increment or decrement the duty cycle so you can test control of the motor drive electronics via the MCU manually. You may also need to setup and test the ADC or Timer/Comparator modules for obtaining input on the motor speed. (If you utilize the on-board reflective photo detector, you *might* use the comparator to "clean up" the signal (although there are other alternatives) and the timer to determine the shaft speed by timing some number of revolutions. If you use the back-emf method, you could use the A/D converter to sample the induced voltage during the "off" part of the drive cycle and estimate the rpm that way.) You may need to initialize and test other modules as well, depending on your overall design, but verifying their basic operation one step at a time is still a good strategy to avoid concurrent problems in troubleshooting later.
- 6) Install your MCU on your prototype board and make the basic connections. (If you are ultimately going to derive power from the same source as the motor drive, you

should consider regulation for the MCU.) Program the flash memory on the MCU using one of the programmers available for sign-out from the Technicians' office (Microshp's PICkit 2). Programming instructions and set up are described in more elsewhere in this document.

3.4 Part 2: Design and testing of an electronic governor

For the remainder of this lab, you will be completing your design and building and testing your electronic governor. The restrictions you will be required to work within are as follows:

- You can assume you are working from a 5 - 10VDC source of energy (remember not to overdrive the 6VDC motors!).
- The MCU available to you is, as stated above, the PIC16F886 from Microchip Technology
- A small 6VDC brushed electric motor is available as a sample target motor. The board on which the motor is mounted also contains a Fairchild QRD1114 Reflective Object Sensor.
- Microchip's development software has been installed on the Lab computers for your use. Hi-Tech's tool suite has also been installed (plug in), and provides a convenient interface and compiler. Compatible programmers (PICkit 2) that will plug into a USB port are available for use to program your MCU.
- remaining components must be from laboratory stock (A reasonable selection of suitable components is available. Some parameters may have to be adjusted slightly to conform to component spec's.)

Additional information on these components will be available from the manufacturer, on the EE391 web site, or from your instructors.

3.4.1 Design Objective:

Your objective will be to design and build a motor controller that will keep the shaft speed of the target electric motor within $\pm 2\%$ of its set speed. The set speed may be determined in one of two ways (your choice): a particular rpm may be pre-selected by the operator through some type of user interface after which the control circuit should keep it within the specified margin, or the user may adjust the speed manually by some method and then "lock" this speed in (e.g. by pressing a button). The control should maintain the set speed through changes in input voltage (5 -10VDC) and changes in load (+50% motor current). After a step change in supply voltage or load, the motor shaft speed must be within the specified limit after 1 second. Serious consideration should be given to a "fail safe" design, where failure of a component or an unresponsive (i.e. hung) MCU will not cause the target motor to rev uncontrolled (in practice, the particular application would dictate whether this would mean stopping the motor altogether or controlling the speed at some safe but reduced rpm).

3.4.2 Report

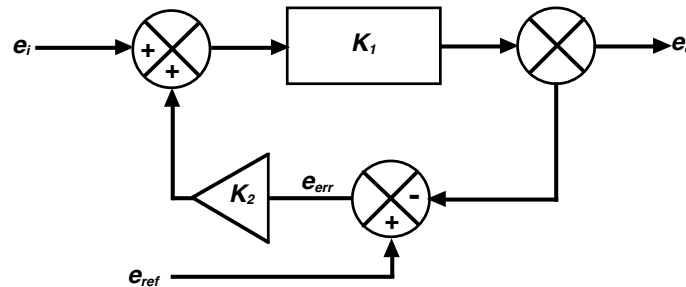
You may be required to submit a report of your results (semi-formal report format). This report should cover your initial characterization data and verification results, as well as your design objectives, restrictions, decisions, code and results. Each student must write their own report, even though much of the data may be in common with your lab partner.

The due date for this report will be announced in the Lab and posted on the EE391 web site. Note: Material covered in this laboratory will be examinable.

3.5 Background Theory:

As suggested in the introduction, this motor control problem is an application of basic control theory. While it is difficult to model practical systems accurately, a basic understanding of the principles involved will be beneficial when it comes to designing an appropriate algorithm for a feedback loop as required in this design.

A simple closed-loop (feedback) control system, like that depicted below, drives an output based on an error signal derived by comparing a reference value and a representation of the output.



Such systems are often modeled as second order systems, often in Laplace-transform or Z-transform terms. In an electro-mechanical system such as we're have in the case of a motor, the inertia of the masses, viscous loading of the parts and any applied load all affect the parameters of the system. Although it is possible to model such a system approximately, empirical fine tuning will undoubtedly be required for a final design.

