Handbook for Learning "Circuit Theory"

Mani Soma, soma@ee.washington.edu

Revision date: June 2001

This Handbook is designed for you as a guide to learn the circuit analysis and design skills in Circuit Theory. The document is organized based on the topics to be taught, not based on any specific textbook or lesson structure. After the introductory sections listing the topic coverage and suggested textbooks, the guide for each topic is presented in three major classifications:

1. Overall learning notes for this specific topic.
2. Emphasis: key issues that you should pay attention to, suggestions on techniques to practice skills, etc.
3. Once Over Lightly: topics that you need to have some familiarity with but are not expected to master by the end of the course. In most cases, the instructor might decide to skip these topics if time runs out in the course.

1. Course contents

1.1 Prerequisites

1.1.1 Skills and topics

This course assumes these prerequisite skills and knowledge from previous courses, typically Mathematics courses and one introductory circuit course in electrical engineering. You should already know how to:

- Use Kirchhoff's current and voltage laws (KCL and KVL).
- Apply Ohm's law.
- Apply efficient circuit theorems to speed up analysis of circuits containing parallel or series combinations of elements, voltage dividers, and current dividers.
- Use Thevenin and Norton equivalent circuits to simplify the analysis process.
- Work with controlled voltage and current sources.
- Use linearity and superposition to write current and voltage equations resulting from node analysis and mesh (or loop) analysis.
- Analyze circuits containing capacitors and inductors, in addition to resistors.
- Analyze first-order and second-order circuits in the time domain. Integrate and differentiate common functions.
- Solve first and second order linear differential equations.
- Manipulate complex numbers (add, subtract, multiply, divide, complex conjugate, absolute value, phase (argument), etc.).
- Manipulate vectors and matrices up to dimension 3 or 4.

If your skills in these topics are rusty, the best time to catch up and review is the first week of the course. Go over the materials you have had: the best way to review is to solve problems based on the old materials. Most textbooks have answers to problems or you can check your previous answers when they took the course covering these prerequisite topics. Keep practicing since these skills underline this present course and all future courses in circuit design.

1.1.2 Background survey

Your instructor is likely to conduct a background survey to evaluate your skills in prerequisite courses and topics. An example of a web-based background survey is enclosed. If you are not sure of your skill level after taking this survey, practice problem solving again using homeworks from previous courses. Your instructor will not be able to spend a lot of time going over old materials since he/she has new materials to cover.

1.2 Learning outcomes

At the end of this course, you will be able to:

- Analyze a circuit given sinusoidal inputs.
- Compute average power consumed or supplied by a circuit.
- Design simple circuits for maximum power transfer to a load.
- Apply Laplace transform techniques to simplify the analysis of complex circuits.
- Analyze circuits in the frequency domain.
- Use several alternative techniques in time-domain and frequency-domain to analyze the same circuit.
- Design simple circuits from time-domain and frequency-domain specifications.
- Use two-port models and parameters to simplify the analysis of large circuits.
- Use SPICE as a computer tool to verify a design, and to confirm time-domain and frequency-domain analysis results.
- Use basic laboratory instruments: oscilloscope, power supply, function generator, and multimeter.
- Measure basic signal parameters: amplitude, frequency, etc.
- Measure and compute basic circuit parameters from measurements.

These objectives are not necessarily listed in the order in which they will be accomplished during the course.

1.3 Topic coverage

The topics to be covered to meet the learning objectives above are:

- Sinusoidal excitation (15% of the course): how to analyze and design simple circuits (containing R, L, C, and opamps) when the signals are sinusoidal.
- Average power and maximum power delivery (5% of the course): how to compute average power dissipated in a load and how design a circuit to maximize power delivery to the load.

- Laplace transformation techniques (30% of the course): techniques to work with simple circuits when the signals are more complicated than a step or a sinusoidal signal.
- Transfer functions (25% of the course): techniques to analyze and design simple circuits to distinguish signal from noise or to select a specific signal.
- Frequency response and simple filters (25% of the course): applications of transfer function techniques to a specific class of circuits.
- Simulation of circuits using SPICE simulator, with some design examples: how to use computer-aided simulation tool to verify analysis and design.
- Basic EE laboratory, components and instrumentation: how to design, build, and test simple circuits to verify concepts from lectures and to validate simulation results.

