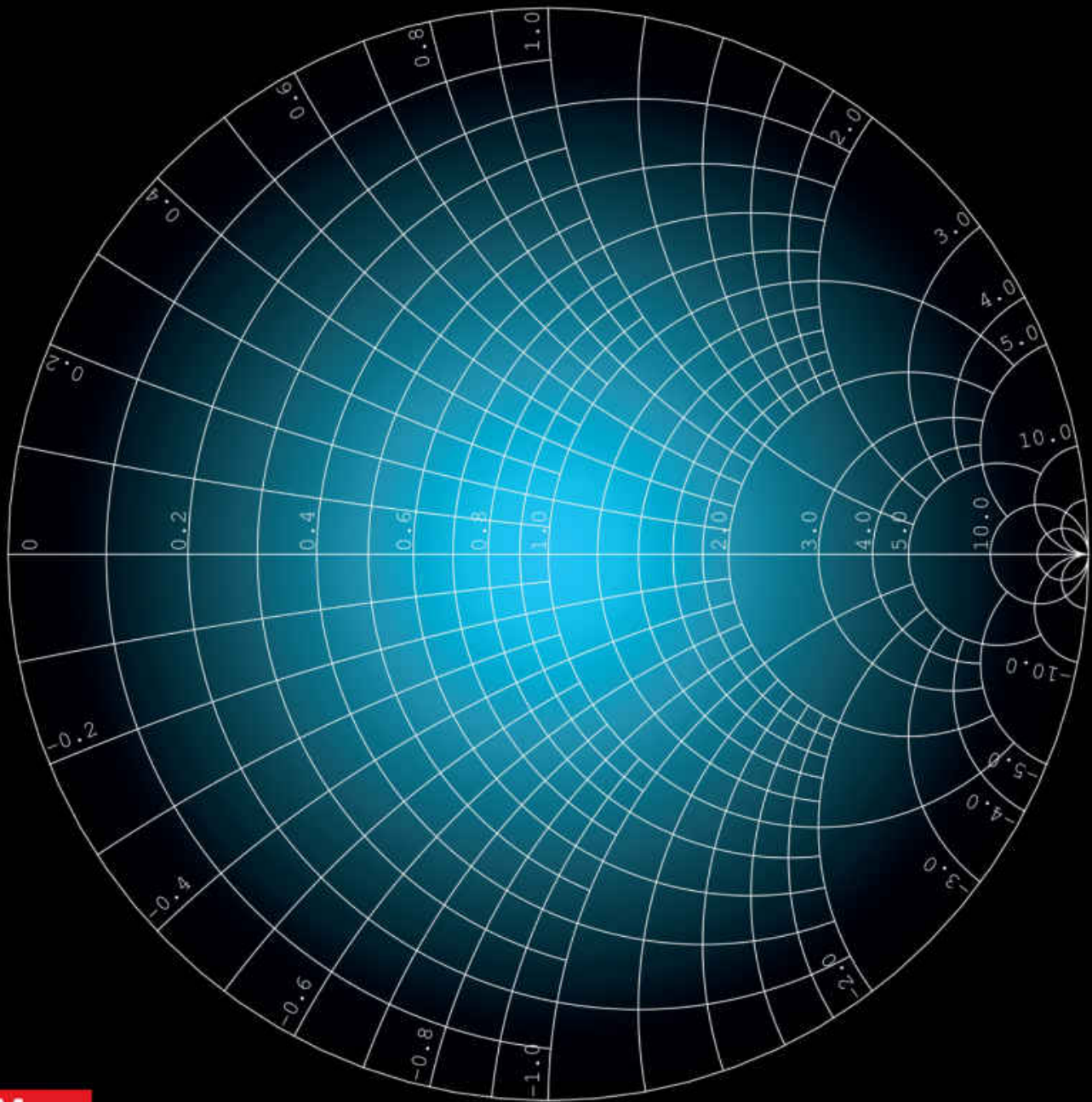


EXPERIMENTS AND ACTIVITIES MANUAL FOR

PRINCIPLES OF

FIFTH EDITION

ELECTRONIC COMMUNICATION SYSTEMS



**Mc
Graw
Hill**

LOUIS E. FRENZEL JR.

**Experiments and Activities
Manual *for***

**Principles of
Electronic
Communication
Systems**

Fifth Edition

Louis E. Frenzel Jr.

**Mc
Graw
Hill**



EXPERIMENTS AND ACTIVITIES MANUAL FOR PRINCIPLES OF
ELECTRONIC COMMUNICATION SYSTEMS, FIFTH EDITION

Published by McGraw Hill LLC, 1325 Avenue of the Americas, New York, NY 10019. Copyright ©2023 by McGraw Hill LLC. All rights reserved. Printed in the United States of America. Previous editions ©2016, 2008, and 2001. No part of this publication may be reproduced or distributed in any form or by any means, or stored in a database or retrieval system, without the prior written consent of McGraw Hill LLC, including, but not limited to, in any network or other electronic storage or transmission, or broadcast for distance learning.

Some ancillaries, including electronic and print components, may not be available to customers outside the United States.

This book is printed on acid-free paper.

1 2 3 4 5 6 7 8 9 LKD 27 26 25 24 23 22

ISBN 978-1-260-78937-9

MHID 1-260-78937-3

Cover image: *Audrius Marfeldas/Shutterstock*

All credits appearing on page or at the end of the book are considered to be an extension of the copyright page.

The Internet addresses listed in the text were accurate at the time of publication. The inclusion of a website does not indicate an endorsement by the authors or McGraw Hill LLC, and McGraw Hill LLC does not guarantee the accuracy of the information presented at these sites.

mheducation.com/highered



Contents

Introduction v

Organization of this Manual v

Activities v

Lab Experiments v

How to Use this Manual v

Lab Equipment v

Breadboards and Trainers v

Virtual Instrumentation Trainers vi

Meters vii

Oscilloscopes vii

Signal Generators vii

Frequency Counters vii

RF Equipment vii

Other Equipment vii

Components viii

Kits viii

Commercial Lab Equipment viii

Introduction to Electronic Communication 1

1-1 Exploring the Regulatory Agencies 1

1-2 Becoming Familiar with the FCC Rules and Regulations 2

Electronic Fundamentals for Communications 3

- 2-1 *LC* Low-Pass Filter Design and Testing 3
- 2-2 *RC* Active Filter Design and Testing 5
- 2-3 Ceramic Band-Pass Filter 6
- 2-4 Switched Capacitor Filters 8

Amplitude Modulation Fundamentals 11

- 3-1 Measuring the Percent of Modulation 11
- 3-2 Spectrum Analyzer 17
- 3-3 AM Radio Experience 19

Amplitude Modulation and Demodulation Circuits 23

- 4-1 Diode Modulator and Mixer 23
- 4-2 Differential Amplifier Modulator 27
- 4-3 Diode Detector 33
- 4-4 Double-Sideband Balanced Modulator 37

Fundamentals of Frequency Modulation 41

- 5-1 Frequency Modulation 41
- 5-2 FM Radio Experience 44

FM Circuits 45

- 6-1 Phase-Locked Loop Operation 45
- 6-2 Frequency Demodulation with a Phase-Locked Loop 49
- 6-3 Pulse-Averaging Discriminator 52

Chapter 7 has no experiments and therefore is not included in this Experiments and Activities Manual 55

Radio Transmitters 57

- 8-1 Class C Amplifiers and Frequency Multipliers 57
- 8-2 Crystal Oscillator 61
- 8-3 Building and Testing a Radio Transmitter 63
- 8-4 Measuring Transmitter Output Power 65
- 8-5 Impedance Matching 68

Communication Receivers 73

- 9-1 Noise and Its Measurement 73
- 9-2 PLL Frequency Synthesizer 77
- 9-3 Mixer Operation 83
- 9-4 Building and Testing a Communications Receiver 86

page iv

Digital Data Transmission 87

- 10-1 Serial Digital Data 87
- 10-2 Frequency Shift Keying 93
- 10-3 Binary Phase-Shift Keying (BPSK) 95

Multiplexing, Duplexing, and Multiple Access 101

- 11-1 Pulse Amplitude Modulation and Time Division Multiplexing
101

Chapter 12 has no experiments and therefore is not included in this Experiments and Activities Manual 107

Transmission Lines 109

- 13-1 Measuring SWR 109
- 13-2 Measuring Coax Cable Loss 111
- 13-3 Measuring Complex Load Impedance on a Transmission Line
112
- 13-4 Transmission Lines and Time Domain Reflectometry (TDR)
113
- 13-5 Smith Chart Tutorial 120

Fundamentals of Networking, Local Area Networks, and Ethernet 121

- 14-1 Exploring Ethernet 121

Chapter 15 has no experiments and therefore is not included in this Experiments and Activities Manual 123

Antennas and Wave Propagation 125

- 16-1 Dipole Radio Antenna 125
- 16-2 Plotting Directional Antenna Patterns 127

Internet Technologies 135

- 17-1 Review of Internet Protocol 135
- 17-2 Review of OTN 136

Microwave and Millimeter-Wave Communication 137

- 18-1 Exploring Millimeter Waves 137

Satellite Communication 139

- 19-1 Exploring GPS 139
- 19-2 GPS Experience 140

Optical Communication 141

- 20-1 Optical Fiber Data Transmission 141
- 20-2 Infrared Remote Control 145
- 20-3 Fiber Optic Tutorial 147

Cell Phone Technologies 149

- 21-1 Cell Phone Technology in Depth 149
- 21-2 Long Term Evolution and OFDM 150
- 21-3 Small Cell Networks 151

Wireless Technologies 153

- 22-1 Comparing Short-Range Wireless Technologies 153
- 22-2 Wireless Monitoring 154

Communication Tests and Measurements 157

- 23-1 RF Test Equipment Familiarization 157

Appendix A Breadboarding 159

- Rules and Tips for Breadboarding 161



501-1111

Introduction

This manual is to accompany the textbook *Principles of Electronic Communications Systems* (PECS), fifth edition. That text was written to support a one- or two-semester course in the fundamentals of communications systems including wireless and networking, both analog and digital. The text is written at the technology level and provides coverage for both 2-year community college and technical school programs through 4-year bachelor of technology programs. This manual serves as a supplement to the text and provides practical student activities as well as laboratory experiments, given the appropriate equipment.

There are always limits as to the extent of coverage of any textbook. Given practical page limits in addition to the time constraints of any college course, a text can cover only so much. The scope and breadth of coverage has been balanced with depth of coverage. However, this manual serves the instructor and the student in providing not only validation of principles in the text but also the opportunity to dig deeper on a variety of subjects with both Internet access and hands-on lab experiments.

Organization of this Manual

This manual is divided into two types of projects, activities and lab experiments. The projects are targeted at specific chapters and are numbered as such. For example, Project 3-2 is the second project related to Chapter 3.

Activities

The activities are primarily Internet searches or Web page accesses to acquire additional information on specific topics. Some activities

provide questions to answer from the material found. With Web access and search being the dominant method of information gathering on the job in the real-world today, the activities provide students the chance to practice the process they will use considerably in their future employment. Most projects have specific questions to answer by referencing the accessed materials.

The projects typically give relevant websites to avoid excessive search time. Use these websites first to answer the questions. Then use a search via Google, Yahoo, or Bing, using specific search terms to find other answers not in the referenced material.

Lab Experiments

Lab experiments are projects using actual electronic and/or computer hardware to demonstrate and validate key theoretical concepts. The goal is to provide a minimum of true hands-on practice with real test equipment, components, and related hardware.

How to Use this Manual

The projects assume that you have read the related chapter and/or have had one or more lectures on the subject. Instructors will assign relevant projects best fitting their goals for the course. Otherwise, students should feel free to select projects of interest and pursue them.

Lab Equipment

Most colleges have adequate lab equipment for teaching basic electronics as well as digital circuits and microcomputers. However, few have suitable lab equipment for teaching communications hardware. It is typically more expensive than basic instruments and trainers and can certainly be more difficult to use. Working with RF circuits and equipment is particularly challenging.

For these reasons, the lab experiments in the manual have been simplified as much as possible without compromising the teaching of the fundamentals. Furthermore, low cost was a key consideration in choosing the lab experiments. One way to do this is to use circuits that operate at much lower frequencies. Instead of VHF, UHF, and microwave signals that are the most common today, this manual uses

low RF and in many cases audio and midrange frequencies below 1 MHz. This makes the circuits less expensive, less critical, and more adaptable to traditional breadboards. An effort has been made to use the older DIP ICs rather than the more common surface-mount devices so common in wireless and communications gear today. Finally, by using the lower frequencies, commonly available test equipment is typically acceptable.

Breadboards and Trainers

Standard breadboards and trainers may be used for most of the experiments described in the manual. The key to making the circuits work is to keep the component leads and interconnecting wires short and the components grouped closer together to minimize the stray inductance, capacitance, and coupling. All power supplies should be bypassed with 0.1- μ F ceramic capacitors where they attach to the breadboard to minimize noise, coupling, and other problems.

A good choice for a general-purpose trainer is the National Instruments (NI) ELVIS. See Figure 1. It contains the usual breadboarding sockets as well as multiple power supplies and a function generator. And it is designed to connect to a PC or laptop running NI LabVIEW software that converts the computer into a group of virtual instruments (VIs), which include a multimeter and oscilloscope. This combination is hard to beat since the complete lab station, breadboard, trainer, power supplies, generators, and instruments are contained in two units, the ELVIS trainer itself and the PC. All of the experiments have been tested on the ELVIS trainer. I highly recommend it, as the availability of the LabVIEW software provides useful virtual instruments and allows the demonstration of DSP and other concepts that are impossible with common hardware.

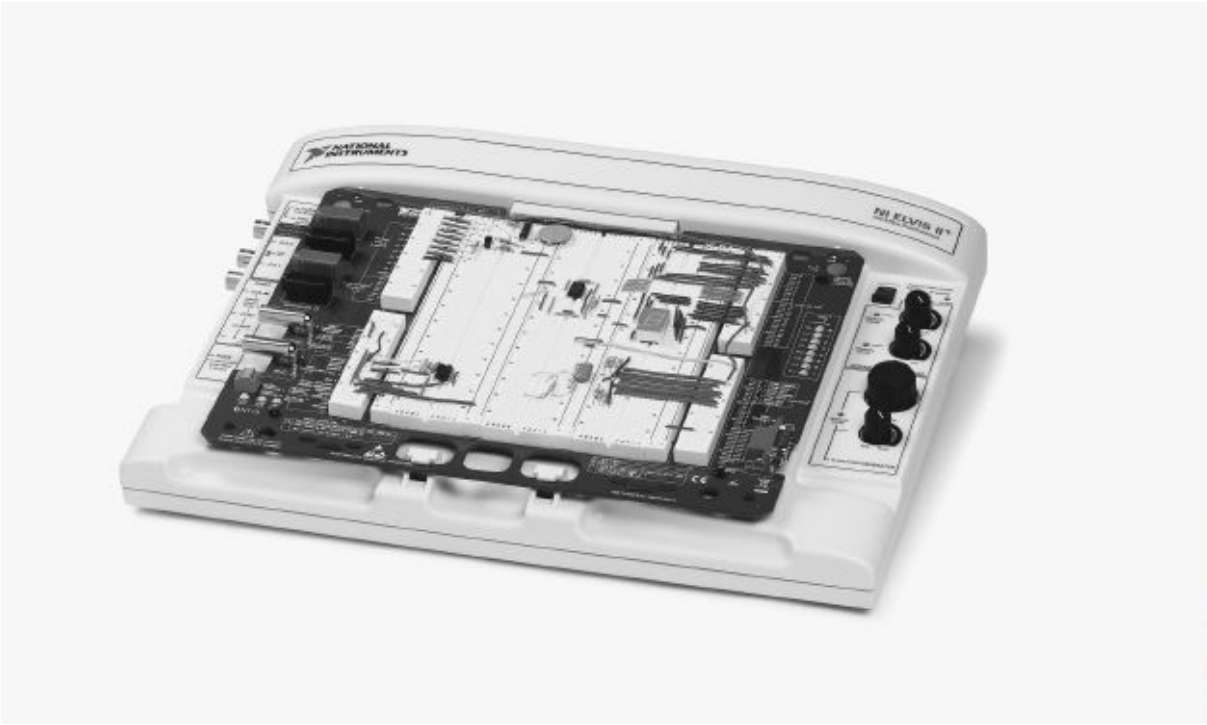
Virtual Instrumentation Trainers

Some colleges are now using virtual test instruments instead of the traditional bench test instruments. A virtual instrument is one that is implemented on a computer. Special circuitry and software is used to create a DMM, oscilloscope, function generator, spectrum analyzer, and even power supplies by using a desktop or laptop computer. The computer screen serves as the readout for the meter, scope, and any other instruments available. On-screen knobs, switches and other controls are manipulated with the mouse.

The ELVIS trainer uses virtual instruments. Two other less expensive virtual instrument products are the National Instruments myDAQ and the Digilent Discovery. See Figure 2. The myDAQ connects to the computer via a standard USB port and derives power to supply DC voltages of 5 V and ± 15 V to experimental circuits. In addition, the software referred to as LabVIEW implements a DMM, scope, function generator, and a few other instruments. A connector on the side of the myDAQ provides connections to the power supplies, the instruments, and an external breadboarding socket.

Another virtual instrument is the Digilent Discovery. It too has built-in power supplies, meter, scope, and function generator. The device also uses a USB connection to the computer. Software called *Waveforms* is used on the computer to implement the instruments. Wire leads connect to the breadboarding socket where the experimental circuit is built.

A positive feature of these products is that they are very affordable and typically cost less than a textbook. Many students can afford them. If a student owns a laptop, he or she may want to invest in one of these devices as then they will be able to run many of the experiments in this manual at home, in the dorm, or anywhere when it is convenient. These trainers work well in the lab, and they are affordable if the lab already has computers.

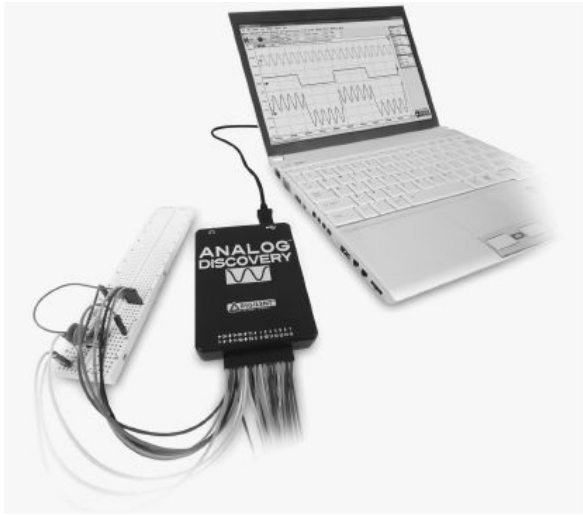


©National Instruments

Fig. 1 The ELVIS trainer. Courtesy National Instruments.



©National Instruments



©Digilent, Inc.

Fig. 2 The National Instruments myDAQ (left) and Digilent Analog Discovery (right) trainers use virtual instruments on a laptop to provide a low-cost base for most lab experiments.

Meters

Any standard digital multimeter is adequate for making tests and measurements in these experiments. It should be capable of measuring dc and ac voltage and current and resistance. A good analog meter is also suitable.

Oscilloscopes

Almost any oscilloscope used in college labs is suitable for use with the experiments in this manual. A dual trace/channel scope is recommended. The bandwidth should be as broad as affordable. I used both 25-MHz and 100-MHz scopes in creating these experiments. However, the virtual oscilloscope in the NI ELVIS trainer, despite its upper bandwidth limit restricted by the sampling rate of the data acquisition card used, is certainly adequate for some low-frequency experiments. Good scope probes are essential for proper results.

Signal Generators

Most of the experiments require one or more signal sources. Since most schools have general purpose function generators that can produce sine and square waves up to about 2 MHz, these are specified in most cases. Some experiments have built-in signal sources to minimize the need for excessive equipment. The ELVIS trainer contains a function generator capable of generating sine, square, and triangular waves up to 250 kHz.

An RF signal generator is highly recommended for some experiments. In general, any good RF signal generator will work, but most of them are very expensive. I strongly recommend the MFJ Enterprises MFJ-269. This instrument is designed for antenna and transmission line tests and measurements. However, it contains a versatile signal generator that covers the range from 1.8 MHz to 170 MHz and 415–470 MHz. It also has a built-in frequency counter and circuits for measuring SWR. It can measure, calculate, and display complex impedances. Its low cost makes it a good investment for any communications course and lab.

Frequency Counters

Any standard frequency counter will work fine with the experiments described in this manual. The higher the frequency capability, the

better. A counter with an upper frequency limit of 200 MHz or more is recommended.

RF Equipment

Two common pieces of RF equipment are recommended, an RF power meter and an SWR meter. In some cases, these units are combined. A recommended low-cost unit is the MFJ-816. The older Bird 43 watt-meters also work very well and are still available. Newer digital watt/SWR meters work well but are not generally worth the expenditure for the few experiments given here.

Other Equipment

One of the most often used instruments in RF and wireless work is the spectrum analyzer. It is, in fact, used more than the oscilloscope in the real wireless world. However, few if any colleges own one. They are hugely expensive. They can be rented, and lower cost used units are available for sale. Even so, the price is high even for a unit with a limited bandwidth.

The experiments in this manual have been designed not to require a spectrum analyzer, but it is highly educational when it can be used to view harmonics, sidebands, and the effects of modulation and noise. I highly recommend that the college invest in at least one spectrum analyzer and use it in as many of the experiments as possible to give the students the experience that will serve them well later on the job.

Components

The components required are listed at the beginning of each experiment. Only commonly available, low-cost components are specified. They are available locally from distributors in the larger cities or from mail order sources like Digi-Key and Jameco Electronics. These latter two sources are highly recommended for their low prices and rapid delivery.

Some of the experiments are design projects that will require values that cannot be previously determined. As a result, it is good lab practice to buy and keep on hand complete kits of standard-value resistors and capacitors. One-quarter-watt resistors are satisfactory for most designs. Ceramic and polystyrene capacitors are

recommended. Both the mail order suppliers mentioned above stock standard kits that are very affordable and very handy in any lab.

Kits

A few of the projects require the purchase of kits, assemblies of parts designed to build a specific piece of equipment. Kits provide a low-cost way to implement the more complex experiments without problems. They also provide students with experience in building actual circuits on printed circuit boards, using real ICs and other components.

The kits recommended for these projects are very low in cost and many can be purchased for a given class. The students may even want to purchase them for themselves. In any case, building the kits can be part of the lab experience. Later, the assembled kits may be retained and used in future classes without the need to build them.

Commercial Lab Equipment

Another good choice for a communications laboratory is to use commercial training equipment. Listed below is a sampling of companies and some of their communications lab products worth considering.

- EMONA tims (www.emona-tims.com)
 - Stand-Alone Telecoms Trainer
 - Telecoms Trainer (uses NI ELVIS)
 - Fiber Optics Trainer (uses NI ELVIS)
 - Signal-Processing Trainer (uses NI ELVIS)
- Feedback Instruments (www.feedback-instruments.com)
 - Mobile Phone Trainer
 - Bluetooth Trainer
 - ZigBee Trainer
 - USB Trainer
 - RFID Trainer
 - Embedded Internet Trainer
 - CAN Bus Trainer
 - Analog and Digital Telecommunications Workstation
 - Amplifiers and Oscillators Workboard

- Tuned Circuits and Filters Workboard
- Modulation and Coding Workboard
- Microwave Trainer
- MIDE Microwave Design Software
- Antenna Lab
- Complete Microstrip Trainer
- Antenna System Demonstrator
- Transmission Line Demonstrator
- LAN-T Trainer
- Fiber Optics Trainer
- Optical Fiber Trainer
- Optical Fiber System
- Keysight Technologies (formerly Agilent Technologies, www.keysight.com)
 - TDR Probe Kit
 - Lightwave Laboratory Kit
 - RF Laboratory Kit
 - RF Transceiver and Courseware
 - Series 1000 Oscilloscope Training Kit
 - Analog Electronics Lab Station
 - Digital Signal Processing Lab Solution
 - Digital RF Communications Training Kit
- Lab-Volt Systems Inc. (www.labvolt.com)
 - Radar Training Systems
 - Satellite Communications Training Systems
 - Antenna Training and Measuring System
 - Telephony Training Systems
 - Microwave Technology Training System
 - Communications Technologies Training Systems
 - Digital Communications Training Systems
 - Analog Communications Training Systems
 - Microwave Technology Training System
- National Instruments (www.ni.com)
 - ELVIS Computer-Based Instrumentation
 - Universal Software Radio Peripheral (USRP) Platform
- Nida Corporation (www.nida.com)

- 360S Systems Trainer
- Training Console
- Basic Microwave
- Microwave Communications
- Microwave Standing Wave Ratio Measurement
- Microwave Reflection
- Radio Trainer



NAME

page 1

DATE



Introduction to Electronic Communication

Project 1-1 Exploring the Regulatory Agencies

Objective

Become familiar with those U.S. government agencies responsible for managing and regulating electronic communications.

Introduction

Both the Federal Communications (FCC) and the National Telecommunications and Information Administration (NTIA) are those government agencies designated to control and regulate electronic communications and manage the wireless spectrum in the United States. Everyone working in the communications field must absolutely be familiar with these organizations, their purpose, and procedures. This project provides an introduction to both.

Required

PC with Internet access.

Procedure

1. Go to the website of the FCC at www.fcc.gov.
2. Explore the website to see what it contains.
3. Repeat the procedure by going to the NTIA website at www.ntia.doc.gov.
4. Explore the website to see what it contains.
5. Using both websites as your information sources, answer the questions below. Use the search feature available on each site to find things you are interested in.

Questions

1. To whom does the FCC report within the government organization?
2. To whom does the NTIA report within the government organization?
3. Describe the purpose and function of the FCC.
4. Describe the purpose and function of the NTIA.
5. What is the FCC CFR Title 47?
6. Search the FCC website for information on licensing. Locate the information on commercial operators' licenses. What is the GROL? Who is required to have one?
7. What is the procedure for getting a GROL?

page 2

Project 1-2 Becoming Familiar with the FCC Rules and Regulations

Objective

Access the FCC rules and regulations pertaining to wireless and other forms of electronic communications and learn how to find specific answers to regulatory questions.

Introduction

The FCC rules and regulations are extensive and anyone working in communications needs to be familiar with them. It is easy to violate these rules, and that can lead to major troubles for you personally or

your company or its vendors, clients, or customers. Fines and even jail time can occur for really serious violations. Lack of knowledge of these rules and regulations is no excuse for their violation.

This project will let you see just how large and complex the regulations are. The goal is mainly to provide you with the knowledge and skills of accessing the rules and applying them to your work.

Required

PC with Internet access.

Procedure

1. Go to the FCC website www.fcc.gov and click on the rules and regulations link on the left side of the page.
2. From the next page, access the FCC rules Title 47 of the Code of Federal Regulations (CFR). This will bring you to a list of the different parts of the rules that pertain to specific sectors of communications. Scan through this list and become familiar with the different categories.
3. Using the rules as your source of information, answer the questions below.

Questions

1. What is contained in Subpart A of Part 2? Will you find that useful?
2. In Part 2, state what communications services can occupy the spectrum from 108 to 138 MHz?
3. What part pertains to commercial radio operators and licensing?
4. Give a brief overview of the kinds of devices covered by the Part 15 regulations.
5. Give a brief overview of the kinds of devices covered by the Part 18 regulations.
6. What part covers TV broadcasting?
7. Is cable TV regulated by the FCC? Which part?
8. Under what part(s) is radar covered?

9. Which parts cover CB and FRS personal radio services. What is the operating frequency ranges and power limits for each service?
10. List all of the communications modes and modulation methods allowed under the amateur radio regulations.
11. Under which part is radio-frequency interference and electromagnetic interference explained?
12. Which parts cover cell phones and wireless local area networks?

NAME

page 3

DATE



Electronic Fundamentals for Communications

Project 2-1 *Lc* Low-Pass Filter Design and Testing

Objective

Design, build, and test a low-pass *LC* filter.

Introduction

There are many different types of filters used in wireless and other communications applications. Newer types such as SAW and DSP are widespread, but the older traditional *LC* filters are still widely used in many designs. They are so widespread that you must have some familiarity with their design, application, and testing. Providing design details is beyond the scope of this book, but there are Internet resources that can perform this task. In this project, you examine several filter design websites and resources, then use one to design a filter. You will then build that filter with real components and test it. Finally, you will test the filter to become familiar with general specifications and measurement practices.

Required

- 1 PC with Internet access.
- 1 breadboard trainer
- 1 function generator
- 1 oscilloscope
- Resistors, capacitors, and inductors as specified by the designs

Procedure

1. Do a Web search on the term “filter design.” Scan through the recovered sites to see what is available. What two types of filters are the most commonly referred to in these sites?
2. Do a Web search on the phrase “How do *LC* filters work.”
3. Do a Web search on the phrase “on-line *LC* Filter Design Tool.”
4. Use one of the on-line *LC* filter design tools to design a low-pass *LC* filter with the following specifications.
 - a. Cutoff frequency of 32 kHz.
 - b. 1-k Ω input and output impedances.
 - c. Pi-type Butterworth design.
 - d. Three poles or an attenuation of at least 30 dB at 100 kHz.
5. Design the filter and print out the schematic and parts list.
6. Use the analysis feature to test the filter and print out the frequency plot.
7. Build the filter, using the closest standard component values. In most cases, the design will produce nonstandard values of capacitance and inductance. In some cases, you can get closer to the design values by combining two or more standard-value capacitors in parallel.
8. Test the filter by running a frequency-response curve, using the function generator and oscilloscope. Plot your frequency curve on standard logarithmic graph paper. You can also use the Bode plot feature in ELVIS if you are using NI LabVIEW.
9. What is the passband insertion loss of this filter at 30 kHz? Find that value on the breadboarded circuit.
10. What is the roll-off rate of this filter?

page 4

page 5

Project 2-2 *Rc* Active Filter Design and Testing

Objective

Design, build, and test an *RC* active band-pass filter.

Introduction

For audio and below 1-MHz applications, *LC* filters are no longer widely used because of the high cost, size, and weight of inductors for these low frequencies. At these frequencies, *RC* active filters based on op amps are popular and commonly used. They are relatively easy to design and build. In this experiment, you will use a common technique to design a typical *RC* active filter and then breadboard it and test it.

Required

- 1 breadboard trainer
- 1 dual ± 12 -V power supply
- 1 multimeter
- 1 oscilloscope
- 1 function generator
- 1 op amp IC (any common device will work such as the 741, 1458, TL072, or 301)
- Resistors and capacitors as determined by the design

Procedure

1. Go to the website www.ti.com. Locate Application Report SLOA093 (December 2001). Its title is *Filter Design in Thirty Seconds* by Bruce Carter. Print out a copy of this paper and read it.
2. Use the procedure described to design a narrow bandpass filter for 19 kHz. Use the procedure for a dual (\pm) supply op amp. Acquire the parts.
3. Breadboard the circuit.
4. Using a function generator and oscilloscope, plot the frequency-response curve of the filter. You can also use the Bode plot feature in LabVIEW if you are using it and the ELVIS trainer.
5. Make any necessary measurements to determine the following:
 - a. Filter insertion loss (or gain) at the center frequency.

- b. Filter bandwidth.
- c. Calculate filter Q .
- d. What is the maximum value of input voltage that can be applied before clipping (distortion) occurs?

Project 2-3 Ceramic Band-Pass Filter

Objective

To demonstrate the operation and selectivity of a typical ceramic filter and to show a method of plotting frequency-response curves.

Required

- 1 oscilloscope
- 1 function generator capable of sweep frequency operation
- 1 frequency counter
- 1 455-kHz ceramic filter with 8-kHz bandwidth (Toko Type CMF2 or equivalent)
- 2 2-k Ω resistors
- 2 10-k Ω resistors
- 1 1N34 or 1N60 germanium diode
- 1 0.1- μ F ceramic capacitor

Introduction

Most communications receivers, regardless of size or application, are superheterodyne types. In addition, most of the gain and virtually all of the selectivity of the receiver is obtained in the IF section. The gain is easily obtained with transistor or IC amplifiers. The selectivity is more difficult as the IF passband must be carefully selected to match the signals being received and must, in most cases, have very steep skirt selectivity. In older receivers, selectivity was obtained with cascaded tuned transformers whose coupling was adjusted to give the desired bandwidth and steepness of response. These IF transformers were difficult to tune and adjust.

Today, most IF selectivity is obtained with crystal filters or ceramic resonators. Some designs even use DSP filters. At very high frequencies, SAW filters are used. These filters are fixed during their

manufacture to furnish a desired frequency response. Very steep selectivity is readily obtained in a very small package at low cost, and no tuning or adjustment is required.

In this experiment, you will demonstrate the operation of a typical IF ceramic filter. You will show its response by tuning and show its attenuation of the signal. Then you will observe a method that lets you plot the frequency-response curve directly on the oscilloscope.

Procedure

1. Build the circuit shown in Fig. 2-1.
2. Connect an oscilloscope across the output. Turn on the function generator and set its frequency to approximately 455 kHz. Adjust the input voltage to the filter to 4 V_{P-P}.
3. Tune the function generator frequency until you notice a peak in the output voltage across the load resistor. Tune the frequency slowly and carefully until maximum output voltage is obtained. Measure the output voltage and record the value below. Measure the frequency at which the peak output voltage is obtained. Be sure to use a frequency counter for best accuracy.

$f =$ _____

$V_0 =$ _____

Does the filter have more than one peak?

4. Calculate the gain or loss of the filter in dB.
dB = _____
5. Slowly tune the function generator frequency above and below the filter frequency of 455 kHz. Note how fast or how slow the output voltage changes. Describe in your own words how the output changes with frequency.
6. Refer to Fig. 2-2. Add the diode and its 10-k Ω load and the low-pass filter made up of the 10-k Ω resistor and the 0.1- μ F capacitor.

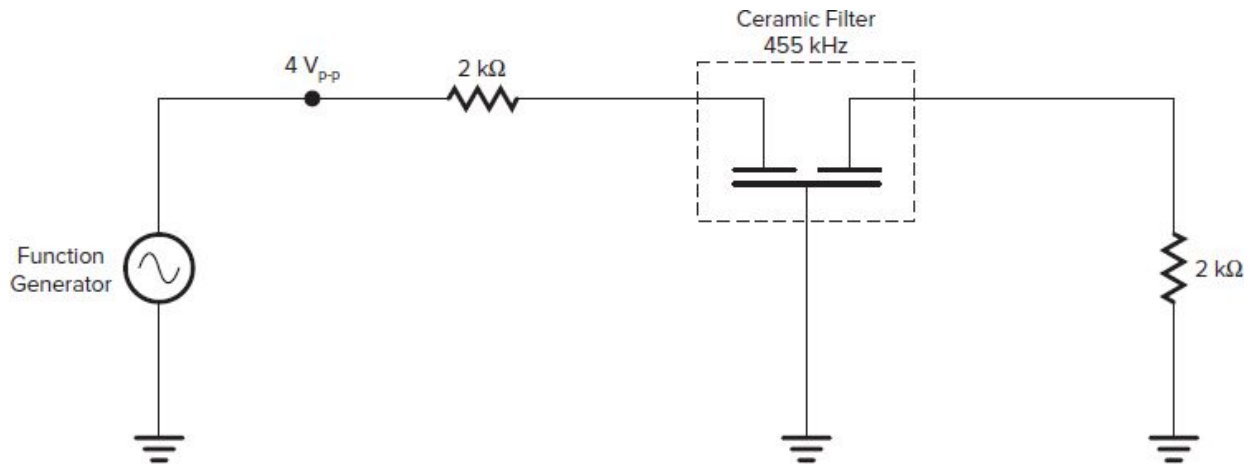


Fig. 2-1 Filter measurement circuit.

7. Set the function generator frequency to a value of about 400 kHz. Then, set up the generator for sweep frequency operation. Refer to the generator operation manual or have your instructor explain the sweep mode. Most generators frequency modulate their internal oscillator with a linear sawtooth wave, which varies the output frequency from some lower value continuously to some higher value. You will apply this linearly increasing frequency to the filter. You will observe the output of the diode detector at the filter output.
8. While observing the output of the diode detector, adjust the sweep characteristics and the oscilloscope time base until you display a waveform that is the frequency-response curve of the filter. Draw what you see. What is it?

Questions

1. Explain how the diode detector and swept frequency combine to plot the frequency-response curve of the filter.
2. How could you determine the 6-dB bandwidth of this filter? Describe the process and then implement the procedure on the experimental circuit and state the bandwidth you measure. Be sure to use a frequency counter for your measurements.
3. Describe the shape of the response curve in the pass band. Is it a single peak, a flat horizontal line, or does it have some variation?

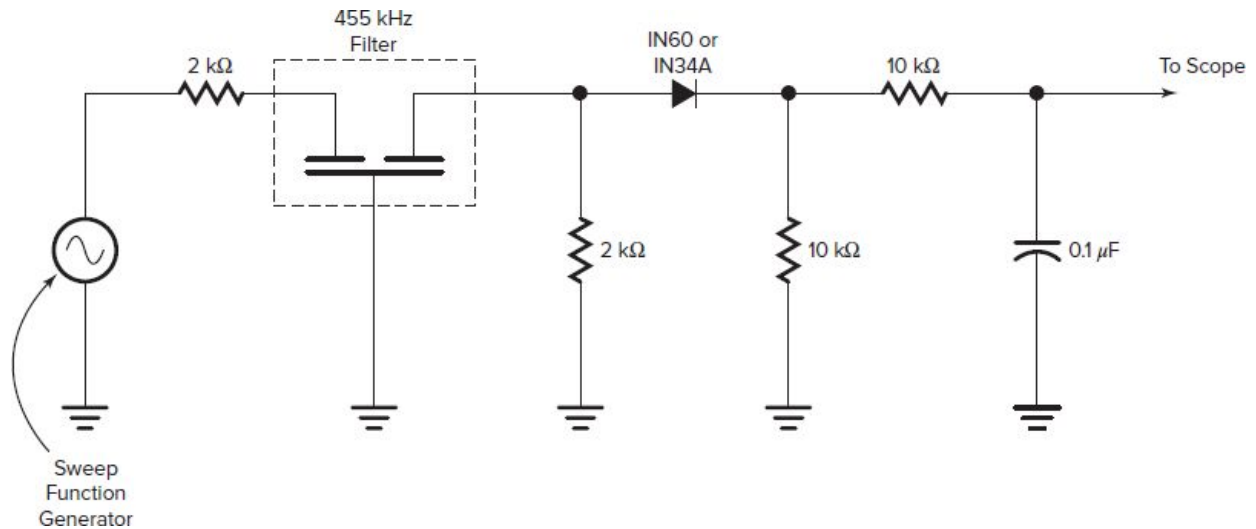


Fig. 2-2 Diode detector for sweep display.

Project 2-4 Switched Capacitor Filters

page 8

Objective

To demonstrate IC switched capacitor filters.

Required

- 1 oscilloscope (dual-trace, 20-MHz minimum bandwidth)
- 3 function generators
- 1 frequency counter
- 1 dc power supply capable of supplying ± 5 V
- 1 general-purpose op amp IC (741, 411, etc.)
- 3 100-k Ω resistors
- 1 MF10 programmable switched capacitor filter ICs (National Semiconductor, now Texas Instruments, or Maxim)
- 1 74LS93 TTL binary counter IC
- 1 22-k Ω resistor
- 1 33-k Ω resistor
- 2 220-k Ω resistors
- 2 330-k Ω resistors
- 2 0.1- μ F capacitors

References

1. *Principles of Electronic Communication Systems*, 5th ed., Chapter 2.
2. MF10 data sheet (National Semiconductor, now Texas Instruments, or Maxim).

Introduction

An alternative to conventional RC or LC passive filter or RC active filters is switched capacitor filters (SCFs). This IC filter uses a capacitor network switched by fast MOSFETs. When combined with op amps in a feedback arrangement, the result is a very selective filter. Most SCFs can be programmed to function as a low-pass, high-pass, band-pass, or notch filter. This all-electronic filter is smaller and lighter than conventional LC filters, and it can be programmed for a specific frequency and bandwidth.

SCFs are sampling filters in that they take in an analog signal and sample it at a rate many times the highest frequency content. Samples of the signal voltage are stored in the capacitor network. A clock oscillator sets the sampling rate. The frequency of the clock sets the center or cutoff frequency of the filter.

The characteristics of a SCF are fully adjustable by setting the clock frequency. The Q and therefore the bandwidth (BW) of the filter are also programmable. A variety of low-cost SCFs are available in IC form. They can be used to form low-pass, high-pass, and band-pass and notch filters in the range 0 to 1 MHz. You will demonstrate the popular MF10 SCF in this experiment.

The MF10 consists of two filters in a 20-pin-DIP IC. Externally connected resistors select the filter mode and operational characteristics. The design procedures are beyond the scope of this experiment, but more details can be obtained from the manufacturer's data sheet. In this experiment, the MF10 is programmed as a band-pass filter (BPF). Its Q is determined by the external resistor ratios. As in any BPF, the bandwidth (BW) is

$$BW = \frac{f_{CLK}}{Q}$$

The filter center frequency is f_0 .

The center frequency of the BPF is determined by the clock frequency (f_{CLK}) applied to the internal MOSFET sampling switches:

$$f_0 = \frac{f_{\text{CLK}}}{50}$$

page 9

For example, if the clock is set to 30 kHz, the BPF center frequency is 30 kHz/50 = 600 Hz. The clock signal is a TTL-compatible 50 percent duty cycle square wave. Its maximum value is 1 MHz. This sets the upper frequency limit of the center frequency to

$$f_0 = \frac{1000 \text{ kHz}}{50} = 20 \text{ kHz}$$

In this experiment, you will demonstrate the filter operation and characteristics.

Procedure

SCF Operation

1. Construct the MF10 filter circuit shown in Fig. 2-3. Remember that there are two filters in the IC and you will test both of them. The input to the first filter is labeled V_{inA} at the 220-k Ω resistor R_1 . The output of the first filter is at pin 2 and is labeled V_{oA} . The input to the second filter is labeled V_{inB} at the 330-k Ω resistor. The output of the second filter is at pin 19 and is labeled V_{oB} . The gain of the filter is the ratio of $-R_3/R_1$. The Q of the filter is the ratio of R_3/R_2 .

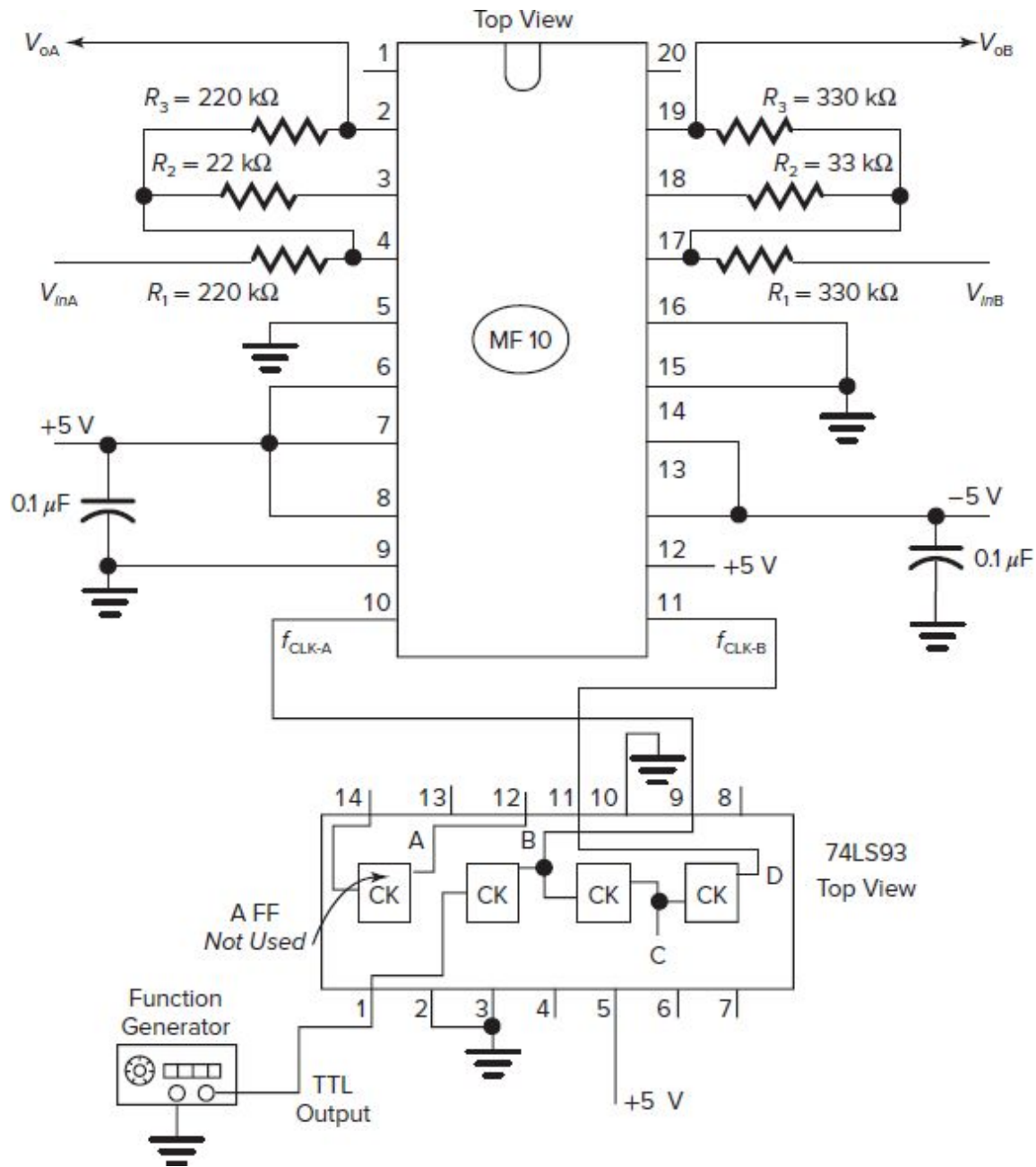


Fig. 2-3 MF10 SCF BPF wiring.