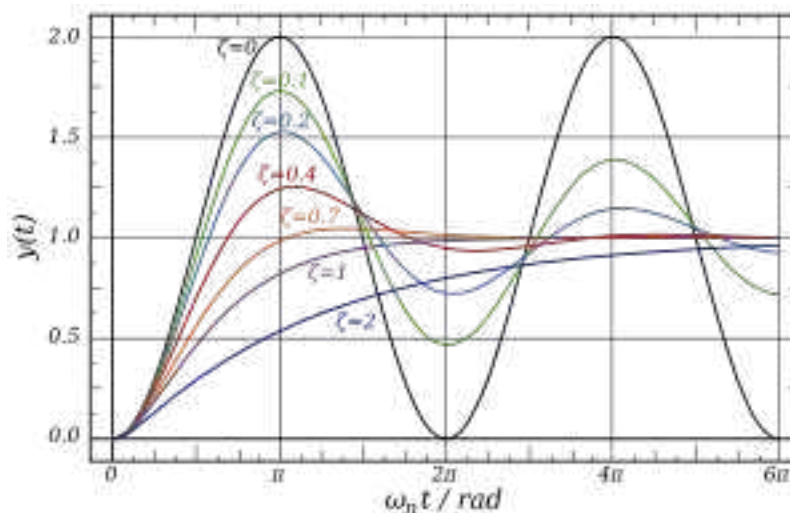
Consider an intuitive explanation of the feedback control system we are considering here. Let us assume we have a motor driven by a voltage source, and we wish to control the speed of the motor in order to keep it at a constant rpm if either the load varies or the input supply voltage sags. There are a number of things we need to design. First we need to devise some way to control the voltage applied to the motor in order to control its speed. Next we would figure out a way to measure the rpm of the output shaft and convert that into some type of signal we could use as feedback. Then we would create some type of an adjustable reference that we could compare to the fed back rpm. If we found the rpm was lower than we desired, we would want to increase the voltage to bring it back up. If it was higher, we would want to reduce it. Of course, how, and how much, we increase or decrease the drive voltage gets a little more complicated. In any system, there is some delay between the time a change occurs in the input and the time the response to that change shows up in the output. This delay is due to a number of factors like the "inertia" in the system; the mass in an armature or load takes some time to accelerate. There is also often a delay in getting the feedback information from the output back, through the feedback channel, to the input. If we use a very simple system and apply a very large increase (or decrease) in drive voltage for even a very small change (i.e. "error") in rpm, then by the time we notice that the rpm is caught up and reduce the drive, the rpm will likely "overshoot" and then become too high (especially if there is also some delay in our feedback loop). Our simple system would then drastically reduce the drive and the motor will slow down, again probably too far, and fall below the set point. It would then apply a much higher drive voltage to bring the rpm back up etc. The final result is that with the inertia in the system and the delays in response and feedback, our motor could continually be speeding up and then slowing

down over and over and never “settle” at a reasonably constant speed. In some circumstances, the overshoot/undershoot errors would gradually become larger and larger until the system was “out of control”. Having a high amplification of the “error” signal lets us use the maximum drive we have available to bring the rpm back up as fast as possible, but the system could go into uncontrollable oscillations if there is too much gain or too much delay in the feedback loop! One thing we could do is to reduce the amplification of the error signal and thus the magnitude of the increase (decrease) in the drive voltage. This would help “stabilize” the system, but it may take a very long time after a change in load for the desired rpm to again be realized. We need to find a compromise between correcting any speed errors quickly and the risk that the system will oscillate excessively. Somehow finding this compromise between speed of response and stability is the bane of the control engineer.

A mathematical representation of a transfer function of a simplified form of a second order system can be given by the following (LaPlace) expression (Ogata, p228):

$$\frac{e_o}{e_i} = \frac{\omega_n^2}{s^2 + 2\xi\omega_n s + \omega_n^2},$$

where ω_n is the “undamped natural frequency” of the system (i.e. the rate at which it will oscillate if it goes unstable), and ξ is the “damping ratio” (how aggressively the system will ‘squash’ any oscillation that does occur). The natural frequency and damping ratio are the critical parameters of a system that must be considered in order to end up with a stable yet responsive closed-loop system.

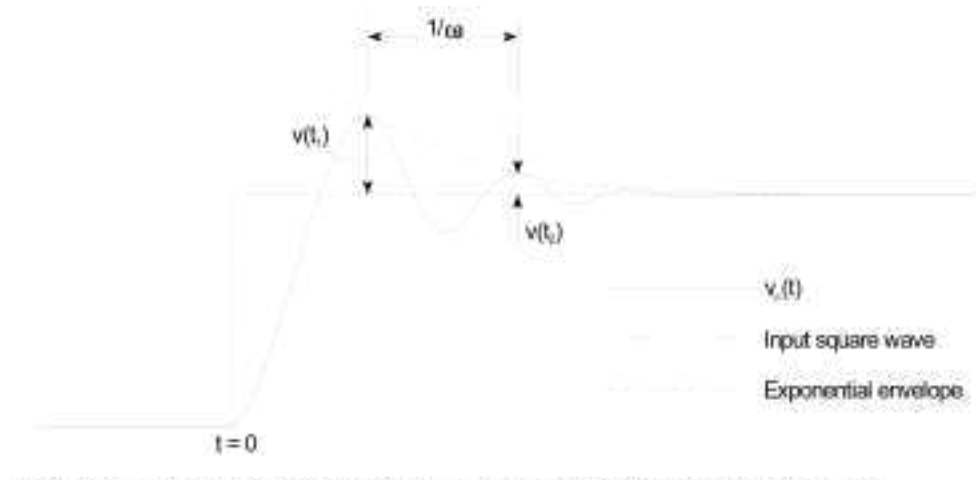


The graph above (en.wiki) shows the output response of a feedback-based 2nd-order controlled system at various levels of damping. Notice that a ξ of 0 means no damping whatsoever, and the system will oscillate at its ‘natural frequency’ (ω_n) with the right average, but never the right value! As we increase the damping ratio from 0 towards 1, you can see that the oscillations die down faster, but at the expense of a longer time to get back to the desired output value (normalized as 1.0 on the graph). A ξ of 1 is known as “critically damped; it is the fastest recovery from an error with *no overshoot* at all. A $\xi > 1$ is called overdamped, and is very stable, but takes a long time to recover from a change. Usually a small amount of overshoot is acceptable to gain a quicker

response, but care must be exercised to ensure that unanticipated changes in system parameters don't inadvertently lead to an unstable system.