Some of these topics are intricately linked (e.g. transfer functions and frequency response) and maybe covered as one single topic depending on the instructor's preference.

These topics should be covered in a 10-week quarter. If the course is taught in a semester system (16 weeks), more topics need to be added. This decision will be made by your instructor.

1.4 Suggested textbooks

There are numerous textbooks for this course. A representative non-exhaustive list includes:

- J.W. Nilsson and S.A. Riedel. *Electric Circuits*. Prentice Hall.
- R.C. Dorf and J.A. Svoboda. *Introduction to Electric Circuits*. John Wiley & Sons.
- C.K. Alexander and M.N.O. Sadiku. *Fundamentals of Electric Circuits*. McGraw-Hill.

This Handbook does not recommend any specific text and may be used with any text containing materials for the topics in section 1.3 above. The topics map onto different chapters and sections in a specific textbook, and sometimes the coverage of materials might not be in the order of the topics listed above. Ask your instructor or check the course web site for the specific chapters and sections to be covered in the textbook selected for the course.

1.5 Applicable ABET outcomes

ABET (Accreditation Board for Engineering and Technology) is a national agency which reviews all universities' degree requirements for accreditation. This accreditation is extremely important since it validates your Bachelor degree. Otherwise, the degree would not have any real value when you apply for a job. ABET sets criteria that all degree programs must meet to offer the Bachelor of Science in a particular field of engineering.

With respect to ABET EC 2000 criteria, this course meets the following outcomes:

- Outcome 3 (a) Apply math, science and engineering knowledge.
- Outcome 3 (b) Conduct experiments, as well as to analyze and interpret data.
- Outcome 3 (c) Design simple RC and opamp circuits to meet desired needs.
- Outcome 3 (d) Function and contribute various individual skills in laboratory teams.
- Outcome 3 (e) Identify, formulate, and solve basic RC and opamp circuit problems.

- Outcome 3 (g) Communicate effectively via written laboratory reports.
- Outcome 3 (k) Use the techniques, skills, and modern engineering tools.

2. General learning notes

2.1 Too much math!

The course, like other fundamental electrical engineering courses, contains significant mathematical techniques. However, it is not an extension of mathematical courses. The emphasis is to use mathematics as a tool to understand how circuits work and how to design them. Once you master the fundamental circuits, where mathematics is used a tool, you will learn how to develop a "feel" for circuits and will be able to determine how a complex circuit works without having to resort to messy equations. This "feel" comes with practice and the more you practice, the better you get at analysis and design and the less you have to worry about mathematics. As circuits get more complex, the only way to understand how they work is to employ this "feel" developed by experience. This experience is why "experienced designers" are paid more than novices.

2.2 Learn by doing

Learning how to analyze and design circuits is not a passive activity: you simply do not just sit and listen to the instructor, or do not just read the textbook and somehow expect to be able to design circuits and systems. Acquiring circuit skills is just like acquiring swimming or skiing skills: when you learned how to swim, you did not sit in a classroom listening to lectures for hours, right? You did not read a swimming textbook for weeks, right? You got some instructions and then you were told to jump into the pool and practice!! You need to approach engineering learning the same way: practice, practice, practice. Solving problems is the best way to practice: many textbooks have problems with complete solution procedures and answers provided so you can check your own work. Doing more problems than assigned in the homework is another way. The instructor will be glad to give you solutions for the extra problems you practice at home. As Thomas Edison, a famous electrical engineer, put it, "Genius is 1% inspiration and 99% perspiration."

2.3 Dealing with the textbook

The textbook provides most of the materials and almost all the homework problems for the course. You should learn how to read textbooks (they are written for students!) even though the reading might be dry and boring compared to your other readings. Developing skills in reading technical materials is very important since you will need it when you work in a job: there will be no instructor to help you read technical materials in a job. In this course, if you have problems understanding the textbook materials, you can always ask the instructor as you develop the reading skills. Textbooks sometimes have mistakes: a derivation is not really rigorous, an answer to a problem is incorrect, etc. If you encounter these instances, check with your instructor. It is rather frustrating to see these mistakes in textbooks but they are a fact of life and you might as well learn now how to deal with it. Trust the textbook most of the time but also be ready to question it: you need to develop the ability to judge the materials you read and to decide when they are correct and when they are wrong.