The 74LS93 is a TTL binary counter used as a frequency divider. The clock signal is derived from an external function generator that drives the counter. The output of the first flip-flop in the counter (pin 9) divides the clock signal by 2 to ensure a 50 percent duty cycle signal to the f_{CLK-A} input that sets the center frequency of the first filter. The output of pin 11 on

$$f_0 = \underline{\hspace{2cm}}$$

$$f_0 = \underline{\hspace{2cm}}$$

$$V_{oA} = \underline{\hspace{2cm}}$$

$$\text{Voltage at -3 dB down} = \underline{\hspace{2cm}}$$

$$f_L = \underline{\hspace{2cm}}$$

$$f_U = \underline{\hspace{2cm}}$$

$$\text{BW} = \underline{\hspace{2cm}}$$

$$Q = \underline{\hspace{2cm}}$$

Questions

1. Assume a MF10 filter with $R_1 = 47 \text{ k}\Omega$, $R_2 = 22 \text{ k}\Omega$, $R_3 = 47 \text{ k}\Omega$. The clock frequency is 72 kHz. Find the filter center frequency, Q , and the bandwidth.
2. What would be the effect of cascading the two filters in the MF10?
3. How is the center frequency of the filter varied?
4. (True or False) The MF10 can be used for low- and high-pass filtering.

NAME

page 11

DATE

Chapter

3

Amplitude Modulation Fundamentals

Project 3-1 Measuring the Percent of Modulation

Objective

To learn two methods of expressing and measuring the percent of modulation of an AM signal and to observe the condition of overmodulation.

Required

- 1 oscilloscope with provision for external horizontal input
- 1 frequency counter
- 1 12-V dc power supply
- 1 function generator with sine output
- 1 XR-2206 Exar function generator IC
- 1 0.002- μ F capacitor
- 1 0.47- μ F capacitor
- 1 1- μ F capacitor
- 1 10- μ F electrolytic or tantalum capacitor
- 1 150- Ω resistor
- 2 4.7-k Ω resistors
- 1 6.8-k Ω resistor
- 1 10-k Ω resistor

- 1 47-kΩ resistor
- 1 1-kΩ pot
- 1 100-kΩ pot

References

Principles of Electronic Communications Systems, 5th ed., Chapter 3.

Introduction

Optimum amplitude modulation occurs when the peak-to-peak value of the modulating signal is equal to the peak value of the carrier. In that way, the modulating signal causes the carrier amplitude to vary from twice its unmodulated value to zero, as Fig. 3-1 illustrates. That is called a modulation index m of 1, or 100 percent. Of course, a modulation index of 0, or 0 percent modulation, occurs when the modulating signal amplitude is 0 and a constant-amplitude carrier signal results. Intermediate values of modulation index or percent can be determined by measuring the amplitudes of the modulated signal, as shown in Fig. 3-2. The following formulas can then be used to compute the percent of modulation:

$$\text{Modulation index } m = \frac{V_{\max} - V_{\min}}{V_{\max} + V_{\min}}$$

$$\text{Percent modulation} = m \times 100$$

page 12

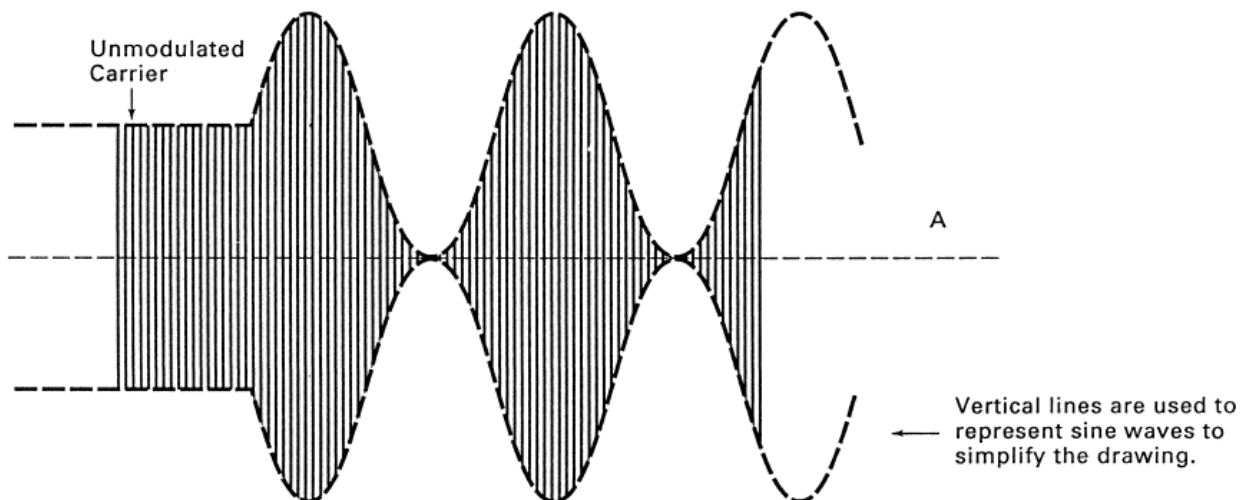


Fig. 3-1 100 percent modulation.

The most important criterion is to attempt to achieve 100 percent modulation but not exceed it. If the modulating-signal amplitude is greater than the carrier amplitude, overmodulation will occur and the information signal will be distorted. Note how the peaks in Fig. 3-3 are clipped.

Another way to measure the modulation index or percent of modulation is to use a trapezoidal pattern that can be displayed on an oscilloscope. Such a pattern is shown in Fig. 3-4. To create it, the AM signal is connected to the vertical input as usual. The modulating information signal is connected to the external horizontal input and is used as the sweep signal. The scope is put into the external sweep or *x vs. y* display mode, and the vertical and horizontal gains are adjusted until the pattern appears. The V_{\max} and V_{\min} values can then be read from the pattern and used in the previously given formula to calculate the modulation index and percent of modulation.

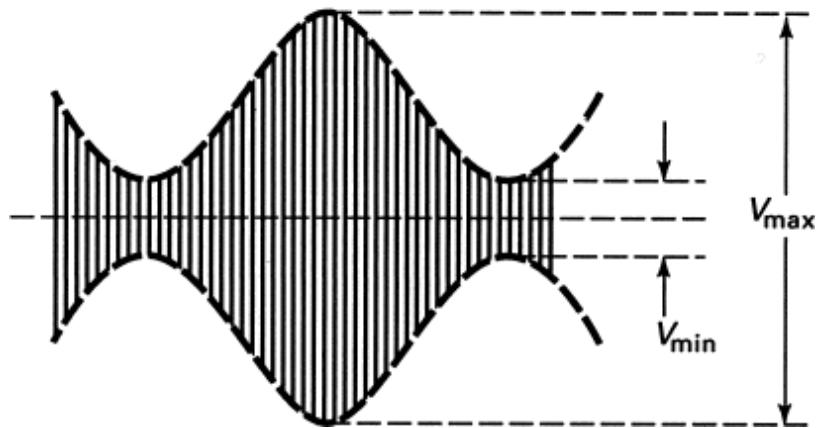


Fig. 3-2 Measuring voltages to calculate modulation index.

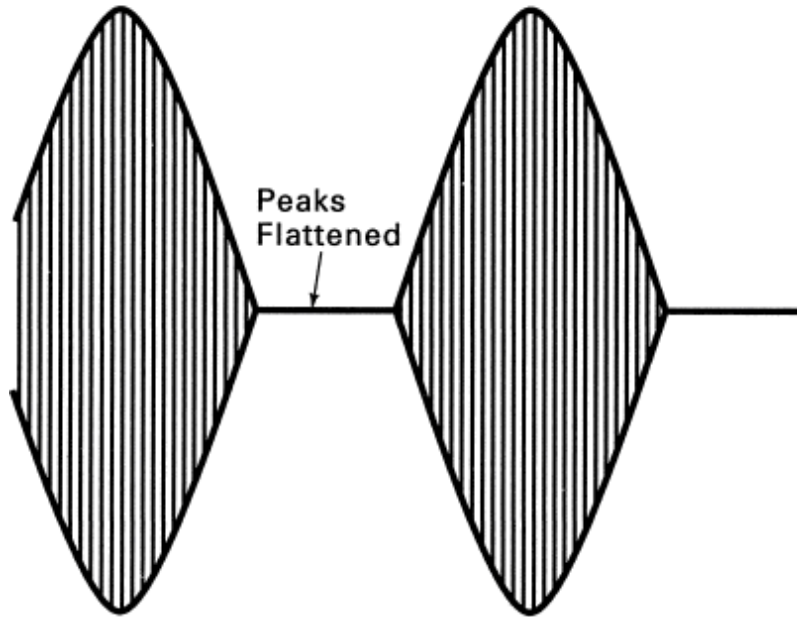


Fig. 3-3 Overmodulation-caused distortion.

The trapezoidal pattern method is usually preferred because measurement of V_{\max} and V_{\min} is easier. Further, the slanted sides of the trapezoid readily show distortion. Straight sides mean no distortion; nonlinear sides indicate overmodulation or other distortion.

To demonstrate AM, you will use the XR-2206 Exar function generator IC. This chip contains an oscillator that produces sine, square, triangle, and sawtooth waves. The frequency is set by an external resistor and capacitor. The oscillator can also be voltage-controlled, which permits its frequency to be varied. This allows sweep frequency or frequency modulation to be produced, but that function will not be used here.

The chip also contains an amplitude modulator circuit. The internal oscillator serves as the carrier, which can be amplitude-modulated by an external input signal. The modulator produces excellent AM, but the modulating signal amplitude required is higher than the theoretical value necessary to produce a given degree of modulation.

Procedure

1. Construct the circuit shown in Fig. 3-5. You will use the external signal function generator with a 1-kHz output to modulate the

carrier produced by the 2206 IC function generator. The 2206 chip contains a built-in amplitude modulator.

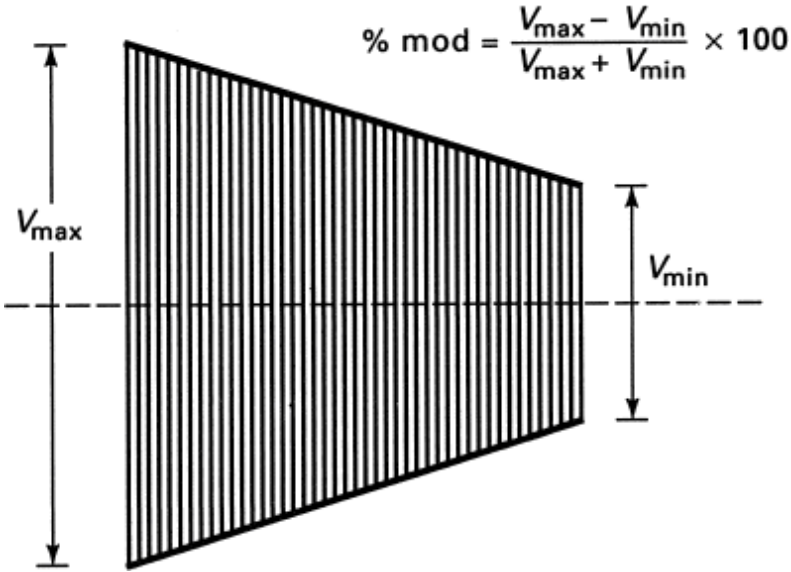


Fig. 3-4 Trapezoid pattern for measuring the percent of modulation.

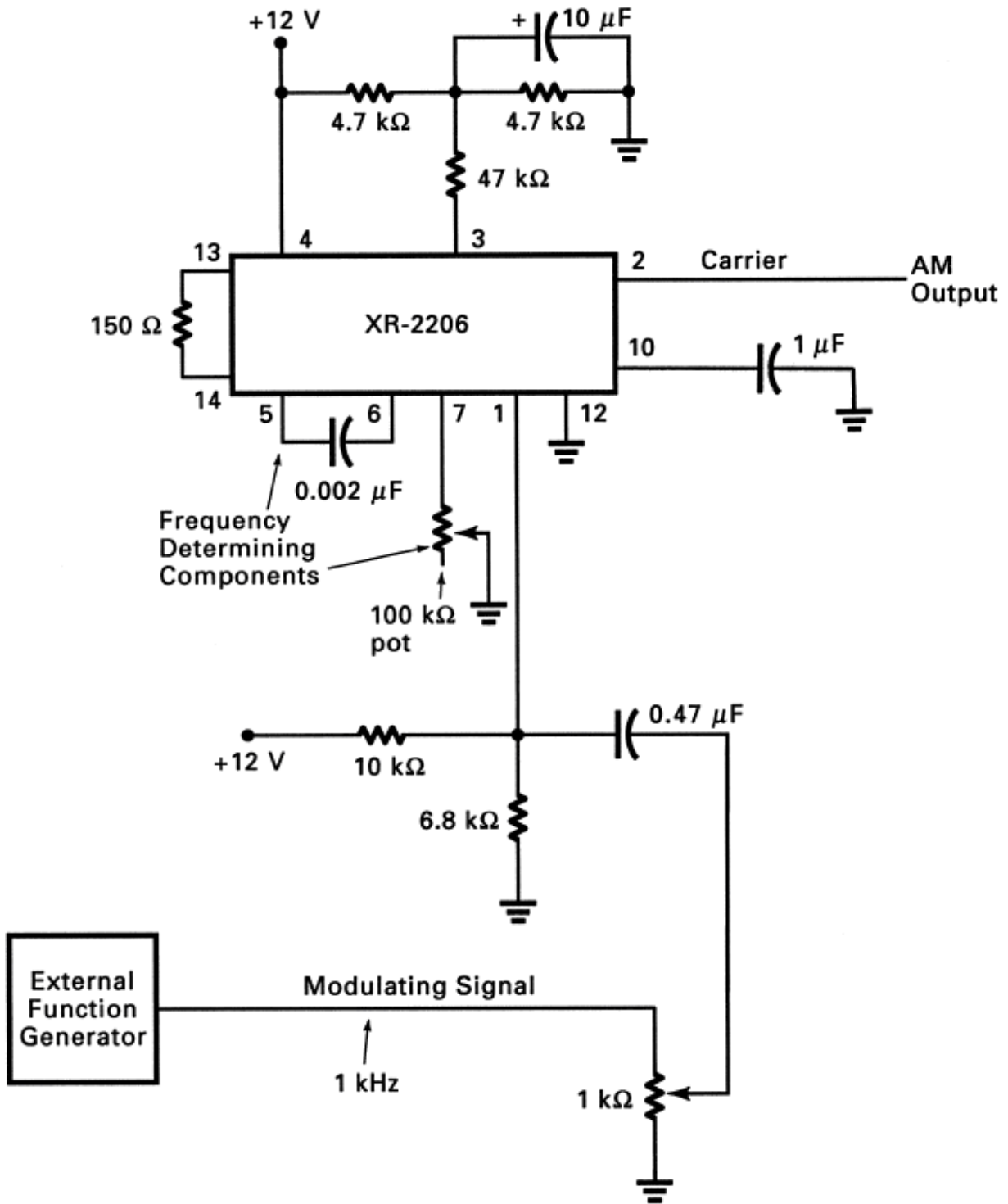


Fig. 3-5 Amplitude modulation of the 2206 function generator IC.

2. Apply power to the circuit. Adjust the dc supply voltage to 12 V. Observe the waveform at the output at pin 2 of the carrier

function generator IC. Reduce the modulating signal input from the external function generator to zero with the 1-k Ω pot or the amplitude control on the function generator. While measuring the output at pin 2 with a frequency counter, adjust the 100-k Ω pot for a frequency of 40 kHz. (*Note:* If no counter is available, use the calibrated horizontal sweep of the scope to set the frequency.)

3. Measure and record the carrier output voltage, peak and peak-to-peak, with the oscilloscope.
4. Use the value obtained in step 3 to calculate and record the amount of voltage required to 100 percent modulate this carrier.
5. Vary the amplitude of the modulating signal from the external function generator by using the 1-k Ω pot and/or the external amplitude control from zero to about 5 V_{p-p} . Note the effect on the AM signal.
6. Adjust the modulating signal amplitude for maximum value just prior to the occurrence of distortion. If clipping like that shown in Fig. 3-3 occurs, reduce the input signal amplitude until the distortion just disappears. Measure and record V_{max} and V_{min} .
7. Calculate and record the modulation index and percent of modulation. Can 100 percent modulation be achieved without clipping?
8. Measure and record the amplitude, both peak and peak-to-peak, of the modulating signal voltage. How does this value compare to the value you calculated in step 4?
9. Display the trapezoidal pattern to measure percent modulation. Leave the vertical scope input on the AM signal at pin 2. Connect the modulating signal at the 1-k Ω pot output to the external horizontal (sweep) input. Set the time base control or other switch on the scope for external input sweep. Adjust the scope vertical and horizontal gain controls to display the trapezoid.
10. Vary the modulating signal amplitude and note the variation in the trapezoid. Vary the horizontal and vertical gain controls on the scope to keep the pattern centered.
11. Temporarily set the scope back to normal time base operation. Observe the AM wave. Adjust the modulating signal amplitude for 100 percent modulation without distortion.

12. Return the scope to the previous condition with external horizontal sweep. Observe the pattern. What is the shape of the pattern with 100 percent modulation?
13. Return the scope to normal time base operation. Observe the AM wave.
14. Adjust the amplitude of the modulating signal to zero so that the output is a constant-amplitude sine wave. Measure and record the carrier amplitude, both peak and peak-to-peak.
15. Set the input modulating signal amplitude to $0.5 V_{p-p}$.
16. Calculate and record the modulation index and modulation percent you would expect from the voltages given in steps 14 and 15.
17. Now view the AM signal on the oscilloscope and determine V_{max} , V_{min} , and the percent of modulation. Record your observations.
18. Compare the values you obtained in steps 16 and 17. Explain any discrepancies.
19. While observing the AM signal at pin 2 on the 2206 IC, increase the amplitude of the modulating signal to achieve 100 percent modulation. Then continue increasing the modulating signal amplitude until distortion occurs. Describe the effect you observe when overmodulation occurs.
20. Do not disassemble the modulator circuit, because you will use it in future experiments. For now, turn off the power.

Questions

1. On an AM wave, $V_{max} = 2 V$ and $V_{min} = 0.5 V$. The modulation index is _____ .
 - a. 0.25
 - b. 0.4
 - c. 0.6
 - d. 0.85
2. For 100 percent modulation, what is the relationship between the carrier voltage V_c and the modulating signal voltage V_m ?
 - a. $V_m = V_c = 0$
 - b. $V_m = V_c$
 - c. $V_m > V_c$

- d. $V_m < V_c$
3. When overmodulation occurs, _____ .
- a. $V_m = 2V_c$
- b. $V_m = V_c$
- c. $V_m < V_c$
- d. $V_m > V_c$
4. When you are using a trapezoidal pattern to measure modulation percentage, 100 percent modulation is indicated by what shape of pattern?
- a. Triangular
- b. Rhomboid
- c. Square
- d. Rectangle
5. (True or False) The frequency of the modulating signal affects the percent of the modulating signal.

page 16

page 17

Project 3-2 Spectrum Analyzer

Objective

To demonstrate the operation and application of a spectrum analyzer.

Required

All the equipment and circuitry used in Project 3-1 plus a spectrum analyzer capable of measuring and displaying low-frequency signals.

Introduction

A spectrum analyzer is a test instrument designed to measure and display signals in a frequency-domain format. Like an oscilloscope, the spectrum analyzer uses a CRT or LCD display, but unlike a scope, it displays signal amplitude with respect to frequency rather than time. Complex waveforms made up of many signals are shown as vertical lines whose heights indicate signal strength and whose horizontal positions indicate frequency. In other words, the spectrum analyzer effectively displays a complex signal as its individual sinusoidal components.

The spectrum analyzer is an ideal test instrument for use with modulated signals because it allows you to see the carrier and sidebands resulting from the modulation. It makes measurement and troubleshooting faster and easier.

In this experiment, you will become familiar with spectrum analyzer operation and use. You will display the spectrum produced by the 2206 amplitude-modulated function generator you analyzed in the preceding experiment.

Procedure

1. Acquire the operation manual for the spectrum analyzer you will be using. Take some time to read and study it. Concentrate on the following:
 - a. How does the spectrum analyzer work? Analyze its operation in terms of a block diagram of the major circuits.
 - b. How do you use the spectrum analyzer? What are the various controls, inputs, and output? What are the procedures for setting up, calibrating, and making measurements with the unit?
 - c. What are the unit's specifications?
2. Set up the spectrum analyzer and become familiar with the controls and how to display a signal.
3. What display would you expect to see on the spectrum analyzer display if a 1-kHz square wave is connected? Sketch the frequency domain display you would see.
4. Connect a 1-kHz square wave from a function generator to the spectrum analyzer. Adjust the controls to see the display. Draw what you see. Is it the same as you predicted in step 3? Explain.
5. Apply power to the 2206 function generator IC you constructed in Lab Experiment 2-1. Adjust the output for a frequency of 35 kHz. Set the modulating signal frequency to 1 kHz. While observing the output at pin 2 of the 2206 (see Fig. 3-5), increase the modulating signal amplitude until 100 percent modulation is achieved. Predict what signal the spectrum analyzer will display and sketch it. (*Note:* If your spectrum analyzer will not accommodate the 35-kHz carrier, reduce the carrier frequency by

connecting a larger capacitor to the XR-2206 in place of the 0.002- μ F unit on pins 5 and 6.)

6. Apply the AM signal to the spectrum analyzer and adjust the controls to display the signal. How does it compare to the prediction you made?
7. Reduce the modulating signal amplitude and note the effect it has on sideband amplitude.
8. While observing the AM signal, increase the modulating signal amplitude until overmodulation occurs. The signal should be as clipped and distorted as possible. Observe the spectrum analyzer display. Explain how the distortion is indicated.

page 18

Questions

1. (True or False) The spectrum analyzer displays the amplitudes of the individual sinusoidal components of a complex signal as determined by the Fourier theory.
2. With 60 percent modulation and a 500-mV carrier, the sideband amplitude for a sine-wave modulating signal is _____ .
 - a. 150 mV
 - b. 250 mV
 - c. 300 mV
 - d. 500 mV
3. The largest component in an AM signal as displayed on a spectrum analyzer is the _____ .
 - a. Upper sideband
 - b. Lower sideband
 - c. Carrier
 - d. Modulating signal
4. Distortion from overmodulation is displayed by a spectrum analyzer as _____ .
 - a. Greater carrier power
 - b. Higher sideband amplitudes
 - c. Lower carrier and sideband amplitudes
 - d. Additional harmonic-related sidebands
5. The horizontal sweep calibration of a spectrum analyzer is given in units of _____ .

- a. Time
- b. Frequency
- c. Voltage
- d. Power

Project 3-3 AM Radio Experience

Objective

To tune and listen to radio stations using amplitude modulation.

Required

- 1 AM broadcast radio
- 1 receiver capable of receiving shortwave (SW) signals*
- 1 receiver capable of receiving citizen band (CB) stations*
- 1 receiver capable of receiving VHF aircraft radio stations (scanner radio)
- 1 receiver covering the amateur radio bands at 80, 40, or 20 m and capable of receiving SSB transmissions (built-in BFO and product detector)*
- Antennas as required

*Shortwave (SW) radio.

Introduction

AM is the oldest form of modulation and, despite its disadvantages, it is still used in many communications applications. AM broadcast radio is the most common example. AM is also the most common modulation for shortwave (SW) broadcast radio. Two examples of two-way radio are citizen band (CB) radio and aircraft radio. CB radio is used by individuals for car-to-car communications and by truckers because it is effective and very inexpensive. And it is available to anyone with the money to purchase a transceiver. AM is also still used in aircraft radios. These are the VHF radios used from tower to airplane and from plane to plane. Again, they are low-cost and very effective. A common use of the SSB form of AM radio is amateur radio communications.

The purpose of this experiment is to listen to typical AM transmissions and gain experience using typical communications equipment.

Procedure

1. State the frequency range of the AM broadcast band and the frequency spacings of stations.

2. Use an AM radio to listen to the broadcast band. Do this during daylight hours. Tune the entire extent of the band, count the number of stations you hear, and record their frequencies.

3. Listen for weak stations. What is the main problem in hearing weak stations?

4. Repeat steps 2 and 3 at night. What are the main differences between day and night reception?

5. What is the frequency range of the CB band? How many channels are available?

6. Use the receiver in a CB transceiver or a scanner radio to monitor the CB channels. Which channels are most widely used?

7. Try out the squelch and noise blanker controls on the receiver.

8. Monitor the channels and try to determine the main use of the CB band. What are some common applications?

9. State the frequency range of the aircraft radio band.

10. Use a scanner radio to monitor these aircraft frequencies. Listen to as many different conversations as you can. Describe briefly what you hear.

11. Try out the squelch and noise blanker controls on the receiver.

12. Use an amateur radio transceiver or receiver to listen to SSB transmissions. A good example is the continuous communications on the amateur 20-m band. Tune the 14.2- to 14.3-MHz range with the receiver BFO on. As you tune the receiver slowly, note how the tuning affects the voice. Explain what you are hearing.

13. If a USB/LSB switch is available on the receiver, try out both positions to see which one produces the most signals.

14. Tune the ham bands for Morse code signals. Use the BFO. Some frequency ranges to try are 3.5–4, 7–7.3, 10.0–10.15, 14–14.3, 18.068–18.168, and 21–21.5 MHz.

15. Turn off the BFO. Tune the 5- to 18-MHz range on a SW radio. Look for foreign broadcast radio stations. The frequencies near 6, 9, and 13 MHz are particularly rich in SW BC stations. Look for as many English-language stations as possible. Record their call letters and locations if possible.

16. Tune for the U.S. time station WWV at 10, 15, or 20 MHz. Listen to this station about 5 min before the hour. Note what happens at the turn of the hour.

page 21

Questions

1. What is the main disadvantage of AM over FM for most communications applications? Did you experience it during your monitoring?
2. Name two other AM applications in communications.
3. Explain the need for and operation of the squelch and noise blanker controls.
4. Why is a BFO needed to receive SSB signals?
5. (True or False) CW signals are a form of AM. _____

page 22

NAME

page 23

DATE

Chapter

4

Amplitude Modulation and Demodulation Circuits

Project 4-1 Diode Modulator and Mixer

Objective

To demonstrate how a diode can be used to perform amplitude modulation and mixing.

Required

You will use all of the components and test equipment in Project 3-1 plus the following:

- 1 741 IC op amp
- 1 1N4149 or 1N4148 silicon diode
- 1 2-mH inductor
- 3 0.1- μ F capacitors
- 1 0.01- μ F capacitor
- 1 150- Ω resistor
- 4 10-k Ω resistors
- 1 15-k Ω resistor
- 1 100-k Ω pot

Introduction

Amplitude modulation and mixing are actually the same process. Two or more signals are combined in a circuit that produces new signals at different frequencies. The amplitude modulator generates sidebands whose frequencies are the sum and difference of the frequencies of the carrier of the modulating signal. A mixer produces similar signals. The output of a mixer consists of the two original input signals plus the sum and difference frequencies, which are, of course, the same as the sidebands.

One of the simplest ways to perform mixing or amplitude modulation is to combine the two signals linearly in a resistive network or op-amp summer and then feed them to a diode rectifier. The output of the rectifier consists of a tuned circuit or filter that selects the desired output frequency. In this experiment, you will demonstrate the use of a diode for amplitude modulation and mixing.

Procedure

Modulator

1. Modify the function generator circuit you used in Project 3-1; the revised circuit is shown in Fig. 4-1. First remove the 10- and 6.8-k Ω resistors connected to pin 1 of the 2206. Also remove the 0.47- μ F capacitor and the 1-k Ω pot. Then connect pin 1 of the 2206 to ground. This modification removes the amplitude modulation capability of the 2206. In this experiment, you will use the function generator simply as a carrier oscillator.

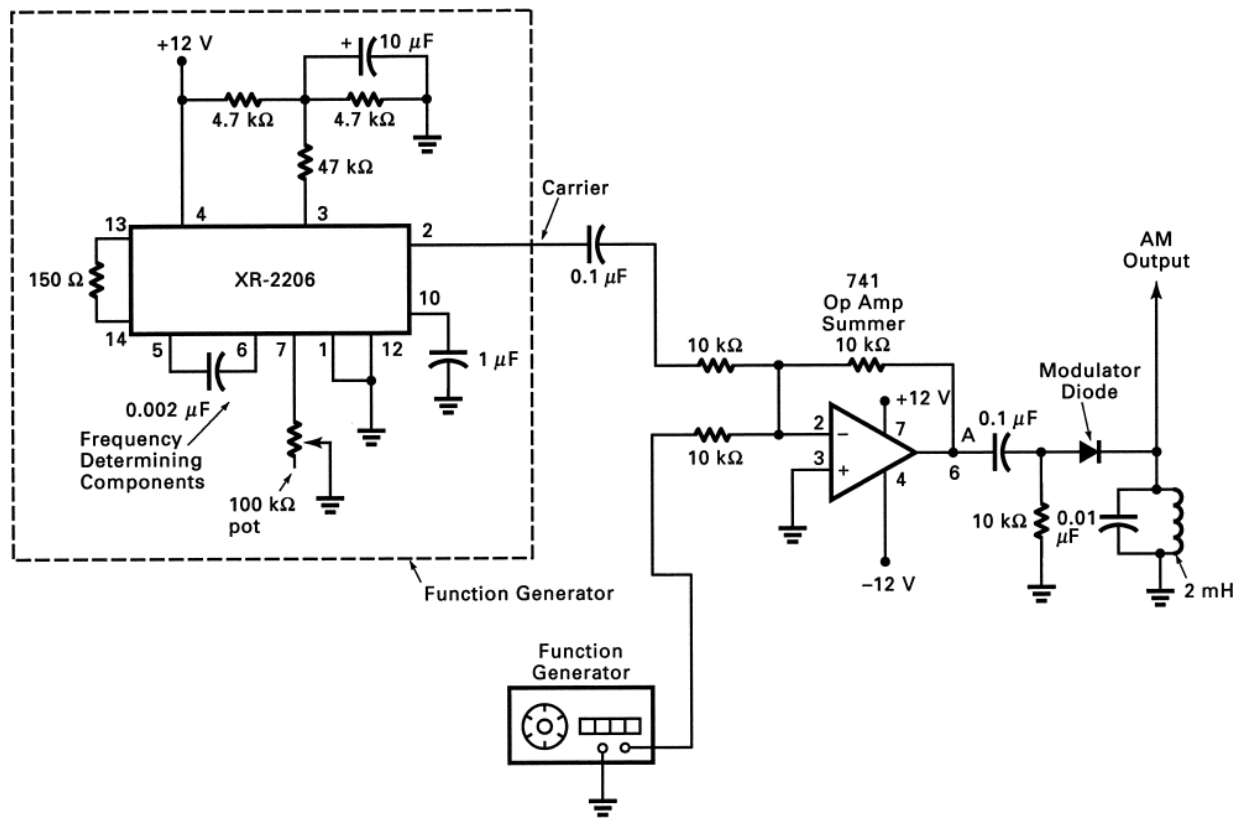


Fig. 4-1 Diode (AM) modulator circuit.

2. Construct the remaining portion of the circuit shown in page 24 Fig. 4-1. The circuit consists of an op amp summer used as a linear mixer and a diode modulator mixer circuit. This first circuit will be used for amplitude modulation.
3. Apply power to the circuit. Monitor the output of the 2206 function generator at pin 2. You should observe a sine wave output with an amplitude of approximately $4 V_{p-p}$. The input signal from the external function generator should be zero at this time. Next, connect the frequency counter to the function generator output. Then adjust the 100-k Ω potentiometer for an output frequency of 40 kHz. If you do not have a frequency counter, simply use the calibrated horizontal sweep of the oscilloscope to help you in setting the correct frequency.
4. Connect the oscilloscope to pin 6 of the 741 op amp summer. You should observe the 40-kHz signal. Now begin increasing the amplitude of the external signal from the function generator. This is the modulating signal, which can have almost any desired

frequency; set it to 1 kHz for this exercise. Adjust the amplitude of the 1-kHz signal for about $5 V_{p-p}$ at the op amp summer input.

5. Observe the output signal that appears at pin 6 of the 741 op amp summer. Sketch the waveform you obtained. Is this an amplitude-modulated signal? If not, explain what it is.
6. Connect the oscilloscope across the 2-mH inductor. This inductor and the $0.01\text{-}\mu\text{F}$ capacitor in parallel with it form a resonant circuit that will select the desired output signal. In this case, it will pass the carrier and sidebands of an AM signal. Calculate and record the resonant frequency of this tuned circuit.
7. Reduce the amplitude of the modulating signal to zero. With the oscilloscope across the parallel-tuned circuit, adjust the $100\text{-k}\Omega$ frequency potentiometer on the 2206 function generator. Tune the frequency for maximum output voltage on the oscilloscope. In doing so, you are adjusting the frequency of the carrier to the resonant frequency of the tuned circuit. Measure and record the output frequency with the frequency counter. How does the measured frequency compare to the frequency you calculated above?
8. While observing the output across the tuned circuit, begin increasing the amplitude of the modulating signal from the external function generator. Set the amplitude of the modulating signal to approximately $1 V_{p-p}$. Now observe the signal across the tuned circuit. Is this an AM signal?
9. Continue to increase the amplitude of the modulating signal while observing the effect on the output waveform. Note the effect that it has on the output. Can this circuit achieve 100 percent modulation?
10. You will now modify the circuit and demonstrate its operation as a mixer. First turn off the power. Remove the 2-mH inductor and the associated $0.01\text{-}\mu\text{F}$ capacitor from the output. Replace them with a $15\text{-k}\Omega$ resistor in parallel with a $0.1\text{-}\mu\text{F}$ capacitor as shown in Fig. 4-2.
11. Apply power to the circuit. Observe the output of the 2206 function generator with the oscilloscope. Connect the frequency counter to the output of the 2206. Adjust the $100\text{-k}\Omega$ pot on the 2206 for a frequency of 20 kHz. This is one input to the mixer.

12. Next, observe the output from the external function generator at the second input to the op amp summer. With the frequency counter, adjust the external function generator output for a frequency of 21 kHz. Set the output amplitude of the external function generator to a value of $1 V_{p-p}$. This is the second input to the mixer.

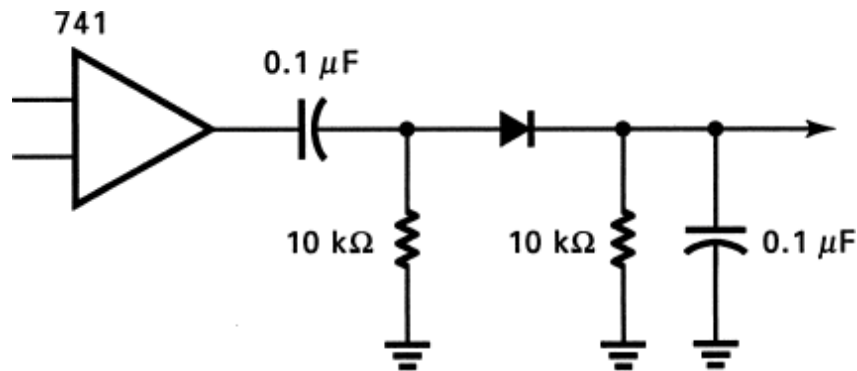


Fig. 4-2 Modifying the diode circuit for mixer operation.

13. From what you have learned about mixers, calculate and record the output frequencies that would occur at the diode mixer output.
14. Considering the components connected to the output of the diode mixer, which of the output signals you predicted in step 13 would appear at the output?
15. Connect the oscilloscope and the frequency counter to the 15-kΩ load. Observe the signal at the output of the mixer. What are its shape and frequency?

Questions

1. What mathematical process is involved in linear mixing?
 - a. Addition
 - b. Subtraction
 - c. Multiplication
 - d. Division
2. What mathematical process is involved in amplitude modulation or nonlinear mixing?
 - a. Addition

- b. Subtraction
 - c. Multiplication
 - d. Division
3. (True or False) Nonlinear mixing and AM are the same process.
 4. What is the function of the $0.1\text{-}\mu\text{F}$ capacitor shown in Fig. 4-2?
 - a. Resonant circuit
 - b. Noise elimination
 - c. High-pass filter
 - d. Low-pass filter
 5. If the circuit shown in Fig. 4-2 is replaced with the one shown in Fig. 4-3, what output signal will you expect to see?
 - a. Difference
 - b. Sum
 - c. Carrier
 - d. Modulating signal

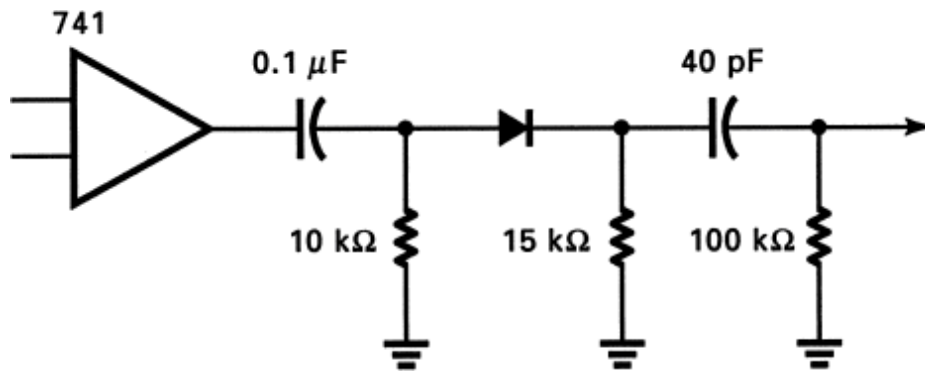


Fig. 4-3 Circuit for question 5.

Project 4-2 Differential Amplifier Modulator

page 26

Objective

To demonstrate the use of a differential amplifier as an amplitude modulator.

Required

You will need the 2206 function generator circuit and all of the other components and test equipment listed for Project 3-1. In addition, you

will need:

- 3 NPN transistors (2N4401, 2N4124, 2N3904, MPSA20, etc.)
- 1 741 op-amp
- 1 2-mH inductor
- 3 0.1- μ F capacitors
- 1 0.47- μ F capacitor
- 1 150- Ω resistor
- 1 4.7-k Ω resistor
- 1 6.8-k Ω resistor
- 2 15-k Ω resistors
- 1 47-k Ω resistor
- 1 10-k Ω pot

Introduction

A differential amplifier makes an ideal amplitude modulator for low-level circuits, and a standard circuit of one is shown in Fig. 4-4. The differential transistors Q_1 and Q_2 are connected so their emitters are common. Inputs are applied to the base of one transistor or the bases of both transistors. Two outputs at the transistor collectors are provided. A load can be connected between the two collectors to provide a differential output. Loads can also be connected between the two collectors for balanced operation or between either collector and ground to provide single-ended operation.

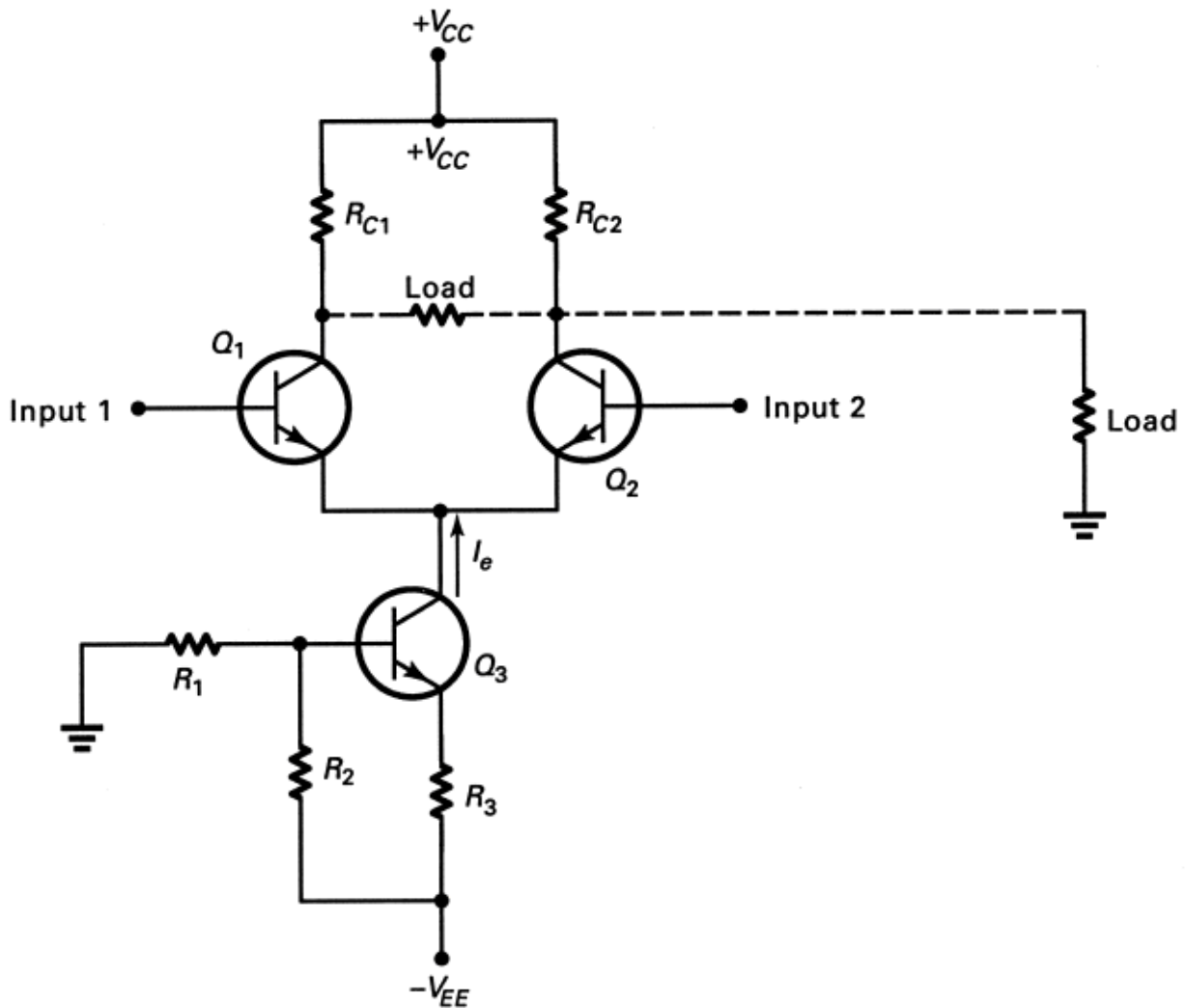


Fig. 4-4 Differential amplifier.

The common current I_e to the differential transistors is page 27 supplied by a constant-current source, transistor Q_3 . Bias resistors R_1 through R_3 and the negative supply voltage V_{EE} set the current value. The emitter current I_e remains constant despite any input circuit variations. The current splits between the two differential transistors. As the base inputs to Q_1 or Q_2 vary, the emitter current divides between the two transistors in proportion to the signal applied. The sum of the currents through Q_1 and Q_2 remains constant and equal to the emitter current.

The approximate single-ended gain A_s of a differential amplifier can be computed with the formula

$$A_s = \frac{R_c}{2.5 R_e}$$

This gain is the ratio of the input voltage connected to one of the base inputs and the output voltage taken from either collector to ground. R_c is the value of the collector resistor, and R_e is $25 \text{ mV}/(I_e/2)$, where $I_e/2$ is the current through each differential transistor.