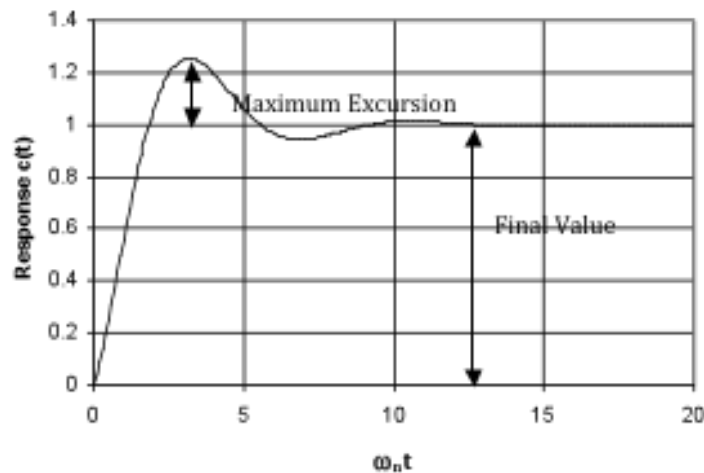
You may be able to take a couple of empirical measurements on your actual system to estimate ω_n and ξ , but you should design your control algorithm to make it easy for you to adjust these parameters in your software so you can fine tune the system for actual usage conditions. The procedure described in "Second Order Systems" may provide some insight into how this might be accomplished. Some excerpts from that source are reproduced below for convenience.

The figure below show a typical, underdamped system's response to a step change.



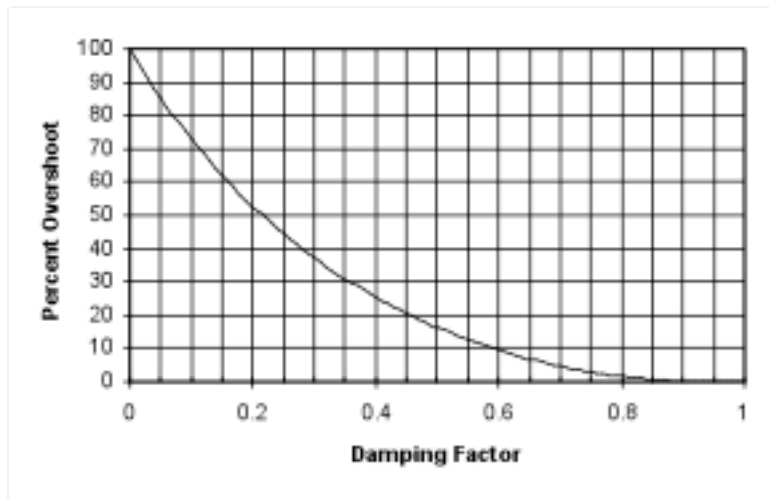
We can observe the overshoot and damped natural frequency. The damped natural frequency, ω_d , is easily found from $v_c(t)$ as shown above. The damping factor (ratio), ξ , and the damped natural frequency, ω_n are related by: $\xi = \sigma/\omega_n$ and σ is found as follows: at the two points shown, $v(t_1)$ and $v(t_2)$, the $\sin()$ component = 1. Therefore, $v(t_1) = Ke^{-\sigma t_1}$ and $v(t_2) = Ke^{-\sigma t_2}$. The ratio of $v(t_1)$ to $v(t_2)$ is therefore: $\frac{v(t_1)}{v(t_2)} = e^{-\sigma(t_1-t_2)}$.

The damping factor of a 2nd order system may be determined by measuring step response overshoot as indicated in the next figure.



Percent overshoot is defined as the ratio of the maximum excursion to the final, settled value. In this case, the percent overshoot is approximately 25%. The maximum overshoot M_p is related to the damping factor ξ through $M_p = e^{-\frac{\pi}{\sqrt{1-\xi^2}}}$ (Second...)

After measuring the percent overshoot, the damping factor can also be determined using the following graph.



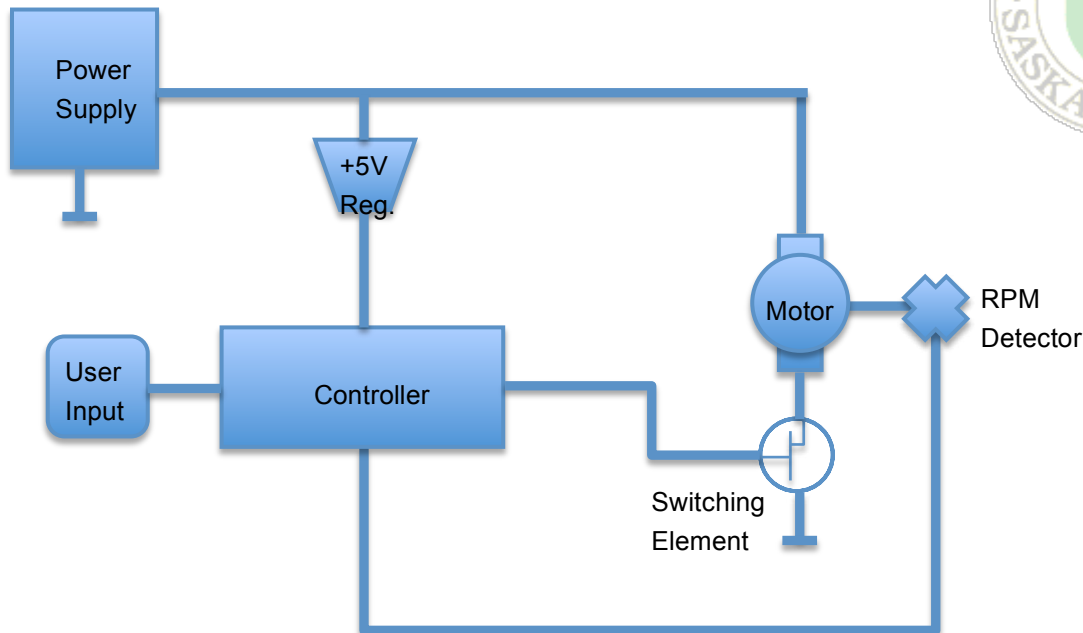
On a small DC motor, it may be difficult to measure its response to a step input effectively, but with some creativity, a solution may arise.