Sometimes a textbook skips a few steps in a derivation (the famous "proof is left as exercise for the reader" quote). Try to fill in these missing steps: it gives you another opportunity to practice and you know the correct answer in advance! So the key is to fill in the procedure, which is the essential skill in problem solving. Check with your instructor if you want to validate your own derivation.

2.4 The "right" way to solve problems

As you develop more skills in analysis and design, you will find out that there is no one "right" way to solve a problem. There are usually several different methods to solve a problem, each with its merits and drawbacks. So you might solve a problem one way and find out that your friend or the instructor solves it in a different way. This is acceptable. The more methods you know, the better. In fact, after you solve a problem, try to solve it again using a different method and see which one you prefer. Compare your solutions with the textbook solution or with the instructor's solution and pick up more problem-solving skills this way.

In real-life engineering, a "trade-off" study is conducted to compare various methods to solve a specific problem, depending on constraints such as cost, resources, legal aspects, etc. Trade-off analysis is a fundamental aspect in engineering design.

2.5 Reflection as a part of learning

After you solve a problem, do not rush to move to the next one. Go over your solution again and see whether you can optimize your procedure. Could you have solved it faster using a different technique? Could you have used a different method (maybe less mathematically messy)? Could you write a different set of KCL and KVL and end up with fewer variables? This type of reflection is important since it reinforces your knowledge and shows you in hindsight other possible procedures.

If you make mistakes in a solution, reflection is even more important. What mistakes did you make and why? How do you avoid these same mistakes in the future (especially in an exam)? What systematic procedure should you follow to avoid these same mistakes? How do you check your own work? Develop your own procedures ("bag of tricks") as you solve problems.

3. Sinusoidal analysis and design

3.1 Overall learning notes

1. Use the slides file sine.pdf as a guide to learn this topic, together with the more extensive materials from the textbook. Please review this file as you follow the notes below.
2. This topic seeks to teach you how to analyze (to understand how circuits work) and to design (to create circuits for specific tasks) circuits. You are expected to develop both analysis and design skills.
3. Some students are better with node analysis (using KCL), some better with mesh analysis (using KVL). Use your preferred methods as long as you write correct equations. However, after you solve a problem by one method, try it again with another method. You should know both methods and should be able to mix and match them in problem solving.

4. Practice the analysis of small circuits (the network KVL and KCL are straightforward to write) as much as possible to build skills. These practice exercises are from the textbook, with complete step-by-step solutions so you can check your own works.

3.2 Emphasis

1. Check how sinusoidal signals are used in your home stereo and computer systems: CD stereo circuits, computer-generated speech circuits, web-based audio playback, speakers, telephones, etc. Dig up as much information as you can to relate to signal amplitude, frequency, and phase, which are the 3 key parameters of a sinusoidal signal.
2. Can you figure out, based on specifications, why one set of speakers or one CD player costs more than another? Check the literature that comes with the speakers and CD player and compare them.
3. In the analysis process, the key parameters to evaluate, given a circuit and an input sinusoidal signal, are the amplitude and phase of the output of interest. The simple circuits (R, L, C, and opamps) do not change the frequency of the input signal, so the frequency of the output signal is known in advance. This is also the central tenet of the phasor concept (see Method 3 below): the phasor representation explicitly shows the amplitude and phase of a signal, leaving the frequency as an implicit parameter.
4. Practice these 3 methods to solve circuit problems after you have applied KCL / KVL to write the differential equation (DE) with one unknown (usually the output of the circuit). For each method, use one example circuit and make sure you know a step-by-step procedure to solve the circuit based on that method.

    - Method 1: look up Math handbook or use Math software (e.g. Maple or Mathematica) to find the solution. Advantage: easy, general technique for all circuit analysis. Disadvantages: cannot be performed in a job interview or design review with customers, Math Handbook might not have the answer for this specific DE.

    - Method 2: use techniques learned in Math courses to solve this DE by assuming a known solution form and evaluating the unknown amplitude and phase. They are equivalent and can be transformed into each other. Advantage: plug-and-chug technique, general technique for sinusoidal circuit analysis. Disadvantages: still need to work with DE, technique gets messy with medium-sized or large circuits (set of simultaneous trigonometric equations to be solved).