For example, if $R_c = 10 \text{ k}\Omega$ and $I_e/2 = 1.2 \text{ mA}$,

$$R_e = \frac{25 \times 10^{-3}}{1.2 \times 10^{-3}} = 20.83$$

Therefore, the gain is

$$\begin{aligned} A_s &= \frac{R_c}{2.5 R_e} = \frac{10,000}{2.5(20.83)} \\ &= \frac{10,000}{52.1} = 192 \end{aligned}$$

When the gain formula is rearranged algebraically in terms of I_e , it becomes

$$A_s = \frac{R_c I_e}{125}$$

where I_e is the current from the current source, in milliamperes.

The circuit gain is directly proportional to the emitter current and collector resistance. Under ordinary conditions, both I_e and R_c are fixed values, thereby providing a fixed circuit gain. In amplitude modulation, however, modifications are made to the circuit to cause the emitter current to vary. That is done by applying the modulating or information signal to the constant-current source Q_3 , which causes the emitter current to vary linearly and thereby change the emitter current. Changing the emitter current, in turn, changes the circuit gain. By using the differential amplifier to amplify a higher-frequency carrier signal, modulation is achieved. As the modulating signal varies the circuit gain, the output amplitude of the carrier varies proportionally.

Figure 4-5 shows an amplitude modulator made with a differential amplifier. The circuit is connected in a single-ended mode; the carrier is applied to the base of Q_1 through capacitor C_1 ; and the base of Q_2 is connected directly to ground.

The constant-current source Q_3 is biased into conduction by resistors R_1 through R_3 , and the modulating signal is applied to the base of Q_3 through capacitor C_2 . The input transistor Q_1 is connected as a follower and has no collector load resistor. Q_2 has a load resistor and serves as the amplifying stage. The output is taken from across the collector load resistor R_5 .

As the modulating signal varies, the amplitude of the carrier will vary at the output. The differential amplifier serves as a modulator generating the sidebands and carrier at the output. An appropriate filter or tuned circuit at the output is required to eliminate the low-frequency modulating signal, which also appears in the output. The output network can be a simple high-pass filter as shown in Fig. 4-6(a) or (b) or a tuned circuit as shown in Fig. 4-6(c) and (d). The output signal consists of the carrier and sidebands.

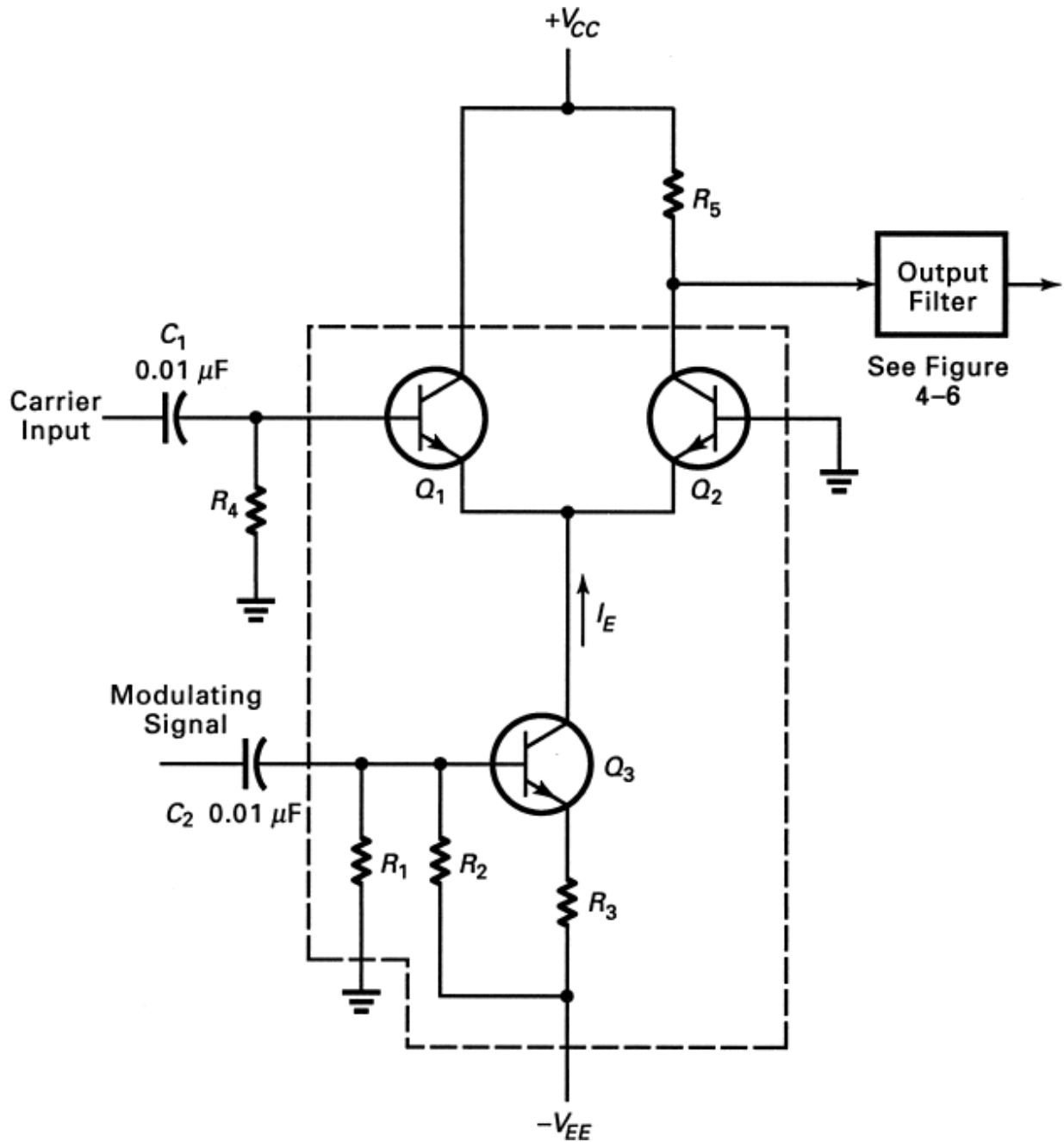


Fig. 4-5 Differential amplifier amplitude modulator.

It is the amplitude of the modulating signal that determines the percent of modulation, and the signal is increased until 100 percent modulation is obtained. The negative-going portion of the modulating signal is capable of reducing the gain of the differential amplifier to zero; the positive-going portions can double the circuit gain. Therefore, the circuit is capable of producing 100 percent modulation.

Excessive modulating signal input will cause clipping and distortion as in other amplitude modulators.

In this experiment, you will demonstrate the use of a differential amplifier as an amplitude modulator. Such a circuit is widely used; for example, one version of it produces AM in the 2206 IC function generator you used in the preceding experiment.

Procedure

1. Wire the differential amplifier amplitude modulator circuit shown in Fig. 4-7. Q_1 – Q_3 can be any common NPN transistor. The carrier is supplied by the 2206 function generator, and the modulating signal is supplied by an external function generator. Initially, do not connect the 0.1- μ F capacitor and the tuned circuit to the collector of Q_3 .

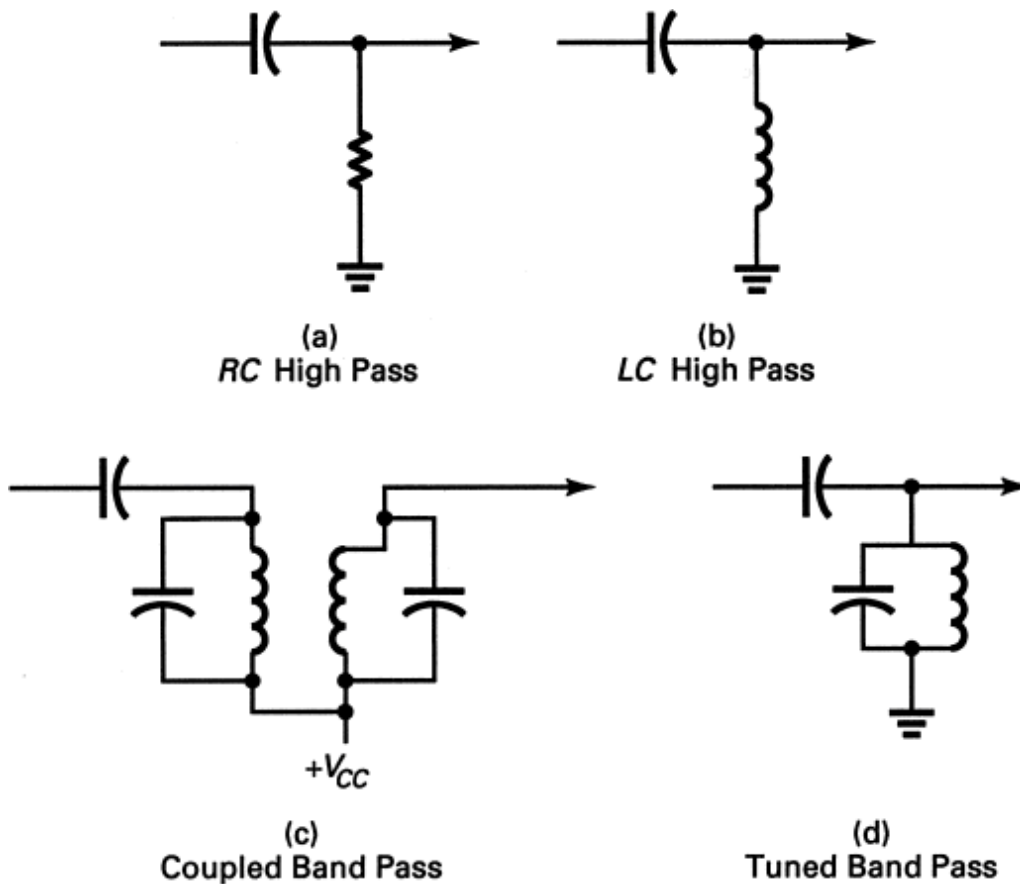


Fig. 4-6 Output filters for differential amplifier amplitude modulator.

2. Reduce the output of the external function generator supplying the modulating signal to zero. With an oscilloscope, check at the base of Q_3 to ensure that no ac signal is present.
3. Observe the carrier at the output of the 741 op amp follower. Using the 100-k Ω pot, adjust the frequency of the 2206 to 12 kHz. Apply approximately 100 mV of carrier signal to the modulator at the base of Q_1 by adjusting pot R_4 . Observe the differential amplifier output at the collector of Q_2 . It should be an undistorted sine wave; if it is not, reduce the input to Q_1 with R_4 until any distortion is eliminated.
4. Connect the 0.1- μ F capacitor and tuned circuit to the collector of Q_3 as Fig. 4-7 shows.
5. Now connect the oscilloscope across the tuned output circuit. Vary the 100-k Ω pot on the 2206 function generator to set the carrier frequency to the resonant frequency. Tune for maximum output voltage.
6. Given the values in Fig. 4-7, calculate the resonant frequency f_r . Measure the carrier frequency with a frequency counter.
7. Set the modulating signal frequency to 300 Hz. While observing the carrier output across the tuned circuit, begin increasing the amplitude of the modulating signal from the external function generator. Note the effect on the output signal. You should see a clean amplitude-modulated wave.
8. Adjust the output waveform for 100 percent modulation.
9. Continue to increase the amplitude of the modulating signal to determine if it is possible to overmodulate the differential amplifier and produce distortion and clipping. Is it?

Questions

1. In a differential amplifier, AM is achieved by varying what characteristic of the circuit with the modulating signal?
 - a. Bandwidth
 - b. Gain
 - c. Resonant frequency
 - d. Distortion

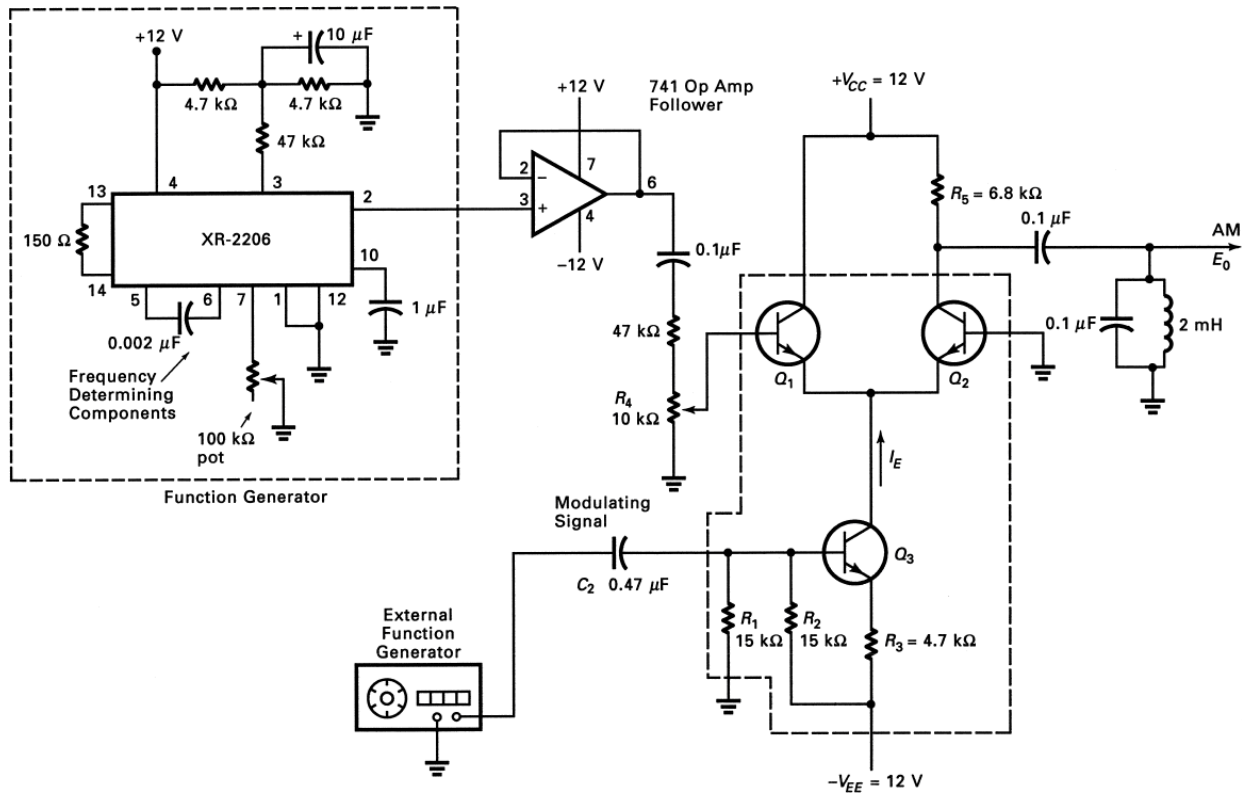


Fig. 4-7 AM with a differential amplifier.

2. The modulating signal causes what parameter in the differential amplifier to vary?
 - a. Base voltage of input transistor
 - b. Emitter resistance of current source
 - c. Collector voltage of Q_1
 - d. Emitter current of the differential pair
3. The output tuned circuit is needed to _____.
 - a. Increase gain
 - b. Suppress the carrier
 - c. Eliminate the modulating signal
 - d. Eliminate the sidebands
4. (True or False) One-hundred percent modulation can be achieved with the differential amplifier modulator.
5. Assuming I_e is constant, what component in Fig. 4-7 would you change to increase the gain of the circuit?
 - a. R_5
 - b. R_2

- c. R_4
- d. R_3

Project 4-3 Diode Detector

Objective

To construct a diode detector circuit and demonstrate its use in demodulating an AM signal.

Required

In addition to the equipment and components you used in Project 3-1, you will need the following:

- 1 741 IC op amp
- 1 silicon signal diode (1N914, 1N4148, 1N4149, etc.)
- 1 470-pF capacitor
- 1 0.005- μ F capacitor
- 2 0.1- μ F capacitors
- 1 150- Ω resistor
- 1 1-k Ω resistor
- 1 10-k Ω resistor
- 2 100-k Ω resistors
- 1 10-k Ω pot

Introduction

The process of recovering the original information signal from an AM waveform is known as demodulation. A variety of circuits can be used to demodulate an AM signal, but the one that is simplest and most widely used is the diode detector. A typical circuit is shown in Fig. 4-8. The rectifier diode D_1 removes one-half of the AM signal, and the capacitor C_2 across the load R_2 acts as a low-pass filter to remove the carrier, leaving only the original modulating signal. The waveforms shown in Fig. 4-9 illustrate this process.

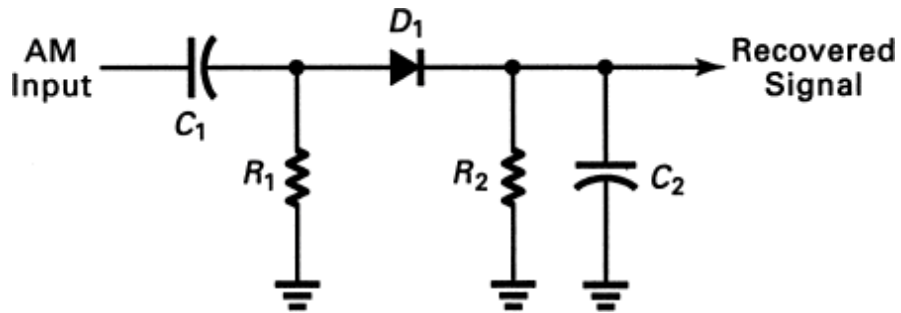


Fig. 4-8 Diode AM detector circuit.

The value of capacitor C_2 is critical in that it determines the degree to which the carrier is filtered out. The capacitor must be large enough to filter the carrier out, but if it is too large, it will cause distortion of the recovered signal. Thus, a compromise value is selected.

In this experiment, you will construct a diode detector circuit, show how it demodulates an AM signal, and demonstrate the effect of different values of filter capacitance on the signal waveshape.

Procedure

1. The modulator circuit you built in Project 3-1 will be used as the AM signal source in this experiment. Rewire the circuit to conform to Fig. 4-10. The modulating signal will come from an external function generator; the 741 op amp amplifies the AM signal to a level suitable for detection; and the 10-k Ω pot is used to adjust the carrier level.

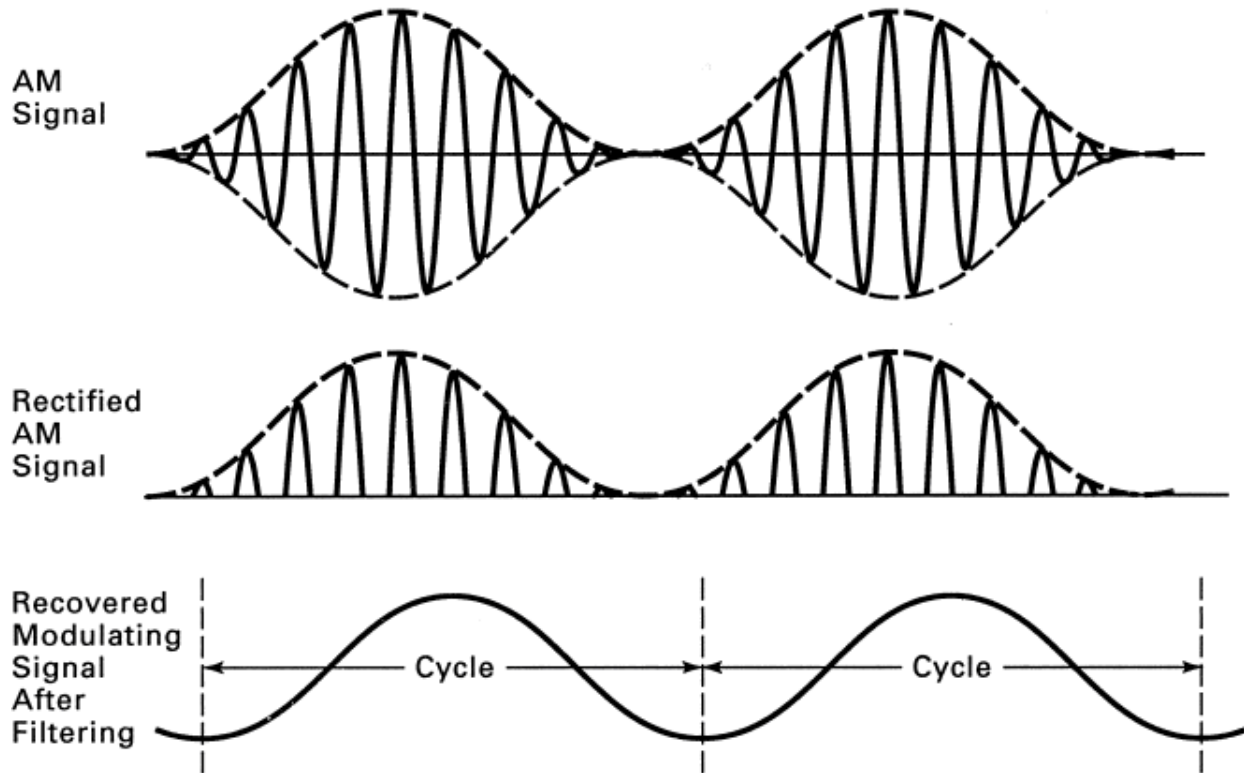


Fig. 4-9 Demodulation of an AM signal.

2. Next, construct the diode detector circuit shown in Fig. 4-11. Note that a filter capacitor is not connected at this time.
3. Apply power to the circuit. Reduce the modulating signal amplitude to zero. While observing the carrier output of the 741 op amp, adjust the 100-k Ω pot on the 2206 for an output frequency of 30 kHz. Then adjust the 10-k Ω pot for an amplitude of 1.5 V_{p-p}. Set the modulating signal frequency to 200 Hz, and then increase the modulating signal amplitude while observing the op amp output until 100 percent AM is obtained.
4. Observe the diode detector output across the 100-k Ω load resistor. Reverse the diode polarity and note the output waveform. Again reverse the diode connections.
5. Connect a 0.005- μ F capacitor across the diode detector load as shown in Fig. 4-12. Observe the diode detector output and sketch the output waveform. If the signal is distorted, reduce the modulating signal amplitude until the distortion is minimized.

6. Connect the 470-pF and the 0.1- μ F capacitors, one at a time, in place of the 0.005- μ F capacitor as filters in the diode detector circuit. Note the resulting output waveforms.
7. Considering the waveforms you observed in steps 5 and 6, explain how small and large values of capacitance affect the output signal.
8. Reconnect the 0.005- μ F capacitor across the 100-k Ω load, and then reverse the connections to the diode. Monitor the output waveform.
9. Compare your results to the result obtained in step 5. Explain the reason for the waveform you observed in step 8.
10. With the oscilloscope, monitor the AM waveform input to the diode detector. Increase the amplitude of the modulating signal to produce overmodulation and clipping.
11. Observe the recovered waveform. Explain the results you obtained.
12. While observing the recovered signal at the output of the diode detector, vary the modulating signal amplitude from zero until distortion occurs. Explain what effect the percent of modulation has on the output signal.

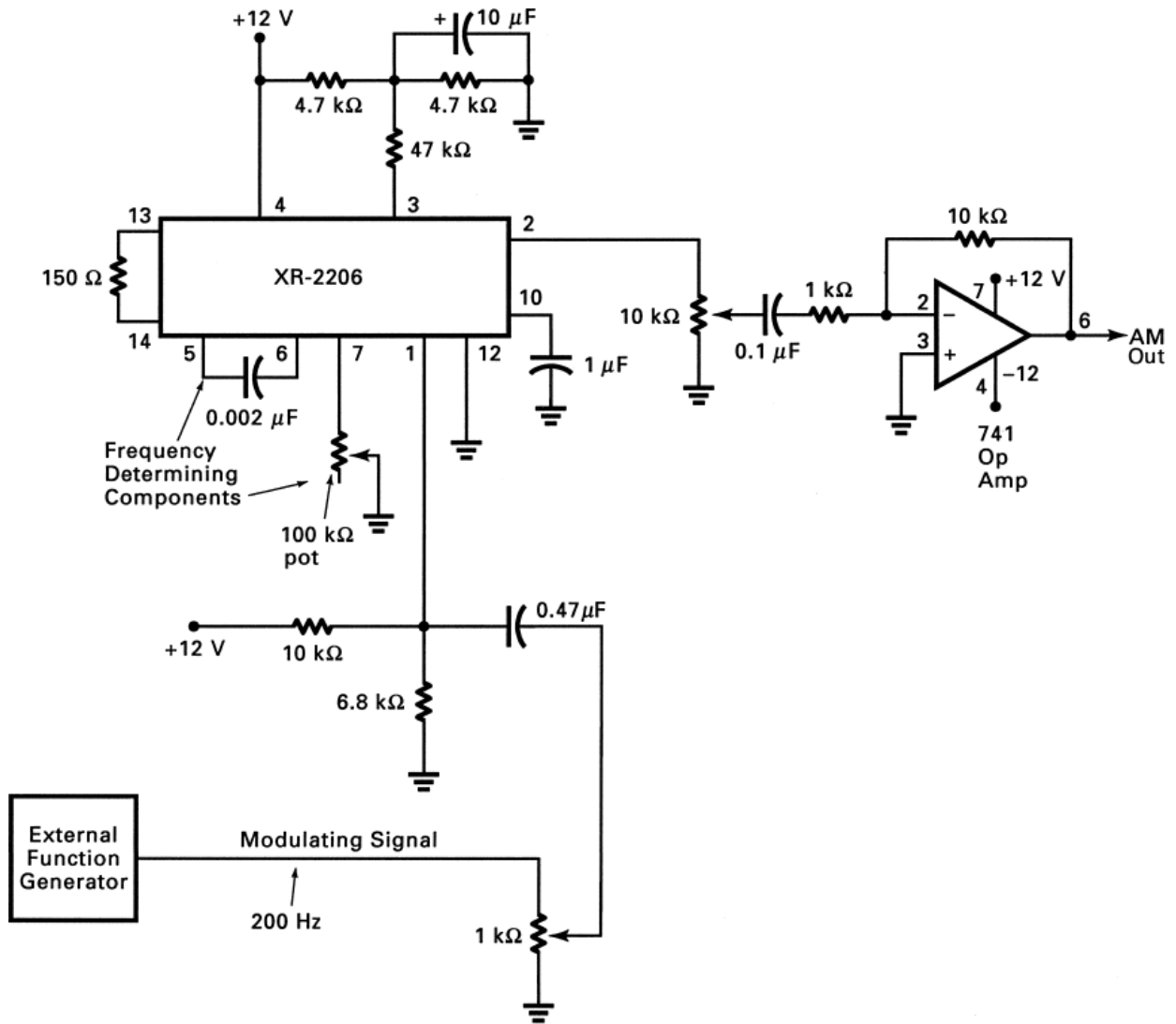


Fig. 4-10 Amplitude modulation of the 2206 function generator IC.

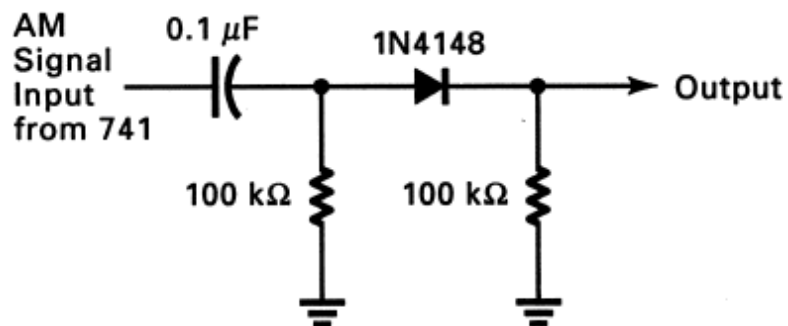


Fig. 4-11 Diode detector without filter capacitor.

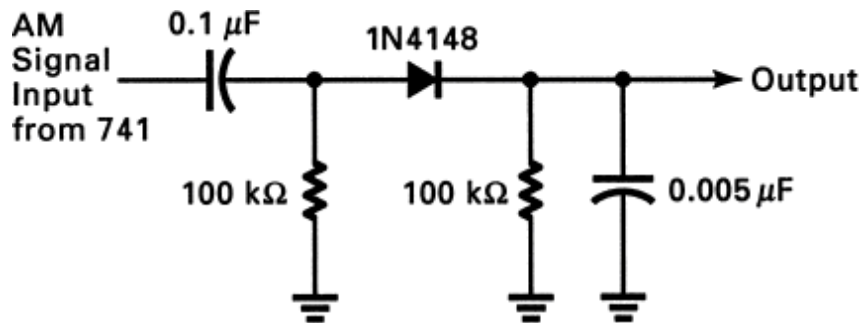


Fig. 4-12 Diode detector with filter.

Questions

1. If the filter capacitor in a diode detector is too small, the _____ .
 - a. Carrier is not removed
 - b. Recovered signal is distorted
 - c. Carrier is distorted
 - d. Signal amplitude is reduced
2. If the filter capacitor in a diode detector is too large, the _____ .
 - a. Carrier is not filtered out
 - b. Recovered signal is distorted
 - c. Carrier is distorted
 - d. Signal amplitude is reduced
3. Decreasing the percent of modulation causes the demodulated signal amplitude to _____.
 - a. Increase
 - b. Decrease
 - c. Remain the same
 - d. Drop to zero
4. (True or False) Distortion produced by overmodulation is filtered out in the diode detector.
5. Reversing the polarity of the detector diode causes the recovered signal to _____.
 - a. Be inverted
 - b. Be lowered in amplitude
 - c. Be increased in amplitude
 - d. Remain the same

Project 4-4 Double-Sideband Balanced Modulator

Objective

To demonstrate the operation of a balanced modulator in producing a double-sideband signal.

Required

In addition to the 2206 function generator circuit that you built previously and all of the test instruments used earlier, you will need the following components:

- 1 741 IC op amp
- 1 1496 balanced modulator IC
- 2 100- Ω resistors
- 1 150- Ω resistor
- 1 560- Ω resistor
- 3 1-k Ω resistors
- 2 3.3-k Ω resistors
- 2 4.7-k Ω resistors
- 1 6.8-k Ω resistor
- 2 10-k Ω resistors
- 2 0.1- μ F capacitors
- 1 100- μ F electrolytic capacitor
- 1 10-k Ω pot
- 1 50-k Ω pot

Introduction

Double-sideband (DSB) modulation refers to amplitude modulation but with the carrier removed. The modulator suppressed the carrier but produces full upper and lower sidebands as may be created by the modulating signal. Single-sideband (SSB) operation refers to amplitude modulation with the carrier suppressed but with one sideband removed. All of the intelligence is carried in either the upper or the lower sideband. SSB operation has the advantage of requiring one-half the spectrum space required by a normal AM or DSB signal.

The key circuit in generating a DSB or SSB signal is the balanced modulator. This circuit produces AM but suppresses the carrier, leaving only the two sidebands in the output. For SSB operation, a highly selective filter is normally used to eliminate one of the sidebands.

A variety of different balanced modulator circuits are used to suppress the carrier. One of the most effective and widely used is the differential amplifier balanced modulator. One such popular integrated circuit is the 1496. It can produce carrier suppression up to 65 dB and has a useful gain up to 100 MHz. It uses specially connected differential amplifiers to cancel the carrier.

In this experiment, you will demonstrate the use of the 1496 to produce a DSB signal. You will also show how this IC can be used to produce conventional amplitude modulation.

Procedure

1. Construct the circuit shown in Fig. 4-13. The 2206 integrated circuit function generator will serve as the carrier source. You will use a 741 op amp inverter to buffer the output signal from the function generator. The output of the 741 op amp becomes the carrier input to the 1496. The modulating signal is supplied by an external function generator. Be particularly careful in constructing this circuit, because there are many components and wiring mistakes are easy to make. Once you have built the circuit, double-check your wiring before applying power.

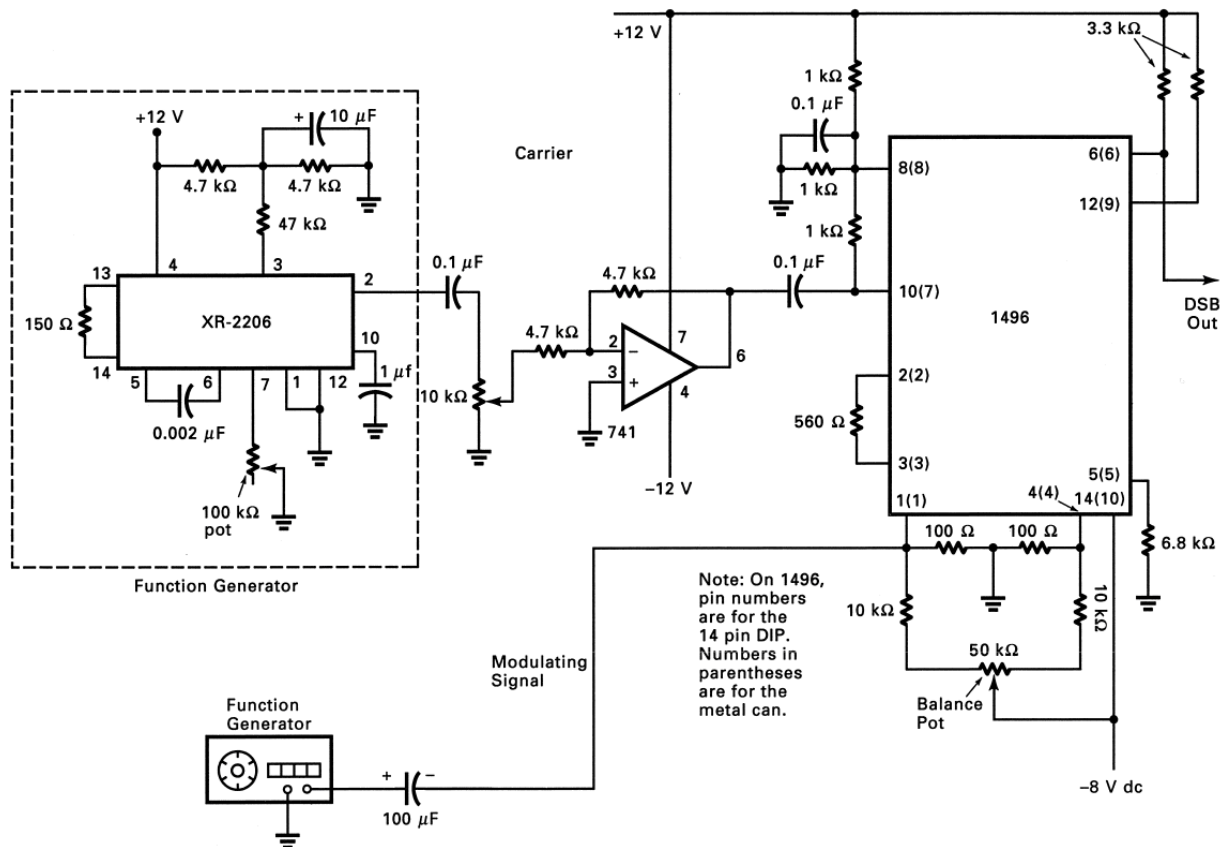


Fig. 4-13 Circuit for demonstrating SSB.

2. While monitoring the carrier signal at the op amp output on pin 6, adjust the 10-kΩ pot for a carrier amplitude of $2 V_{p-p}$. By using the 50-kΩ pot on the 2206, set the carrier frequency to 20 kHz, and then adjust the modulating signal input from the external function generator for a frequency of 400 Hz with an amplitude of 0.5 V.
3. Connect the oscilloscope to pin 6 of the 1496 to observe the double-sideband output signal. The classical DSB signal pattern should appear. Adjust the 50-kΩ balance pot on the 1496 so that the arm is approximately in the center of its rotational range. Then fine-tune the pot adjustment until the positive and negative peaks of the DSB waveform are equal.
4. Readjust the 100-kΩ pot on the 2206 function generator to set the carrier frequency to approximately 6 kHz. The adjustment should not change the shape of the DSB output signal, but you should be able to see the carrier cycles more clearly. Next, by

using the scope horizontal and vertical gain and triggering controls, expand the DSB signal so you can clearly see the portion of the signal where it goes to zero or reaches a null point between the peaks. You should be able to stabilize the waveform for proper viewing by making slight frequency adjustments with the 50-k Ω carrier-frequency pot. In that way, you will be able to clearly see the phase shift that is characteristic of the DSB signal. (Refer to Figs. 3-15 and 4-27 in the text.) This is one way to clearly identify a DSB signal from a conventional AM signal.

5. Readjust the carrier frequency to approximately 20 kHz and again display the DSB signal, and then rotate the balance pot in one direction or the other until a conventional AM signal is obtained. If the AM signal does not appear when you are adjusting the pot in one direction, adjust the pot in the opposite direction until it does. You may have to reduce the amplitude of the modulating signal from the external function generator as well to make the AM signal appear. Then, by adjusting the modulating signal input level, you should be able to obtain any percent of modulating desired. By varying the 100-k Ω pot, you have unbalanced the circuit and thereby allowed the carrier to be inserted. With the pot in the center of its rotation, the internal IC circuitry is carefully balanced so the carrier is suppressed. In the unbalanced condition, the carrier passes through, causing traditional amplitude modulation to occur.

Questions

1. The circuit that produces amplitude modulation but suppresses the carrier is known as a(n) _____.
 - a. Op amp
 - b. Function generator
 - c. Balanced modulator
 - d. Differential amplifier
2. A DSB signal is made up of the sum of the _____.
 - a. Upper and lower sidebands
 - b. Upper and lower sidebands and the carrier
 - c. Carrier and the upper sideband
 - d. Carrier and the lower sideband

3. (True or False) If properly adjusted, an IC balanced modulator can also be used to produce AM.

NAME

DATE

A graphic for Chapter 5 featuring the text 'Chapter 5' in a white, sans-serif font. The text is centered within a dark, irregular, cloud-like shape that has a gradient from black to grey. The number '5' is significantly larger than the word 'Chapter' and is positioned directly below it.

Chapter

5

Fundamentals of Frequency Modulation

Project 5-1 Frequency Modulation

Objective

To demonstrate frequency modulation with a voltage-controlled oscillator.

Required

You will need the 2206 function generator circuit you constructed previously plus an oscilloscope, a frequency counter, and an external function generator. You will also need a 10- μ F capacitor and a 100-k Ω resistor.

Introduction

In frequency modulation, the modulating signal varies the frequency of the carrier. The carrier amplitude remains constant, but the increasing and decreasing amplitude of the modulating signal causes the carrier to deviate from its center frequency. The amount of deviation is a function of the amplitude of the modulating signal. The

rate of deviation is proportional to the frequency of the modulating signal.

There are many different ways to produce frequency modulation. Today, one of the easiest ways is simply to apply a modulating signal to a voltage-controlled oscillator (VCO), and numerous IC VCOs are available. The 2206 function generator IC you have used in past experiments uses a VCO as the primary signal-generating circuit. This oscillator can easily be frequency-modulated simply by applying the appropriate modulating signal. In this experiment, you will demonstrate frequency modulation with the 2206.

Procedure

1. The circuit for this experiment is shown in Fig. 5-1. This is the same 2206 function generator you have used previously, but note that you will add a 10- μ F capacitor and a 100-k Ω resistor to pin 7. To the capacitor you will connect an external function generator to serve as the modulating signal.
2. Apply power to the circuit. Reduce the modulating signal amplitude from the external function generator to zero, and then observe the carrier output at pin 2 of the 2206. Adjust the 100-k Ω pot on the 2206 for a frequency of approximately 30 kHz. Set the horizontal sweep controls on the oscilloscope to display approximately three cycles of the carrier wave.
3. Slowly increase the amplitude of the modulating signal from the external function generator and set the frequency of the signal to approximately 200 Hz. As you increase the modulating signal amplitude, note the display on the oscilloscope. You should see the carrier wave begin to “vibrate,” which indicates a change in frequency with the modulating signal. You should observe a waveform that looks approximately like the one shown in Fig. 5-2. Continue slowly increasing the modulating signal amplitude and note the effect on the carrier wave. If you have difficulty obtaining a stable display, use the triggered sweep function on the oscilloscope. Manipulate the trigger and horizontal frequency controls until a stable waveform is obtained. What you are seeing is the carrier frequency being instantaneously changed by the modulating signal. While varying the amplitude of the modulating

signal and observing the resulting FM output, determine the relationship between the frequency deviation and the modulating signal amplitude.

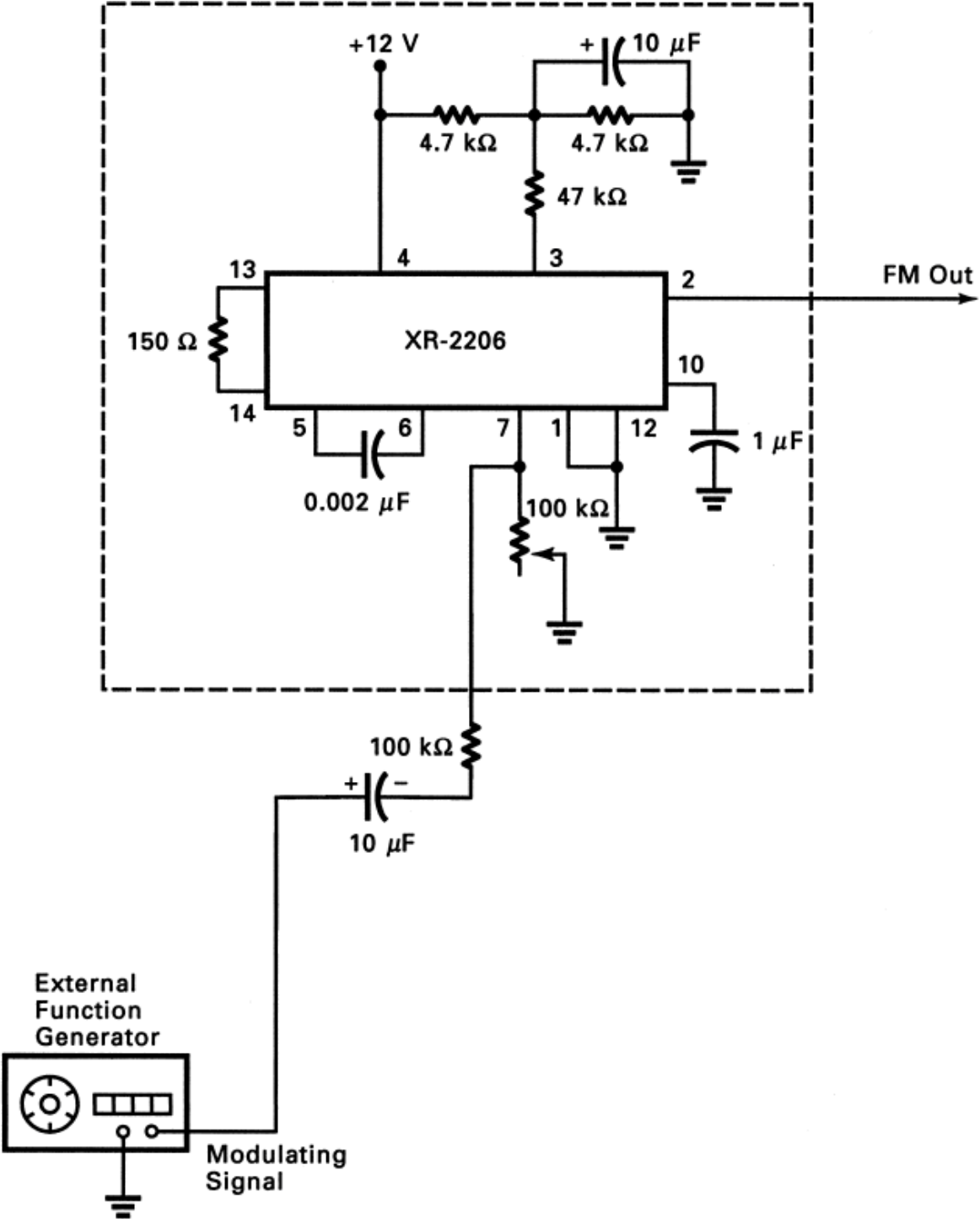


Fig. 5-1 Frequency modulation of the 2206.

4. Set the amplitude of the modulating signal at the 10- μ F capacitor to 2 V_{p-p}, and then observe the carrier output on the oscilloscope. You will see the waveform shown in Fig. 5-2, indicating frequency deviation. Now, while observing the waveform, vary the frequency of the modulating signal. Note the effect on the waveform. How does the deviation change with respect to the modulating signal frequency?