Remember, you have a target specification for output rpm error and the amount of time the system is allowed to be outside these limits. As mentioned earlier, you will want to create a control algorithm that will allow a quick recovery from a parameter change while not allowing too much overshoot. Consider this carefully when creating your MCU code.



3.6 Design Examples:

The following block diagram shows the basic elements of one reference design.



The following code examples give some possible solutions which can be used as reference when developing your own design.

3.6.1 Appendix A – Code Examples

```
// Governor_lab.c
//*****
//
// This program has the following functionality that should help you become familiar with the
// PIC microcontroller. – Rlee Prokopishyn 2008, modified and commented by James Kowalski, 2009.

//1.This program is suitable for a code template showing the basic structure required, variable and
function
//declarations, use of header files, configuration bits, IO setup, etc.
//2.The A/D converter is set up on RA5 (pin #7). It uses vdd and vss for reference.
//3.The PWM is set up to output on RC2 (pin #13). It is controlled by the result from the A/D converter
//ie VDD applied to the A/D will produce 100% pwm duty cycle and VSS will produce 0%.
//4.The comparator is set up with its inverting input on RA0 (pin #2) and reference at about 1.25v. The
//comparator outputs on RA4/C1OUT (pin #6) so you can check its operation.
//5.Timer0 is set up with a 256 prescale so it interrupts every 65536us and pulses RB0 (pin #21).
//6.There is a test function to show you how to set up functions, when it is called RB4 (pin #25) is pulsed.

//*****

/*processor header file*/
```




```
#include <htc.h> //loads right header file for the chip in use
//in this case the header file is pic16f887.h

#include <pic.h> //needed for some compiler library functions

/*xtal freq for ms and us delays*/
#define _XTAL_FREQ 4000000

/*Configuration bits*/
// the header file pic16f887.h explains the function of these
// Config Reg 1 and 2. data sheet page 210
__CONFIG(INTCLK & WDTDIS & PWRTDIS & MCLRDIS & UNPROTECT & DUNPROTECT & BORDIS
& IESODIS & FCMDIS & LVPDIS & DEBUGDIS & BORV21);

//declare some variables
unsigned int testvar; // 16 bit variable //variables declared outside functions and main are global
unsigned char testvar1; // 8 bit variable //they should be declared volatile to use in isr.

//declare functions here for use later
void testfunction(void);

//interrupt code
interrupt void isr(void)
//you can check which interrupt here
//if T0IF flag is set then timer 0 interrupt
//in this case only timer 0 is configured for interrupt
{
    //Clear timer 0 interrupt flag, Intcon Reg Data sheet page 31.

    T0IF = 0;
    //Reload timer 0
    TMR0=0x00;
    //if timer 0 interrupts pulse RB0/AN12 (IC pin 21).
    RB0=1;
    __delay_ms(5);
    RB0=0;
}

//main is here
void main()
{
//Set up port I/O and peripherals
```



```
//Internal oscillator frequency is defined previous at 4MHz
//OSCCON = 0X70; TO set Osc. at 8MHz, data sheet page 62.

//Set ANO to AN4 = 0 digital I/O. TRISA,PORTA and ANSEL Reg page 39.
ANSEL &= 0b11100000;
//Set ANS13 TO ANS8 = 0 digital I/O. Page 48.
ANSELH &= 0b11000000;

//set up PWM unit, data sheet page 123.
//Disable O/P driver RC2/P1A/CCP1 pin 13,set as digital input
//This is just good programming. So motor will not start to turn as PWM is setup.
//Pin 13 will be an O/P after setup.
TRISC |= 0b00000100;
//Set PWM period by loading PR2 reg with 254 0xfe timer2 prescale at 1
PR2 = 0xfe;
//CCP1 Reg, single o/p P1A modulated, PWM mode.
//DC1B0=0, LSB (bit 0) of PWM duty cycle in CCP1 Reg.
//DC1B1=0, LSB (bit 1) of PWM duty cycle in CCP1 Reg.
CCP1CON = 0X0C;
//Set PWM duty cycle at high to start initialize to start up motor
//This motor won't start if the voltage is too low
CCPR1L = 0xff; //MSBs of PWM duty cycle
//Configure and start timer 2
TMR2IF=0; //clear interrupt flag
//Timer 2 control reg page 81, prescaler at 1, timer on.
T2CON = 0X04;
//Enable O/P driver RC2/P1A/CCP1 pin 13,set as digital output.
TRISC &= 0b11111011;
//Now PWM can be adjusted for 0-100% by writing 0 - ff to CCPR1L

//Set up A/D converter page 99 to page 110.
//RA5/AN4(pin 7) set as analog I/P, ANSEL Reg page 39.
ANSEL |= 0b00010000;
//RA5/AN4(pin 7) is input used so disable output driver
TRISA |= 0b00100000;
//ADCON REG 0. Conversion clock fosc/8, AD Ch4, ADC enabled
ADCON0 = 0b01010001;
//ADCON REG 1. Data format(left justified = 0), Voltage ref(vdd and vss)
//ADRESH (MSB/bit7 to bit0). 10 to 3 bits of 10-bit of A/D Result.
//ADRESL (bit7 to bit6/LSB). 2 and 1 bits of 10-bit of A/D Result.
ADCON1=0x00;
```



```
//Wait till Sampling circuit stabilizes
__delay_ms(1);
//Set GO/DONE bit(1) in ADCON0 Reg to start conversion and poll it
//When GO/DONE is zero conversion is complete

//Set up comparators
//CM1CON0 Reg. page 88,
//Comparator 1 on, C12IN0(PIN 2)to C1Vin-, C1Vref O/P to C1Vin+, O/P=C1out(pin6)=0 when
C1Vin+ < C1Vin-
CM1CON0 = 0b10100100;
//Set up CVRef For comparator at about 1.25 volt. VRCON Reg page 97.
//Use equation 8-1 page 94 to calculate CVref(CVRef meas < calc). VRCON Reg(CVRef equation
wrong)page 97.
//Enable CVRef and CVRef voltage on pin 4.
VRCON = 0b11000000;
//Set C1out/RA4(pin 6)and CVRef(pin 4)as digital O/P. TRISA,PORTA and ANSEL Reg page 39.
TRISA &= 0b11101011;
//comparator output is available on pin #6 as well as by monitoring C1OUT in CM1CON0

//Set up Timer 1 16 bit timer page 76.
//set for 8 prescale, internal clock Fosc/4, timer off.
//Fosc/4 = 4MHz/4 = 1uS period, 16 bit = 65536, prescaler = 8, 1uS x 65536 x 8 = 524280uS
T1CON = 0b00110100;
//clear timer
TMR1H = 0;
TMR1L = 0;
//Just set TMR1ON to start timer, reset TMR1ON to stop it.

//Set up Timer 0, 8 bit timer to interrupt on overflow
// Option Reg page 30
//Timer 0 will interrupt (because of prescaler) every 65536 us, timer increments every instruction cycle
T0CS = 0; //internal clock inc every intruction cycle Fosc/4
PSA = 0; //prescaler assigned to timer 0
OPTION |=0b00000111; //OR Option Reg with 0b00000111, result in Option Reg. prescaler set at
256
//Fosc/4 = 4MHz/4 = 1uS period, 8 bit = 256, prescaler = 256, 1uS x 256 x 256 = 65536uS
//Turn on interrupts, Intcon Reg Page 31
GIE=1; //Global interrupt enable.
T0IE=1; //Timer 0 interrupt enable
//load timer 0
TMR0 = 0x00;
T0IF=0; //timer0 interrupt flag cleared
```



```
//Set RB0/AN12 (pin 21) as digital output.  
//Testfunction will pulse RB4/AN11 (pin 25) so make it an digital output.  
    TRISB &= 0b11101110;
```

```
//End of initialization.
```

```
for(;;)  
{  
    //use a/d convertor results to test pwm  
    GODONE = 1;  
    while (GODONE==1)//when godone goes low conversion is finished  
        {  
        }  
    CCPR1L = ADRESH;
```

```
    testfunction();
```

```
}
```