    - Method 3: use complex numbers and phasors to solve problems. You should observe that the analysis can be performed without any DE at all. If you are sick of DE, this is the method for you! Advantages: no DE, techniques suitable for medium-sized or large circuits (set of simultaneous algebraic equations can be solved using matrix techniques). Disadvantage: technique works only for sinusoidal signals.

5. Item 4 is the core of this topic: systematic procedures to analyze circuits with sinusoidal signals, and the emphasis is on Method 3. Practice problem-solving skills as much as possible, keeping in mind that math is used only as a tool, not the central idea of this topic.
6. As you develop analysis skills, try to work with larger circuits with these techniques:

- Combine parallel or series elements (prerequisite topics) to simplify the circuit before the analysis begins.
- Use Norton equivalent or Thevenin equivalent (prerequisite topics) to simplify the circuit before the analysis begins.
- Use superposition to analyze circuits with 2 or more input signals (apply one signal at a time). This topic is also from prerequisite courses.

7. Your instructor will teach you how to design a circuit by demonstrating various examples. Make sure you understand and can follow the step-by-step simplest design process (slide “Analysis vs. Design,” row labeled “Design (easy)”) where the input signal, the circuit diagram, and the required output signal are given, and the only components to be designed are the passive components. “Design” in this case means “select component values” to meet the given specifications. Your instructor will explain how to select component values and justify his / her selection.
8. Observe these critical points in circuit design and compare them with circuit analysis:
   - Design process is just as systematic as analysis process.
   - Design usually has many possible solutions (many component values make a design work), while analysis has only one possible answer. The components must be selected from existing catalogs. Component catalogs are available from many sources: your school’s electronic store, DigiKey, the local RadioShack stores, and many web-based electronic stores.
   - Design might not have any solution at all if the specifications are not realistic. In this case, there are no components in catalogs that can be used to build the circuit. Your instructor will demonstrate one example of this type of design requirements.

3.3 Once Over Lightly

1. The graphical representation of a phasor is not too critical. Some students like it; some do not. The graphical representation generally does not add much understanding. The step-by-step analysis process is much more important.
2. Skip all materials not related to sinusoidal signals. Some textbooks might contain these materials in this topic, probably to confuse the students.
3. The computation of the RMS value of signals will be covered in later topics. No need to worry about it now.
4. Some textbooks use phrases like “phasor transform” or “inverse phasor transform.” Ignore these terms. They are not conventional. The key is to use phasors as a problem-solving method.
5. The proofs that KVL and KCL work in phasors are strictly mathematical and detract from the focus on circuit analysis and design. You do not need to prove them.
6. Skip all transformer materials.

4. Average power

4.1 Overall learning notes

1. Use the slides file power.pdf as a guide to learn this topic, together with the more extensive materials from the textbook. Please review this file as you follow the notes below.
2. This topic seeks to teach you methods to analyze and design circuits to deliver power from a source signal to the load.
3. Practice the process of working with power as much as possible to build skills. These practice exercises are from the textbook, with complete step-by-step solutions so you can check your own works.

4.2 Emphasis

1. Check your monthly power bill at home or at your apartment: how much do you pay per month? How do you think the company measures the amount of power you use to charge you for it? What does the power meter actually measure? Try to find out more information.
2. Check your stereo system information for power rating of your speakers, and power delivery capability of your stereo receiver. Do they match? What would happen if they do not?
3. The textbook and the lecture materials describe several alternative methods to compute average power in case of sinusoidal signals (plug in formulas, compute from instantaneous power definition, use RMS formula, etc.). Practice these methods and show that for the same problem, they provide the same answer.
4. Observe that the RMS formula is more general and can apply for any signal type, not just sinusoidal.
5. Practice designing circuits to maximize power delivery to the load, using exercises or problems in the textbook. Re-examine your stereo system specifications to see if the receiver really delivers the maximum possible power to your speakers. If not, try to explain the mismatch.

4.3 Once Over Lightly

1. Skip all materials related to reactive power, 3-phase power (except as motivating examples related to your power bill), complex power, etc.

5. Laplace transform technique

5.1 Overall learning notes

1. Use the slides file laplace.pdf as a guide to learn this topic, together with the more extensive materials from the textbook. Please review this file as you follow the notes below.
2. This topic seeks to teach you methods to analyze and understand circuits when the input signal has more complicated form (e.g. an FM signal from a radio station, a speech signal, etc.).
3. Many textbooks go from sinusoidal analysis directly into transfer functions and filters before Laplace transform. Your instructor may choose his/her order to teach these topics.