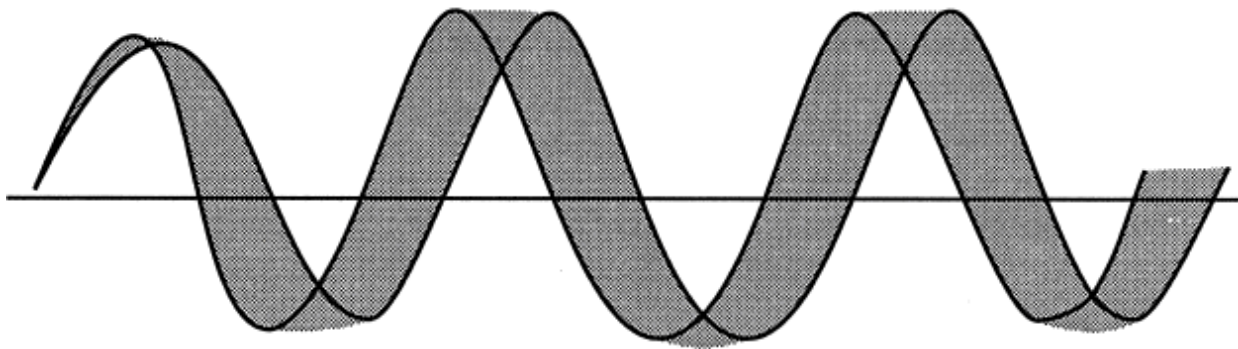


Fig. 5-2 How FM looks on an oscilloscope.

Questions

1. Which of the following is the most correct statement with regard to the relationship between the carrier frequency deviation and the amplitude of the modulating signal?
 - a. Decreasing the modulating signal amplitude increases the carrier deviation.
 - b. Decreasing the modulating signal amplitude decreases carrier deviation.
 - c. Carrier deviation is not affected by the amplitude of the modulating signal.
 - d. The frequency deviation varies in direct proportion to the modulating signal frequency.
2. Varying the frequency of the modulating signal causes the frequency deviation of the carrier to _____ .
 - a. Increase
 - b. Decrease

- c. Remain the same
 - d. Drop to zero
3. The type of modulation produced by the VCO in the 2206 IC is _____ .
- a. Frequency modulation
 - b. Phase modulation
 - c. Indirect FM
 - d. FSK
4. What is the deviation ratio of an FM system in which the maximum permitted frequency deviation is 10 kHz and the maximum modulating frequency is 3 kHz?
- a. 0.3
 - b. 1
 - c. 3
 - d. 3.33
5. The number of sidebands produced by a sine-wave carrier being frequency-modulated by a single-frequency sine-wave tone is _____ .
- a. 1
 - b. 2
 - c. 4
 - d. Infinite

Project 5-2 FM Radio Experience

Objective

To tune and listen to radio stations using frequency modulation.

Required

- 1 FM broadcast radio
- 1 receiver capable of receiving two-way FM communications in the 140- to 170-MHz and 420- to 470-MHz ranges. (A common scanner radio with a frequency range of 50 to 1300 MHz is the best way to listen to all of the signals in this range.)
- 2 family radio units
 - Antennas as required

Introduction

Most two-way radio communications use FM. These include police, fire, taxi, public service, marine, and many others. Most of this communication takes place in the VHF range from 150 to 170 MHz. The 420- to 470-MHz frequency range is also widely used. You can hear most of the conversations in these ranges using a scanner receiver. In this experiment, you will experiment with a scanner receiver and use it to listen to as many different types of FM communications as possible.

Procedure

1. Read the instruction manual for the scanner receiver.
2. State the frequency range of the FM broadcast band and the frequency spacing of stations.
3. Use an FM radio to listen to the broadcast band. Tune the entire extent of the band and count the number of stations you hear and record their frequency. Describe what you hear.
4. Set the scanner receiver to receive stations in the 152- to 155-MHz range. You should hear a variety of signals. Describe what you hear, including the source of the transmissions.
5. Try out the squelch control on the receiver.
6. Monitor the channels in the 156- to 158-MHz range. Describe the type of stations you hear, if any.
7. Listen to stations in 162- to 163-MHz range. What do you hear?
8. Set the scanner to listen to stations in the 146- to 148-MHz range. Again, describe what you hear.
9. Set the scanner to scan the 450- to 470-MHz range. Describe some of the conversations you hear. What are the sources?
10. Scan the 46- to 47-MHz and 49- to 50-MHz ranges. What do you hear, if anything? What should you hear?
11. Try out the family radio units. Can you change frequencies? Explain. List all controls on the unit.
12. Experiment with the maximum communications range, both indoors and outdoors. Record your results and explain.

Question

1. What is the main advantage of FM over AM for most communications applications?

NAME

DATE

Chapter

6

FM Circuits

Project 6-1 Phase-Locked Loop Operation

Objective

To demonstrate the operation and determine the lock and capture ranges of a phase-locked loop (PLL).

Required

- 1 oscilloscope (dual-trace)
- 1 frequency counter
- 1 function generator (sine wave)
- 1 power supply (± 12 V dc)
- 1 565 IC PLL or an equivalent PLL IC
- 1 0.001- μ F capacitor
- 1 0.01- μ F capacitor
- 1 0.1- μ F capacitor
- 1 10- μ F electrolytic
- 2 1-k Ω resistors
- 1 2.7-k Ω resistor

Introduction

A PLL is an electronic feedback-control system used in a variety of applications. It consists of an error detector, a loop filter, and a voltage-controlled oscillator (VCO). See Fig. 6-1.

A phase-locked loop will track or follow the frequency of an input signal. The input signal frequency is compared to the VCO frequency in the phase detector, and an error signal is generated. This error signal from the phase detector is filtered into a dc voltage that controls the VCO. Changes in the input frequency develop an error signal that will cause the VCO's output frequency to change in such a direction as to follow the input signal frequency exactly. Frequency changes of the input are interpreted as phase changes that cause the VCO to readjust. The VCO output frequency is equal to the PLL input frequency, but there is a phase shift between the input and VCO signals. It is this phase shift that generates the error signal and the dc voltage that keeps the phase-locked loop locked. In this experiment, you will demonstrate PLL operation by using the popular 565 IC phase-locked loop.

In the 565 integrated circuit phase-locked loop, the VCO is of the multivibrator type and produces both triangular and square-wave outputs. The frequency is determined by external values of resistance and capacitance. The approximate free-running frequency is computed with the formula

$$f_0 = \frac{1.2}{4R_1C_1}$$

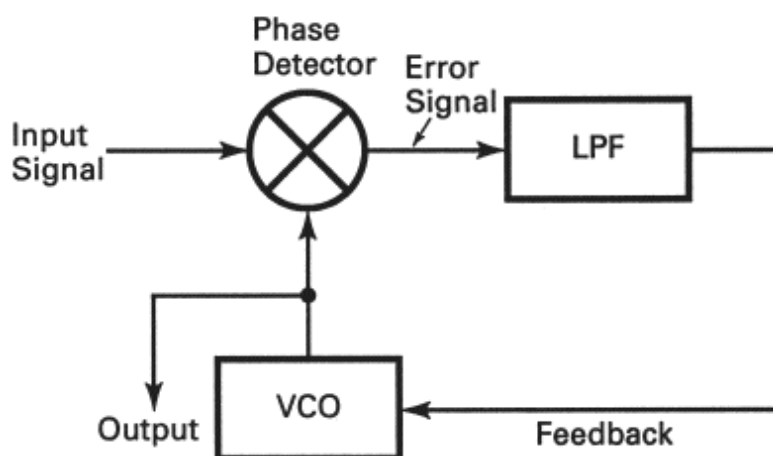


Fig. 6-1 Phase-locked loop.

Refer to Fig. 6-2, which shows the 565 and the related components.

The frequency range over which the VCO is capable of tracking the input signal is known as the lock range. It is primarily a function of the range of the VCO, and it is centered around the free-running frequency of the VCO. For the 565 PLL, it is computed with the formula

$$f_L = \frac{\pm 8f_0}{V_s}$$

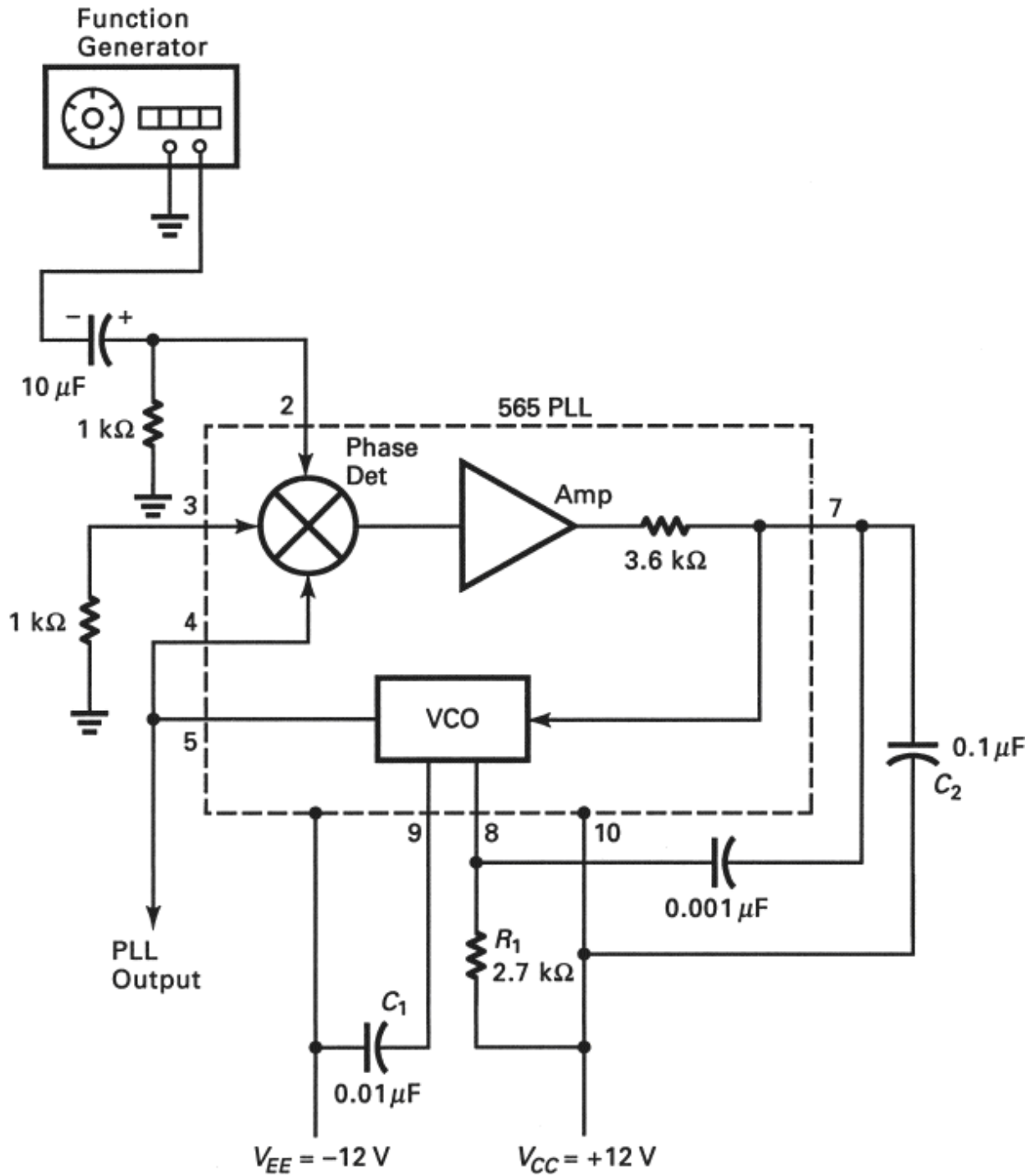


Fig. 6-2 Phase-locked loop circuit.

where f_L is one-half the lock range, f_0 is the free-running VCO frequency, and V_s is the total supply voltage. In this circuit, there is a $+V_{CC}$ supply and a $-V_{EE}$ supply for a total of $12 + 12 = 24\text{ V}$. The total lock range is $2f_L$.

The capture range is the range of frequencies over which page 47 the PLL will lock onto an input signal. When an input signal is applied to the VCO, it must be within the capture range in order for the circuitry to “capture” the signal. A signal whose frequency is outside the capture range will simply cause the PLL to remain unlocked, and the VCO will operate at its free-running frequency. The capture range is usually narrower than the lock range, which causes the PLL to act like a very selective band-pass filter.

In this experiment, you will demonstrate PLL operation and measure the phase shift between the input signal and the VCO during the locked state. You will also demonstrate how the VCO output tracks a variable-frequency input. Finally, you will calculate and measure the free-running frequency and the lock range of the 565 PLL.

Procedure

1. Connect the circuit shown in Fig. 6-2. An external function generator with variable-frequency capability is connected as the PLL input. Reduce the amplitude to zero for now.
2. Use the formula for the free-running frequency of the VCO, given earlier, to calculate that frequency by using the values of R_1 and C_1 in Fig. 6-2.
3. Apply power to the circuit shown in Fig. 6-2. By using a frequency counter, measure the output frequency of the VCO at pin 5 on the 565 IC. Observe the output signal with an oscilloscope.
4. Compare your computed and measured values of free-running frequency and explain any difference.
5. Adjust the function generator input to the PLL input for an amplitude of 1 V each peak to peak of sine wave. Set the external function generator for a frequency approximately equal to the free-running frequency of the VCO.
6. Use an oscilloscope and/or frequency counter to measure the function generator output frequency and the VCO output frequency at pin 5. Are the two frequencies the same? Why or why not?

7. Use a dual-trace oscilloscope to display the function generator and VCO output signals simultaneously. Measure the amount of phase shift between the two. You can do that by measuring the time shift t between corresponding parts of the two waveforms and then using the formula

$$\text{Phase shift, in degrees} = \frac{360t}{T}$$

where T is the period of the signals.

8. While continuing to observe the function generator and VCO output signals on the dual-trace oscilloscope, vary the function generator frequency above and below the free-running value. Note the effect on the VCO output. Describe the relationship between the function generator input and the VCO output signal frequencies.
9. Use the formula given earlier to compute the lock range for the PLL. Record your value.
10. Assuming that the lock range is centered on the free-running frequency of the VCO, calculate the upper and lower lock range limits. Is the difference between the upper and lower lock values equal to the lock range?
11. By using the circuit shown in Fig. 6-2, you will now measure the lock range. First, however, set the function generator input frequency to the PLL free-running frequency to ensure initial lock.
12. Begin decreasing the function generator input frequency page 48 while observing the VCO output. At some point, you will notice a frequency variation or jitter. Stop decreasing the frequency exactly at that point and measure the function generator frequency. It is the lower lock limit of the PLL. Record it.
13. Increase the input frequency while observing the VCO output. Lock will occur. Keep increasing the frequency. Again you will reach a point where the VCO output begins to jitter. Reduce the input frequency to just below the point at which the jitter ceases. Then measure the function generator output frequency. This is the upper lock limit of the PLL. Record it.

14. By using the experimental data you collected in steps 11 through 13, calculate and record the measured lock range. How does it compare to your calculated value?

Questions

- Varying the PLL input frequency causes the PLL output to _____.
 - Track the input
 - Remain constant at the free-running frequency
 - Vary inversely
 - Drop to zero
- The error signal is generated by what characteristic of the input and VCO signals?
 - Frequency
 - Phase shift
 - Amplitude difference
 - Rise time
- What is the relationship between the capture f_c and lock f_L ranges of the PLL?
 - $f_c = f_L$
 - $f_c > f_L$
 - $f_c < f_L$
 - $f_c = 2f_L$
- To increase the free-running or center frequency of the PLL, what changes should be made in R_1 and/or C_1 ?
 - Increase R_1
 - Increase C_1
 - Decrease R_1
 - Decrease C_1
 - Both **a** and **b**
 - Both **c** and **d**
- If the input frequency to the PLL is outside the capture and lock ranges, the VCO output is the _____.
 - Upper lock frequency
 - Free-running frequency
 - Lower lock frequency

Project 6-2 Frequency Demodulation with a Phase-Locked Loop

Objective

To demonstrate the operation of a phase-locked loop as a demodulator for FM signals.

Required

In addition to an oscilloscope, frequency counter, function generator, and power supply, you will need the 2206 FM modulator you built in Project 5-1 and the 565 PLL circuit you used in Project 6-1. You will also need a 0.001- μ F capacitor and a 10-k Ω pot.

Introduction

Perhaps the best demodulator circuit for FM is the phase-locked loop. Because of its excellent fidelity, wide range response, and noise suppression, the PLL produces a clean and faithful reproduction of the original modulating signal. Because PLLs are available in low-cost IC form, they are widely used for this purpose, particularly in critical applications.

To use the PLL as a frequency demodulator, you apply the FM signal to the phase detector input. A recovered signal is taken from the output of the loop low-pass filter (LPF). See Fig. 6-3.

As the carrier varies in accordance with the modulating signal, it produces a varying phase when compared to the VCO. The error signal developed then forces the VCO output to follow the FM input. Remember that, in order for the PLL to remain locked, the two inputs to the phase detector must be the same. The VCO output will track the FM input.

In order for the VCO to duplicate the input FM signal exactly, the signal applied to the VCO control input from the loop filter must be the same as the modulating signal. The output of the loop filter is therefore the recovered intelligence signal.

In this experiment, you will use the 565 IC PLL to demonstrate FM demodulation.

Procedure

1. The complete circuit for this experiment is shown in Fig. 6-4. It is made up of the 2206 IC frequency modulator you used in the preceding experiment, and the demodulator circuit itself is the 565 PLL you used in that experiment. Connect the output of the frequency modulator to the 565. In the PLL circuit, replace the $0.01\text{-}\mu\text{F}$ capacitor with a $0.001\text{-}\mu\text{F}$ capacitor. Replace the $2.7\text{-k}\Omega$ resistor R_1 with a $10\text{-k}\Omega$ pot.

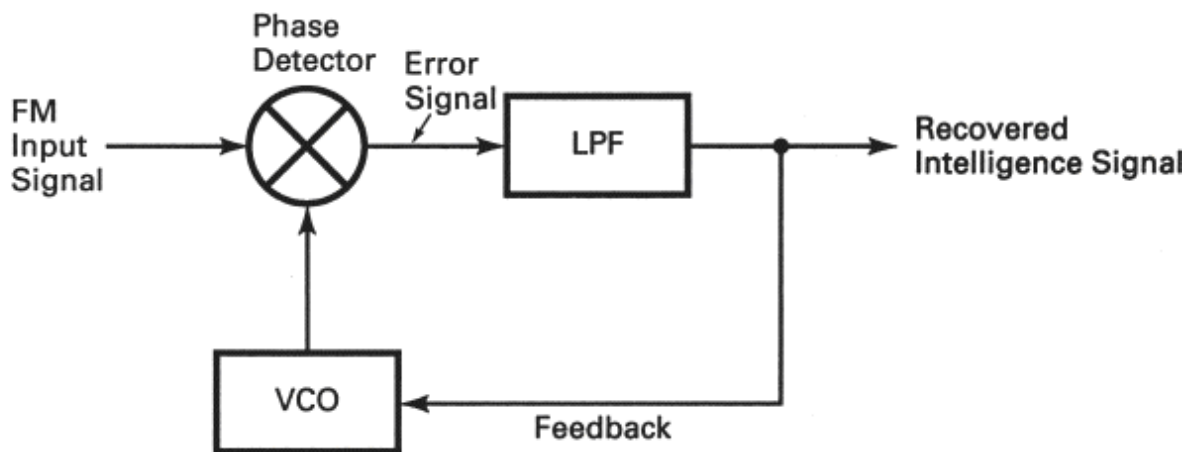


Fig. 6-3 Phase-locked loop used as an FM demodulator.

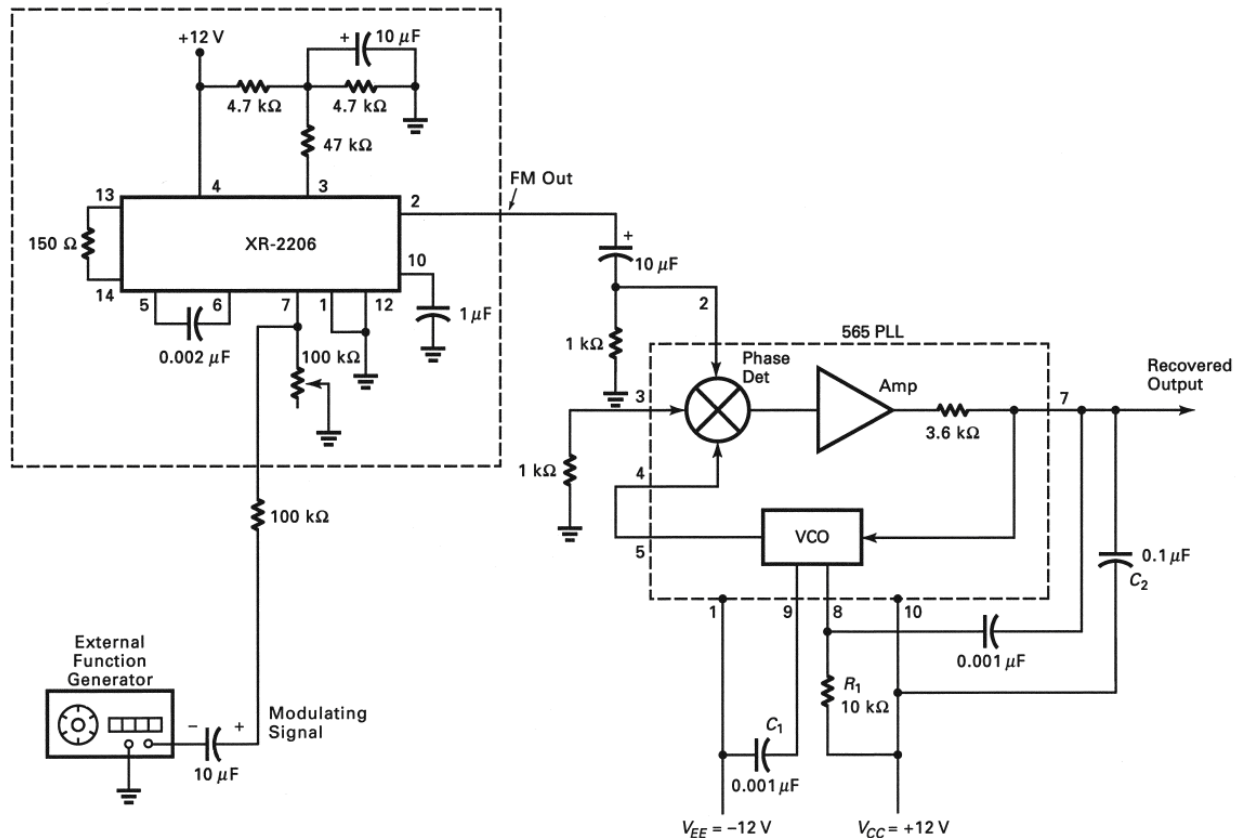


Fig. 6-4 Using a PLL for FM demodulation.

2. Connect a frequency counter and oscilloscope to the 2206 output at pin 2. Reduce the modulating input signal to zero. Adjust the 100-k Ω pot for a carrier frequency of 30 kHz. page 51
3. Connect the frequency counter and oscilloscope to the PLL output at pin 5 on the 565. Adjust the 10-k Ω pot for a frequency of 30 kHz. At this time, the 565 PLL should be locked to the 2206 output; check that by observing the 565 output at pin 5 while varying the 100-k Ω pot on the 2206. The output should track. Readjust the 100-k Ω pot for an output of 30 kHz.
4. Apply a modulating signal from the external function generator to the 2206 FM generator. Use a frequency of 200 to 400 Hz. While observing the output of the 2206 at pin 2, increase the amplitude of the modulating signal until you obtain an FM signal.
5. Observe the PLL loop filter output at pin 7 to see the recovered signal. Compare that signal to the modulating signal applied to the 2206.

6. While observing the recovered output at pin 7 on the 565 PLL, vary both the frequency and amplitude of the modulating signal. Note the effect on the recovered output.

Questions

1. The recovered output from a PLL demodulator appears at the output of the _____.
 - a. Phase detector
 - b. Loop filter
 - c. VCO
 - d. *RC* timing network
2. (True or False) The PLL VCO output signal tracks the FM input.
3. The signal controlling the VCO in the PLL is the _____.
 - a. FM signal
 - b. Carrier
 - c. Phase detector output
 - d. Original modulating signal
4. If the modulating signal is removed from the modulator circuit, the loop filter output of the PLL will be _____.
 - a. The carrier
 - b. The modulating signal
 - c. The free-running VCO frequency
 - d. Zero
5. To what frequency should the PLL VCO free-running be set when the demodulator is used in a superheterodyne receiver?
 - a. The input carrier frequency
 - b. The manufacturer's recommended frequency
 - c. The receiver intermediate frequency
 - d. Any frequency higher than the highest-frequency modulating signal

Project 6-3 Pulse-Averaging Discriminator

Objective

To demonstrate the operation of a pulse-averaging discriminator in demodulating FM signals.

Required

You will need the 2206 FM generator you used in Project 5-1 plus the oscilloscope and frequency counter. In addition, you will need the following parts:

- 1 74123 or 74LS123 dual TTL one-shot (The 74121 and 74122 TTL ICs also will work in this application)
- 1 0.01- μ F capacitor
- 2 0.02- μ F capacitors
- 1 1-k Ω resistor
- 1 4.7-k Ω resistor
- 2 15-k Ω resistors
- 2 Power supplies of +12 V and +5 V

Introduction

There are numerous types of FM demodulator circuits that use the principle of averaging pulses in a low-pass filter to recover the intelligence signal. For example, the quadrature FM detector produces output pulses of varying widths that, when averaged in a low-pass filter, produce the original modulating signal. The pulse-averaging discriminator generates fixed-amplitude, fixed-width pulses, one per cycle of the FM signal. These pulses, when averaged in a low-pass filter, reproduce the original intelligence.

In this experiment, you will demonstrate that concept by constructing a pulse-averaging discriminator made up of a one-shot multivibrator. When triggered by the FM signal, the one-shot, or monostable, produces one fixed-width pulse per cycle. The pulses are fed to a low-pass filter to reproduce the original signal.

Procedure

1. Construct the experimental circuit shown in Fig. 6-5. The output from the 2206 function generator is taken from pin 11; it is a 5-V square wave compatible with the TTL logic circuitry in the 74123 one-shot. The signal is also frequency-modulated and follows exactly the sine-wave output that appears at pin 2 on the 2206.

2. Connect the frequency counter to the 2206 output at pin 11, and then adjust the 100-k Ω pot for a center frequency of 20 kHz. While you do that, be sure to reduce the modulating signal input from the external function generator to zero.
3. Connect the oscilloscope output to pin 13 of the 74123 one-shot. You should observe a chain of pulses, and the pulse width should be approximately 15 μ s. By using the 100-k Ω pot on the 2206, vary the frequency over a narrow range above and below the 20-kHz point. Note that the frequency of the pulses changes but the pulse width remains constant.
4. Again set the 2206 function generator for a center frequency of 20 kHz with the 100-k Ω pot, and then apply the modulating signal to the 2206 from the external function generator. Set the modulating signal frequency to 200 Hz and monitor the output of the 2206 at pin 11. You should see a frequency-modulated square wave. This is the signal applied to the one-shot.
5. Connect the oscilloscope to the output of the low-pass filter. You should observe the recovered 200-Hz sine wave. Vary the amplitude of the modulating signal and note the effect on the output.

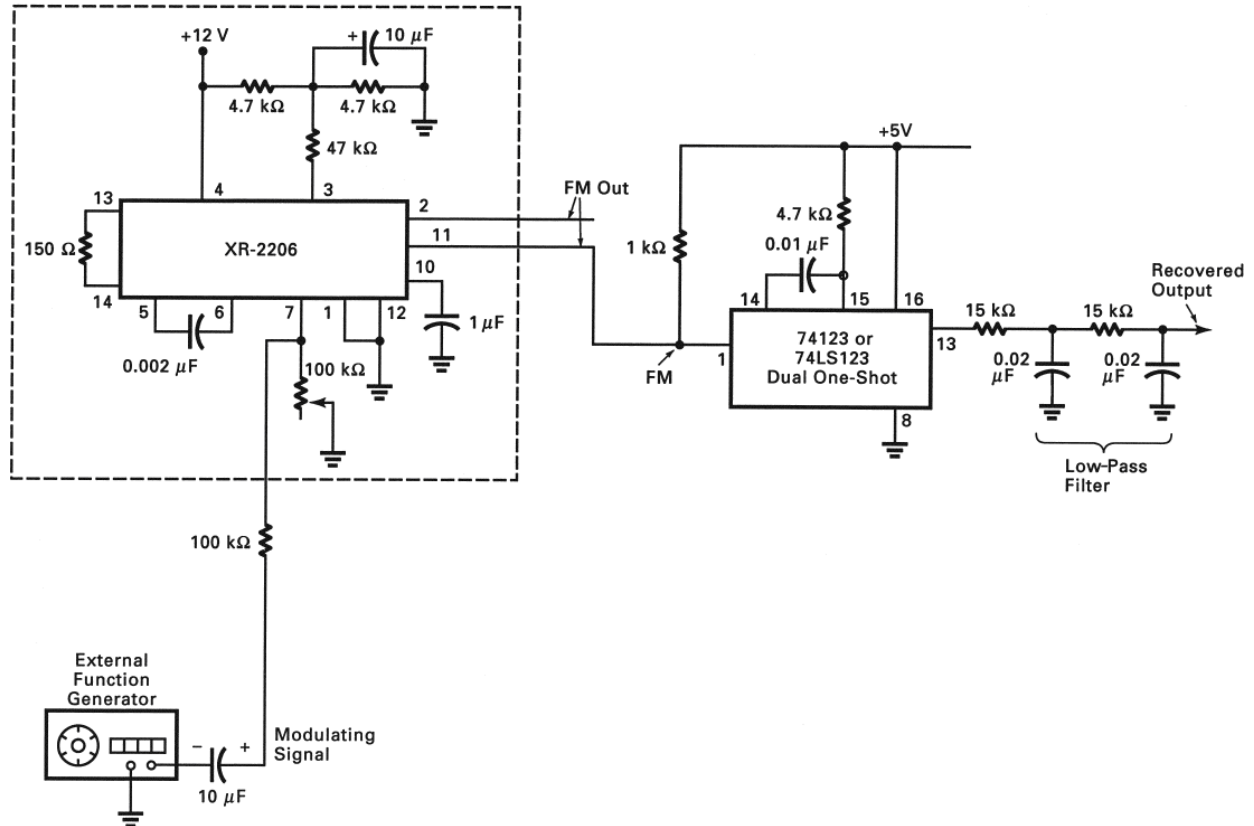


Fig. 6-5 Pulse-averaging discriminator or demodulator.

page 53

6. While observing the demodulating output, increase the modulating signal frequency from 200 Hz slowly up to approximately 2 kHz. Note the effect on the output. How does the demodulated signal vary with frequency? Explain the reason for the action you observe.

page 54

Questions

1. The circuit that generates fixed-width fixed-amplitude pulses is known as a(n) _____.
 - a. Astable multivibrator
 - b. Pulse generator
 - c. Oscillator
 - d. One-shot multivibrator
2. The circuit that smooths the pulses into the original intelligence signal is the _____.
 - a. Low-pass filter
 - b. High-pass filter

- c. One-shot multivibrator
 - d. Quadrature detector
3. Another FM demodulator that uses the principle of averaging pulses in a filter is the _____.
- a. Phase-locked loop
 - b. Foster-Seeley discriminator
 - c. Quadrature detector
 - d. Ratio detector
4. The amplitude of the recovered intelligence signal is directly proportional to _____.
- a. Carrier frequency
 - b. Carrier deviation
 - c. Pulse amplitude
 - d. Pulse width
5. As the frequency of the modulating signal increases, the recovered signal output amplitude _____.
- a. Decreases
 - b. Increases
 - c. Remains the same
 - d. Drops to zero

NAME _____

page 55

_____ DAT

E _____

Chapter 7 has no experiments and therefore is not included in this Experiments and Activities Manual.

page 56

NAME

DATE

Chapter

8

Radio Transmitters

Project 8-1 Class C Amplifiers and Frequency Multipliers

Objective

To demonstrate the operation and use as a frequency multiplier of a class C amplifier.

Required

- 1 function generator
- 1 oscilloscope
- 1 frequency counter
- 1 digital multimeter
- 1 NPN transistor (2N3904, 2N4124, 2N4401, 2N2222, etc.)
- 1 2-mH inductor
- 1 0.02- μ F capacitor
- 1 0.1- μ F capacitor
- 1 10-k Ω resistor
- 1 power supply for +5 V

Introduction

Some radio transmitters use class C amplifiers for power amplification, and that is particularly true of FM transmitters. Most AM transmitters or those transmitting SSB use linear amplifiers. Class C amplifiers are usually preferred because they are far more efficient than linears, but they distort the AM/SSB signals. They can be used in FM transmitters, however, which means that they generate more output power vs. input power than any other kind of amplifier.

A transistor class C amplifier is one that conducts for less than one half-cycle (180°) of the input sine wave signal. Class C amplifiers conduct only during the half-cycle when the emitter-base junction in the transistor is forward-biased. Most class C amplifiers use some form of base bias that keeps the transistor cut off until the input signal polarity and amplitude are high enough to cause it to conduct. Typically, the bias is set up so that conduction occurs during the 90° to 150° range. Since conduction is less than one half-cycle, the collector current is, of course, highly distorted. However, since a tuned circuit is connected in series with the collector, the output signal is a complete sine wave. The flywheel effect of the tuned circuit supplies the missing half-cycle.

Class C amplifiers are also used as frequency multipliers. By connecting a tuned circuit in the collector whose resonant frequency is some integral value of the input frequency, frequency multiplication occurs. That provides a simple and low-cost way to increase the signal frequency. In this way, stable low-frequency circuits can be used to generate the original signal, and multipliers are used to increase the signal to its final, higher RF value.

The class C amplifier is a good frequency multiplier page 58 because of the distortion it generates. The distortion, of course, means that the output signal contains many harmonics. The tuned circuit in the collector is used to select the desired harmonic.

In this experiment, you will demonstrate a simple transistor class C amplifier and show its use as a multiplier.

Procedure

1. Construct the class C amplifier circuit shown in Fig. 8-1. Initially connect the function generator but be sure that the input amplitude is reduced to zero initially.

2. Observe the tuned circuit in Fig. 8-1 and calculate the resonant frequency by using the values given. Record the frequency.
3. Connect an oscilloscope to the class C amplifier collector output. While observing the signal there, begin increasing the function generator input signal. Apply approximately $1.5 V_{p-p}$ to the input capacitor. The function generator frequency should be set to the resonant frequency value you computed in step 2. While observing the output, carefully tune the function generator frequency for maximum output voltage at the collector.
4. Measure and record the output frequency with the frequency counter. Your measured value should be close to the computed resonant frequency.
5. While observing the collector output on the oscilloscope, vary the function generator input voltage from zero until the output signal increases. At just the point where the output signal amplitude reaches its maximum output, stop increasing the input voltage.
6. By using a digital multimeter, measure the voltage across the 10-k Ω base resistor. Record the polarity and voltage.
7. While monitoring the dc voltage across the 10-k Ω resistor, continue increasing the input voltage from the function generator. Increase it to a value of about $4 V_{p-p}$ and note how the bias voltage changes. Does it increase, decrease, remain the same, or otherwise change?

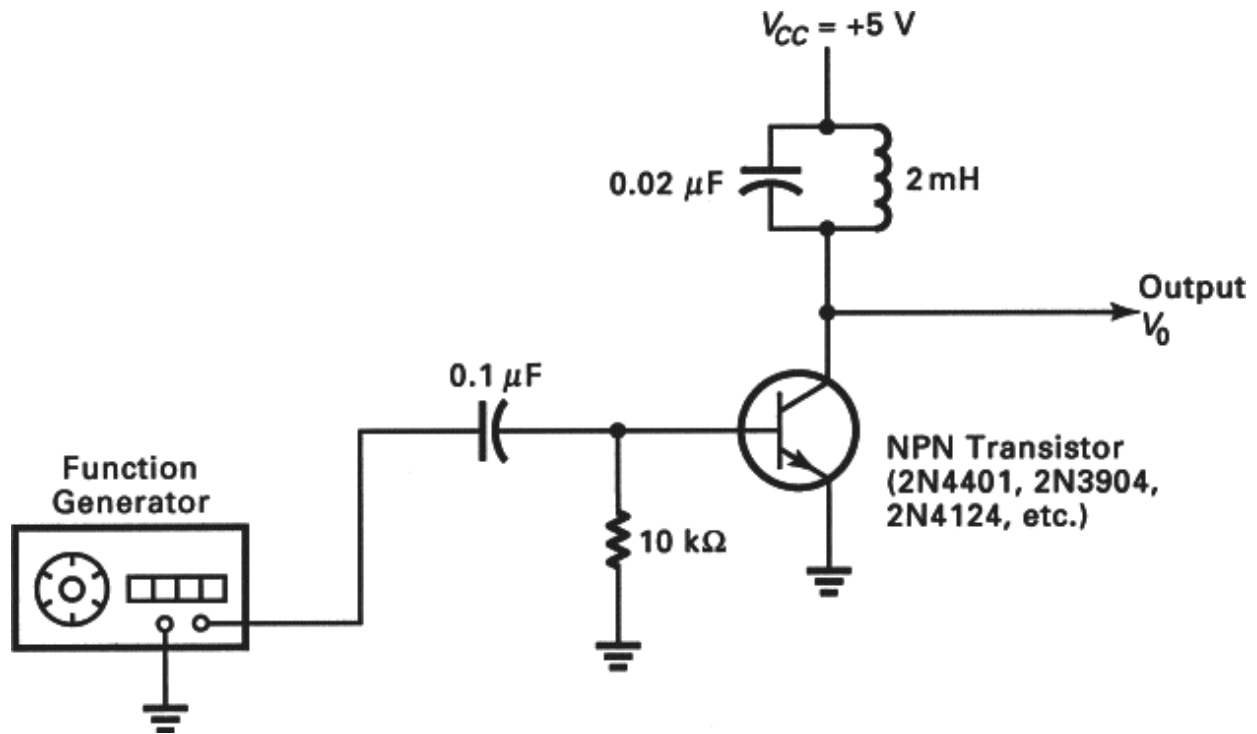


Fig. 8-1 Class C amplifier.

8. Again reduce the function generator input voltage to zero and monitor the collector output. Begin increasing the input voltage slowly and note the point where the output voltage rises to its maximum point. At that time, measure the input voltage. Explain why it is necessary for the input voltage to reach that level before the output signal appears.
9. While observing the output, continue increasing the input and note any change in the output. Measure and record the output voltage. Compare the output voltage you measured to the supply voltage V_{CC} and explain how the output value is obtained.
10. Reduce the input from the function generator to the point where the output signal at the collector just begins to drop off its maximum value. Then insert an ammeter in series with the collector supply. Measure and record the collector current.
11. Calculate and record the input power of this class C amplifier.
12. Assume now that the class C amplifier will be a frequency multiplier. The output will be the resonant frequency of the tuned circuit. Calculate all of the input frequencies of which the

resonant frequency will be a harmonic. Calculate and list all the frequencies from the second through the tenth harmonics.

13. While observing the collector output, adjust the function generator input level to just where the output voltage reaches its maximum amplitude. The output level should be approximately $10 V_{p-p}$.
14. Now adjust the function generator input frequency to each of the input frequencies you calculated above. Tune the generator carefully until the output signal peaks, and then measure the input frequency with the frequency counter. Repeat that process for all of the harmonic values you calculated in step 12.
15. As you continue to lower the input frequency and produce higher and higher harmonics, note the effect on the output voltage. Does the output increase or decrease as the harmonic number increases?

Questions

1. The type of bias used on the class C amplifier shown in Fig. 8-1 is known as _____ .
 - a. Self-bias
 - b. External bias
 - c. Signal bias
 - d. Variable bias
2. If the transistor in the class C amplifier were a PNP, the base bias with respect to ground would be _____ .
 - a. Positive
 - b. Negative
 - c. Zero
 - d. Insufficient information is given to answer this question
3. In a properly adjusted class C amplifier, the collector output voltage is _____ .
 - a. V_{CC}
 - b. $2V_{CC}$
 - c. $V_{CC}/2$
 - d. A value that depends on the input signal amplitude
4. As the frequency multiplication factor increases, the output signal voltage and power _____ .

- a. Decrease
- b. Increase
- 5. The class C amplifier multiplier deliberately distorts the signal to generate _____.
 - a. Higher output voltage
 - b. Higher power
 - c. A clean sine wave
 - d. Harmonics

page 60

page 61

Project 8-2 Crystal Oscillator

Objective

To build and test a crystal oscillator circuit.

Required

- 1 oscilloscope
- 1 frequency counter
- 1 power supply, 10 V dc
- 1 2N3904 (or other NPN transistor, 2N2222 or 2N4401, etc.)
- 1 12-MHz quartz crystal (other frequencies from 10 to 20 MHz OK)
- 1 1-k Ω potentiometer
- 1 1-k Ω resistor
- 1 220-k Ω resistor
- 1 47-pF ceramic capacitor
- 1 100-pF ceramic capacitor
- 2 0.001- μ F ceramic capacitors

Introduction

The source of sine-wave carrier in most transmitters is a crystal oscillator. In simple transmitters, a single crystal is all that is needed. In some cases, two or more crystals may be switched to permit multiple transmission frequencies. In more sophisticated transmitters, a frequency synthesizer is used. However, the frequency of the output and its frequency stability is set by a crystal oscillator

reference. And, of course, crystal oscillators are also widely used as local oscillators in receivers.

In this experiment, you will build and test a common crystal oscillator. The circuit is shown in Fig. 8-2. This is a Colpitts-type oscillator, as the positive feedback is derived from a voltage divider made up of capacitors C_1 and C_2 . R_1 biases the transistor into the linear region for maximum gain and least distortion. The frequency of the circuit is set by the crystal that uses its parallel resonant mode. The output, taken from the emitter, is a low-power sine wave.

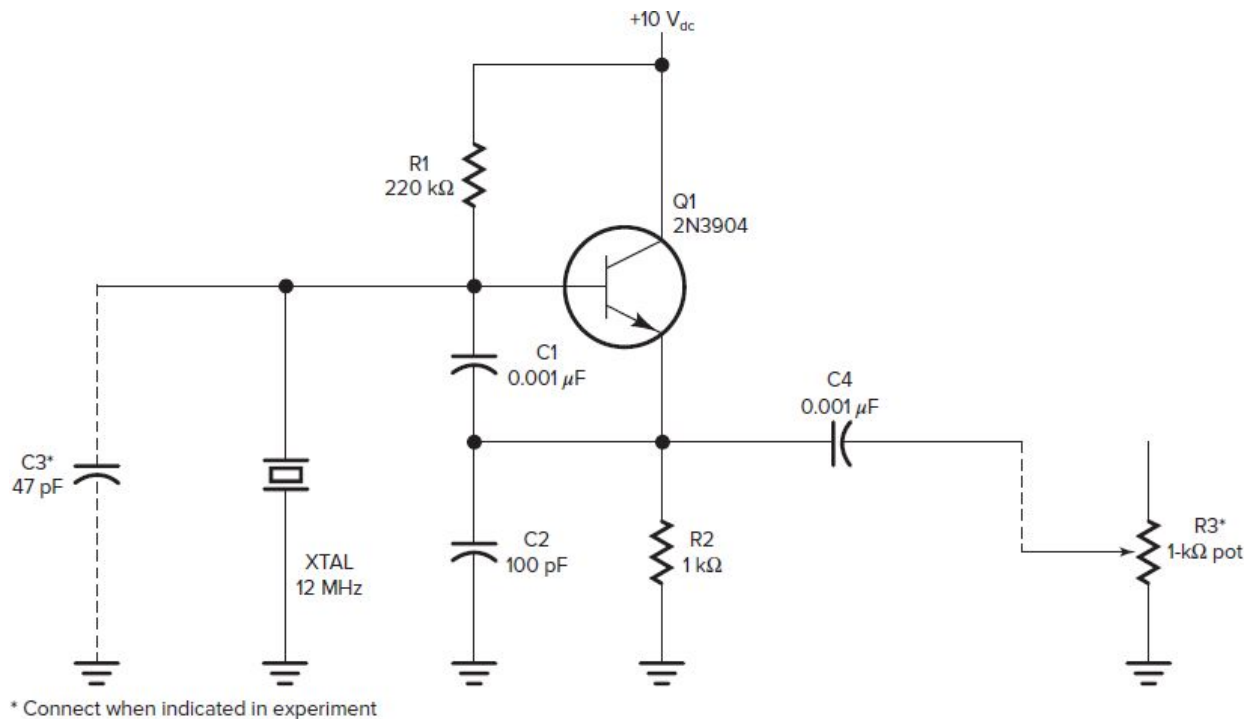


Fig. 8-2 Colpitts-type crystal oscillator.