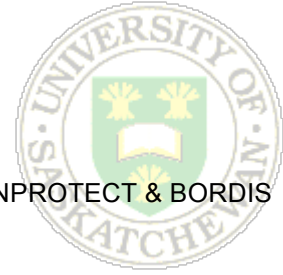
```
}
```

```
void testfunction(void)//function goes here
```

```
{  
    RB4=1;  
    __delay_ms(1);  
    RB4=0;  
}
```

```
//Jame?s Sol?n  
//Works like a cruise control  
//One possible solution for the governor lab.  
//set point is set by pot in real time  
//pushing button locks in speed
```

```
/*processor header file*/  
#include <htc.h> //loads right header file for the chip in use  
    //in this case the header file is pic16f887.h  
#include <pic.h> //needed for some compiler library functions
```



```
/*xtal freq for ms and us delays*/
#define _XTAL_FREQ 4000000

/*Configuration bits*/
// the header file pic16f887.h explains the function of these
__CONFIG(INTCLK & WDTDIS & PWRTDIS & MCLRDIS & UNPROTECT & DUNPROTECT & BORDIS
& IESODIS & FCMDIS & LVPDIS & DEBUGDIS & BORV21);

//declare some variables
unsigned int tach_old;//setpoint tach value
volatile unsigned int tach;//current tach value modified in interrupt so volatile

//interrupt code
interrupt void isr(void)
{
//this routine use Timer 1 to measure the period of the tach pulse
//a comparator interrupt will turn the clock on and off
//if timer 1 is off turn on to start timing pulse
    //Global interrupt disable. INTCON Reg page 31.
    //Disabled so glitches triggering comparator do not trigger interrupt.
    GIE=0;
//Two Comparator transition occur due to noise. C1out/RA4(pin 6).
    //Delay from the start of the 1st transition to the end of the 2nd.
    //This allows clear of comparator interrupt flag without a sudden set of flag right after clear.
    __delay_ms(1.5);
//clear comparator interrupt flag
    C1IF=0;
    if (TMR1ON==0) //Timer 1 OFF
    {
        TMR1ON=1;
    }
    else
    //If timer 1 was running turn off, save tach value and clear timer
    {
        TMR1ON=0;
        tach=(TMR1H<<8)+TMR1L;
        TMR1H=0;
        TMR1L=0;
    }
    GIE=1;          //Global interrupt enable. INTCON Reg page 31.
}

//main
```



```
void main()
{
//*****
//;initialization code
//
//Set ANO to AN4 = 0 digital I/O. Page 40.
    ANSEL &= 0b11100000;
//Set ANS13 TO ANS8 = 0 digital I/O. Page 48.
    ANSELH &= 0b11000000;

//set up PWM unit, data sheet page 123.
    //Disable O/P driver RC2/P1A/CCP1 pin 13,set as digital input
    //This is just good programming. So motor will not start to turn as PWM is setup.
    //Pin 13 will be an O/P after setup.
    TRISC |= 0b00000100;
    //Set PWM period by loading PR2 reg with 254 0xfe timer2 prescale at 1
    PR2 = 0xfe;
    //CCP1 Reg, single o/p P1A modulated, PWM mode.
//DC1B0=0, LSB (bit 0) of PWM duty cycle in CCP1 Reg.
//DC1B1=0, LSB (bit 1) of PWM duty cycle in CCP1 Reg.
    CCP1CON = 0X0C;
    //Set PWM duty cycle at high to start initialize to start up motor
    //This motor won't start if the voltage is too low
    CCPR1L = 0xff; //MSBs of PWM duty cycle
//Configure and start timer 2
    TMR2IF=0; //clear interrupt flag timer 2.
//Timer 2 control reg page 81, prescaler at 1, timer on.
    T2CON = 0X04;
    //Enable O/P driver RC2/P1A/CCP1 pin 13,set as digital output.
    TRISC &= 0b11111011;
    //Now PWM can be adjusted for 0-100% by writing 0 - ff to CCPR1L

//Set up A/D converter page 99 to page 110.
    //RA5/AN4(pin 7) set as analog I/P, ANSEL Reg page 39.
    ANSEL |= 0b00010000;
    //RA5/AN4(pin 7) is input used so disable output driver.
    TRISA |= 0b00100000;
    //ADCON REG 0. Conversion clock fosc/8, AD Ch4, ADC enabled
    ADCON0 = 0b01010001;
    //ADCON REG 1. Data format(left justified = 0), Voltage ref(vdd and vss)
    //ADRESH (MSB/bit7 to bit0). 10 to 3 bits of 10-bit of A/D Result.
    //ADRESL (bit7 to bit6/LSB). 2 and 1 bits of 10-bit of A/D Result.
    ADCON1=0x00;
    //Wait till Sampling circuit stabilizes
```



```
__delay_ms(1);
//Set GO/DONE bit(1) in ADCON0 Reg to start conversion and poll it
//When GO/DONE is zero conversion is complete