4. The greatest challenge in learning this topic is to relate it to electrical engineering. Textbook materials are almost strictly mathematics. The key is to use the Laplace transform and inverse transform for realistic engineering signals or functions, especially signals that appear frequently in circuit analysis and design. Keep your focus on signals and circuits if you feel you are getting lost in the math.
5. For every theorem in the text, try one circuit example so you can see how the mathematics relates to circuit analysis.
6. Your background in calculus and differential equations is sufficient to understand the proofs for the theorems in the text. Try to follow the proofs since they will help in the future when you need to deal with very complicated signals.
7. After you are familiar with using the integral definition to find the transform, you may use the tables and apply transform theorems. You need to learn how to look things up and apply theorems properly.
8. Practice the process of finding transform and inverse transform of signals as much as possible to build skills. These practice exercises are from the textbook, with complete step-by-step solutions so you can check your own works.

5.2 Emphasis

1. How do you analyze a circuit where the input signal is not a DC signal, a step signal, or a sinusoidal signal? Analyze a small circuit in the textbook and assume the input signal is a product of two cosine signals. You will have to use the differential equation technique again! The Laplace method offers an alternative for circuit analysis, especially when the signals are complicated, without differential equations.
2. Go over the slides and make sure you understand the step-by-step process of employing the Laplace transform in circuit analysis before jumping into all the mathematical equations.
3. Check the textbook to see which lower limit it uses to define the transform: 0 or 0-. If you do not understand the difference between these two possible values for the lower limit, ask your instructor.
4. The Laplace transform used in this course is called "one-sided" transform since its lower limit is 0 or 0-. There is also the "two-sided" Laplace transform used in signal analysis, with the lower limit as $-\infty$. In circuit analysis and design, we are more interested in the signals from the time the circuit is turned on (t=0), so this is one reason why only the one-sided transform is used. Can you think of other reasons?
5. Try to relate the impulse function to something more physical so you can understand it better. For example, the short intense pain caused by hitting your knee or elbow against a sharp corner is similar to an impulse. What other examples can you think of?
6. The best way to review partial fraction expansion in inverse transform is to do exercises. For the repeated root case, try working with roots repeated twice or three times for practice.
7. Transform simple circuit components without and with initial conditions so that you can use circuits to demonstrate how the Laplace transform works in solving problems. Observe the advantage of automatic inclusion of initial circuit conditions in the Laplace transform

technique. You previously had to solve for the output signal first then apply the initial conditions. The Laplace transform technique does it in one single process.

8. You can use the Laplace transform in conjunction with the techniques to deal with large circuits (combination of elements, Thevenin's and Norton's equivalents, and superposition). Now you know how to analyze complex circuits with any type of input signals!!
9. Use the Laplace transform in circuit design. Pick one or two exercises from the text (with answers so you can check your own work) and apply the design procedure taught in section 3, but this time use the Laplace transform to derive the equations relating the input to the output. Select the component values as before. Does your design work?

5.3 Once Over Lightly

1. Skip all existence and uniqueness proofs of the transform or inverse transform. Real-life signals in circuits do have transforms and they are unique.
2. Do not try to use the integral definition for the inverse transform. For circuits, the partial fraction expansion process is sufficient to find the inverse transform.
3. Skip all materials related to mutual inductance.
4. KCL and KVL work in the s-domain just like in the time domain. No need to worry about the proofs.
5. Skip all materials related to mutual inductance.
6. Many textbooks show circuits with impulsive signals, which do not exist in real life. Impulsive signals are used mostly as a way to derive circuit models.

6. Transfer functions: circuit analysis with Laplace transform

6.1 Overall learning notes

1. Use the slides file transfer.pdf as a guide to learn this topic, together with the more extensive materials from the textbook. Please review this file as you follow the notes below.
2. This topic seeks to teach you methods to analyze circuits using the Laplace transform and compare these methods with time-domain methods. These methods are complementary and you should practice all of them.
3. Design with transfer function will be covered as part of the frequency response topic.
4. It might be somewhat difficult to totally decouple this topic with the topic on frequency response (see section 7). Your instructor might mix materials between these two topics.
5. Practice the analysis of circuits using transform technique as much as possible to build skills. These practice exercises are from the textbook, with complete step-by-step solutions so you can check your own works.