While the frequency of the output is set by the crystal, any page 62 capacitance appearing in series or parallel with the crystal will also affect the frequency. However, a variable capacitor like that shown in Fig. 8-2 is often connected in parallel with the crystal to permit some variation in the output frequency.

Procedure

1. Construct the circuit shown in Fig. 8-2. If you are using a breadboarding socket, be sure to keep the component leads as

short as possible to ensure best performance. Initially, *do not* connect the 47-pF capacitor C3 or the 1-k Ω pot. Apply +10 V dc to the circuit. Connect an oscilloscope to the output and observe the signal.

- a. What is the shape of the signal? Is it what you could call distorted?
 - b. What is the peak-to-peak output voltage? The rms output voltage?
2. Connect a frequency counter to the output. Set the counter for the best resolution of measurement. Measure the output frequency. How does it compare to the frequency marked on the case of the crystal?
 3. Compute difference between the marked and measured frequencies. Express this as a percentage of the marked frequency.
 4. Connect the 47-pF capacitor across the crystal as shown in Fig. 8-2. How did the frequency change? Explain.
 5. Disconnect the 47-pF capacitor.
 6. Again measure the no-load output voltage.
 7. Connect the 1-k Ω pot load across the output as shown in Fig. 8-2. Set the pot for maximum resistance of 1 k Ω . Measure the output voltage. How did the voltage change? Explain.
 8. Adjust the pot until the output voltage drops to a value exactly one-half of the output voltage with no load. Record that value.
 9. Turn off the power. Disconnect the pot from the circuit. Measure the resistance of the pot with an ohmmeter. What is the significance of this value?
 10. Using the values you determined in steps 8 and 9, answer these questions:
 - a. What is the output power across the load with the pot set as in step 8?
 - b. What is the output impedance of this circuit?

Questions

1. (True or False) The frequency of a crystal oscillator can be changed with an external capacitor.
2. The crystal oscillator in this experiment operates in which class?

- a. A
 - b. AB
 - c. B
 - d. C
 - e. E
3. A distorted output means that:
 - a. The circuit is not operating correctly.
 - b. The circuit is operating off frequency.
 - c. Harmonics are contained in the output.
 - d. Output power will be degraded.
 4. The circuit in Fig. 8-2 operates with a
 - a. Common emitter
 - b. Common collector
 - c. Common base
 5. The circuit in this experiment uses which mode of crystal operation?
 - a. Parallel
 - b. Series

Project 8-3 Building and Testing a Radio Transmitter

Objective

To build a typical radio transmitter from a commercial kit and to perform tests for correct operation.

Materials

- 1 Pacific Antenna
(<https://www.qrpkits.com/ezseries.html#eztransmitter>) Easy Transmitter
 - Soldering iron and basic hand tools (side cutters, needle nose pliers, wire stripper)
 - Dummy loads (two 100- Ω 2-W resistors and MFJ-260C)
 - RG-58/U coax cable
 - RCA phono plugs

- SO-239 UHF plugs
- 1 12–15-V power supply, 4 to 5 A
 - 1 oscilloscope (25 MHz or greater vertical bandwidth)
 - 1 RF power meter (Bird 43, MFJ-816 or equivalent)

Introduction

A commercial kit is one of the fastest, easiest, and lowest-cost ways to examine and test a contemporary HF transmitter. The one recommended here is designed for low-power (QRP) amateur radio use. It is available for the 40 meter (7 MHz) amateur radio band. Please keep in mind that if you do not have a valid FCC radio amateur license, you may not connect this transmitter to an antenna. Serious fines or even jail time can result from operation without a license. In this project, you will use a resistor as a dummy antenna load so no radiation will take place.

The transmitter is made up of a crystal oscillator, a buffer power amplifier, and a final amplifier that can deliver about 2 to 2.5 W to a 50- Ω load. The transmitter kit calls for a BNC connector for the antenna. A BNC or RCA phono connector can be used.

To test the transmitter, you will need a dummy load that you can build with 100- Ω 2-W resistors. You should use a commercial dummy load for the power amplifier such as the MFJ-260C.

Procedure

1. Build the transmitter kit first following the instructions provided in the kit's manual.
2. Build the dummy load as illustrated in Fig. 8-3.

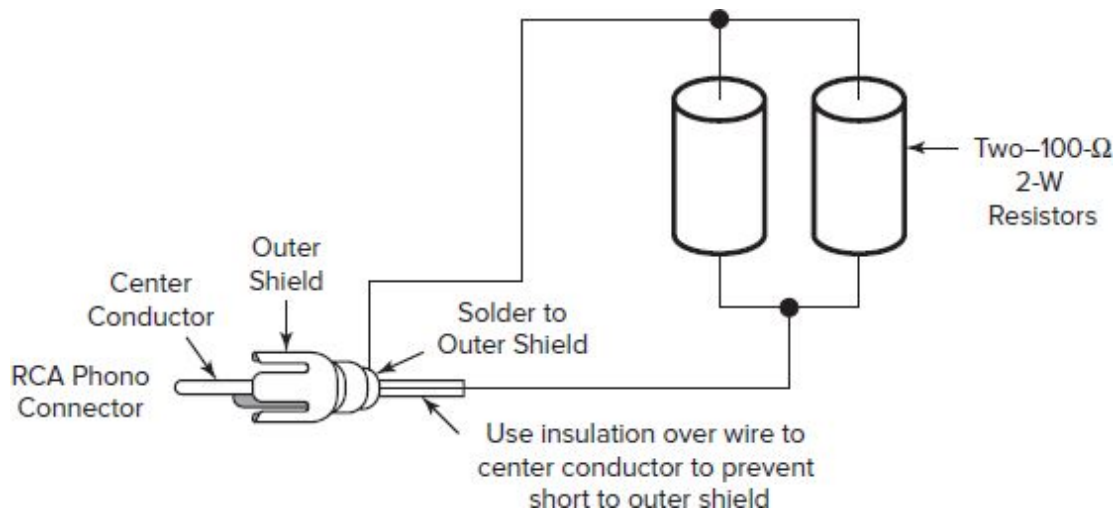


Fig. 8-3 Building a 50- Ω dummy load for testing the transmitter.

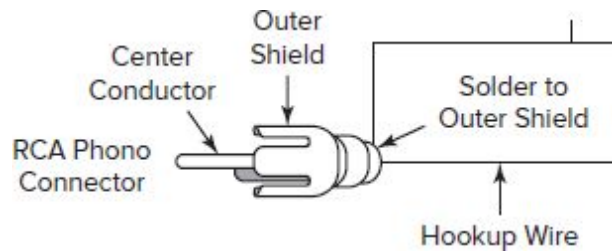


Fig. 8-4 Building a short to key the transmitter.

3. Build a shorting key as shown in Fig. 8-4. This transmitter is designed to be turned off and on by a telegraph key. The key is nothing more than just a convenient hand-operated switch. For this project, you can just use a shorted RCA phono plug. Just solder a short piece of hookup wire between the tip and shell of a phono plug. Plug the short into the KEY jack on the transmitter at the appropriate time.
4. Connect the dummy load to the phono jack labeled ANT.
5. Connect the transmitter to the power supply. Be sure to set the voltage to 12 V if this is a variable supply before you turn the power on. The current drawn by this transmitter is about 200 mA with full output power.
6. Turn on the power and follow the instructions for testing provided in the kit's manual.
7. Plug the shorted phono plug into the KEY jack on the transmitter.

8. To verify the output power, connect an oscilloscope across the 50-Ω load and measure the voltage. The circuit should be connected like the diagram in Fig. 8-5.
9. Remembering to use rms voltage, calculate the output power ($P = V^2/R$).
 $P = \underline{\hspace{2cm}}$ watts
10. Describe the shape of the output signal.
11. Turn off the power. Remove the shorting plug.

Questions

1. Referring to the schematic diagram of the transmitter in the kit's manual, state the type of crystal oscillator used.
2. Which component varies the frequency of this transmitter?
3. The buffer/driver stage utilizes a _____ transistor and the final amplifier stage utilizes a _____ MOSFET.
4. Describe the type of circuit required, so the transmitter can share an antenna with a receiver.
5. Referring to the circuit description in the kit's manual, what type of low pass filter is used to ensure a clean signal?
6. Referring to the schematic drawing of the transmitter, what is the purpose of the 78L09 IC?

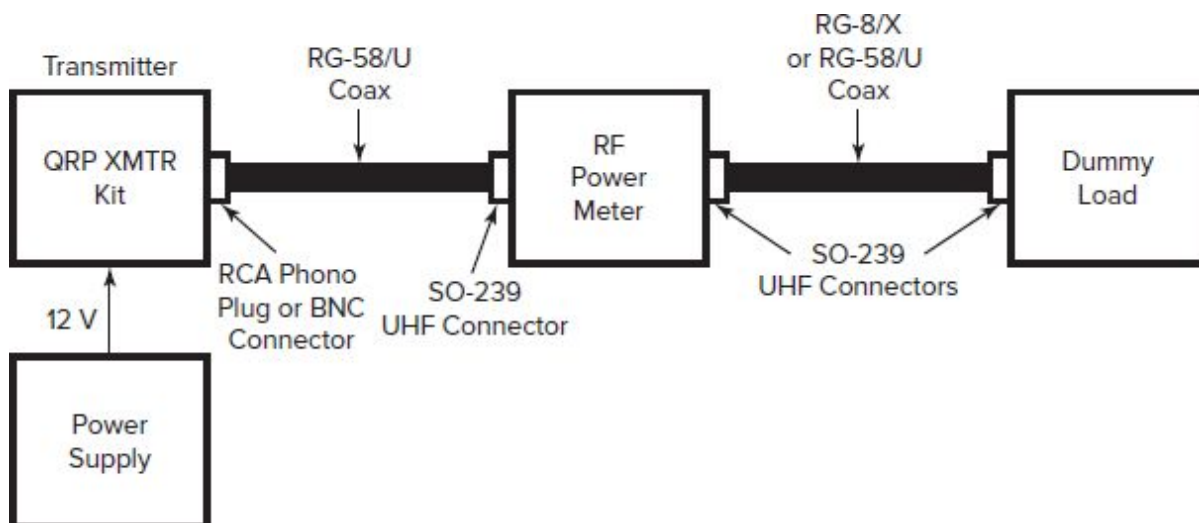


Fig. 8-5 Test setup for transmitter power measurement.

Project 8-4 Measuring Transmitter Output Power

Objective

To demonstrate how to measure transmitter output power and compute SWR.

Required

- 1 oscilloscope
- 1 RF power wattmeter (Bird model 43, MFJ-816, or equivalent, capable of forward and reverse power measurements, and of measuring 5 to 100 W up to 30 MHz)
- 1 radio transmitter portion of an amateur radio transceiver, capable of generating 1 to 100 W output in the 2- to 30-MHz range; alternatively, radio transmitter kits such as the Easy Transmitter QRP 40 meter kit.
- 2 dummy loads, 50 Ω and 100 Ω , capable of dissipating power equal to the transmitter power
- 2 coax cables each about 2 or 3 ft long (RG-8/U or equivalent with 50- Ω impedance) with appropriate connectors such as N-type or PL-259

Introduction

The output impedance of a typical transmitter is usually 50 Ω . To achieve maximum power output, a 50- Ω antenna load is connected. If the antenna represents some other impedance, as is often the case, maximum power will not be sent to the load. What happens is that the power sent to the load is not completely absorbed by the load when the impedances are mismatched. This causes any unabsorbed power to be reflected back toward the transmitter. The forward power sent to the load and the reflected power add algebraically on the transmission line and produce what are called standing waves. Standing waves are patterns of voltage and current along the line.

The reflected power represents lost power. A measure of this lost power is the standing-wave ratio (SWR). You can easily determine the SWR through your power measurements. All you have to do is to

measure the forward (P_F) and reflected (P_R) power and plug those values into the following formula to find SWR:

$$SWR = \frac{1 + \sqrt{P_R/P_F}}{1 - \sqrt{P_R/P_F}}$$

You can also compute SWR by finding the ratio of the load resistance (R_L) to the output impedance of the transmitter (Z_0):

$$SWR = \frac{Z_0}{R_L} \quad \text{or} \quad \frac{R_L}{Z_0} \quad (\text{SWR must be greater than 1})$$

If the load is matched (equal) to the output impedance of the transmitter, the SWR is 1 and maximum power transfer is achieved. There is no reflected power. If the load impedance does not equal the transmitter output impedance, there will be some reflected power and the SWR will be greater than 1. Most transmitters can accommodate some mismatch, but few can accommodate an SWR of greater than 2, as damage may result from the reflected power.

In this experiment, you will learn to measure the output power of a transmitter. You will also measure the reflected power and compute SWR. Refer to Chapter 13 of the text for more details, or wait and run this experiment when you have completed Chapter 13.

Procedure

1. Read the operating manual for the transceiver or have your instructor explain it.
2. Set the transceiver to transmit in the CW mode. Set the frequency to a common ham band frequency, such as 7.05 MHz.
3. Refer to Fig. 8-6(a). Connect the 50- Ω dummy load to the transmitter output.
4. Turn on the power and tune the transmitter for maximum output as described in the transceiver manual.
5. Connect the scope across the dummy load. Measure the voltage across the load. (*Note:* Be very careful, as you could receive a shock or serious RF burn!)
6. Calculate the output power.

7. Turn off the power and connect the wattmeter into the line between the transmitter output and the dummy load as shown in Fig. 8-6(b).
8. If you have not already done so, familiarize yourself with the wattmeter by reading the instruction manual or asking your instructor. There should be provision for setting the power level to be measured and selecting a forward or reverse power measurement.
9. Turn on the power. Set the wattmeter to read forward power. Record the maximum output power you observe. How does this value compare to the value you computed in the step above? Explain.
10. Set the wattmeter to measure reflected power. Record the value of reflected power you measure.
11. Compute the standing wave ratio on the coax cable.
12. Turn off the power and disconnect the 50- Ω dummy load. In its place, connect the 100- Ω dummy load.
13. Turn on the power and measure the forward and reflected power output of the transmitter. Can you obtain the same power level as with the 50- Ω load? Explain your results.
14. Calculate the SWR.

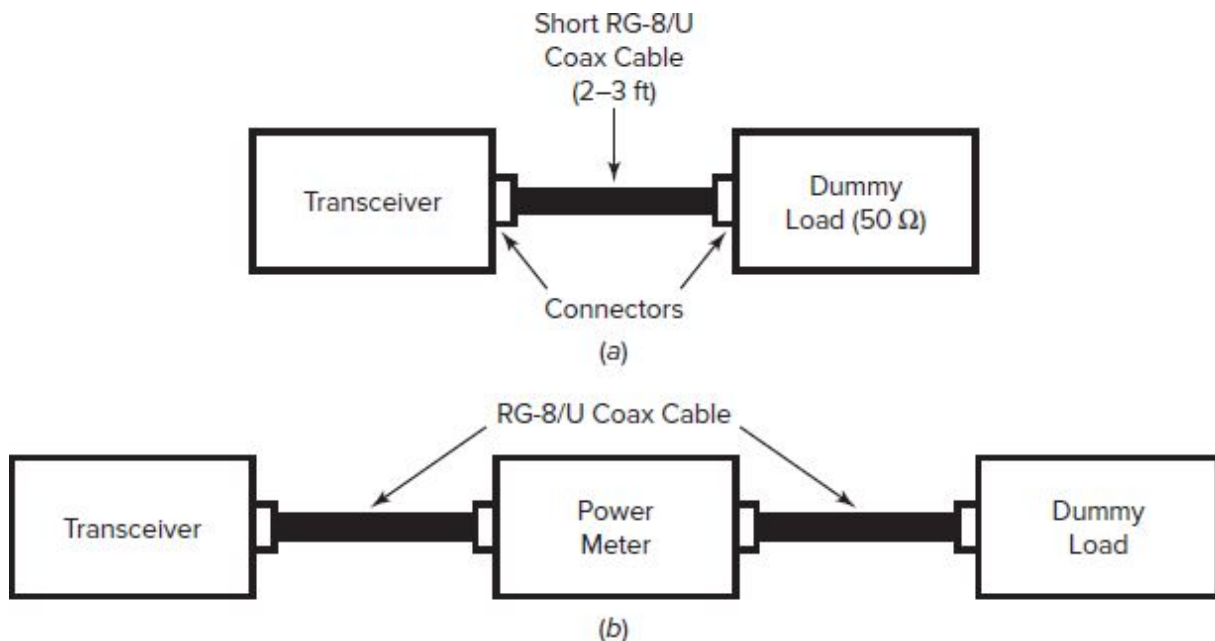


Fig. 8-6 Connections for measuring power.

Questions

page 67

1. What is the most desirable level of reflected power in a transmission line?
2. If the transmitter is designed for a $75\text{-}\Omega$ output load and the antenna has an impedance of $20\ \Omega$, what is the SWR?
3. The most desirable value of SWR is
 - a. 0
 - b. 1
 - c. 2
 - d. Infinity
4. Assume a forward output power of 150 W and a reflected power of 30 W . What is the SWR?
5. What is the maximum allowable SWR in most transmitters?
 - a. 0
 - b. 1
 - c. 2
 - d. Infinity

page 68

Project 8-5 Impedance Matching

Objective

To design, build, and test an L network to match a transmitter to an antenna.

Required

- 1 oscilloscope
- 1 function generator ($50\text{-}\Omega$ output impedance and capable of a frequency of 2 MHz)
- 1 $51\text{-}\Omega$ resistor
- 1 $22\text{-}\Omega$ resistor
- 1 each, 0.0018- , 0.002- , 0.0022- , and $0.0027\text{-}\mu\text{F}$ capacitors
- 1 each, 1.8- , 2.2- , and $4.7\text{-}\mu\text{H}$ inductors

Introduction

One of the most important parts of any transmitter is the output impedance-matching network. Various L , T , and pi (π) networks are used between transmitter stages and between the final amplifier and the antenna to ensure maximum power transfer. As a technician, you will not likely be involved with interstage impedance-matching networks, but you will often encounter situations where it is necessary to create a special impedance-matching network between a transmitter and an antenna. An L network is frequently used to match the transmitter to the antenna impedance. This provides minimum SWR and maximum power transfer to the antenna. Commercial impedance-matching units are available for this purpose. They are generally referred to as antenna tuners and contain a variety of variable capacitors and inductors that can be connected and adjusted in a variety of ways to achieve an impedance match.

Although antennas are designed to be resistive loads, in most cases they are not. If they are operated at a frequency even slightly different from the design center frequency, their impedance will be reactive. Furthermore, the overall antenna impedance depends on the height above the ground and the effects of any nearby structures and objects. To improve the impedance match between transmitter and antenna, an L network is introduced between the transmitter output and the coax cable to the antenna.

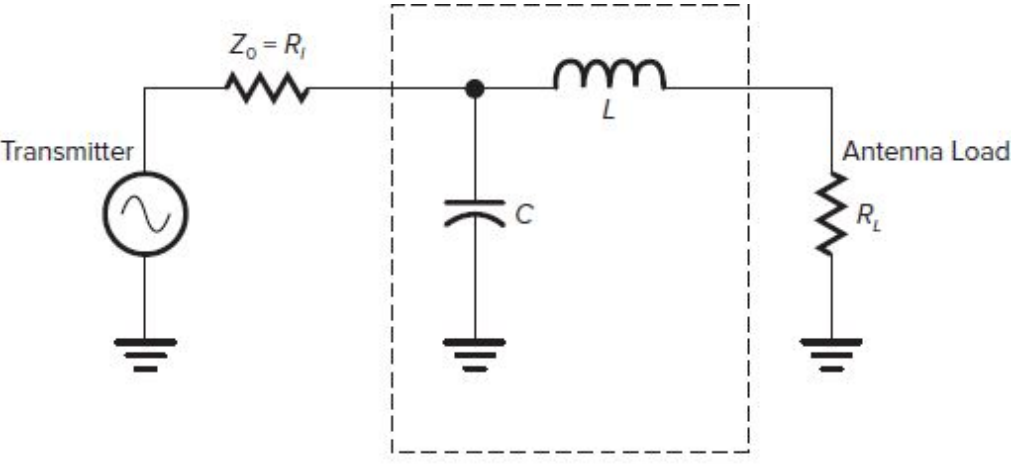
A commonly used L network is shown in Fig. 8-7(a). This variation of the L network is used when the load impedance is less than the output impedance of the transmitter, which is a common occurrence. It is also a low-pass filter, so it provides some attenuation of unwanted harmonics that almost always appear in the output of a transmitter. Values of L and C can be computed to make the antenna impedance appear to be equal to the transmitter output impedance.

If you redraw the circuit as shown in Fig. 8-7(b), you can see how this is done. The coil and capacitor actually form a parallel resonant circuit where the resistive antenna appears in series with the coil and becomes the coil's apparent series resistance. The goal of the calculations is to select values of L and C such that the parallel circuit looks resistive at the frequency of operation and so that its impedance is equal to the transmitter output impedance.

Recall that you can find the impedance of a parallel resonant circuit (Z_p) with either of the following formulas:

$$Z_p = \frac{L}{CR_s}$$

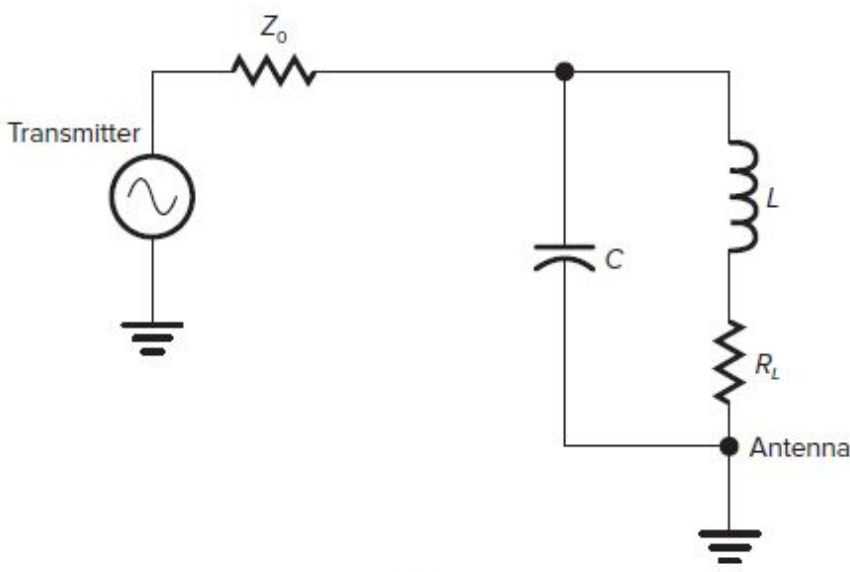
$$Z_p = R_s(Q^2 + 1)$$



$Z_0 = R_i$ = transmitter output impedance or internal resistance

R_L = load resistance (antenna)

(a)



(b)

Fig. 8-7 *L*-matching network: (a) basic *L* network; (b) equivalent circuit of *L* network is a parallel resonant circuit.

where R_s is the winding resistance of the coil plus the antenna (load) resistance. Using the formula in the textbook, you will compute the values of L and C that will make the above relationship come true.

A common occurrence is that the antenna also contains a reactive element. For example, assume that the equivalent impedance of the antenna is capacitive. Refer to Fig. 8-8(a). The antenna looks like a series RC circuit. Note that this antenna capacitance appears in series with the inductor of the parallel tuned circuit. The effect of this capacitance may be cancelled out by simply making the inductive reactance higher than the value computed with the formulas by the amount of the capacitive load reactance at the frequency of operation. See Fig. 8-8(b). If this is done, the antenna capacitance is effectively cancelled out. The resulting circuit is the same as in Fig. 8-7(b). If the antenna was inductive, this inductance would simply add to the L -network inductance value. The total value of inductance must be equal to the value that will give the reactance calculated with the formulas.

In this experiment, you will design and build an L network to demonstrate this concept. You will use a common function generator as the transmitter and discrete components to simulate the antenna and build the L network.

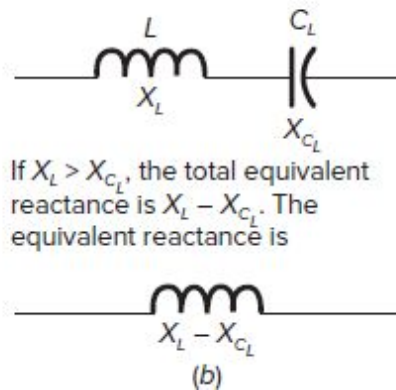
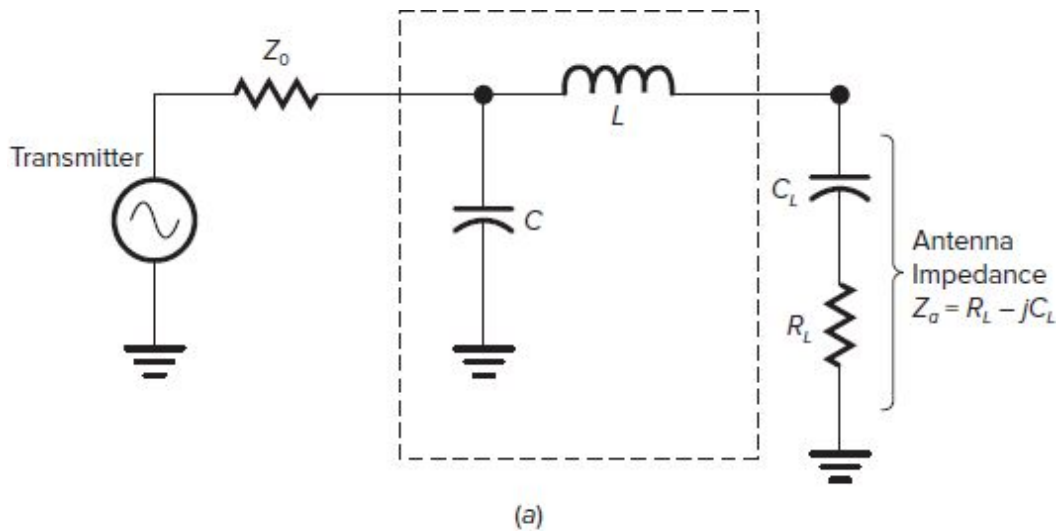


Fig. 8-8 *L*-matching network: (a) the antenna is a reactive load; (b) equivalent series reactance.

Procedure

1. Set the function generator to supply a 2-MHz sine wave of 10 V_{p-p} with no load connected. Then connect a $51\text{-}\Omega$ resistor as the load. See Fig. 8-9. The cable connecting the function generator will be a coax cable with a characteristic impedance of $50\ \Omega$. Measure the voltage across the $51\text{-}\Omega$ load.
 - a. Calculate the power in the load.
 - b. Will this arrangement supply maximum power to the load?
2. Remove the $51\text{-}\Omega$ load and reset the function generator no-load output voltage to 10 V_{p-p} . Then connect a $22\text{-}\Omega$ resistor as the

load. Again, measure the load voltage and calculate the power. Is maximum power being provided?

3. Design an L network to match the $22\text{-}\Omega$ load to the $50\text{-}\Omega$ generator, using the formulas in the text.

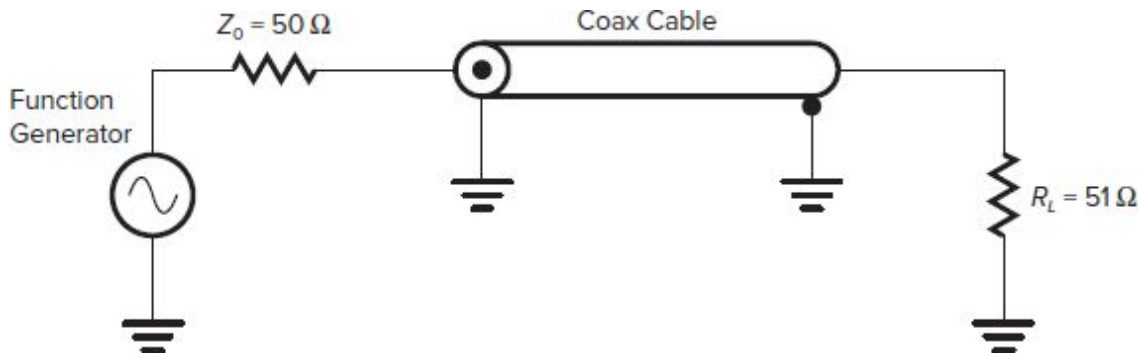


Fig. 8-9 Connecting the function generator to the load.

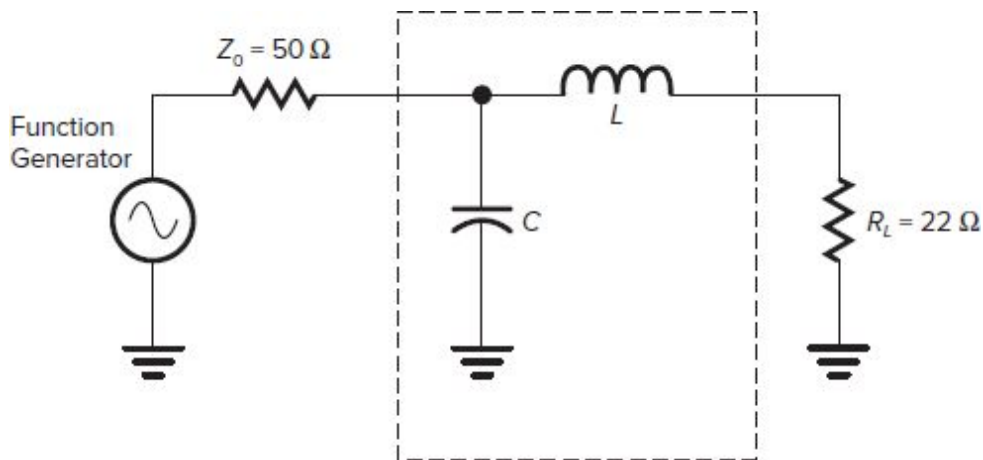


Fig. 8-10 Calculate L and C , using formulas in textbook.

4. Then build the circuit as shown in Fig. 8-10.
5. Before you connect the generator to the L network, set its no-load output to $10\text{ V}_{\text{p-p}}$ as before. Then connect the L network and the $22\text{-}\Omega$ load. Measure the voltage across the $22\text{-}\Omega$ load and compute the power. Is maximum power now being supplied? Also measure the voltage across the entire L network at the generator output. Is maximum power dissipated here? Explain.
6. Using one of the parallel impedance formulas given in the introduction, calculate the impedance of the circuit you designed

above to see that the value matches the generator impedance. Does it match? Explain.

7. Assume that the load is now capacitive with a complex impedance of $22 - j36$. The operating frequency is 2 MHz. What value of capacitance does this represent?
8. Connect the circuit as shown in Fig. 8-11. Set the open-circuit generator output to $10 V_{p-p}$ before you connect the load. Measure the voltage across the resistor and calculate the output power. Is maximum power being delivered?
9. Design the L network to match the generator to this load. Use the procedure you followed in step 3 but factor in the load capacitance. Draw the final circuit, labeling all component values.
10. Build the circuit and test as before. Be sure to reset the open-circuit output voltage to $10 V_{p-p}$ before you connect the circuit. Again measure the resistor output voltage across the combined load and compute the power. Does the L network provide maximum power?

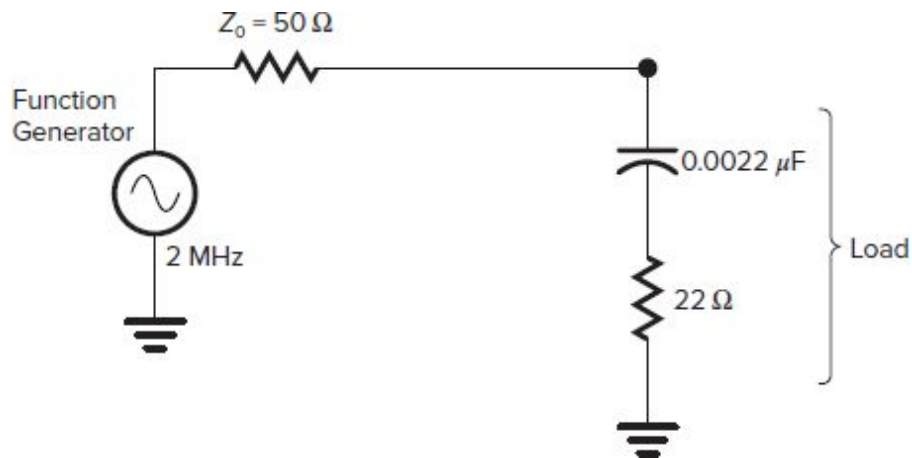


Fig. 8-11 Transmitter with reactive load.

Questions

1. What is the Q of the L networks you built and tested in this experiment?
 - a. < 1
 - b. < 5
 - c. > 10

- d.** > 100
- 2.** If the antenna is inductive and its reactance is greater than the reactance you compute with the formulas, how can you build this circuit and make it work?
 - a.** This circuit cannot be used.
 - b.** Connect a capacitor in parallel with the antenna.
 - c.** Connect a capacitor in place of the inductor in the L network and make its value such that the correct inductive reactance is obtained.
 - d.** Use the other form of L network with the capacitor across the load.
- 3.** Name a way that the L network could be made more useful over a wide range of possible load impedances.
- 4.** Explain how you could make a variable inductor with a fixed inductance and a variable capacitor.

NAME

page 73

DATE

Chapter

9

Communication Receivers

Project 9-1 Noise and Its Measurement

Objective

To show that noise is present in electronic circuits, its effect on low-level signals, and a way to measure noise amplitude.

Required

- 1 oscilloscope (dual-trace) with direct (X1) probes
- 1 function generator
- 1 power supply, ± 5 V dc
- 1 1458 dual or amp IC (individual 741 op amps can also be used)
- 1 MC3403 or TL074 BIFET op amp
- 1 0.47- μ F capacitor
- 2 0.001- μ F capacitors
- 2 2.2-k Ω resistors, both carbon composition and film types
- 2 220-k Ω resistors, both carbon composition and film types
- 1 1-M Ω resistor
- 1 10-k Ω pot

Introduction

Noise consists of random signal variations that interfere with electronic intelligence signals. It comes from a variety of sources in electronic equipment, but most of it is generated by thermal agitation in resistors and other components. Shot noise in semiconductor devices also contributes to noise. The noise itself is a random change of voltage; it varies widely in amplitude and frequency. One way to look at noise is that it contains an infinite number of frequencies mixed together in constantly changing amplitudes.

Although noise signals are low in amplitude, usually several microvolts, they can still interfere with an intelligence signal that you may want to amplify and process. Radio signals are particularly low in amplitude and, therefore, are easily interfered with by noise. In some cases, the noise is greater than the signal and completely masks it. In order for a communications system to function properly, the signal-to-noise ratio (S/N) must be high enough that the signal is intelligible. That means transmitting with sufficient power and using low-noise receivers.

The best way to ensure a good signal-to-noise ratio is to design circuits so that the noise they generate is as low as possible. Since most of the noise produced by electronic circuits is caused by thermal agitation in resistors, steps can be taken to minimize it. The noise itself is a function of the resistance at the input to the circuit, the temperature, and the bandwidth of the circuit. The larger the resistance, the higher the temperature, and the wider the bandwidth, the greater the noise that will be present. Reducing any one of the three factors will decrease the amount of noise.

Keeping input impedance low and using small values of page 74 resistance in the front end of amplifiers of electronic equipment will greatly reduce the amount of noise generated. Further, film resistors produce much lower noise voltage than the more common carbon composition resistors. Noise can be minimized by reducing the temperature and limiting the bandwidth of the circuitry. Since noise is broadband in nature, filtering it will reduce its amplitude. By restricting the bandwidth, many higher and/or lower noise frequencies are reduced or eliminated.

The types of components used in an amplifier also determine the noise. In general, JFETs and MOSFETs produce lower noise than

bipolar transistors. BIFET amplifiers, those with FETs in the front end and bipolar transistors elsewhere, are good for use in low-noise applications.

In this experiment, you are going to observe actual noise in an electronic circuit by viewing it on an oscilloscope. Because noise voltages are very low in amplitude, two operational amplifiers set for high gain will be used to increase the noise level. The small amount of noise generated at the amplifier input will be multiplied by the gain of the amplifier to a level that will permit convenient viewing on an oscilloscope. You will then show how noise can be significantly reduced by using film rather than carbon resistors and by filtering. You will also show how FET amplifiers produce less noise than bipolars.

You will learn a special technique for measuring the amount of noise in a circuit. Because noise is random, it is difficult to measure and quantify. The tangential noise measurement method of using a dual-trace oscilloscope gives a surprisingly accurate measurement of rms noise voltage.

Procedure

1. Construct the circuit shown in Fig. 9-1. Use the carbon composition rather than the film resistors when constructing the circuit. For the initial steps, omit the $0.001\text{-}\mu\text{F}$ capacitors from across the $220\text{-k}\Omega$ feedback resistors. Keep all leads as short as possible to reduce 60-Hz power line noise. Adjust the $10\text{-k}\Omega$ pot so that the input voltage is zero. You will observe the output from the second op amp at pin 7.
2. Knowing the gain formula of an operational amplifier inverter, calculate and record the gain of each op amp stage in Fig. 9-1 and the overall circuit gain.
3. Connect an oscilloscope to the output of the second op amp. Use a direct ($\times 1$) probe instead of the usual $\times 10$ attenuator probe. An attenuator probe contains a voltage divider that reduces the input signal by a factor of 10, and that reduction could prevent your observing the noise on the oscilloscope. Use the direct probe and select one of the most sensitive vertical gain positions on the oscilloscope input.

- Apply power to the circuit. Be sure that you have adjusted the positive and negative supply voltages to ± 5 V. Observe the noise at the amplifier output. Adjust the oscilloscope vertical gain and sweep controls until you observe the random varying pattern on the screen. If you have access to the intensity control on the oscilloscope, increase the trace intensity to a higher value to permit you to see more of the noise. Because many of the noise elements have a short duration and high peaks, you may not be able to see them at the lower intensity setting. To get a more accurate noise measurement, increase the intensity so you see the major peaks more clearly. Measure the average peak-to-peak value of the noise. Since the amplitude is varying considerably, simply take an eyeball average and estimate the peak-to-peak amplitude of the noise. Record that value.

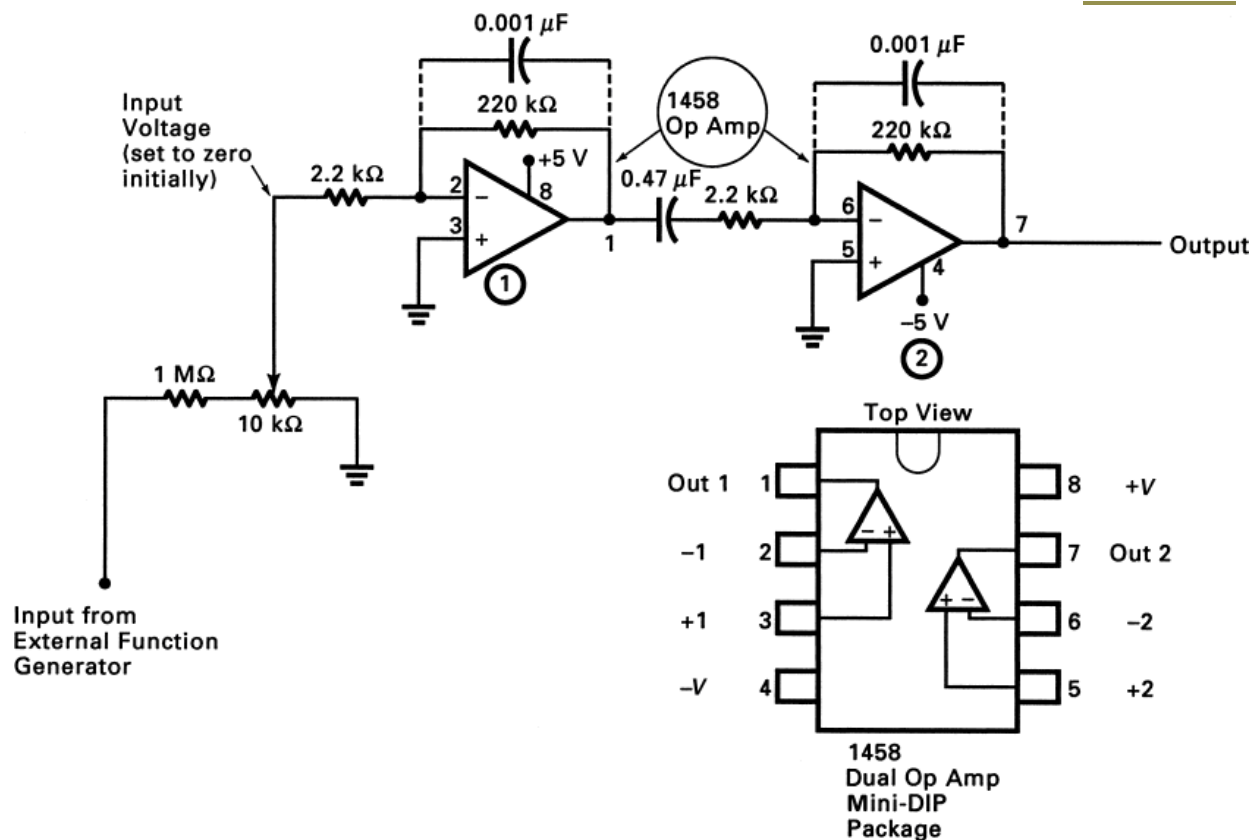


Fig. 9-1 Amplifier circuit to demonstrate noise and its measurement.

5. Compute the noise voltage level at the amplifier input by dividing the average noise output voltage by the total amplifier gain.
6. Apply a 1-kHz signal from the external function generator to the amplifier input. Adjust the 10-k Ω pot slowly while observing the amplifier output. You may also have to adjust the function generator amplitude control. At some point, you should be able to see the 1-kHz signal appear above the noise level. Continue to slowly increase the input signal amplitude. At some point, the signal will be clearly visible but with a considerable amount of noise riding on it. Continue increasing the input signal amplitude until it is significantly higher than the noise itself. You can increase the signal amplitude until it reaches the distortion or clipping level of the amplifiers. Note that the higher the input voltage, the cleaner the signal looks and the less effect that noise has upon it. You can easily see why a high signal-to-noise ratio is desirable.
7. Reduce the input signal amplitude to zero and again display the noise on the oscilloscope. Next, connect the 0.001- μ F capacitors across the 220-k Ω feedback resistors in the amplifier circuit. Keep the leads short to minimize 60-Hz power line pickup. Measure and record the average peak-to-peak noise amplitude.
8. Turn off the power. Remove the 0.001- μ F capacitors from across the op amp feedback resistors. Replace the resistors in each op amp circuit with the film resistors of the same value. Keep the leads short.
9. Reapply power to the circuit. Now observe the noise at the amplifier output. Measure and record the average peak-to-peak noise output voltage.
10. Use the overall gain of the amplifier to calculate the equivalent noise input with the film resistors. Record your answer.
11. Turn off the power. Rewire the circuit with the composition resistors.
12. You will now demonstrate a method of obtaining a more accurate measurement of actual noise voltage by using a calibrated dual-trace oscilloscope. The oscilloscope should have an alternate horizontal sweep capability. Connect the direct probes from the two vertical input channels to the amplifier output at pin 7. The

noise will be displayed simultaneously on both channels. Adjust the vertical calibrations to exactly the same settings so that the noise on each channel is displayed at the same level.

13. Next, by using the vertical-position controls on one or both vertical-input channels, adjust the traces so that they begin to merge. Your goal is to adjust the noise waveforms so that the darker background area between the two traces just disappears.
14. Next, remove the oscilloscope connections to the amplifier output and thereby reduce the input voltage to zero on both channels. At this time, the oscilloscope display will simply show two horizontal traces separated by a small distance that is a measure of the noise amplitude. Use the calibrated scope graticule to determine it. The voltage that you measure between traces is 2 times the actual rms noise voltage. Record the actual noise voltage.
15. As before, divide the value in step 14 by the total amplifier gain to obtain the actual input noise level. Record the result.
16. Turn off the power and rewire the circuit with the MC3403 or TL074 BIFET op amps. Refer to the data sheets to obtain pin connections, because they are different from those shown in Fig. 9-1.
17. Apply power to the circuit. Again measure the output noise and then calculate the input noise. Use the tangential noise measurement technique. Record your results. Is the noise produced by the BIFET amplifiers more or less?
18. Turn off power and disconnect the circuit.