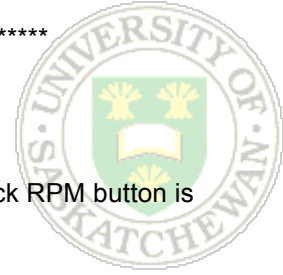
//Set up comparators
//CM1CON0 Reg. page 88,
//Comparator 1 on, C12IN0(PIN 2)to C1Vin-, C1Vref O/P to C1Vin+, O/P=C1out(pin6)=0 when
C1Vin+ < C1Vin-
CM1CON0 = 0b10100100;
//Set up CVRef For comparator at about 3.3 volt. VRCON Reg page 97.
//Use equation 8-1 page 94 to calculate CVref(CVRef meas < calc). VRCON Reg(CVRef equation
wrong)page 97.
//Enable CVRef and CVRef voltage on pin 4.
VRCON = 0b11001101;
//Set C1out/RA4(pin 6)and CVRef(pin 4)as digital O/P. TRISA,PORTA and ANSEL Reg page 39.
TRISA &= 0b11101011;
//comparator output is available on pin #6 as well as by monitoring C1OUT in CM1CON0

//set up the comparator to interrupt when triggered
C1IF=0; //comparator 1 interrupt flag. PIR2 Reg page 35. A change at C1OUT will set
interrupt flag.
GIE=1; //Global interrupt enable. INTCON Reg page 31.
PEIE=1; //Peripheral interrupts enable. INTCON Reg page 31.
C1IE=1; //Comparator 1 interrupt enable. PIE2 Reg page 33.

//Set up Timer 1 16 bit timer page 76.
//Timer 1 used in interrupt.
//set for 1 prescale, internal clock Fosc/4, timer off.
//Fosc/4 = 4MHz/4 = 1uS period, 16 bit = 65535, prescaler = 1, 1uS x 65536 x 1 = 65536uS
T1CON = 0b00000100;
//clear timer
TMR1H = 0;
TMR1L = 0;
//Just set TMR1ON to start timer, reset TMR1ON to stop it

//Set RB0/AN12 (pin 21,LED) as digital output.
TRISB &= 0b11111110;

//Set RA3/AN3 (pin 5) as digital input for lock RPM switch.
TRISA |= 0b00001000;
```



```
//End of initialization
//*****
//This is the start of the main program.
start:

//Perform this routine, adjusting PWM (rpm) with A/D(pot)RA5/AN4(pin 7) until Lock RPM button is
pressed.
//Loop here until Lock RPM button RA3/AN3 (PIN 5) is pushed.

while (RA3==1) //Lock RPM button not pushed.
{
    GODONE = 1; //Start A/D conversion. ADCON0 Reg page 104.
    while (GODONE==1) //When GODONE goes low conversion is finished.
    {
    }
    CCPR1L = ADRESH; //Copy the A/D value to the PWM.
    RB0=0; //LED off
}

//Wait for user to let go of Lock RPM button and debounce.
while (RA3==0)
{
}
__delay_ms(50); //debounce

//Begin code to control motor at locked RPM.
//Record current tach value.
tach_old = tach; //This is the RPM value to control.

RB0=1; //Lock led on.

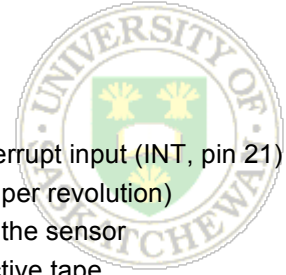
//Loop here until Lock RPM button pressed again.
//Control motor to Lock RPM.
//The tach value is the length of time of the pulse coming from the motor reflective sensor in uS.
//If the motor is running at the same speed as Lock RPM (tach = tach_old), do nothing.
//If the motor is running slower than the Lock RPM (tach < tach_old), increment PWM.
//If the motor is running faster than the Lock RPM (tach > tach_old), decrement PWM.

for(;;) //Loop forever until Lock RPM button pressed again.
{
    if (tach = tach_old)
    {
    }
    else if (tach < tach_old)
```




```
{
    CCPR1L++;
    //check for upper limit
    if (CCPR1L==0xff)
        //If upper limit is reached then problem regulating, something
        //wrong, etc, so turn off motor and try to restart.
        {CCPR1L=0x00;}
    }
    else if (tach > tach_old)
    {
        CCPR1L--;
        //check for lower limit
        if (CCPR1L<5)
            //IF lower limit is reached the shut down the motor.
            {CCPR1L=0;}
    }

    //Go back to start Lock RPM button is pressed again.
    if (RA3==0)
    {
        //Wait for user to let go of Lock RPM button and debounce.
        while (RA3==0)
        {
            __delay_ms(50);    //debounce
            goto start;
        }
    }
}
```



```
//Modifications of Rlee's and James' code by Denard Lynch – Aug 2009
//Works like a cruise control pushing button locks in speed
//One possible solution for the governor lab.
//set point is set by pot in real time
//This example uses the output from the optical reflective sensor directly as an interrupt input (INT, pin 21)