6.2 Emphasis

1. Take a reasonably complex circuit in the text, and analyze the same circuit with 3 or 4 different input signals. You will discover how tedious it is to do so!! The transfer function method offers one way to reduce this tediousness: you only need to analyze the circuit just once to derive a "characteristic behavior" for the circuit. Then, whenever you apply an input

signal, you re-use this "characteristic behavior" to evaluate the output signal very quickly. If you have a different input signal, you can re-evaluate the output using this "characteristic behavior." No need to re-analyze the whole circuit again. This "characteristic behavior" is formally called the "transfer function" of the circuit: it transfers the input to the output.

2. Take any circuit and find the transfer function H(s), using the procedure taught by your instructor or in the textbook.
3. Now apply several different input signals (as you did in item 1 above) and compute the corresponding output signal without re-analysis. This is much faster and less error-prone.
4. You will need to know the convolution integral as part of the analysis of a circuit. Do the direct convolution integral only once or twice at most. For circuits, the use of Vo(s) = H(s)Vi(s) is more common to solve for the output signal Vo(t). Another way to avoid messy mathematics!!
5. Key note: H(s) does not include initial conditions. If a circuit has to be solved with initial conditions, use the standard Laplace method instead of plugging in values in H(s). Remember that the standard Laplace method automatically includes initial conditions in the solution.
6. Practice the technique by using many short examples, making sure you understand clearly time-domain waveforms at both the input and the output of the circuits. Explain the physical operations of the circuits. You need to develop a "feel" how a circuit works instead of blindly relying on math equations.
7. Practice circuit design using the transfer function with examples in the textbook. Derive H(s) from the circuit and use it and the specifications to select component values. Does your design work?

6.3 Once Over Lightly

1. It is up to the instructor to decide how much time to devote to the graphic process of convolution. No more than one lecture session (1 hour) is recommended.
2. Skip all advanced explanations of convolution based on weighting functions.
3. Skip all materials on filters for the time being. See more below in the topic on simple filters.
4. You do not need to prove KCL and KVL in the s-domain. These laws still work in s-domain, similar to the time domain.

7. Frequency response

7.1 Overall learning notes

1. Use the slides file freqresp.pdf as a guide to learn this topic, together with the more extensive materials from the textbook. Please review this file as you follow the notes below.
2. This topic seeks to teach you the relationships between the time-domain output signals and the frequency-domain plots of H(s), and how to use these relationships in circuit analysis and design.

3. Practice the Bode amplitude plot technique as much as possible to build skills. These practice exercises are from the textbook, with complete step-by-step solutions so you can check your own works.

7.2 Emphasis

1. Observe the advantage of the Laplace transform techniques and H(s) in analyzing circuits with sinusoidal signals. With one circuit as example, analyze it with the techniques in section 3 and again with H(s). Discuss similarities and differences. Relate the gain and phase shift in sinusoidal analysis to the amplitude and phase of H(s). Which method is faster? Which method do you prefer?
2. It is very important to understand, for the sinusoidal case, the differences between transient solutions and steady-state solutions. The Laplace method provides both, while the sinusoidal method in section 3 only provides steady-state solutions. If we just turn on a circuit, the transient output is the output after this turn-on and lasts only a short time. After a while, the circuit settles into a steady operation, and the output is the steady-state output.
3. Focus on the steady-state case for the rest of this topic, since the transient output only lasts a short time.
4. Practice the Bode approximate plotting technique using simple examples of H(s). Use textbook examples. You should be good at plotting and at reading the plot data (i.e. read the gain of the circuit at a specific frequency from the plot).
5. Check your calculations of the frequency response of a circuit with PSPICE AC analysis. They should agree with each other very well.
6. Be careful in the definitions of “3-dB point,” “corner frequency,” and “bandwidth.” They mean the same thing in simple H(s) but if there are repeated roots at one frequency, different textbooks use these terms differently. Check the text you use and check to see how your instructor defines these terms.
7. Practice the resonant circuits well and relate the time-domain output signals to the frequency-domain plots of H(s). These circuits are important. If you know the frequency response (where the poles and zeroes are), can you predict the time-domain signals of a resonant circuit? If you see a time-domain signal output of a resonant circuit, can you predict its poles and zeroes? Practice going back and forth between these two domains.
8. Apply these concepts in interpreting the specifications of a realistic opamp and its frequency response. At high frequencies, does the opamp still have high gains? At what frequency does the opamp have gain exactly equal to 1?
9. Practice circuit design given gain, bandwidth, etc. Pick one or two exercises from the textbook and verify that your design is correct. Run PSPICE too to check.