Questions

Write the answer to each question in the space provided.

1. Which of the following items does not directly affect the noise voltage in an electronic circuit?
 - a. Bandwidth
 - b. Temperature
 - c. Resistance
 - d. Signal frequency
2. The 0.001- μ F capacitors added to the circuit cause which component of the noise to be greatly minimized?

- a. High frequency
 - b. Low frequency
 - c. Midrange frequencies
 - d. 60-Hz power line noise
3. (True or False) Carbon composition resistors generate more noise than film resistors.
 4. (True or False) Bipolar transistors produce less noise than FETs.
 5. An amplifier has an input resistance of 75 Ω , a total bandwidth of 36 MHz, and temperature of 30°C. The noise voltage is _____ μV .
 - a. 3.11
 - b. 4.5
 - c. 6.72
 - d. 8.09

Project 9-2 PLL Frequency Synthesizer

Objective

To demonstrate the operation and characteristics of a phase-locked loop frequency synthesizer.

Required

- 1 oscilloscope
- 1 function generator
- 1 frequency counter
- 1 power supply ($\pm 12\text{ V}$)
- 1 565 PLL IC or an equivalent PLL IC
- 2 4029 CMOS binary/BCD counters
- 1 0.001- μF capacitor
- 1 0.01- μF capacitor
- 1 0.1- μF capacitor
- 1 0.47- μF capacitor
- 2 560- Ω resistors
- 1 2.2-k Ω resistor
- 1 3.3-k Ω resistor

Introduction

A frequency synthesizer is a signal generator circuit that produces a highly stable output frequency that can be changed in discrete increments. A phase-locked loop can be used as a frequency synthesizer by connecting a stable crystal oscillator as a reference to the PLL input and taking the output from the VCO; see Fig. 9-2. By connecting a variable-frequency divider between the frequency output and the phase detector input, the output frequency can be changed by simply changing the frequency division ratio.

There are two key factors in the design of the phase-locked loop frequency synthesizer. First, the output frequency of the synthesizer is equal to the reference input frequency times the frequency division ratio N , as follows:

$$f_0 = Nf_r$$

As you can see, changing the frequency division ratio will vary the output frequency. One way to vary the divide ratio is to use a preset down counter. A binary number is loaded into the counter, which is then decremented to zero. Then the cycle is repeated. For example, if the binary number 0110 is loaded, the counter will divide by 6. A variety of variable or programmable IC counters are available to perform division in a PLL synthesizer. In this experiment, you will use the 4029 4-bit up-down counter, which can be set for binary or BCD counting.

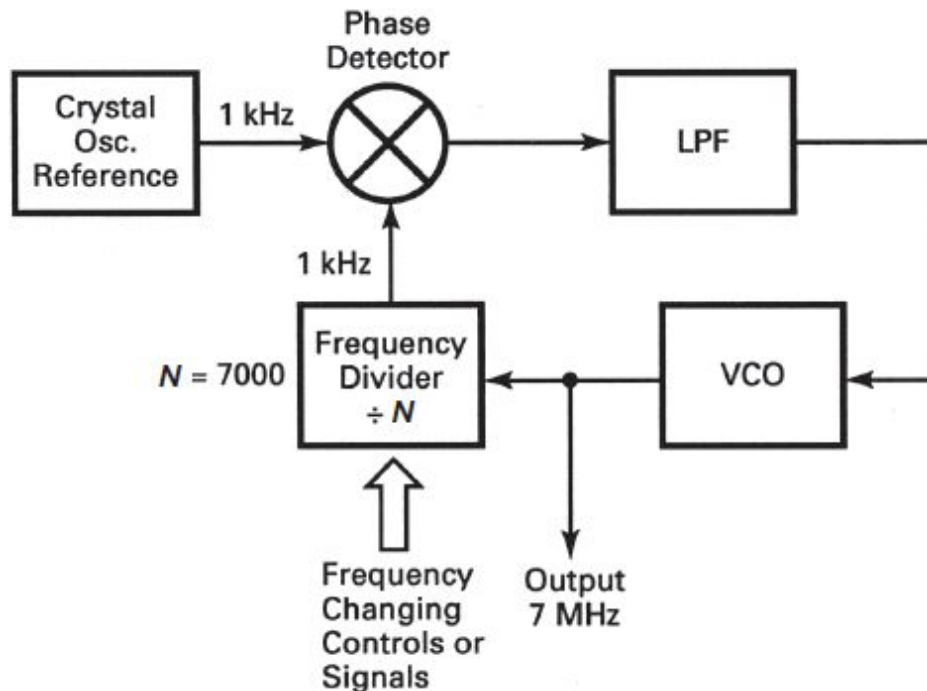


Fig. 9-2 Phase-locked loop frequency synthesizer.

Second, the frequency synthesizer output frequency varies page 78 in increments equal to the reference frequency. For example, if the reference frequency is 100 kHz, the synthesizer output will be varied in 100-kHz steps by varying the frequency division ratio. In some PLL synthesizers, a frequency divider is used between the reference oscillator and the phase detector input to decrease the frequency step size.

In this experiment, you will construct a PLL frequency synthesizer and verify these characteristics.

Procedure

1. Construct the circuit shown in Fig. 9-3. This is the PLL circuit you used in Project 9-1, but note the different R and C values. You will use a 600-Hz input square-wave signal from an external function generator as the reference signal. Initially you will use a frequency divider made up of a 4029 CMOS counter. Pin 9 on the 4029 determines whether the count is pure binary (pin 9 at +12 V) or BCD (pin 9 at ground).

2. Determine the frequency division factor of the 4029. Record the result. Check the state of pin 9 in Fig. 9-3; then assume 4-bit operation with the output taken from pin 2.
3. Examine the circuit shown in Fig. 9-3 and predict the output frequency of the VCO. Record your prediction.
4. By using a frequency counter, set the reference input from the function generator to 1 kHz. Measure and record the VCO output frequency at pin 4 on the 565, which is the synthesizer output. Are your computed and measured values the same?
5. To verify that the PLL is locked, vary the reference input frequency over a narrow range and observe the VCO output. What happens?
6. Change the circuit by moving pin 9 on the 4029 from +12 V to ground. The 4029 now divides by what?
7. Set the reference input to 1 kHz. Predict the VCO output. Record your prediction.
8. Measure and record the VCO output frequency at pin 4 on the 565.
9. Rewire the circuit so that it is as shown in Fig. 9-4. Change the 3.3-k Ω resistor on the 565 between pins 8 and 10 to 2.2-k Ω . Two 4029 ICs are cascaded between the VCO output and phase detector input. Looking at the state of pin 9 on each 4029 determines the individual IC divide ratios and the combined or total divide ratio. What is it? Predict the synthesizer output frequency if the function generator (reference) is set to 100 Hz. Record your prediction.
10. Set the function generator to 10 Hz and apply power. Measure and record the output frequency of the VCO with a frequency counter. Are your computed and measured values the same? If the divide ratio of the 4029 IC were variable, in what steps would the synthesizer frequency change?
11. Rewire the circuit so that it is as shown in Fig. 9-5. One 4029 is used in the PLL feedback loop, and the other is used between the function generator and the reference input to pin 2 on the 565. Predict the VCO output frequency if the function generator is set to 10 kHz. Record your prediction.

- 12.** Determine and record the incremental amount by which the frequency will vary as the frequency division ratio is changed.
- 13.** Set the function generator to 10 kHz. Measure and record the VCO output frequency. Compare it to the predicted value. Are your measured and calculated values the same?

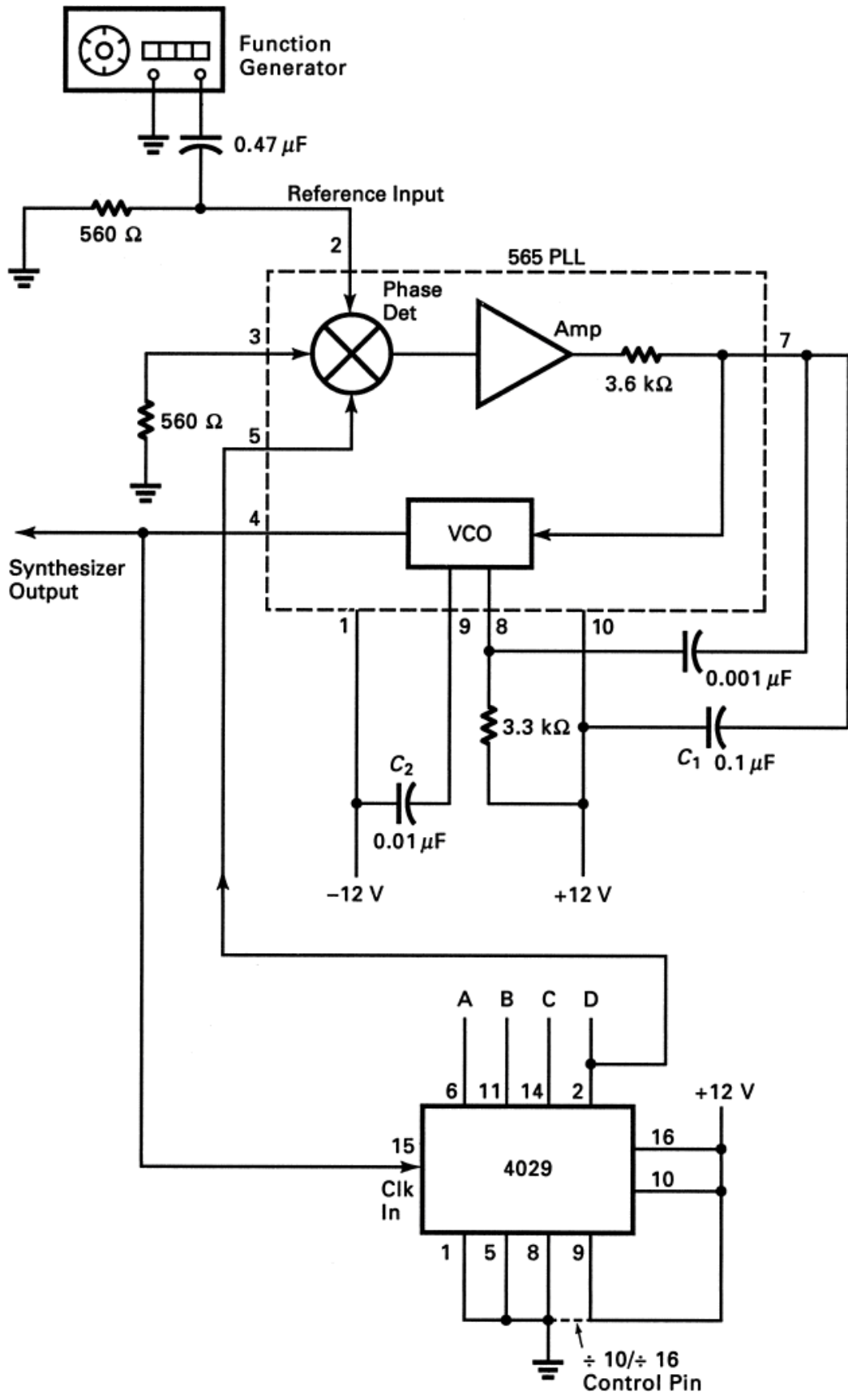


Fig. 9-3 PLL synthesizer experimental circuit.

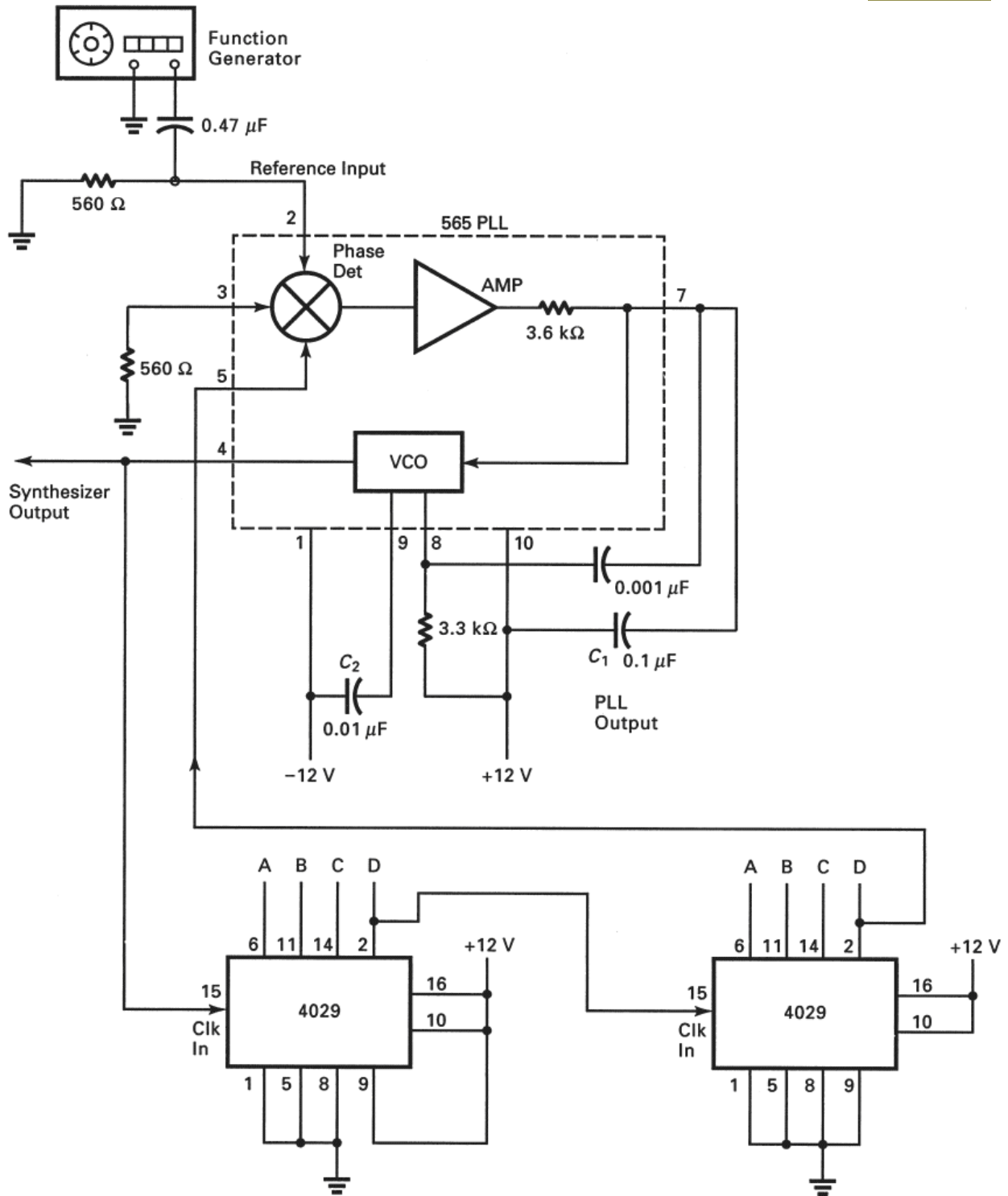


Fig. 9-4 Revised synthesizer experimental circuit.

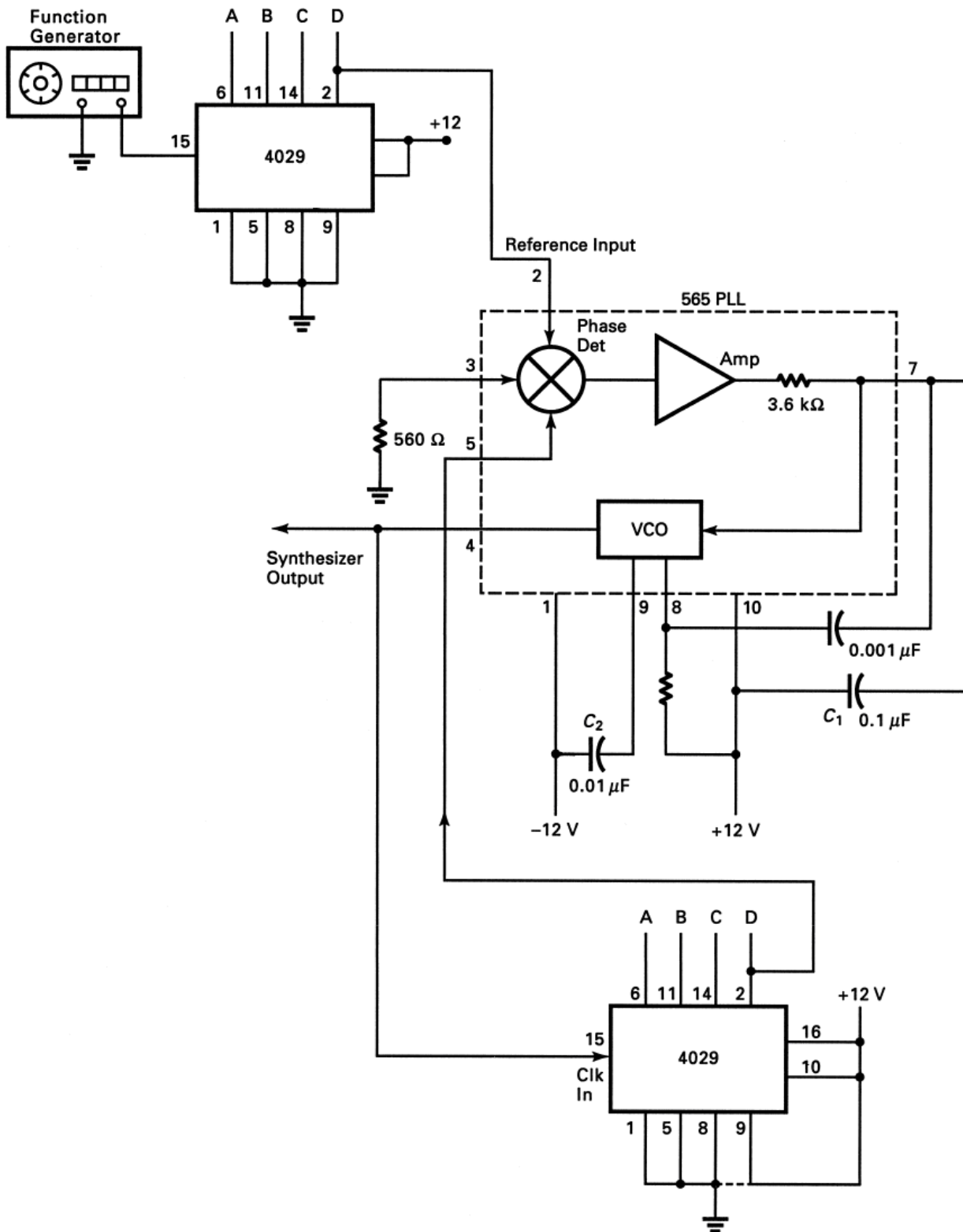


Fig. 9-5 Synthesizer with dividers at the reference input and in the feedback loop.

Questions

1. The frequency variation increment of a PLL synthesizer is determined by the _____ .
 - a. VCO free-running frequency
 - b. Frequency divider ratio
 - c. Both **a** and **b**
 - d. Reference frequency
2. A PLL synthesizer has a 1-MHz reference and a frequency divider of three cascaded BCD counters. The output frequency is _____ .
 - a. 1 MHz
 - b. 10 MHz
 - c. 100 MHz
 - d. 1 GHz
3. In the circuit described in question 2, a two-decade BCD counter is connected between the reference and the PLL phase detector input. The output frequency is _____ .
 - a. 10 kHz
 - b. 100 kHz
 - c. 10 MHz
 - d. 100 MHz
4. (True or False) The PLL synthesizer can function as a frequency multiplier.
5. To function properly, the VCO in the PLL must have its free-running frequency set approximately to the _____ .
 - a. Desired output frequency
 - b. Output divided by N
 - c. Frequency division ratio
 - d. Reference frequency

Project 9-3 Mixer Operation

Objective

To demonstrate mixer operation using a diode ring, lattice mixer.

Required

- 1 crystal oscillator circuit from Project 8-2
- 1 signal generator with an output frequency range from 11 to 13 MHz (MFJ-269 or comparable)
- 1 frequency counter
- 1 Mini-Circuits SBL-1-1 + frequency mixer
- 1 Toko 455-kHz ceramic filter from Project 2-3
- 2 2-k Ω resistors
- 1 0.001- μ F ceramic capacitor

Introduction

There are literally dozens of mixer configurations used in modern communications equipment. From a simple diode to a complex Gilbert cell IC, all have their place. One of the best and most widely used mixers is the diode ring or lattice modulator. While it does introduce conversion loss, its performance is superior to most other types of mixers. This type of mixer is a balanced modulator type, meaning that it suppresses the carrier or, in this application, the local oscillator signal in the output. This leaves only the sum and difference frequencies in the output. The desired frequency is then selected by a filter. In this project, you will show the operation of such a mixer and how the desired output is selected with a ceramic filter.

Procedure

1. Construct the circuit shown in Fig. 9-6. Be sure to wire the mixer as shown in Fig. 9-7. The circuit uses the crystal oscillator circuit from Project 8-2 that produces a 12-MHz output. This will be used as the local oscillator (LO). An external signal generator will be used for the signal source. This generator should be variable so that its frequency can be varied. A frequency counter is needed to measure the signal frequency from the generator. If the MFJ-269 is used, the counter is internal and a separate unit is not needed. The mixer is a commercial diode ring-type mixer designated as the Mini-Circuits SBL-1-1+. The 455-kHz band-pass ceramic filter used in Project 2-3 will serve to extract the sum or difference frequency.
2. Looking at the circuit, determine what the input signal frequency from the generator should be to produce a difference

intermediate frequency (IF) of 455 kHz. There should be two frequencies. Also compute the related sum frequencies of this mixing result.

3. Power up the circuit and increase the signal generator output to its maximum. While monitoring the filter output with an oscilloscope across the 2-k Ω load, tune the signal generator from about 10 MHz upward until you see a peak output voltage. Note the frequency. Continue to increase the generator frequency while looking for a second input frequency that produces a peak output voltage. Again note that frequency.
4. Do the frequencies you measured in step 2 correspond to those you measured in step 3? Explain. Do the sum frequencies appear in the output?

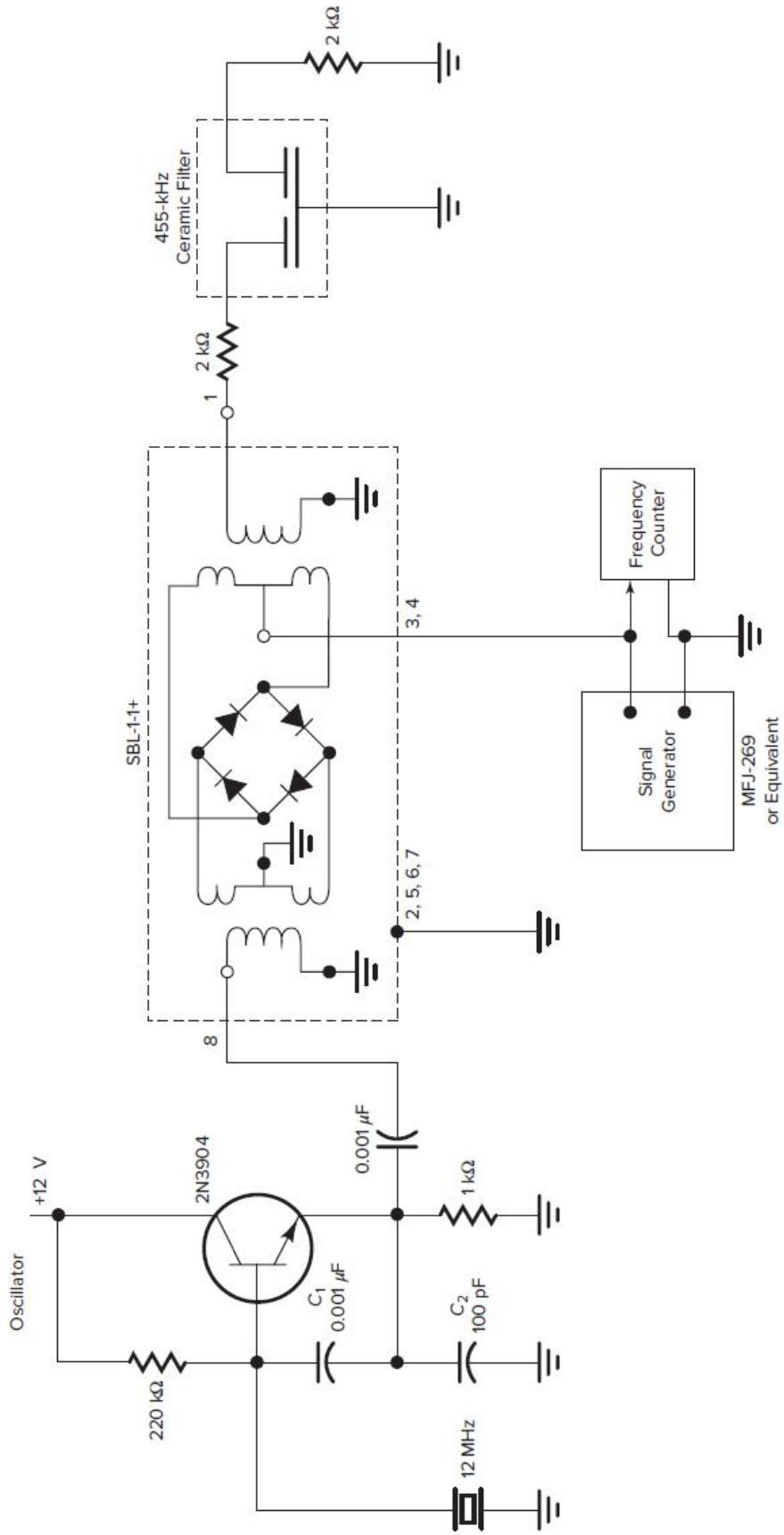


Fig. 9-6 Circuit for demonstrating mixer operation.

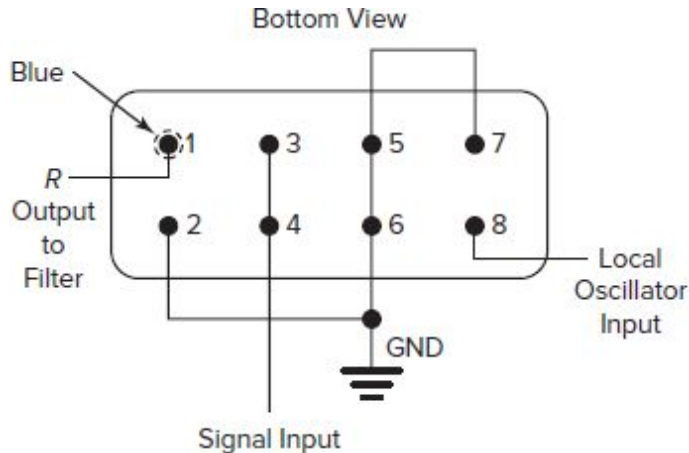


Fig. 9-7 Wiring for the SBL-1-1 +.

Project 9-4 Building and Testing a Communications Receiver

Objective

Gain hands-on experience with actual communications components, equipment, and testing procedures by building a commercial kit receiver.

Required

- 1 Pacific Antenna
(<https://www.qrpkits.com/ezseries.html#ezreceiver>) Easy Receiver kit
- Soldering iron (<35 W), solder, hand tools.
 - Hookup wire

Introduction

There is probably no better way to understand and appreciate the complexity of modern communications equipment than by actually working with it. One way to do this inexpensively and in a relatively short period of time is to build a kit. A kit is a collection of components

that, when assembled and tested, forms a complete workable end product. There are a variety of electronic kits that can be quickly and easily assembled with a soldering iron and some basic hand tools. The end product is a workable electronic product. Many radio receiver kits are available, from simple AM and FM radios to more sophisticated amateur radio transceivers. This project recommends a specific communications receiver, the Easy Receiver by Pacific Antenna. It is a direct conversion design, with a bandpass filter. The receiver covers approximately 75 kHz on the 40 meter amateur radio band (7 MHz).

In this project, you will build the kit, analyze the receiver circuit, examine some of the integrated circuits in detail, test and tune the receiver, and use it.

Procedure

1. Read through the parts list in the kit's manual and familiarize yourself with the various components.
2. To aid in your understanding of the receiver circuit, it will be helpful if you will find and print out the data sheets for the integrated circuits used in this receiver. You can find these data sheets on the IC manufacturer's websites:
NE602 National Semiconductor (now Texas Instruments)
LM386 National Semiconductor (now Texas Instruments)
Reading the data sheets will help you in the circuit analysis.
3. Once you have familiarized yourself with the parts, proceed with the construction of the receiver. Be sure to follow the instructions carefully.
4. Once you have built the kit, test it using the procedure described on page 9 of the Easy Receiver kit's manual. A troubleshooting guide is located on page 10 of the manual, if needed.

Questions

1. What is the function of the LM386 IC?
2. What is the main function of the NE602 IC?
3. What is the function of C6?
4. What is the function of R2?
5. What is the function of the in-line fuse and diode D3?

6. Where is the function of R5?
7. This is a direct conversion receiver. What are some of the key benefits of a direct conversion receiver?
8. What are some of the disadvantages of a direct conversion receiver?
9. If the DC supply voltage is 9 V, where does the 6.2-V supply line come from?

NAME

DATE

A graphic for Chapter 10 featuring the text "Chapter 10" in a white, sans-serif font. The text is centered within a dark, irregular, cloud-like shape with a gradient from black to grey.

Digital Data Transmission

Project 10-1 Serial Digital Data

Objective

To demonstrate one way to develop serial binary data and how to interpret an oscilloscope display of that data.

Required

- 1 dual-trace oscilloscope
- 2 4035B 4-bit CMOS shift register ICs
- 1 555 timer IC
- 1 10-bit DIP switch
- 11 4.7-k Ω resistors
- 1 22-k Ω resistor
- 1 0.01- μ F capacitor
- 1 100- μ F electrolytic capacitor

Introduction

In virtually all data communications applications, binary data is transmitted serially. That is, rather than transmit all bits of a binary word simultaneously or in parallel, the data is transmitted one bit at a

time. The reason for this in communications applications is obvious. To transmit parallel data, multiple paths or channels, one per bit, are required. In most communications systems, only a single channel is available. For that reason, multiple binary words are transmitted one after another, one bit at a time.

On the other hand, in electronic equipment that is not used for communications, binary data is usually generated and manipulated by parallel methods. This is particularly true of computers. For this reason, the processes of parallel-to-serial and serial-to-parallel data conversions are important.

The circuit that is most commonly used to perform parallel-to-serial and serial-to-parallel data conversions is the shift register. A shift register is a digital circuit made up by cascading D or JK flip-flops. To translate a parallel binary number into a serial number, the parallel number is loaded into the shift register flip-flops, and then a clock signal is applied simultaneously to all of the flip-flops. The resulting data is shifted one bit to the right (or to the left in some cases), thereby generating a serial train of data at the output flip-flop.

The process of parallel-to-serial conversion is illustrated in Fig. 10-1. An 8-bit parallel binary number is loaded into the flip-flops of an 8-bit shift register, and then clock pulses are applied to all flip-flops simultaneously. When the first clock pulse occurs, all of the bits in the binary number are shifted one position to the right. The bit in the right-hand flip-flop is shifted out. That bit is then applied to some other circuit or transmitted over the communications channel. An example is the modulator in a modem for transmission over telephone lines.

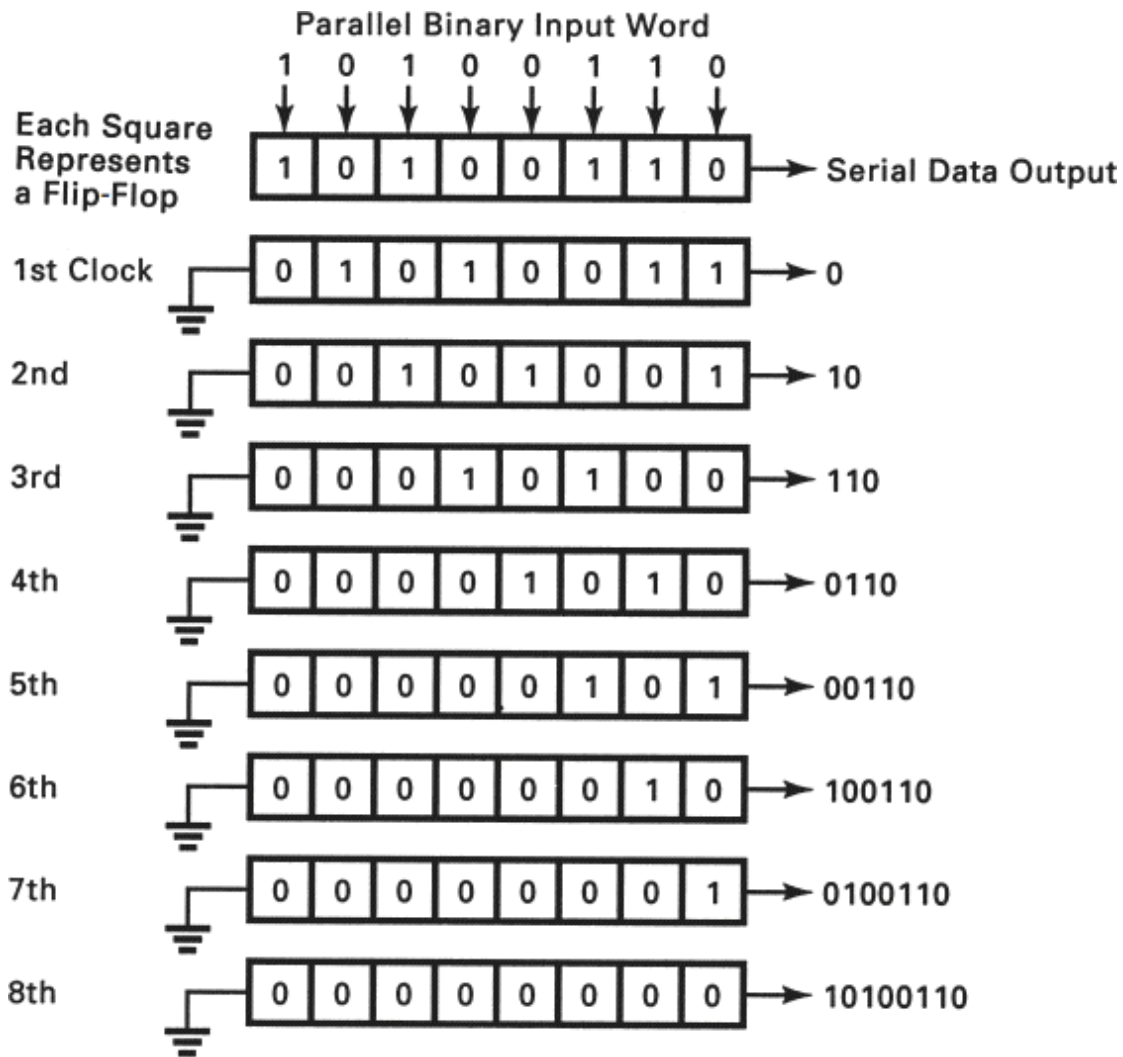


Fig. 10-1 Shift register used for parallel-to-serial data conversion.

When the second clock pulse occurs, all of the bits are again shifted one position to the right. The bit in the rightmost position is shifted out. With each succeeding clock pulse, the bits in the shift register are moved to the right as the number is shifted out. By the time the eighth clock pulse occurs, all bits have been shifted. Note that the input to the left-hand flip-flop of the shift register is connected to ground. That represents a binary 0. As the data is shifted to the right, binary 0s are shifted into the flip-flops.

If you were to observe the serial data output of the shift register of Fig. 10-1, you would see a signal like that shown in Fig. 10-2. The upper waveform is the clock, a series of periodic pulses. The clock

pulses are numbered to correspond to the clock signals designated in Fig. 10-1.

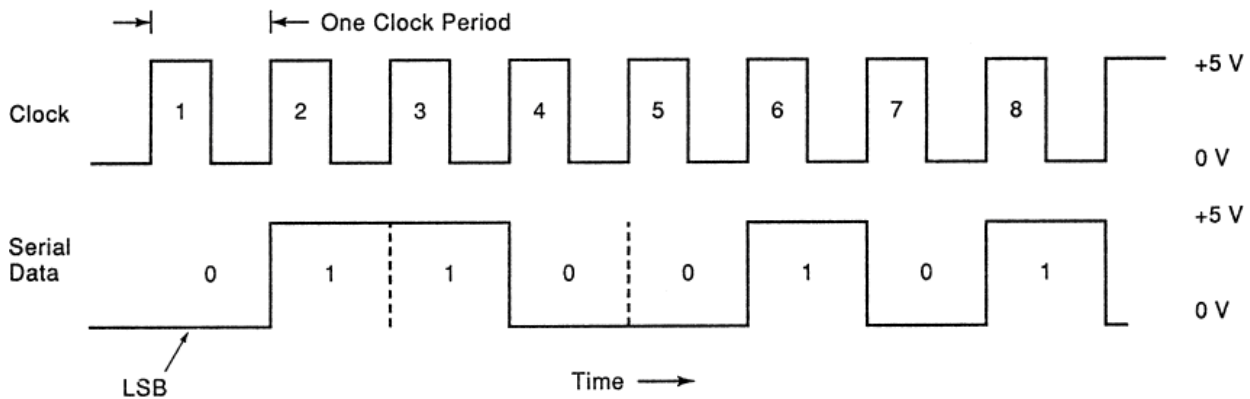


Fig. 10-2 The serial binary waveform.

The serial data shows the voltage waveform at the output page 89 of the right-hand flip-flop in the shift register. After the first positive-going transition of clock pulse 1, a binary 0 output appears. When the second positive-going transition of clock pulse 2 occurs, a binary 1 appears at the output. Note the sequence of the bits in Fig. 10-2 and compare it to the sequence of the bits in Fig. 10-1. They appear to be reversed. The reason is that an oscilloscope displays voltage levels with respect to time, with time proceeding from left to right. You will see the binary data displayed from left to right on the screen as it is shifted out.

An important consideration in transmitting serial data is which bit of a binary word is transmitted first, the least significant bit (LSB) or the most significant bit (MSB). This is important to know so that you can interpret the data after it has been transmitted. Various schemes are used in data communications systems, in which either the MSB or LSB is transmitted first. You must become familiar with the protocols and the system specifications to determine this information.

For example, when ASCII data is transmitted, the LSB is usually transmitted first. Recall that the ASCII system represents letters of the alphabet, numbers, punctuation symbols, and other characters as 7-bit binary numbers. The serial word in Fig. 10-2 might represent the ASCII 0100110. The LSB in that case would be the zero at the left-hand side of the waveform. The eighth bit, which is a binary 1, will

usually represent a parity bit. The ASCII number in Fig. 10-2, 0100110, therefore represents the ampersand (&). The eighth or parity bit being a binary 1 means that an even-parity system is used because there are an even number (4) of binary 1s in the total transmitted value. Note in Fig. 10-2 that, since the LSB is transmitted first, it appears on the left because it occurs earlier in time than the other bits.

In this experiment, you are going to demonstrate the generation of serial binary data by using a shift register. Two 4-bit CMOS shift registers are combined to form an 8-bit shift register. Parallel data will be loaded into this register by way of a set of DIP switches. An oscillator made up of a 555 timer IC will be used to produce the clock pulses to cause the data to be shifted out. In order that you can view the data on an oscilloscope, the serial data output from the shift register is connected back to the serial input of the shift register. In other words, as the data is shifted out, it is also shifted back into the same register. That allows the word to be shifted out and then reloaded again and again for repetitive viewing on an oscilloscope. By loading in different binary values and displaying the clock pulses and serial data output simultaneously, you will be able to determine the relationship between the parallel input bits and the serial output bits.

Procedure

1. Construct the circuit shown in Fig. 10-3. Because of the complexity of the circuit, be extremely careful in making the interconnections to avoid wiring errors.

The pin-out diagram for the 4035B shift register is shown in Fig. 10-4; that is how the pins on the 16-pin DIP are numbered when you view the IC from the top. The P1 through P4 pins accept a 4-bit parallel binary input word. The individual flip-flop outputs are designated Q1 through Q4; Q1 is the flip-flop nearest the serial input. The serial input is applied to the J and K NOT inputs on pins 3 and 4. Usually these two pins are tied together. Pin 6 accepts the CLOCK input. Pin 5 is the reset (RST) line. Bringing this line high or to binary 1 causes all flip-flops to be simultaneously reset. The LOAD input at pin 7 is used to cause

the parallel binary inputs to be loaded into the shift register flip-flops. Normally this pin is held low, or binary 0, for normal shifting operations. To load the parallel word, pin 7 is brought high, or binary 1. The occurrence of the first clock pulse after pin 7 goes high causes the data to be loaded. This pin must then again be brought low in order for shifting operations to take place.

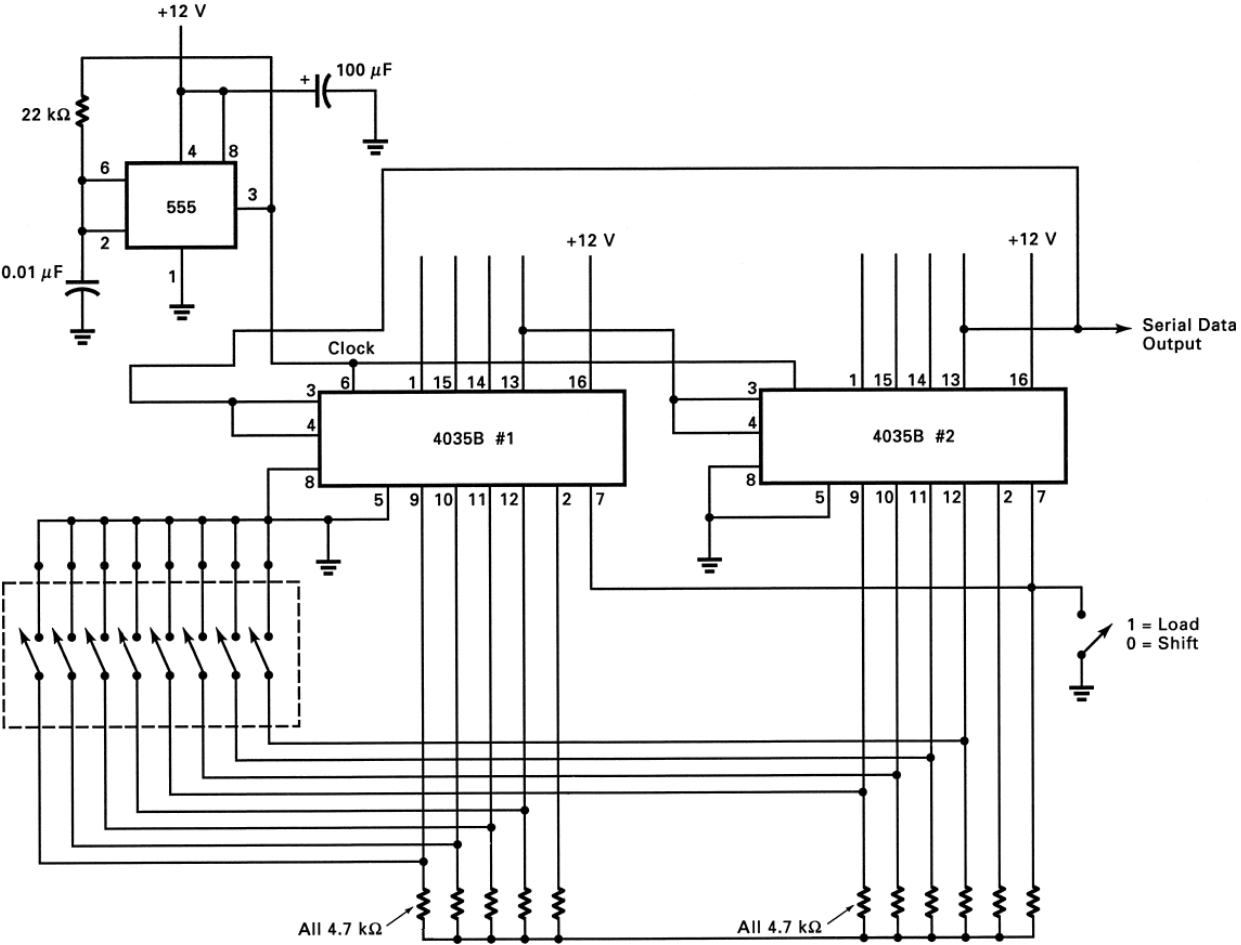


Fig. 10-3 An 8-bit parallel-to-serial data converter.