//It only attempts to modify the speed when it has new input data to use (i.e. once per revolution)
//It also tries to average the input over several revolutions to eliminate "jitter" from the sensor
//The reflective sensor was also modified to cover approx. half the area with reflective tape.
//There are also several bits (e.g. pin 22, 23, ...) used as "test" flags to monitor the code.
//
//
//
/*processor header file*/
#include <htc.h> //loads right header file for the chip in use
                //in this case the header file is pic16f887.h
#include <pic.h> //needed for some compiler library functions

/*xtal freq for ms and us delays*/
#define _XTAL_FREQ 4000000

/*Configuration bits*/
// the header file pic16f887.h explains the function of these
__CONFIG(INTCLK & WDTDIS & PWRTDIS & MCLRDIS & UNPROTECT & DUNPROTECT & BORDIS
& IESODIS & FCMDIS & LVPDIS & DEBUGDIS & BORV21);

//declare some variables
unsigned int tach_set;//setpoint tach value
volatile unsigned int tach;//current tach value modified in interrupt so volatile
volatile unsigned int long sum;//used to calculate time average, needs to handle >> max time
unsigned int count=0;//set up a counting variable
unsigned int time[20] = {0x4F};//set up a 20 element array to store time values over 20 revs and set them
all to 50% to start
unsigned int flag=0;//set up a flag so we know when to regulate
unsigned int n=0;//set up a index count variable

//interrupt code - modified to do a moving average and interrupt every revolution
//
interrupt void isr(void)
{
//this routine uses Timer 1 to measure the period of the tach pulse
//an interrupt on min 21 will run this code each time the wheel does 1 rev
//
                //Global interrupt disable. INTCON Reg page 31.
                //Disabled so it's not re-triggered while we do this routine
                GIE=0;
```



```
INTE=0;

//Delay from the start of the 1st transition to the start of the 2nd.

TMR1ON=0; //Timer 1 should be on, so turn off to do this routine
time[count] = (TMR1H<<8) + TMR1L; //start at element [0] and save the counter value
//RB2 = !RB2;//toggle this bit, pin 23, to show we're entering this ISR

    if (count < 19)
    {
//RB3 = !RB3;
        count++;
    }
    else
    {
//RB4 = !RB4;
        count=0;
    }
    for ( n=0; n<20; n++)//for n=0 to 19 (20 times) do the statement below
    {
//RB5 = !RB5;
        sum += time[n];//sums the 20 elements of array time[] and put it in sum
    }
    tach = (sum / 20);//take the average

// clear the timer registers and enable the timer to start counting again
    TMR1H=0;
    TMR1L=0;
    TMR1ON=1;

    GIE=1; //Global interrupt enable. INTCON Reg page 31.
    INTF=0;//clear the external interrupt flag first, then enable INT for the next rev
    INTE=1;
    flag=1;
    //__delay_ms(5);

}

//main
void main()
{
//*****
//;initialization code
//
```



```
//Set ANO to AN4 = 0 digital I/O. Page 40.
    ANSEL &= 0b11100000;
//Set ANS13 TO ANS8 = 0 digital I/O. Page 48.
    ANSELH &= 0b11000000;

//set up PWM unit, data sheet page 123.
    //Disable O/P driver RC2/P1A/CCP1 pin 13,set as digital input
    //This is just good programming. So motor will not start to turn as PWM is setup.
    //Pin 13 will be an O/P after setup.
    TRISC |= 0b00000100;
    //Set PWM period by loading PR2 reg with 254 0xfe timer2 prescale at 1
    PR2 = 0xfe;
    //CCP1 Reg, single o/p P1A modulated, PWM mode.
    //DC1B0=0, LSB (bit 0) of PWM duty cycle in CCP1 Reg.
    //DC1B1=0, LSB (bit 1) of PWM duty cycle in CCP1 Reg.
    CCP1CON = 0X0C;
    //Set PWM duty cycle at half (127/255) to start up motor
    //This motor won't start if the voltage is too low
    CCPR1L = 0x7f; //MSBs of PWM duty cycle, 50%

//Configure and start timer 2 to set the period
    TMR2IF=0; //clear interrupt flag timer 2.
    //Timer 2 control reg page 81, pre-scaler at 1, timer on.
    T2CON = 0X04;
    //Enable O/P driver RC2/P1A/CCP1 pin 13,set as digital output.