7.3 Once Over Lightly

1. Approximate phase plotting using the Bode technique is complicated for second-order or higher-order circuits. You might want to use computer-aided plotting tools to plot the phase function directly, unless your instructor requires you to perform hand plotting.

2. Approximate amplitude plotting using the Bode technique in case of overshoots at the corner frequency is rather messy. You might want to use computer-aided plotting tools to plot the amplitude function directly, unless your instructor requires you to perform hand plotting.

8. Simple filters

8.1 Overall learning notes

1. Use the slides file filters.pdf as a guide to learn this topic, together with the more extensive materials from the textbook. Please review this file as you follow the notes below.

2. This course covers only a brief introduction to simple filters, mostly passive filters of low-orders (no higher than 3) and active filters with one opamp. The idea is to demonstrate the applications of the transfer function and frequency response methods in analysis and design.

3. Practice with many problems to integrate the analysis and design methods taught in the materials, PSPICE simulation, and laboratory experiments. You need to tie all materials together and demonstrate your skills in analysis and design.

8.2 Emphasis

1. Practice the analysis of all 4 types of filters, using passive RLC circuits. The textbook has all these examples. Observe the differences between ideal filter characteristics and real-life filter characteristics.

2. Given a simple passive filter circuit (order 3 or lower), try to reason (using component impedance and admittance as function of frequency) which filter it is without doing any mathematical analysis. After you make this first classification based on reasoning, do the math to verify your result. You need to develop the skills to do "back-of-envelope" calculations to estimate circuit behavior without a lot of math. Practice this method as many times as possible.

3. Practice to analyze active filter circuits with only one opamp to simplify the process and demonstrate the methods (rather than the complex math). The Sallen-Key filter is a good example for all 4 filter types in one topology. All textbooks have this filter.

4. Practice the design of a simple filter given a filter circuit and specifications. Check your own design by running PSPICE. Is the gain correct? Is the bandwidth correct?

8.3 Once Over Lightly

1. Skip all advanced filter topics. There will be higher-level courses focusing on filter design.

2. Avoid as much math as possible. Focus on methods and link to physical behavior.

9. SPICE

9.1 Overall learning notes

1. Use PSPICE Student Version on the PC as the main tool since it is free and most likely comes in a CDROM with the textbook. The CDROM contains the software and the PSPICE User's Guide. It also has links to the latest web site with updated documentation and free download instruction. Other versions of SPICE (Berkeley SPICE, HSPICE, etc.) are similar.

Check with your instructor to see which version of SPICE is used for the course. SPICE is a standard industry simulation tool and you must learn it well. Do not use non-SPICE tools.

2. Your instructor will demonstrate an example on his computer how to construct the circuit diagram, how to specify components and their values, how to specify the input signal, and how to run a simulation, how to plot the input and output signals, and how to print a hardcopy of the output plot. Follow this example closely since you will need to repeat in as part of the homework and laboratory experiments.
3. The PSPICE User's Guide has a step-by-step example on how to construct a circuit, how to run a simulation, how to get output results. Try this example too to make sure you know how to use PSPICE.
4. Even if the instructor does not assign PSPICE as part of a homework, try it anyway to see if the PSPICE result matches yours. This gives you another check on your answer and provides more practice with PSPICE.
4. This is the first course teaching PSPICE as a computer-aided tool for circuit analysis and design. Go slow at first but you must know how to run PSPICE well by the end of the course. Practice simulations as much as possible.