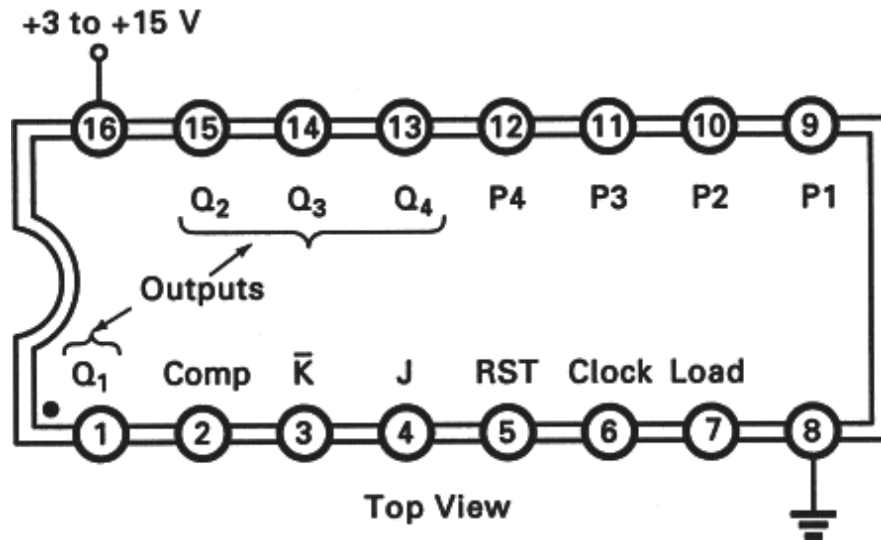


Fig. 10-4 Pin-out diagram of 4035B 4-bit CMOS shift register.

The COMP input at pin 2 is used to control the state of the flip-flop outputs. Recall that flip-flops usually have two outputs: a normal and a complement. When pin 2 is high, or binary 1, the Q_1 through Q_4 outputs are their normal binary values. If pin 2 is brought low, outputs Q_1 through Q_4 are the complement values. In this experiment, the normal outputs are used. DC power is applied to pins 16 and 8.

2. To test your circuit, apply power to it, but first see that the clock is working by checking for a clock waveform at pin 3 at the 555 timer. At that time, use the calibrated horizontal scale of the oscilloscope to measure the period of the clock and compute its frequency. Alternatively, use a frequency counter to measure the clock output.
3. Test the shift register by loading a binary 1 into the LSB flip-flop of the No. 1 4035B shift register. To do that, the DIP switches should be set so that all of them are closed except the one connected to pin 9 of the No. 1 4035B IC. With that switch open, a positive voltage will be applied to pin 9. This positive 12 V represents a binary 1.

The load DIP switch should normally be in the closed position for proper operation. However, to load the binary 1 into the shift register, momentarily switch the DIP switch so that it is open. That applies a binary 1 to pin 7 on both shift register ICs. When

the next clock pulse occurs, that binary 1 will be loaded into the LSB of the shift register. Then reset the load switch to the closed, or binary 0, position.

You can now monitor the serial data output on the oscilloscope by observing the signal at pin 13 on the No. 2 4035B IC. If you have a dual-trace oscilloscope, display the clock waveform on one trace and the serial data output on another trace. In that way, you will be better able to relate the output with the clock pulse occurrences. You should see an output pulse whose width is that of one clock period occurring once every eight clock pulses.

4. The best way to observe the serial data output is to adjust the horizontal sweep frequency control so you display exactly one 8-bit serial word on the screen. Adjust the sweep to display exactly eight complete cycles of the clock waveform on the upper trace of the oscilloscope. One complete 8-bit word of serial data will then be displayed on the lower trace.
5. To see the variation in the output waveform based on page 92 the actual value loaded into the shift register, try loading in different binary numbers and observing the binary output. The following are some examples:

10101010

00110011

11101110

00001111

In each case, set the DIP switches to apply the number to the shift register. The LSB shown above will be the value loaded into what we are calling the LSB of the shift register, which in this case is input pin 9 on the No. 1 4035B IC. The MSB then is pin 12 on the No. 2 4035B IC. Once the DIP switches have been set to the desired number, momentarily apply a binary 1 to pin 7 of the ICs with the load DIP switch. You should immediately note a change in the output pattern on the oscilloscope screen. Be sure that you can relate the binary numbers above to the corresponding oscilloscope patterns that they create.

6. Based on the clock frequency measured in the first step, how long does it take to transmit a single 8-bit binary word?
7. Draw a diagram similar to the one in Fig. 10-1 that illustrates how a serial binary data word is captured by shifting it into a shift register. Illustrate the process in the same number of steps to show how serial-to-parallel data transfer occurs. Use the ASCII code for the letter M with odd parity.

Project 10-2 Frequency Shift Keying

Objective

To demonstrate the generation of a frequency shift keying signal.

Required

- 1 dual-trace oscilloscope
- 1 square wave signal source
- 1 2206 function generator IC
- 1 150- Ω resistor
- 2 4.7-k Ω resistors
- 1 22-k Ω resistor
- 1 47-k Ω resistor
- 1 56-k Ω resistor
- 1 0.001- μ F capacitor
- 1 1- μ F capacitor
- 1 10- μ F capacitor

Introduction

Frequency shift keying is a modulation technique used to transmit binary data over analog communications channels such as the telephone lines. The binary data operates an FSK modulator that translates the binary signals into two different tones or sine waves. The frequencies chosen depend on the bandwidth of the analog channel as well as the speed of the binary data. Because telephone lines have a very narrow bandwidth (approximately 3 kHz), low frequencies are used. For example, in low-frequency modems, the

frequencies of 1070 and 1270 Hz are used to represent the binary values 0 and 1, respectively. In full-duplex systems, the tones 2025 and 2225 Hz are also used to represent binary 0 and 1, respectively. When binary 0 occurs, the low-frequency tone is generated; when a binary 1 occurs, the high-frequency tone is generated. Other schemes have also been used. Because such low frequencies are used, the data rate is restricted; the maximum with such a system is typically in the 300 bits/s region. For higher data rates, other types of modulation such as PSK and QAM are used.

In this experiment you will demonstrate the generation of FSK and observe the output signal. You will use the 2206 function generator IC that you used in preceding experiments. This IC was designed specifically for FSK applications. The serial binary input will be simulated by a square wave.

Procedure

1. Construct the circuit shown in Fig. 10-5. You may have retained the 2206 function generator circuit used in many preceding experiments. If so, modify it so that it appears as shown in Fig. 10-5.
2. Apply a square wave to pin 9 of the 2206 IC; it will simulate a serial binary input. The square wave should have an amplitude that switches between 0 and approximately +5 V. Any TTL data source will be suitable. Set the frequency to approximately 1.5 kHz.
3. Apply power to the 2206. Monitor the FSK output at pin 2. Because the input signal repeatedly switches the output between two different sine-wave frequencies, scope triggering will be somewhat erratic. However, through a combination of scope triggering and horizontal sweep frequency control adjustments, you should be able to stabilize the waveform and see the two different frequencies as the input signal switches between binary 0 and binary 1. Use the variable sweep frequency and trigger level controls to stabilize the waveform. If you have a dual-trace oscilloscope, you can display the binary input signal on the upper trace and the FSK output signal on the lower trace.

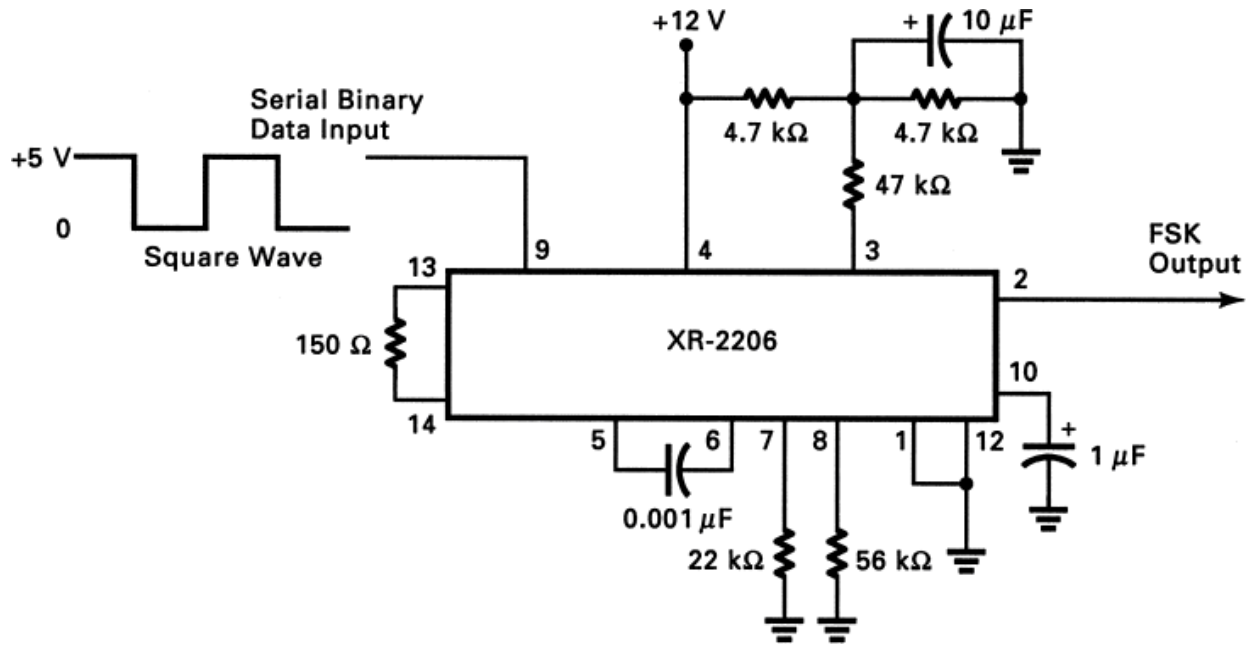


Fig. 10-5 FSK generator.

4. Determine which binary input state produces which output frequency. One way to do that is to remove the square wave from pin 9 and then ground pin 9 to provide a fixed binary 0 input. Measure the 2206 output frequency at pin 2 with a frequency counter or on the oscilloscope. Apply +5 V dc to pin 9 and again measure that 2206 output at pin 2. That will simulate a binary 1 input. Record the information you have collected.
5. The frequencies of the output tones produced by the 2206 are determined by resistor and capacitor values; specifically, the $0.001\text{-}\mu\text{F}$ capacitor connected between pins 5 and 6 and the 22- and 56-k Ω resistors connected to pins 7 and 8 in Fig. 10-5. Knowing that, what steps would you take to reverse the frequency tones you determined in step 4? That is, how would you change the tone obtained with binary 0 so that it is produced when a binary 1 occurs, and vice versa?
6. Suggest a type of circuit that might be used to demodulate an FSK signal.

Project 10-3 Binary Phase-Shift Keying (BPSK)

Objective

To demonstrate a method of generating and detecting a BPSK signal.

Required

- 1 oscilloscope (dual-trace, 50-MHz minimum)
- 2 function generators (sine wave and TTL output)
- 1 frequency counter
- 2 Mini-Circuits SBL-1-1+ diode ring lattice modulators/mixers
- 1 Maxim MAX202CPE RS-232 line driver IC
- 1 74LS13 TTL IC
- 1 0.001- μ F ceramic disk capacitor
- 6 0.1- μ F ceramic disk capacitors
- 1 0.15- μ F electrolytic capacitor
- 1 330- Ω resistor

Introduction

Binary phase-shift keying (BPSK) is one of the most widely used types of modulation in transmitting digital data. The digital data is used to shift the phase of a constant-frequency carrier. In BPSK, a binary 0 is transmitted as a carrier with no phase shift, while a binary 1 is transmitted with a 180° phase shift (phase inversion). This is illustrated in Fig. 10-6.

There are many variations of PSK. For example, quadrature PSK (QPSK) has four phase shifts spaced by 90° ; 8PSK has eight phase-shift values, and so on. These forms of PSK permit each phase to represent multiple bits so that higher data can be achieved. With QPSK, each phase-shift value represents 2 bits of data. With QPSK, the data rate is twice what it is with BPSK.

In this experiment you will see a way to generate BPSK with a diode lattice modulator. The basic circuit is shown in Fig. 10-7. The sine-wave carrier is fed to a transformer, which applies the carrier to a set of diode switches. The diode switches determine the connection of the signal from the secondary winding of T_1 to the primary winding of T_2 . The output is taken from the secondary winding of T_2 .

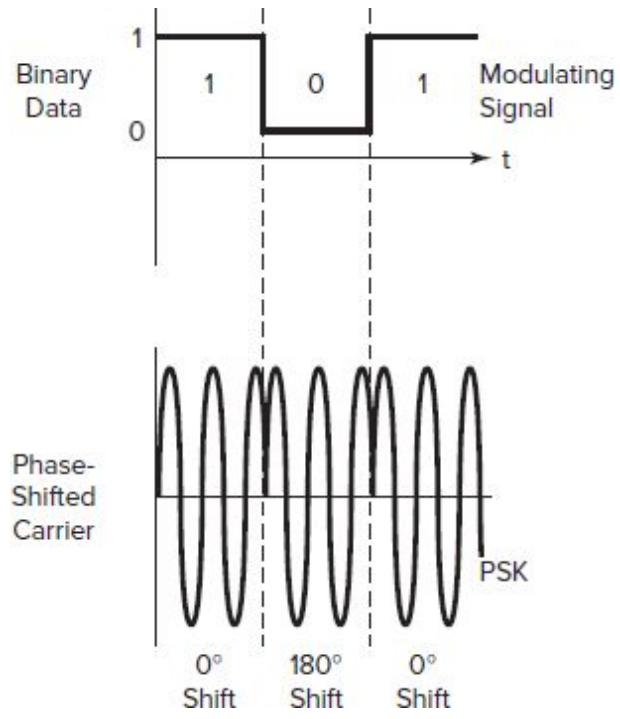


Fig. 10-6 BPSK.

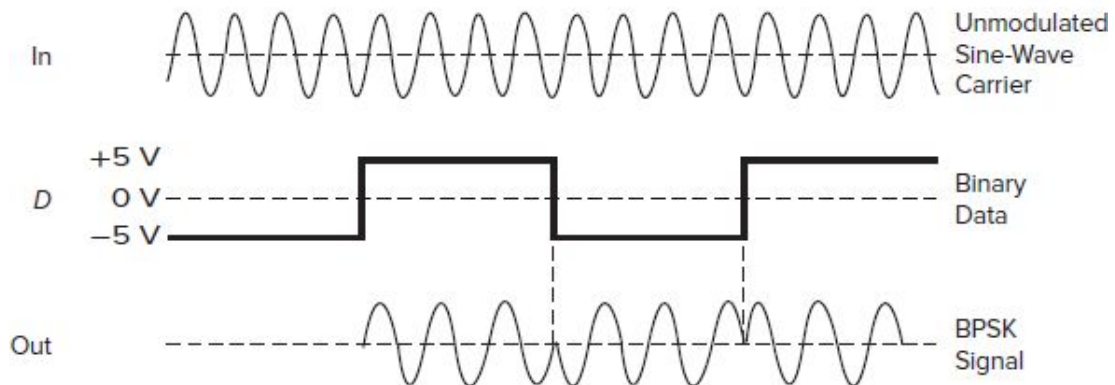
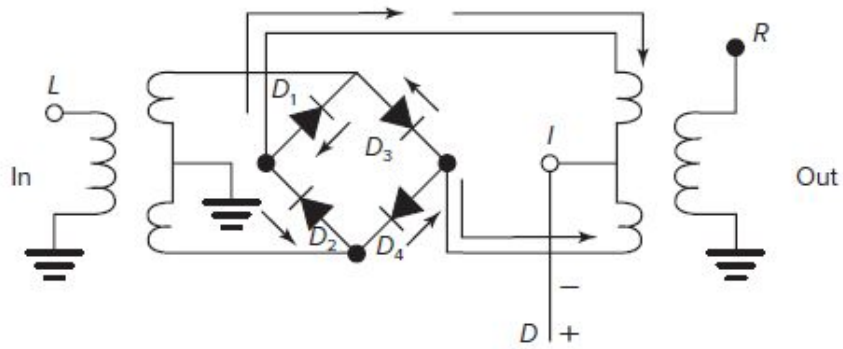
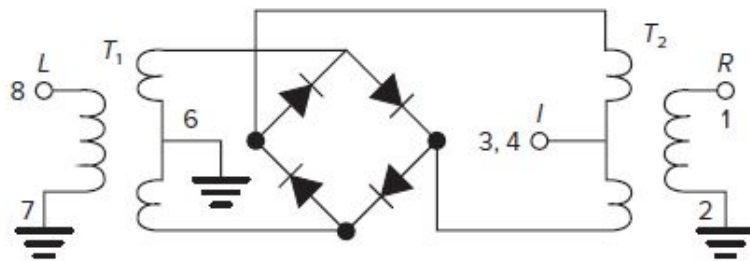
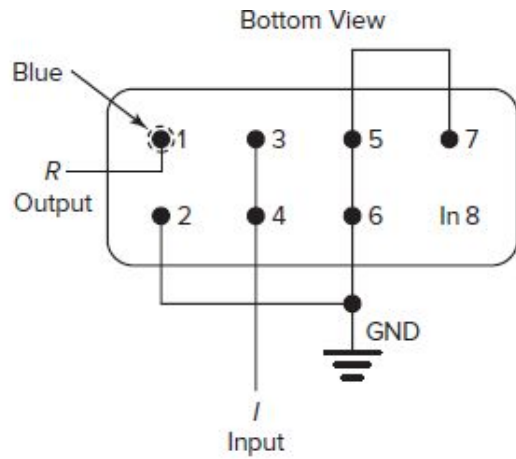


Fig. 10-7 Diode lattice modulator.

The state of the diodes (off or on) is determined by the page 96 binary input signal. This signal must be bipolar. A typical signal is +5 V to +10 V for a binary 1 to -5 V to -10 V for a binary 0, or vice versa.

Although this circuit can be constructed with discrete components, diode lattice modulators are available as a packaged device. One of these is the Mini-Circuits SLB-1-1+, which is also widely used as a mixer. These low-cost circuits have matched transformers and diodes. This circuit is of the form shown in Fig. 10-7. When a binary 1 is applied (+5 V), diodes D_1 and D_4 are forward biased by the input signal and conduct. D_2 and D_3 are cut off. The conducting diodes act like a short circuit and connect the signal from the secondary of T_1 to the primary of T_2 . The windings are arranged so that no phase inversion occurs.

When a binary 0 occurs, the signal applied to the page 97 modulator is -5 V. This reverse-biases D_1 and D_4 , causing them to cut off. D_2 and D_3 conduct and act as a short circuit to connect the upper lead of the T_1 secondary to the lower lead of the primary of T_2 and the lower lead of the secondary of the T_1 to the upper lead of the primary of T_2 . This connection causes T_2 to produce a 180° phase shift at the output.

This experiment uses a special IC (Maxim MAX202CPE) to convert the conventional TTL or CMOS binary voltages into the ±10-V signals necessary for the proper operation of the diode lattice modulator. The circuit uses the +5-V supply voltage to power a circuit that automatically produces +10-V and -10-V sources to power the circuit. No external negative power supply is needed.

Procedure

1. Connect the circuit in Fig. 10-8. Note the bottom view of the SBL-1-1+ in Fig. 10-7, which shows the pin numbers that correspond to the numbers and connections shown in the schematic. Were the SBL-1-1+ first as shown in Fig. 10-7, then connect it as shown in Fig. 10-8. You will use a function

generator to produce the sine-wave carrier. The binary signal can be simulated by any standard digital source. If a second tunable digital function generator is not available, use the simple circuit shown in Fig. 10-9.

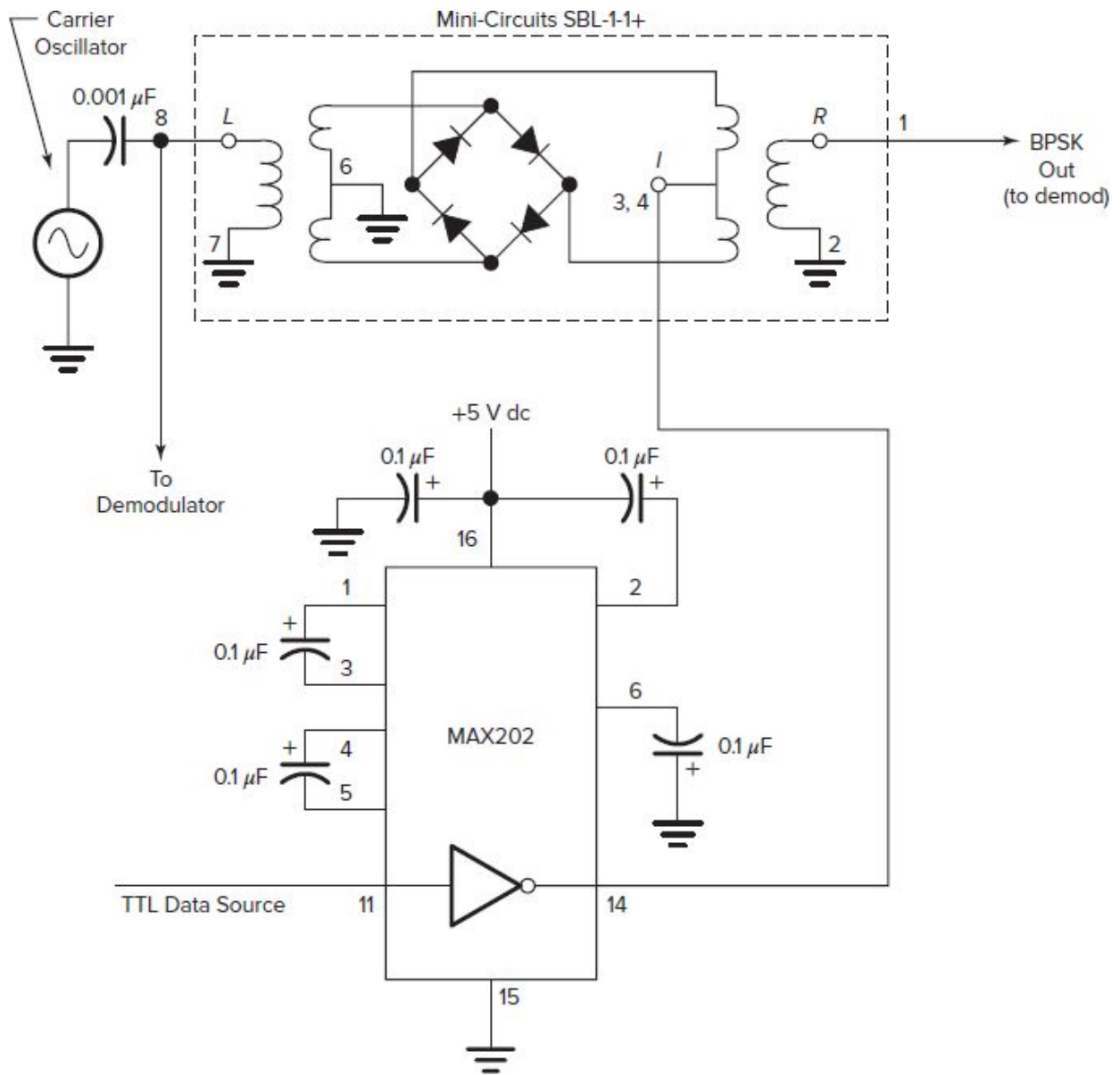


Fig. 10-8 BPSK modulator.

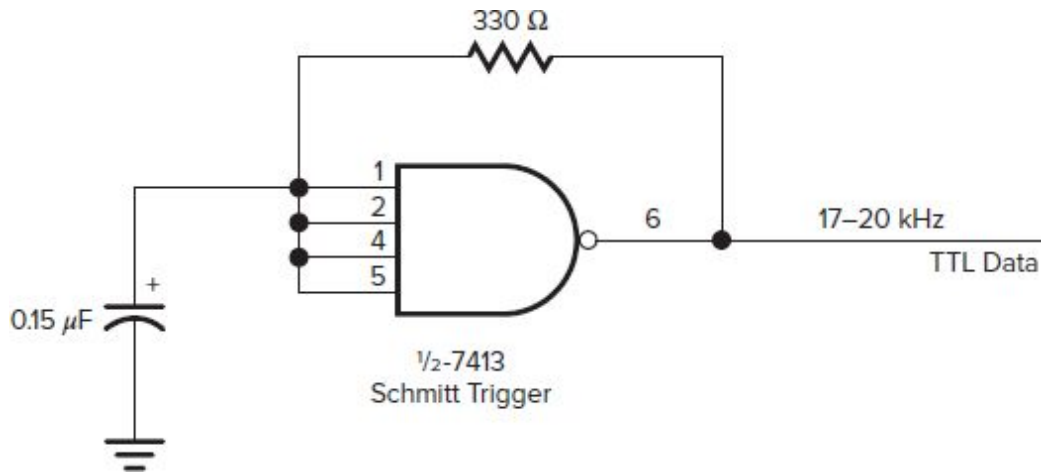


Fig. 10-9 A TTL Schmitt trigger used as a binary data source.

2. Turn on the function generator and set the sine-wave carrier level in the primary of the SBL-1-1+ to $0.5 V_{p-p}$ at a frequency of 300 kHz. Apply dc power to the remainder of the circuit. If you are using a separate function generator for the digital signal, set its frequency to 20 kHz. Use the TTL output from the function generator. If you build the circuit in Fig. 10-9, its frequency is already set to approximately 17 kHz if you used the *RC* values specified.
3. Now, observe the output of the MAX202 IC at pin 14 that is applied to pin 4 of the SBL-1-1+. Measure its voltage levels. Is it bipolar?
4. Next, observe the BPSK output at pin 6 of the SBL-1-1+. Use the second input channel of the scope and display the modulated output along with the modulating signal. Draw the waveforms that you see.
5. Adjust the frequency of the carrier generator until you see a BPSK waveform that is as close to those in Fig. 10-6 as possible. You will need to tune the function generator carefully to synchronize the carrier and modulating signal frequencies. Measure both the carrier and data frequencies with the counter.
6. Next, build the demodulator circuit shown in Fig. 10-10. The BPSK signal from the modulator is applied to the primary of T_1 and the carrier is applied to pin 1 of the SBL-1-1+ in the

demodulator. The output is taken from pin 4 of the SBL-1-1+ in the demodulator.

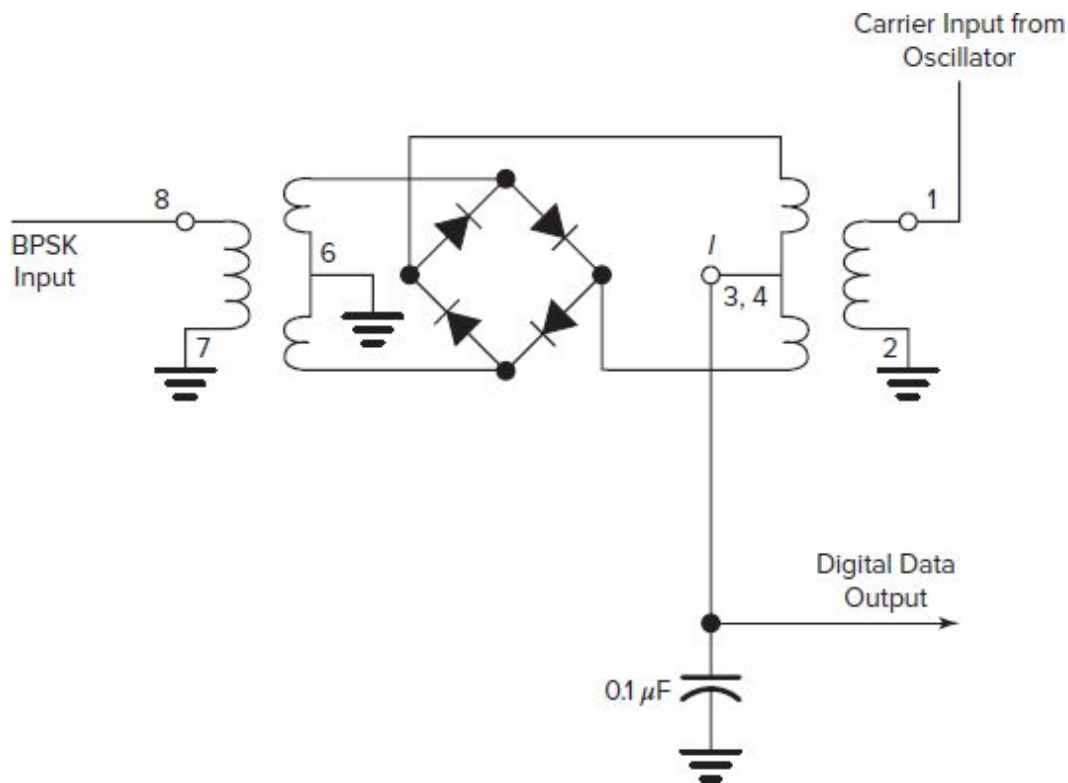


Fig. 10-10 BPSK demodulator.

7. Observe the output of the demodulator. What do you see? Try adjusting the carrier frequency slightly to maximize the output signal. Measure the output voltage. Explain what you see.

Questions

1. Explain why the BPSK output does not start and stop perfectly at the zero crossing of the carrier sine wave. How does adjusting the carrier frequency help correct this condition?
2. Explain how the demodulator works.
3. In this experiment, you used the same carrier signal source for both modulation and demodulation. In separate transmitters and receivers, name the source of the receiver carrier signal and explain how it is derived.
4. What is the capacitor in the demodulator used for?

NAME

DAT

E _____

Chapter

11

Multiplexing, Duplexing, and Multiple Access

Project 11-1 Pulse Amplitude Modulation and Time Division Multiplexing

Objective

To test a circuit that will permit demonstration of both pulse amplitude modulation and time division multiplexing.

Required

- 1 oscilloscope
- 1 sine-wave signal source
- 1 square-wave signal source
- 1 555 timer IC
- 1 4016 or 4066 quad bilateral CMOS switch IC
- 1 NPN transistor (2N3904, 2N4401, etc.)
- 2 10-k Ω pots
- 1 4.7-k Ω resistor
- 2 15-k Ω resistors
- 1 22-k Ω resistor

- 1 0.01- μ F capacitor
- 1 10- μ F electrolytic capacitor

Introduction

In pulse amplitude modulation, the information signal to be transmitted is turned off and on periodically by a gate circuit. The gate, usually some kind of switch, allows segments of the intelligence signal to be passed through. The process is generally known as sampling the intelligence signal. The output of the gate is a series of periodic pulses whose amplitude follows the intelligence signal. This series of pulses is then used to modulate a carrier.

A key design factor in a PAM modulator circuit is the rate of sampling. To represent the intelligence signal accurately, the sampling rate must be at least 2 times the highest frequency content contained in the modulating signal. Higher sampling rates of 10 or more samples for the highest frequency in the intelligence signal are more desirable because they more accurately represent the information being transmitted.

The concept of PAM can be expanded to produce time division multiplexing. A time division multiplexer has two or more inputs or channels, each of which is sampled at a high rate of speed. A gate on each input intelligence signal samples its respective signal for a short period of time. Each of the signals is sampled one at a time, in sequence so that a segment of each signal appears in the output. For example, a four-input time division multiplexer would sample four input signals sequentially, one after the other. The cycling sequence then repeats again and again. The resulting time-division multiplexed output signal is a composite of the four intelligence signals sampled sequentially. The result is that four intelligence signals can be transmitted simultaneously over a single channel.

In this experiment, you are going to demonstrate PAM page 102 and time division multiplexing. You will use a solid-state switch that is operated by an astable multivibrator. You will observe both PAM and time division signals at the output.

The gate or switch used in this experiment is a CMOS switch. It is made of both P- and N-channel enhancement-mode MOSFETs connected in such a way that they act as a simple single-pole, single-

throw switch. When a binary 1 logic signal is applied to the switch, the switch turns on. When a logic signal is binary 0, the switch turns off. During the on period, the intelligence signal is passed. The 4016 and 4066 ICs contain four sampling switches.

One of these switches will be used to demonstrate PAM, and two of them will be connected together to form a two-channel multiplexer. A 555 IC timer connected as an astable multivibrator provides the sampling signal.

Procedure

1. Construct the circuit shown in Fig. 11-1. The 555 IC is used as an astable multivibrator to generate a square wave that will operate the CMOS switch. The resistor and capacitor associated with the 555 sets the 555's oscillating frequency. The 4016 or 4066 IC contains four CMOS switches, and you will use one of them as a gate to produce a PAM modulator circuit. Your signal source can be a function generator or any other source of a sine-wave intelligence signal. Apply the sine wave to the signal 1 input. The 10-k Ω pot will be used to adjust the signal amplitude. The output is taken from across the 4.8-k Ω resistor at the switch output. Check to be sure that each integrated circuit is connected to +12 V and ground before beginning.

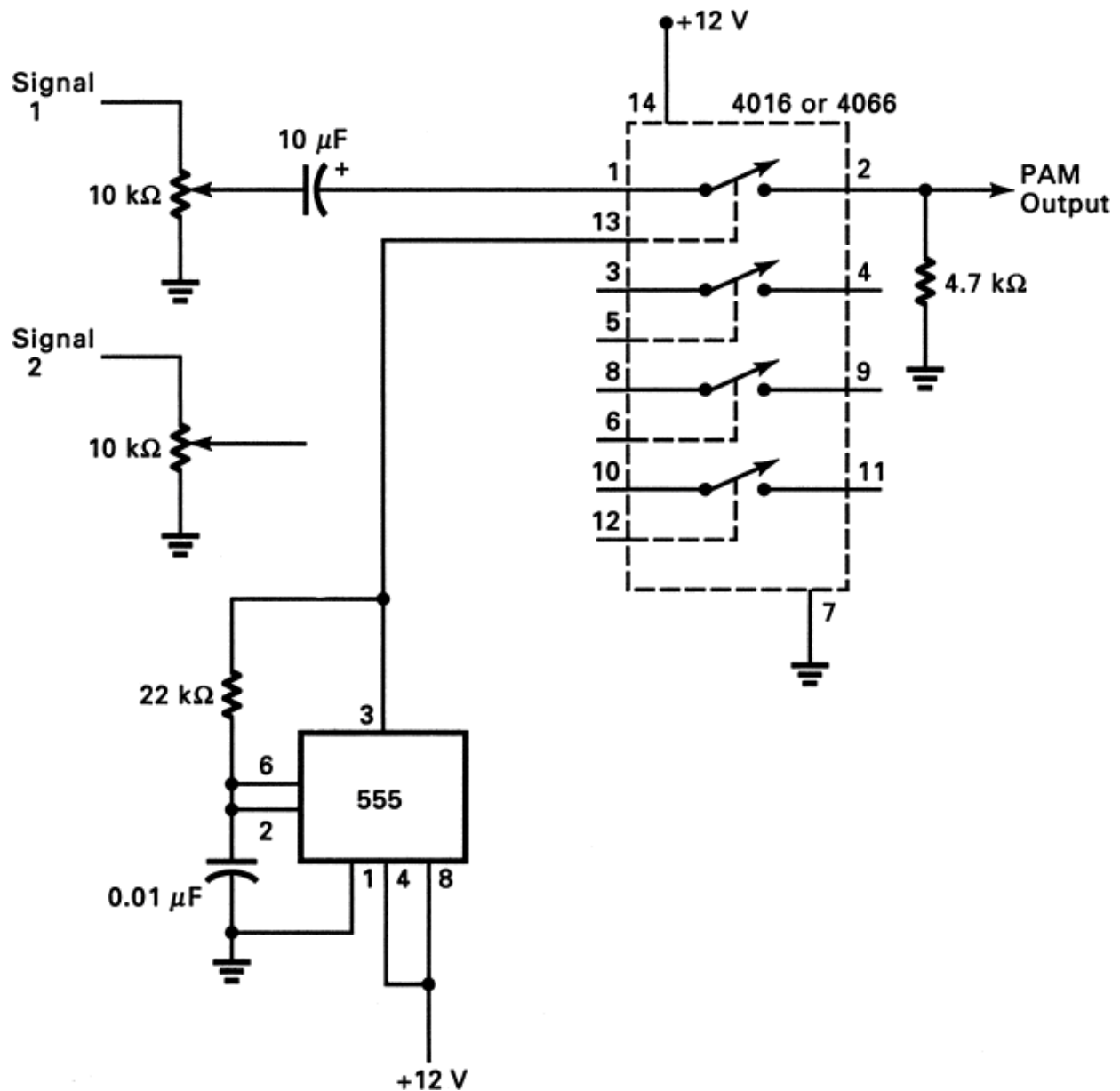


Fig. 11-1 PAM modulator.

2. Set the function generator supplying signal 1 for 150 Hz with an amplitude of several volts peak to peak. Then while monitoring the signal at the arm of the 10-kΩ potentiometer, adjust the pot for a signal amplitude of approximately $0.5 V_{p-p}$.
3. Apply power to the circuit. With an oscilloscope, observe the 555 oscillator output at pin 3. Use the calibrated horizontal sweep on

the oscilloscope or a frequency counter to determine the frequency of the sampling signal. Record your result.

Considering the frequency of the input intelligence frequency and the sampling frequency above, is the intelligence signal sampled fast enough to retain intelligibility?

4. Connect the oscilloscope to the 4.7-k Ω output resistor. You should observe a PAM output signal. Each time the sampling signal is a binary 1, or approximately ± 12 V, the switch closes. That allows a portion of the input signal to appear at the output. When the sampling oscillator produces a binary 0 output, the switch is open, thereby causing the output to be zero. In observing the PAM output, you should note that the switch passes both the positive and negative alternations of the input sine wave.
5. Modify the modulator circuit as shown in Fig. 11-2. You are connecting a pair of 15-k Ω resistors as a dc voltage divider to the gate input, and that provides a dc offset for the input signal. In other words, the ac input signal will be riding on a dc signal, as will the output.
6. Observe the output waveform. Note that the sampled output sine wave is riding on a dc level. The signal is offset from the zero baseline by a voltage level equal to that supplied by the voltage divider. The dc offset at the input to the switch is approximately equal to how many volts?
7. Turn off the power and modify the circuit so that it appears as shown in Fig. 11-3. Remove the two 15-k Ω voltage divider resistors, and then connect the second 10-k Ω pot to another of the solid-state switches in the 4016 or 4066 IC. Connect the switch output to the 4.7-k Ω resistor output as shown. Next, wire the transistor inverter circuit. This switching inverter provides a complement signal to the second gate switch. When the upper switch is on, the lower switch will be off, and vice versa. That means the two input channels will be alternately sampled depending on which switch the binary 1 signal is applied to.
8. Apply power to the circuit. Initially do not connect an input at the signal 2 pot. Set the pot for zero input signal and observe the output. At this time, you should see a signal like that you

obtained with pulse amplitude modulation. You are still observing a sampled version of the 150-Hz sine wave applied to signal input 1. However, during the time that the upper (A) switch is open, the lower (B) switch is closed. The signal at the No. 2 input is 0 V; therefore, you are seeing a 0-V level during the time that the B switch is closed. Applying another signal to the No. 2 input will cause that signal to appear at the output during the time the B switch is closed.

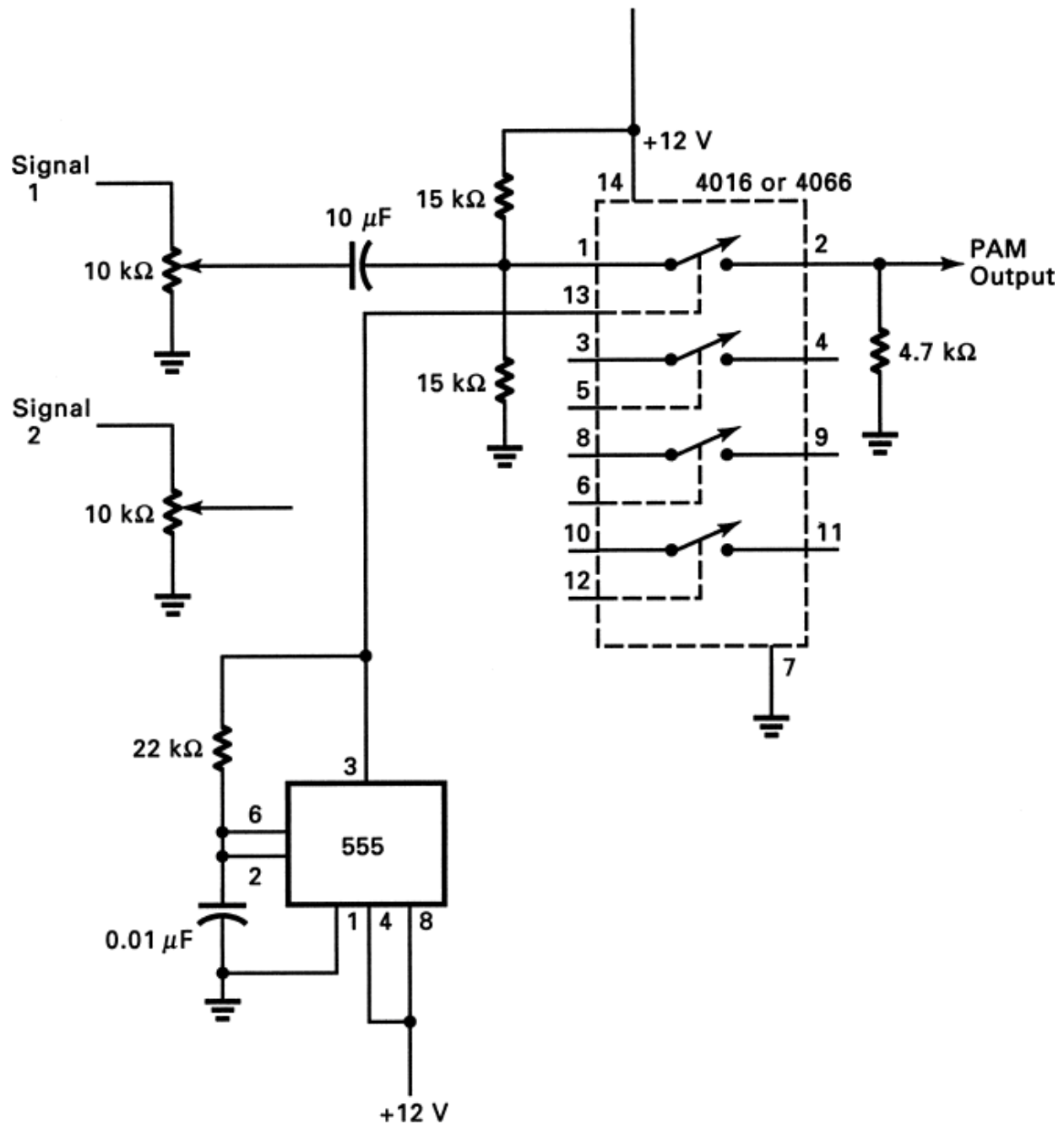


Fig. 11-2 PAM modulator with dc offset.

9. Connect a positive dc voltage to the signal 2 input. Adjust the dc voltage at the arm of the 10-kΩ pot for a value of approximately 0.25 to 0.3 V. Now observe the multiplexed output. What you are observing is the sampled 150-Hz sine wave and a dc voltage level other than zero. While observing the output signal, adjust the 10-kΩ pot on input signal 2 so you can see the output

variation. You can take this one step further and reverse the polarity of input signal 2 to show the effect.

10. Remove the dc voltage from the signal 2 input. Using a second function generator, connect a square wave to the signal 2 input. Set the frequency for approximately 200 Hz and an output that varies between zero and +5 V.
11. Observe the multiplexed output waveform. With all of the different signals involved—the 150-Hz sine wave, 200-Hz square wave, and the sampling oscillator signal—your oscilloscope will go crazy. By using the horizontal frequency and the triggering controls, you should be able to obtain a stable output waveform. Making minor adjustments in the square-wave input frequency will also help to stabilize the waveform for viewing. Of course, what you should see is a sine wave and a square wave appearing in the output, but sampled. The samples are, of course, interlaced, which causes the signals to be merged. If additional input signals were connected, these two would be merged with the composite signal. The resulting output signal is relatively complex and meaningless by itself. However, when it is applied to a demultiplexer circuit, the different signals will be unscrambled and the original modulating signals will be recovered intact.