    TRISC &= 0b11111011;
    //Now PWM can be adjusted for 0-100% by writing 0 - ff to CCPR1L

//Set up A/D converter page 99 to page 110.
    //RA5/AN4(pin 7) set as analog I/P, ANSEL Reg page 39.
    ANSEL |= 0b00010000;
    //RA5/AN4(pin 7) is input used so disable output driver.
    TRISA |= 0b00100000;
    //ADCON REG 0. Conversion clock fosc/8, AD Ch4, ADC enabled
    ADCON0 = 0b01010001;
    //ADCON REG 1. Data format(left justified = 0), Voltage ref(vdd and vss)
    //ADRESH (MSB/bit7 to bit0). 10 to 3 bits of 10-bit of A/D Result.
    //ADRESL (bit7 to bit6/LSB). 2 and 1 bits of 10-bit of A/D Result.
    ADCON1=0x00;
    //Wait till Sampling circuit stabilizes
    __delay_ms(1);
    //Set GO/DONE bit(1) in ADCON0 Reg to start conversion and poll it
    //When GO/DONE is zero conversion is complete
```



```
//Set up Timer 1 16 bit timer page 76.
//Timer 1 used in interrupt.
//set for 1:1 prescale, internal clock Fosc/4, timer off.
//Fosc/4 = 4MHz/4 = 1uS period, 16 bit = 65535, prescaler = 1, 1uS x 65536 x 1 = 65536uS
T1CON = 0b00000100;
//clear timer
TMR1H = 0;
TMR1L = 0;
//Just set TMR1ON to start timer, reset TMR1ON to stop it

//Set RB0/INT (pin 21) as digital input, and RB1 as output for LED instead of RB0, and RB2,3,4,5 as test
bits (out).
TRISB = 0b11000001;
//setup the external interrupt to trigger on falling edge and enable it
INTEDG = 0;
INTF=0;
INTE = 1;
GIE=1;
//Set RA3/AN3 (pin 5) as digital input for lock RPM switch.
TRISA |= 0b00001000;

//End of initialization
//*****
//This is the start of the main program.

start:

//Perform this routine, adjusting PWM (rpm) with A/D(pot)RA5/AN4(pin 7) until Lock RPM button is
pressed.
//Loop here until Lock RPM button RA3/AN3 (PIN 5) is pushed.

while (RA3==1) //Lock RPM button not pushed.
{
    GODONE = 1; //Start A/D conversion. ADCON0 Reg page 104.
    while (GODONE==1) //When GODONE goes low conversion is finished.
    {
    }
    CCPR1L = ADRESH; //Copy the A/D value to the PWM.
    RB1=0; //LED off
RB5 = !RB5;
}
//Wait for user to let go of Lock RPM button and debounce.
while (RA3==0)
{
```



```
    }
    __delay_ms(10);    //debounce

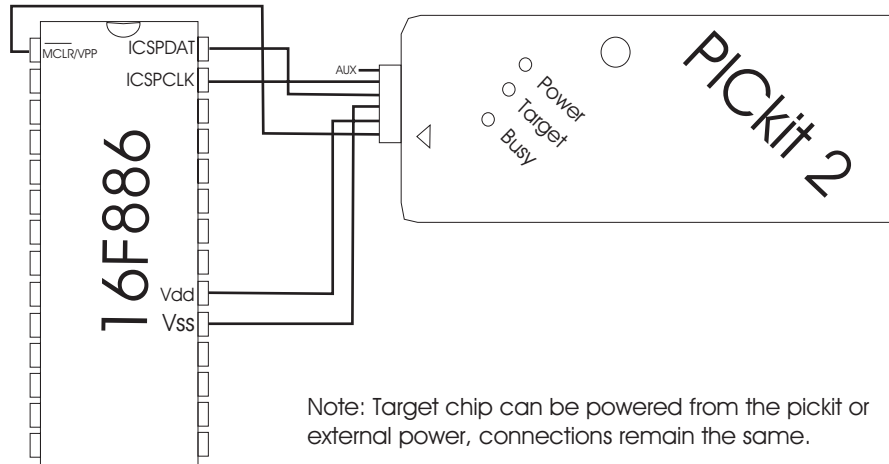
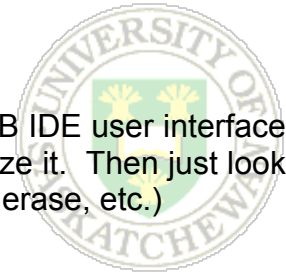
//Begin code to control motor at locked RPM.
//Record current tach value.
    tach_set = tach; //This is the RPM value to control.
    RB1=1; //Lock led on.

//Loop here until Lock RPM button pressed again.
//Control motor to Lock RPM.
//The tach value is the length of time of the pulse coming from the motor reflective sensor in uS.
//If the motor is running at the same speed as Lock RPM (tach = tach_old), do nothing.
//If the motor is running slower than the Lock RPM (tach < tach_old), increment PWM.
//If the motor is running faster than the Lock RPM (tach > tach_old), decrement PWM.

//
//You need some code in here to take the measured speed, compare it to the desired speed,
// and change the duty cycle appropriately.
//
```

3.6.2 Appendix B – PICkit 2 Programmer

To select the programmer, you simply go to “Programmer” in MPLAB IDE user interface and select PICkit 2, the IDE will then look for the PICkit 2 and initialize it. Then just look at the “Program” menu for all the required functions (program, read, erase, etc.)



Pic to PICkit2 Connections

Right angle pin sets area available to easily connect the PICkit 2 programmer to your prototype board

3.7 References:

Wellstead, Peter, and Readman, Mark, "Engine Speed Control", www.control-systems-principles.co.uk, accessed Sept. 2008.

Ogata, Katsuhiko, "Modern Control Engineering", Prentice-Hall, New Jersey, 1970

en.wikipedia.org/wiki/Damping_ratio, accessed Sep, 2008

EE391 Electrical Engineering Laboratory, "Second Order Systems", University of Saskatchewan, 2006



4 Appendix A: Data Sheets

Note: no longer included with manual – please refer to online sources