9.2 Emphasis

1. Use PSPICE to simulate circuits with components (R, L, C) and waveforms (DC, step, sine) covered in the lectures.
2. Perform DC analysis and transient analysis, and relate them to the time-domain solutions that you do by hand. Always compare your manual solutions with PSPICE solutions, and try to explain any differences.
3. Later in the course, perform AC analysis to go along with transfer function and frequency response topics. Always compare your manual solutions with PSPICE solutions, and try to explain any differences.
4. Have you found any instance where PSPICE actually fails, i.e. its output is wrong or it does not provide an output (analysis aborted)? PSPICE, like all computer-aided tools, sometimes fails. If you find an example where PSPICE fails, try to figure out why. It is important to be skeptical of all computer-aided tools.
5. After you design a circuit, use PSPICE to simulate it and verify that the circuit works as designed. This is one of the major applications of PSPICE: to verify a design.
6. Setting the Ground node or Initial Condition on a node takes some work in opening specific component libraries in PSPICE. The User's Guide covers the procedure to open these libraries. Check the Index and learn how to do these settings for their circuits. You have learned the Ground node setting in the example demonstrated by the instructor (otherwise the circuit would not have been simulated!!) and should learn now how to specify Initial condition on a node.
7. Other component libraries contain opamp models. Learn how to open these libraries and use the components. All of these processes (items 6 and 7) can be done in one or two examples.

8. The PSPICE User's Guide is an extensive document (hundreds of pages). There is no need to read it all. Use the examples provided in the Guide and the Index to look up specific topics and procedures. The Acrobat version of the User's Guide is excellent for this purpose.

9.3 Once Over Lightly

1. PSPICE has many other components (bipolar transistors, MOS, mutual inductance, etc.), many other waveforms (pulse, piecewise-linear, etc.) and many other analysis types (Monte Carlo, sensitivity, transfer function, etc.). Do not worry about these topics now. These topics will be covered in subsequent courses.
2. Do not worry about other advanced topics in PSPICE.

10. Laboratory

10.1 Overall learning notes

1. The laboratory materials for the course consist of the motivating experiments (taught in an interactive manner), the laboratory instrumentation manual (how to use the instruments in the lab to apply and measure waveforms), and the verifying experiments (building circuits and measuring their characteristics).
2. The course has several motivating experiments, and the number of motivating experiments to be used depends on the instructor's selection. One motivating experiment (e.g. the DSL modem) might cover all the topics in the entire course, while another motivating experiment (e.g. musical instrument sound characteristics) might cover only one or two topics.
3. The laboratory instrumentation manual is provided for the specific instruments available at the University of Washington, and for the UW-Pandora box. If your course uses different instruments, your instructor will provide the manual to you.
4. The laboratory instrumentation manual and the verifying experiments are posted at this public web site in Acrobat format: http://faculty.washington.edu/manisoma/labs. There are also many laboratory texts suitable for this course. Check with your instructor to see which manual and experiments are used for your course.
5. These experiments are written for general instrument (e.g. "oscilloscope" is used in the description, but not a specific model or manufacturer) so they can be used in local laboratories with any set of instruments.
6. The course must cover at least 4 experiments over 10 weeks. The more the better.
7. Since students work in teams in the laboratory, there is a need to verify that each student has experimental skills. You will be required to take an individual hands-on laboratory exam at the end of the course to demonstrate these skills to the instructor.

10.2 Emphasis

1. Always try to relate the motivating and verifying experiments with the concepts, circuits, analysis procedures, and design practices covered in the lectures. This process reinforces your learning and makes the concepts and the mathematical manipulations more realistic and physical.

2. Later verifying experiments incorporate PSPICE simulations. You need to compare and understand the similarities and differences between your manual calculations, PSPICE results, and experimental data.
3. You must get used to real-world variations in component values and measured values of signals. You have used calculators with a large number of significant digits (e.g. 3.1415926535...) and must realize that in real life, there is no such accuracy. For example, if you buy a $1.00 coffee and pay 8.8% sales tax, you pay a total of $1.09, not $1.088. Two significant digits are usually sufficient in experimental data collection. The number of digits in your calculator and on paper has no meaning unless they relate to real-life measurements and component values.
4. A design must work in the lab (real life). A design that works only on paper or in PSPICE is useless.
5. What if your circuit fails and after consulting with your friends in the same team, you still do not know what happens and how to fix it? Circuit diagnosis and debugging is a difficult task. You should contact your instructor for help. We are developing a more systematic method to help you debug circuits remotely, but it is not available for use yet at this time.

10.3 Once Over Lightly

1. Everything in the laboratory experiments is important since the hands-on work is the key difference between engineering and the ideal world of textbooks, computers, calculators, and paper analysis.