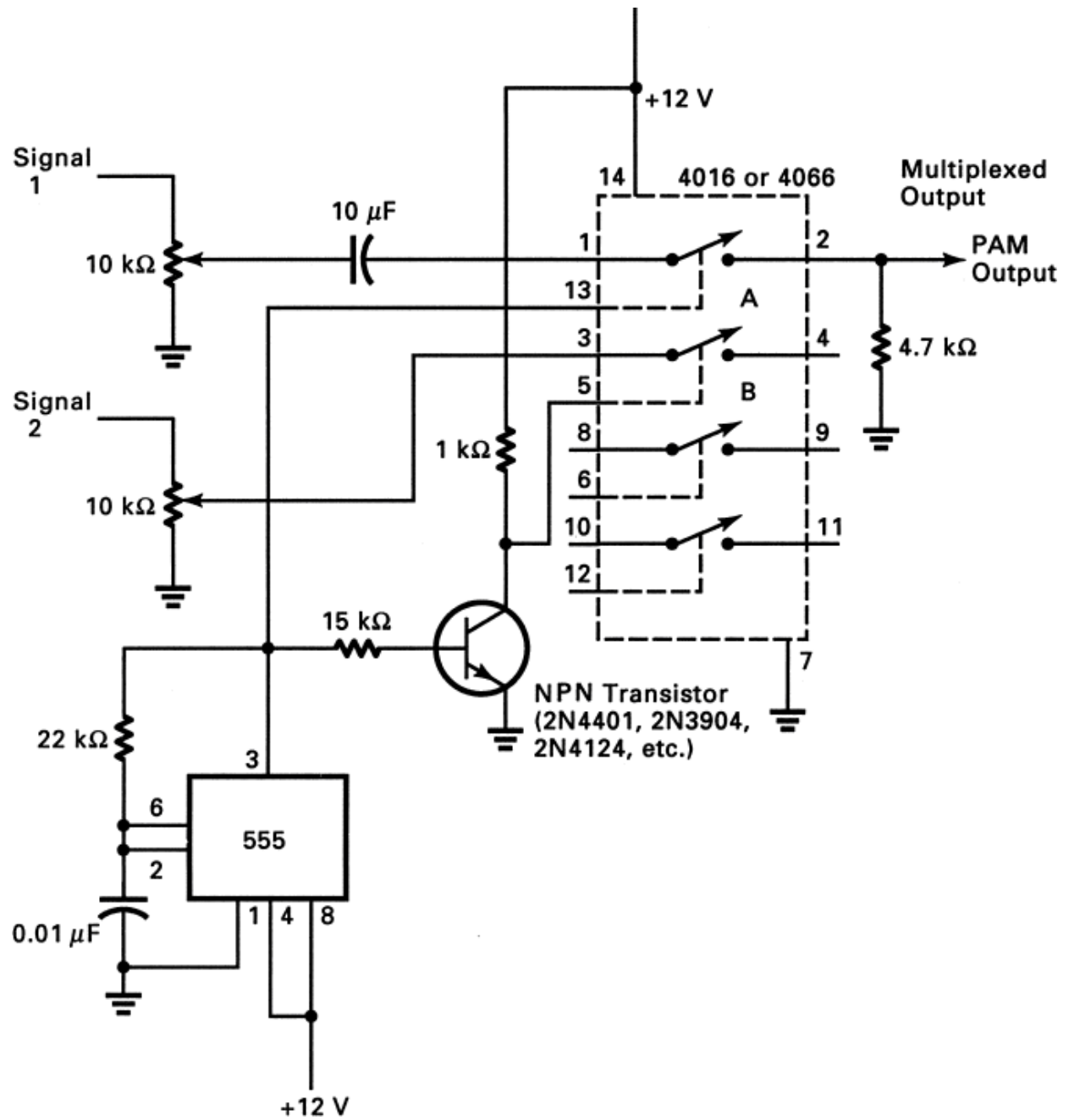


Fig. 11-3 Two-channel time division multiplexer.

NAME _____

page 107

_____ DAT

E _____

Chapter 12 has no experiments and therefore is not included in this Experiments and Activities Manual.

page 108

NAME

DAT

E _____

Chapter

13

Transmission Lines

Project 13-1 Measuring Swr

Objective

To use a standing wave ratio (SWR) meter to measure the SWR of a coax transmission line.

Required

- 1 50-ft length of coax cable (RG-8x or RG-58A/U) with PL-259 UHF connectors
- 1 SO-239 UHF coax connector
- 1 51- Ω resistor
- 1 33- Ω resistor
- 1 100- Ω resistor
- 1 MFJ-269 SWR analyzer
- 1 N-to-UHF coax adapter
- 1 solder lug

Introduction

This project will allow you to use a commercial meter for measuring SWR on a coax cable. That is a common test on antennas. The recommended instrument is the MFJ-269 analyzer, which will measure SWR over a frequency range of 1.8 to 470 MHz. The MFJ-269 has a built-in signal generator for this frequency range, a frequency counter, and the SWR and power measuring circuitry.

To prepare for this project, read the instruction manual for the MFJ-269 to learn how it operates.

Procedure

1. Using the SO-239 coax connector, build a load for the coax cable. Mount a solder lug on one corner of the connector with a #4 screw and nut as shown in Fig. 13-1. Solder the 51- Ω resistor to the center conductor, of the connector and the solder lug as shown. Screw the load onto one of the PL-259 UHF connectors on one end of the coax cable.
2. Given the cable and generator impedances and the load impedance, what load impedance and SWR do you expect to measure?
3. Attach the N-to-UHF adapter to the MFJ-269 output connector, then screw the UHF connector on the free end of the coax cable to this adapter.
4. Apply power to the MFJ-269. Rotate the band switch to the 10–27-MHz range. Then set the frequency to 12 MHz.
5. Read the load impedance and SWR from the MFJ-269 display. How do these figures compare with what you predicted? Explain any variations.

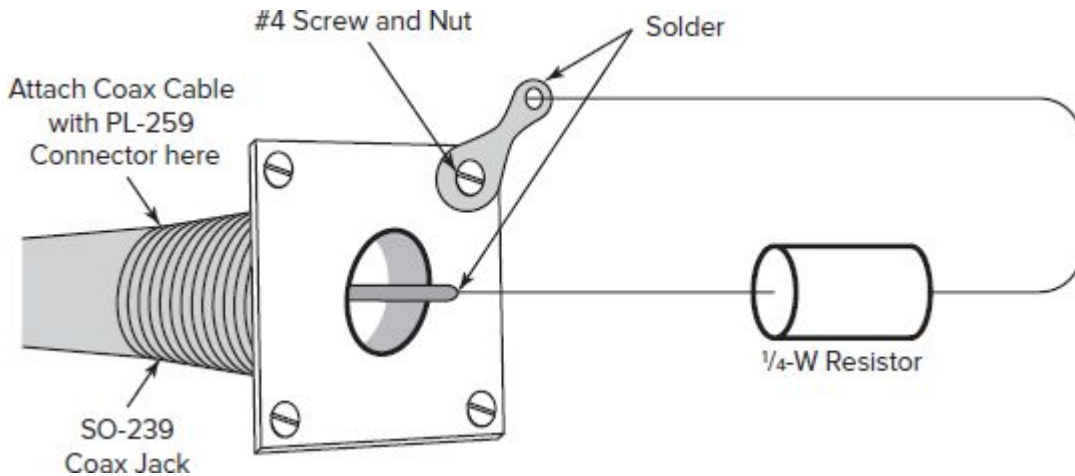


Fig. 13-1 Load for coax cable.

6. Remove power from the MFJ-269. Unsolder the 51- Ω resistor on the load and replace it with a 33- Ω resistor.
7. What impedance do you expect to read? What SWR?
8. Turn on the MFJ-269. Repeat step 5.
9. Turn off the power. Remove the 33- Ω resistor, and in its place solder a 100- Ω resistor.
10. What impedance do you expect to read? What SWR?
11. Turn on the MFJ-269. Repeat step 5.
12. Turn off the power.

Project 13-2 Measuring Coax Cable Loss

Objective

To use a test instrument to measure the dB loss of a coax cable.

Required

- 1 50-ft length of coax cable (RG-8x or RG-58A/U) with PL-259 UHF connectors
- 1 MFJ-269 SWR analyzer
- 1 N-to-UHF coax adapter

Introduction

All transmission lines attenuate the signal they deliver. Coax has a relatively high loss, so it is especially worrisome in most applications. This loss is commonly measured in dB loss per 100 ft. It is specified by the coax manufacturer. This loss also varies by frequency, so the dB loss figures are given at specific frequencies. The most common measurement frequencies are 10, 50, and 100 MHz. For example, the loss of a RG-58A/U cable is 1.4 dB at 10 MHz and 3.3 dB at 50 MHz. The loss of a RG-8x cable is 1.0 dB at 10 MHz and 2.5 dB at 50 MHz.

In this experiment, you will use the MFJ-269 analyzer to measure cable loss.

Procedure

1. Verify the type of cable you are using and, if necessary, measure its length.
2. Using the figures given in the introduction, calculate the estimated loss.
3. Connect one end of the coax cable to the UHF connector on the MFJ-269. The other end of the cable is left open. Do NOT connect it to anything else.
4. Turn on the MFJ-269 and set the frequency to 10 MHz.
5. Depress the Mode button on the analyzer. The cable loss will be displayed.
6. Compare your calculated and measured values. Explain any significant variations.
7. Set the Analyzer frequency to 50 MHz.
8. Repeat steps 5 and 6.

Project 13-3 Measuring Complex Load Impedance on a Transmission Line

Objective

To determine the impedance as seen by a transmitter or generator of a complex load connected to a coax cable using a Smith chart and to measure that impedance.

Required

- 1 4-ft length of RG-58A/U coax cable with a UHF connector on one end
- 1 82- Ω resistor
- 1 82-pF ceramic disc capacitor
- 1 MFJ-269 SWR analyzer
- 1 N-to-UHF coax adapter
- Smith chart graph paper

Introduction

Most antennas and loads in RF circuits and systems are complex, meaning they are resistive and reactive ($R \pm jX$). If the load is connected to a transmitter or other signal source through a coax cable, the cable will change the impedance as seen by the source. The cable translates an impedance to another value based on the cable characteristics and length as well as frequency. You can actually predict what that total impedance will be by using a Smith chart. Then you can measure that impedance with the analyzer.

Procedure

1. Construct the complex load by soldering the 82- Ω resistor and 82-pF capacitor in series to the coax cable as shown in Fig. 13-2.
2. Assuming an operating frequency of 14 MHz, calculate the complex impedance and plot it on a Smith chart.
3. Determine the wavelength of the 4 ft of cable at 14 MHz. Remember that the RG-58A/U cable has a characteristic impedance of 53.5 Ω and a velocity factor of 0.66.
4. Apply the procedure in the textbook to find the impedance as seen by the generator, using the Smith chart. Find the SWR from the chart.
5. Connect the coax cable to the MFJ-269. Turn the unit on and set the frequency to 14 MHz. Note the impedance and SWR values displayed.

6. Compare the calculated and measured values and account for any major variances.

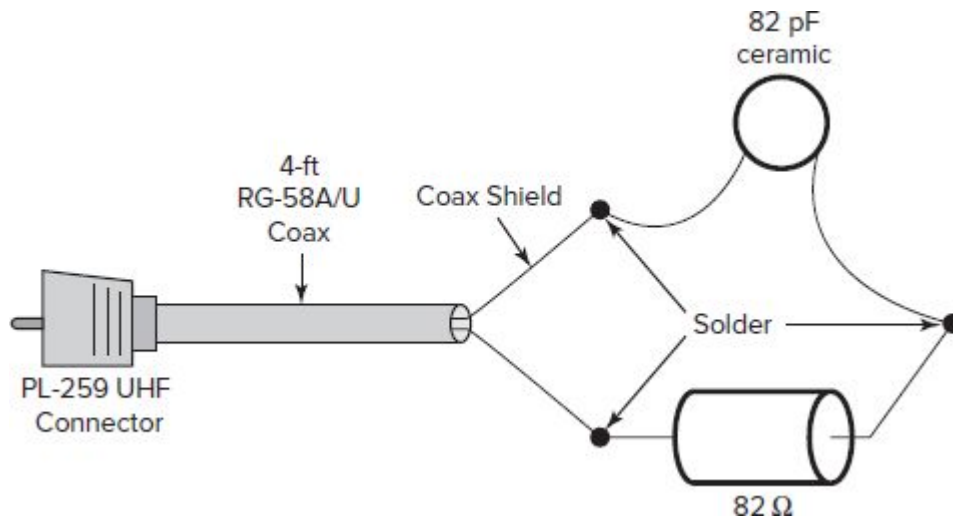


Fig. 13-2 Complex load on a coax cable.

Project 13-4 Transmission Lines and Time Domain Reflectometry (TDR)

page 113

Objective

To demonstrate the response of transmission lines to pulses and to show a method of detecting improperly terminated transmission lines using TDR techniques.

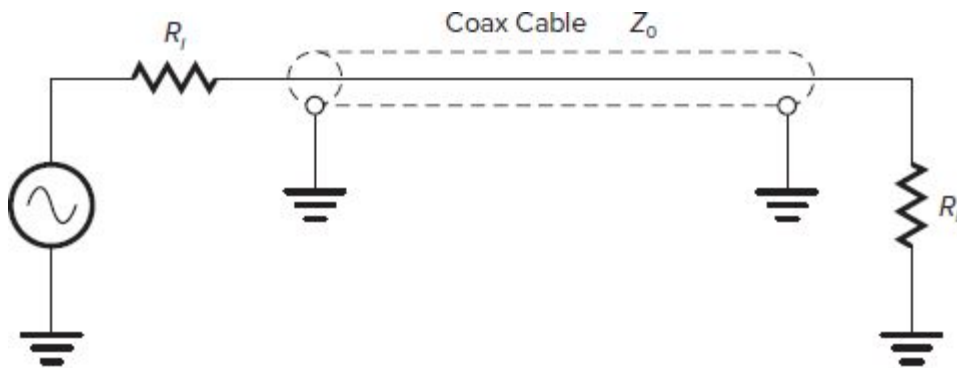
Required

- 1 oscilloscope (25-MHz minimum bandwidth, dual-trace)
- 1 function generator with TTL output square waves at 1 MHz (50 Ω output impedance)
- 1 RG-58/U 53-Ω coaxial cable, 50 ft long
- 1 22-Ω resistor
- 1 51-Ω resistor
- 1 100-Ω resistor

Introduction

Transmission lines such as coax cables and twisted pair lines must be properly terminated with their characteristic impedance to ensure maximum power transfer between the generator (usually a transmitter) and load (typically an antenna). This ideal situation is illustrated in Fig. 13-3. The generator output impedance matches the characteristic impedance of the coax line, which, in turn matches the impedance of the load. With this arrangement, maximum power is dissipated in the load. This value of power is one-half the total power produced by the generator. The other half of the total power is dissipated in the generator internal impedance.

With the line matched to the load, there will be no reflections on the line. If the load is not matched to the line, all of the power sent down the line will not be absorbed by the load. Instead, some of it will be reflected back toward the generator. This produces what we call *standing waves*. Instead of a constant voltage at all points on the line, there will be a pattern of varying voltages distributed down the line. These are the standing waves. Standing waves waste power.



R_T = output impedance of generator
or transmitter

Z_0 = characteristic impedance of
coax cable

R_L = load (antenna) impedance

$R_L = R_T$ for max power transfer
 $Z_0 = R_L = R_T$ for no standing waves

Fig. 13-3 Conditions for maximum power output and no standing waves.

The magnitude of the standing waves is designated by the standing wave ratio (SWR). The standing wave ratio is calculated in several different ways. The most common are

$$\text{SWR} = \frac{Z_L}{Z_0} \text{ or } \frac{Z_0}{Z_L} \text{ (SWR is always greater than 1)}$$

or

$$\text{SWR} = \frac{V_{\max}}{V_{\min}}$$

In the second formula, V_{\max} and V_{\min} are the maximum and minimum voltage values, respectively, of the standing waves on the line. The goal is to achieve an SWR of 1. However, in most practical applications an SWR of 2 or less is usable.

SWR may be measured by a special SWR meter, or it can be computed by knowing the forward and reflected power on the line, as the formula above indicates. RF power meters are available that measure the forward and reflected power.

The characteristic impedance of a transmission line is determined by the physical dimensions of the line, including copper conductor sizes and spacing as well as the dielectric constant of the insulator between the lines if one is present. These factors set the amount of inductance and capacitance per foot of line. Knowing the inductance (L) and capacitance (C) of the line for a given length, you can calculate the characteristic impedance (Z_0) using the formula

$$Z_0 = \sqrt{\frac{L}{C}}$$

The RG-58/U coax cable to be used in this experiment has a characteristic impedance of 53.5 Ω and a capacitance of 28.5 pF/ft.

Because a transmission line is effectively a low-pass filter made up of a distributed shunt capacitance and series inductance, it will filter out the higher frequencies. Further, since it takes a finite amount of time for the cable capacitance to charge and discharge to

voltage input changes, the line introduces a time delay. Any signal applied to the input will be delayed at the load. See Fig. 13-4(a). If the input is a sine wave, the load signal will be phase-shifted (lagged) from the input signal. The amount of time shift depends on the line characteristics. Specifically, the time delay, t , is

$$t = \sqrt{LC}$$

where L and C are the inductance and capacitance per foot of transmission line, respectively. Multiplying this value by the length of the line in feet produces the total line delay.

Knowing the time delay, you can calculate the lagging phase shift in degrees using the formula

$$\theta = \frac{t(360)}{T}$$

In this formula, θ is the phase shift in degrees, t is the time delay of the line, and T is the period of the sine wave, which is $1/f$, where f is the frequency. Refer to Fig. 13-4(b).

The transmission line also attenuates the signal because of the filtering effect and the line losses. The amount of attenuation is set by the type of line and is usually quoted by the manufacturer in terms of dB loss per 100 ft. The RG-58/U coax line used in this experiment has a typical attenuation of 5.3 dB/100 ft at 100 MHz. The attenuation is less at lower frequencies and higher at higher frequencies.

The line also distorts nonsinusoidal signals. A rectangular wave is, according to Fourier theory, made up of a fundamental sine wave and an infinite number of harmonics. The transmission line filters out the higher-order harmonics, thereby rounding the rectangular wave and producing other distortions depending on the degree of impedance matching.

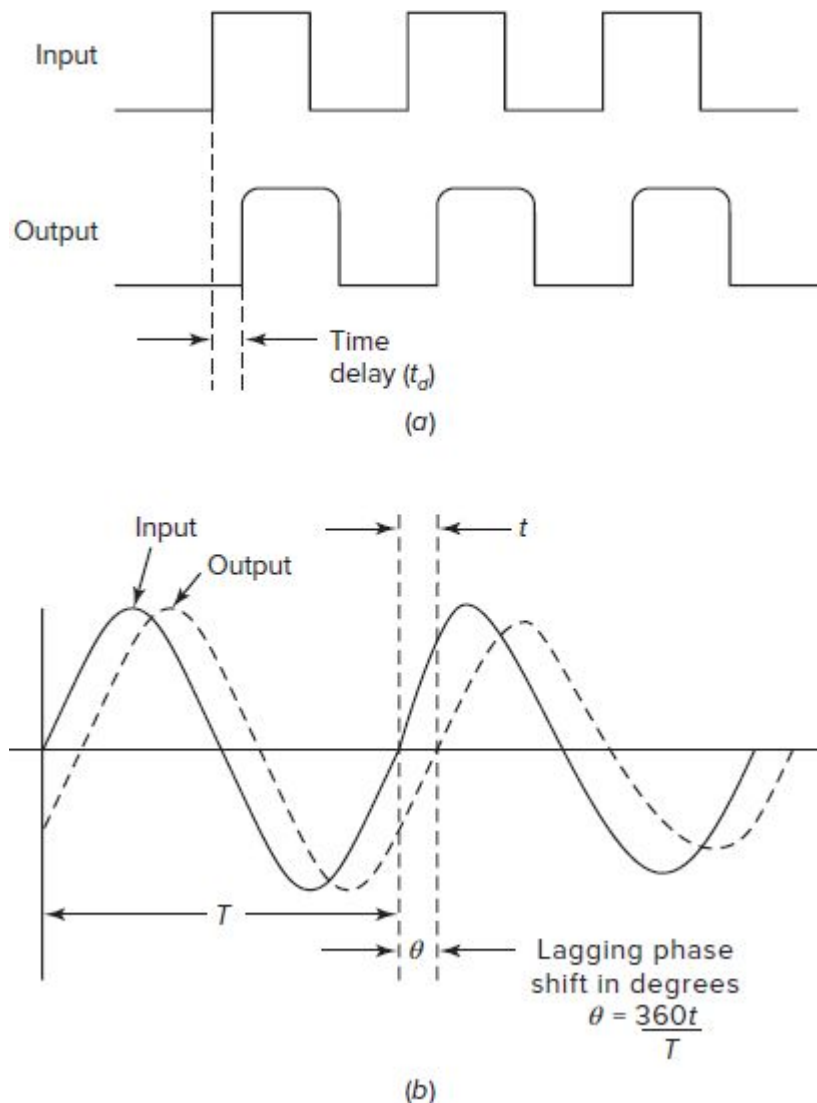


Fig. 13-4 (a) Time delay of a coax cable; note rounding on output signal. (b) Phase shift produced by a coax cable.

A quick way to evaluate the quality of a transmission line is to test it with a square wave. By observing the input and output pulses on an oscilloscope, you can determine the degree of match or mismatch of the line and load on the line. This technique is referred to as time domain reflectometry, or TDR. It makes use of the fact that an improperly matched line will have reflections. The reflected pulse will add to the transmitted pulse and produce a unique pulse shape at the input that can be used to evaluate the load size and mismatch.

Figure 13-5 shows the test setup for TDR measurements. Note that you are monitoring the *input* signal. A pulse generator applies a

square wave to the transmission line. If the line and load are matched to the generator, maximum power is transferred and no reflections occur. The signal across the load will be essentially the same as the input but somewhat rounded due to line filtering, and the signal level will be lower due to the line attenuation. The signal will also be delayed, as described earlier.

If the line is not terminated in the characteristic impedance of the line, reflections will occur. The pulse will travel down the line and be reflected back toward the generator. The returned pulse will add algebraically to the originally transmitted pulse, creating a unique pulse shape that can be used to determine the load status.

Figure 13-6 shows some typical conditions. The input is a dc square wave that varies from 0 V to +1 V. In Fig. 13-6(a), the load is matched to the generator, so no reflection occurs. The signal at the input is not affected, because there is no reflection. In these illustrations, we are assuming no signal attenuation in the cable.

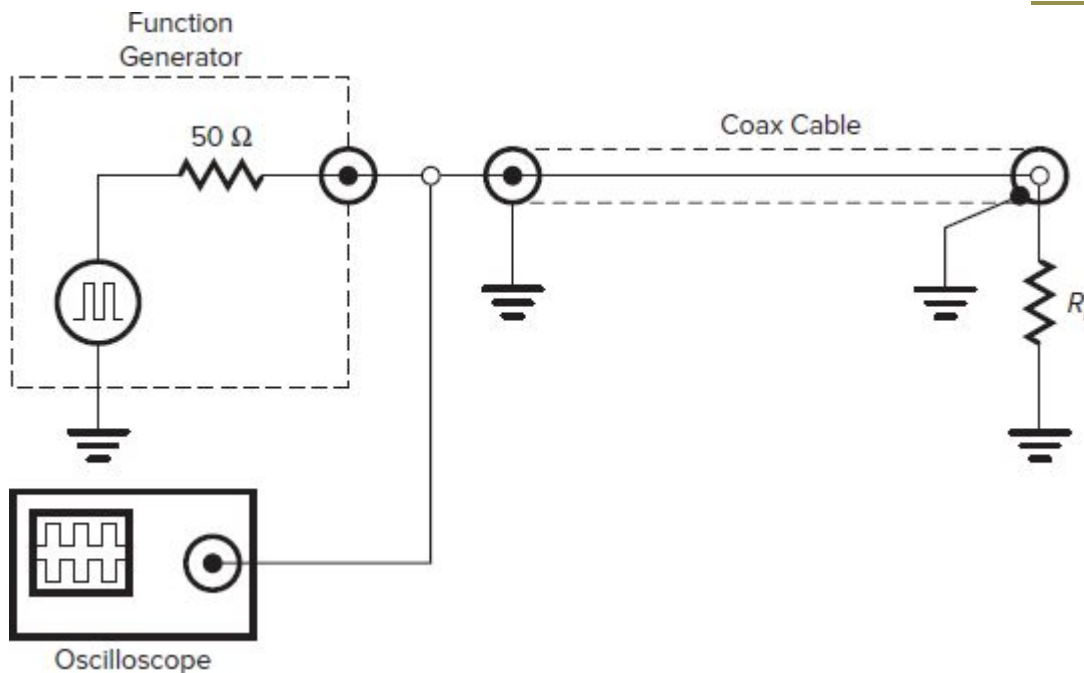


Fig. 13-5 Test setup for TDR. Note that the oscilloscope monitors the input signal.

In Fig. 13-6(b), the line is open at the end, thereby producing 100 percent reflection. The reflected pulse will appear back at the input delayed by the line where it adds to the input pulse. Note the time

shift ($2t_d$) between the generator input signal and the reflected signal. Keep in mind that reflected pulses experience the same delay in traveling from the load back to the input. The reflected signal Y adds to the generator signal X algebraically, producing a composite signal at the cable input.

In Fig. 13-6(c), the line is shorted at the load. This also results in 100 percent reflection, but the returned pulse is inverted in polarity as well as being shifted in time. Note that the resulting signal at the input is bipolar when the reflected signal is added to the generator signal.

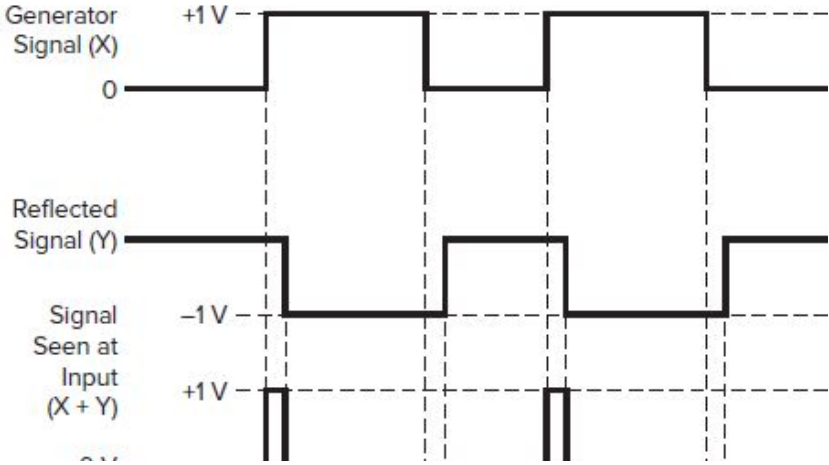
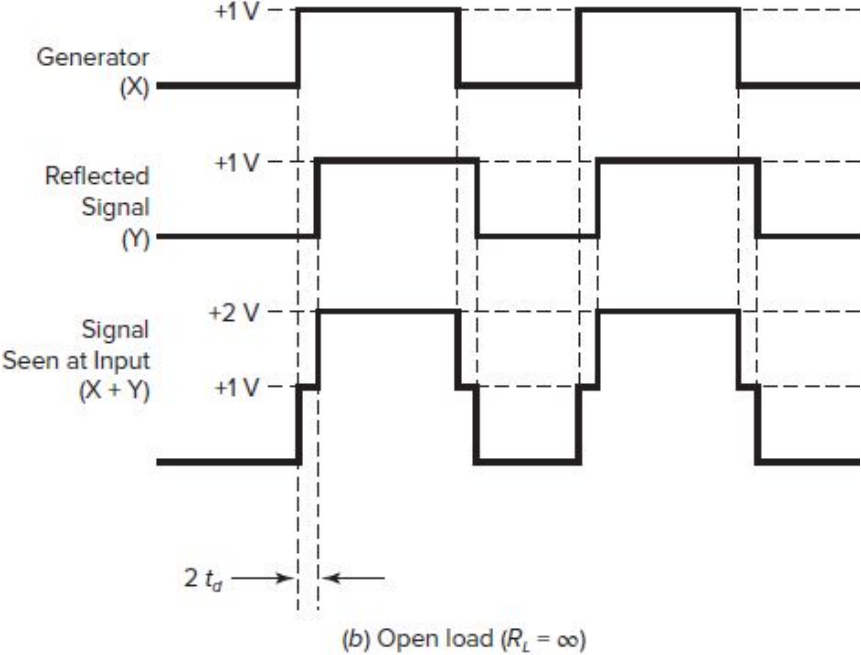
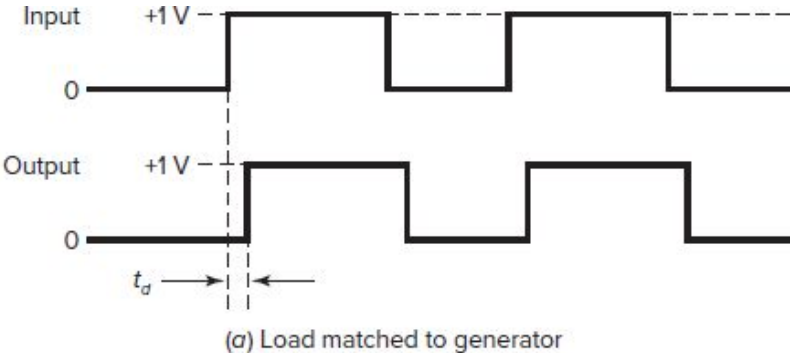
Figure 13-6(d) shows the waveform resulting when the load impedance (Z_L) is greater than the line impedance and generator impedance (Z_0), or $Z_L > Z_0$. Figure 13-6(e) shows the waveform resulting when the load impedance (Z_L) is less than the line impedance and generator impedance (Z_0), or $Z_L < Z_0$. These waveforms are only approximations, as the exact shape of the composite input signal depends on the length of the line.

In this experiment, you will demonstrate these principles and observe the waveforms resulting from TDR.

Procedure

1. Use a piece of RG-58/U coax cable whose length is unknown. The length should be greater than 40 ft but does not have to be longer than about 100 ft. Strip both ends of the cable, exposing the center conductor and the shield braid. Connect small alligator clips to one end of the line to make it quick and easy to connect different values of load resistance. Connect the other end of the cable to the function generator output. You may wish to use a BNC connector on this end of the cable, as most function generators have a BNC output connector. Use a BNC T-connector to provide an output to the oscilloscope. See Fig. 13-7.
2. The function generator should be set to supply 1-MHz square waves. The output should supply dc pulses, and the output impedance of the generator should be 50 Ω . Most function

generators have a TTL output that will be ideal for this experiment.



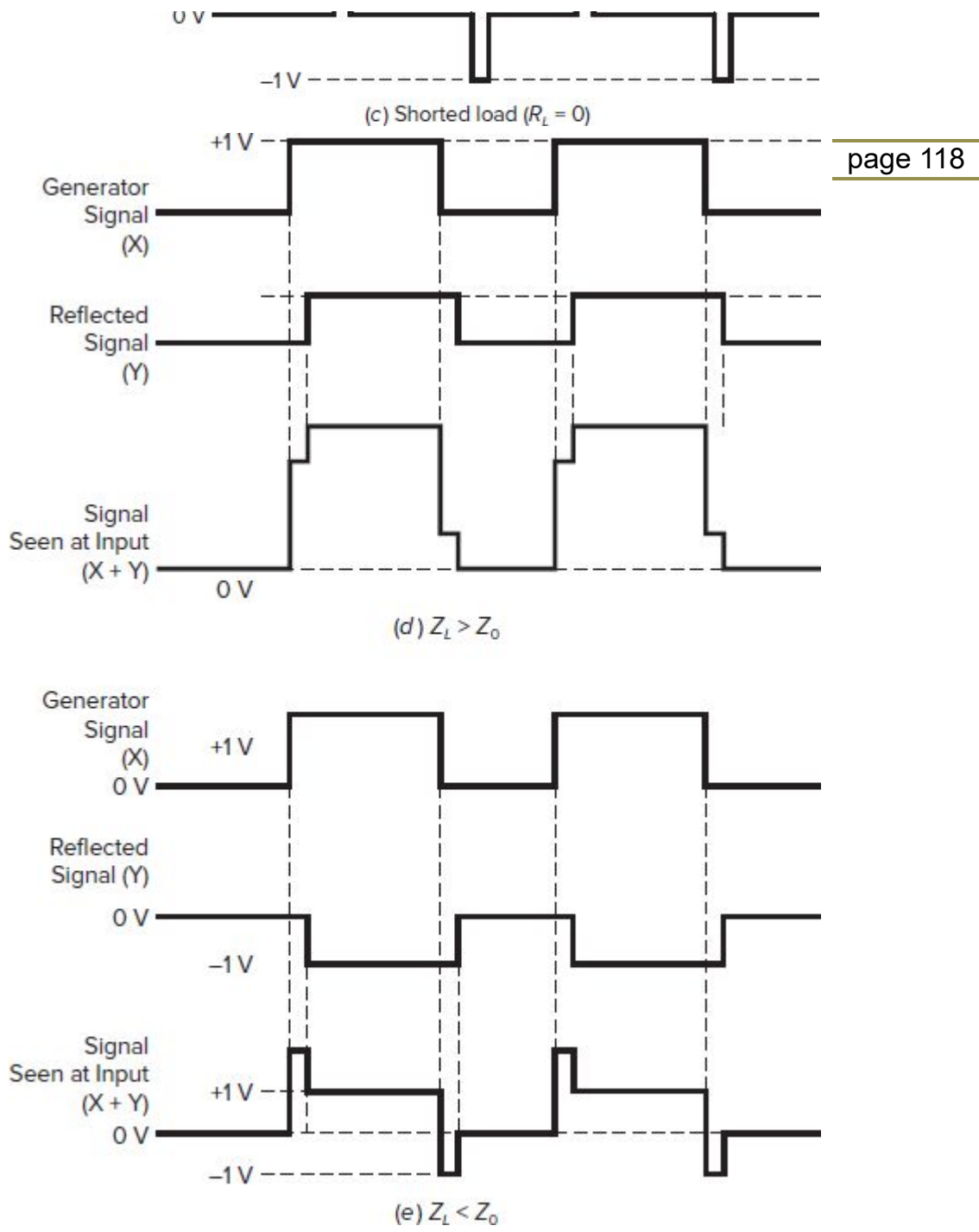


Fig. 13-6 Typical TDR waveforms.

3. Display the function generator output signal on the oscilloscope and measure the generator voltage. it should be somewhere between 3 and 4 V_{p-p} if it is TTL compatible. Record your value below.

$$V_{in} = \underline{\hspace{2cm}}$$

4. Now, connect the circuit shown in Fig. 13-5. Use a 51- Ω resistor for the load. Observe the input signal. Measure the voltage value on the scope and record your value below. Explain what you see. Is the load matched to the line?

$$V_{in} = \underline{\hspace{2cm}}$$

5. Using the second scope input channel, display the signal across the load along with the input signal on the other channel. Measure the output voltage and record below. Is this voltage value correct for a matched line?

$$V_o = \underline{\hspace{2cm}}$$

6. While observing the input and output signals, adjust the trace positions to overlap the signals. Measure the time shift between the two signals.

$$t_d = \underline{\hspace{2cm}} \text{ ns}$$

7. Using the formulas and values given earlier, calculate the equivalent inductance per foot for this cable. Then use the L and C values to calculate the time delay of 1 ft of the line.

$$L = \underline{\hspace{2cm}} \mu\text{H/ft}$$

$$t_d = \underline{\hspace{2cm}} \text{ ns/ft}$$

8. Measure the coax cable length with a tape measure. Then multiply it by the time delay per foot you calculated in step 7 to determine the overall cable delay.

$$\text{Cable length} = \underline{\hspace{2cm}} \text{ ft}$$

$$t_d = \underline{\hspace{2cm}} \text{ ns (total)}$$

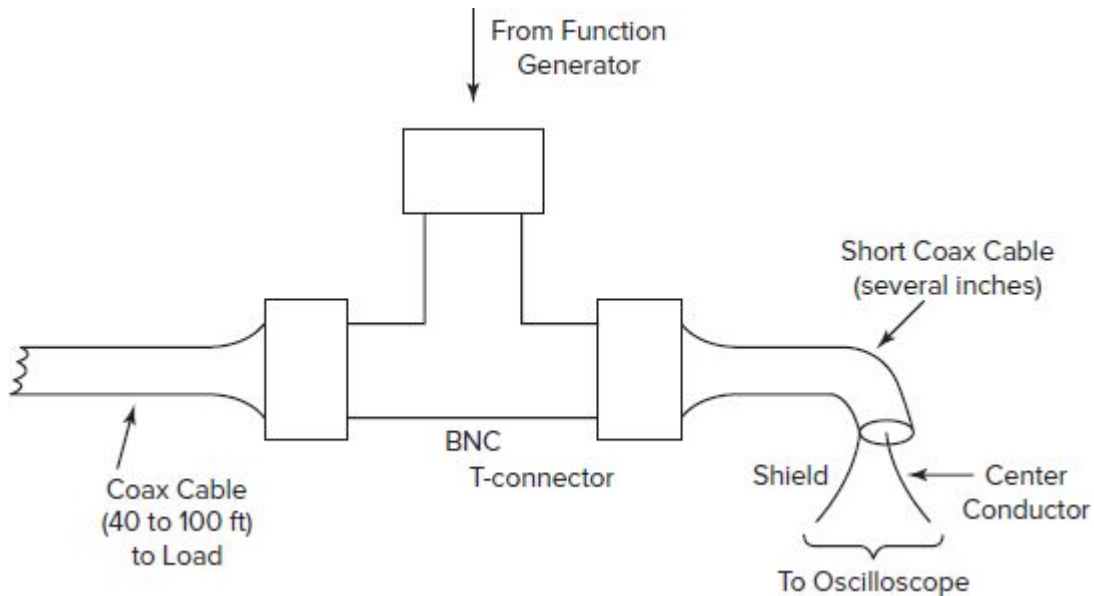


Fig. 13-7 Using a BNC T-connector to make connections between generator, oscilloscope, and cable.

9. Compare your measured and calculated values and account for any differences.
10. Remove the $51\text{-}\Omega$ load. With the line open, observe the signal at the line input. Draw what you see. How does it compare to the waveform given in the Introduction?
11. Next, short the end of the coax. Again, observe the line input. Draw the waveform. How does it compare to the illustrations given in the Introduction?
12. Unshort the line and connect a $100\text{-}\Omega$ resistor. Observe and draw the input waveform. Explain. What is the SWR for this arrangement?
SWR = _____
13. Remove the $100\text{-}\Omega$ load and connect a $22\text{-}\Omega$ load in its place. Once again, look at the input waveform and draw it. Explain. Calculate the SWR.
SWR = _____

Questions

1. Looking at the waveforms you drew in steps 10, 11 and 12 (which should be similar to these in Fig. 13-6), explain how you

could calculate the length of the line from measurements you could make from these waveforms.

2. What is the SWR of an open or shorted transmission line?
3. A 75-ft piece of RG-58/U coax is properly terminated. A sine wave with a frequency of 2 MHz is applied to the line. What is the phase shift between the input and output?
4. A buried coax cable is used to carry cable TV signals from the cable head end to each subscriber. The coax cable between the subscriber's house and the nearest cable amplifier box is accidentally cut when a utility company is digging nearby. Explain a way that the cable company technician could use TDR to estimate the distance to the cable break.

page 120

Project 13-5 Smith Chart Tutorial

Objective

To review the use of the Smith chart.

Introduction

Despite its age (invented in 1938), the Smith chart is still widely used in electronics, especially in the design and application of transmission lines. It is a complex chart that takes some practice to master. The text covers this chart in some detail, and this project gives you the opportunity to review that material and to discover what other materials are available.

Required

PC with Internet access.

Procedure

1. Go to the following websites and explore the materials available. Print out and read any that are interesting. Look for material on how Smith charts are used in impedance matching.

www.sss-mag.com

en.wikipedia.org/wiki/Smith_chart

2. Do a Google, Yahoo, or Bing search on the Smith chart. Also search on Smith chart tutorial. Find and read any tutorials that you find interesting. Again, look for material on how Smith charts are used for impedance matching.
3. Write a short paper on how Smith charts can be used to match one impedance to another. Give one worked-out example.

NAME

DAT

E _____

A graphic for Chapter 14 featuring the text 'Chapter 14' in white. The text is centered within a dark, irregular, cloud-like shape with a gradient from black to grey.

Chapter

14

Fundamentals of Networking, Local Area Networks, and Ethernet

Project 14-1 Exploring Ethernet

Objective

To expand your knowledge of Ethernet, its operation, and its applications.

Required

PC with Internet access.

Introduction

The most widely used LAN technology is Ethernet. And Ethernet has expanded beyond LANs to metro networks and wireless. Ethernet is so ubiquitous that a working knowledge of it is essential to any work that you may do in communications. This project will let you dig deeper into the types of Ethernet, its operation, and its usage.

Procedure

1. For a good general review of Ethernet, go to the following:

- a. www.wikipedia.org/wiki/IEEE_802.3
- b. www.wikipedia.org/wiki/Ethernet
2. Go to the following websites and explore the information available on Ethernet.

grouper.ieee.org/groups/802/3

www.ieee802.org

www.ethernetalliance.org

www.iol.unh.edu/services/testing/ge/training/

www.lantronix.com/learning/networking.html

3. Perform a Web search on the terms Ethernet, Ethernet tutorial, and any related terms you encounter in the questions. Examine the material you find, take any relevant tutorials, and print out information of interest.
4. Answer the questions below about Ethernet.

Questions

1. What organization standardizes Ethernet? What is the standard designation?
2. What is the standard designation (802.?) for Gigabit Ethernet?
3. What does the term MAC mean with regard to Ethernet. Explain the MAC.
4. What are the five main data speeds of Ethernet? What is the primary medium?
5. Can data be transferred over twisted pair at 10 Gbps? Explain.
6. Name the fiber optical versions of Ethernet.
7. What does XAUI mean? Where is it used?
8. What is metro Ethernet?
9. Explain how Ethernet is used in iSCSI and LXI.
10. What is Ethernet in the first mile? What is EPON?
11. List the versions of 100 Gbps Ethernet.
12. State the purpose of the following forthcoming Ethernet standards: 802.3an, 802.3aq, 802.3ap, 802.3at, and 802.3as.
13. Can Ethernet be used as a MAN? WAN?

NAME _____

page 123

_____ DAT

E _____

Chapter 15 has no experiments and therefore is not included in this Experiments and Activities Manual.

page 124

NAME

DAT

E _____

A decorative graphic for Chapter 16, featuring the word "Chapter" in a white sans-serif font above the number "16" in a large, bold, white sans-serif font. The text is centered within a dark, irregular, cloud-like shape with a white glow effect.

Chapter

16

Antennas and Wave Propagation

Project 16-1 Dipole Radio Antenna

Objective

To construct a dipole antenna and test its directional characteristics.

Required

- 1 FM radio with terminal for attaching one of the following external antennas:
 - 300- Ω twin lead (about 8 to 15 ft), or
 - #14 solid copper wire with RG-59/U 75- Ω coax cable and matching RCA phono connector
 - Soldering iron, solder, side cutters, and knife or wire stripper

Procedure

1. Determine what type of antenna terminals are used on the FM receiver. If a 300- Ω twin lead is used, two screw terminals will be available. If coax is used, the connector is most likely the RCA phono type.
2. Use the twin lead to construct a folded dipole antenna for your favorite FM radio station, whose frequency f you must know.

What is it?

3. Cut the antenna following the guidelines in the text. It must be one-half wavelength long, but take into account the velocity factor of twin lead, which is 0.8. What length do you need?
4. Strip both ends of the twin lead and twist the leads together, as Fig. 16-1 shows. Cut into the center of one conductor of the twin lead and attach the 300- Ω transmission line; see Fig. 16-1. The transmission line length is not critical, but it should be great enough to attach the line to the receiver and provide for movement of the antenna. A minimum length is about 5 ft, but more is better.

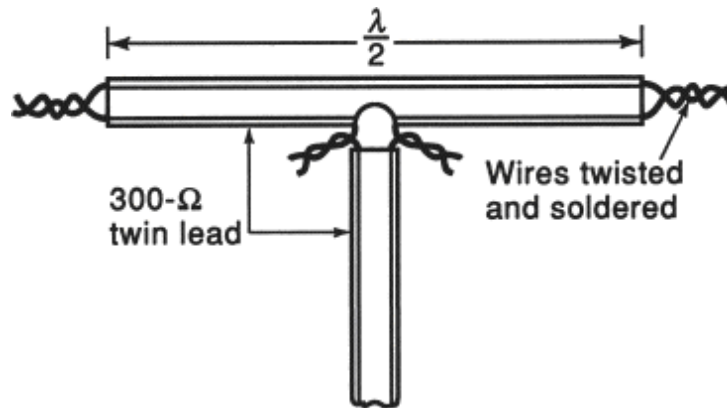


Fig. 16-1 Folded dipole construction with twin lead.

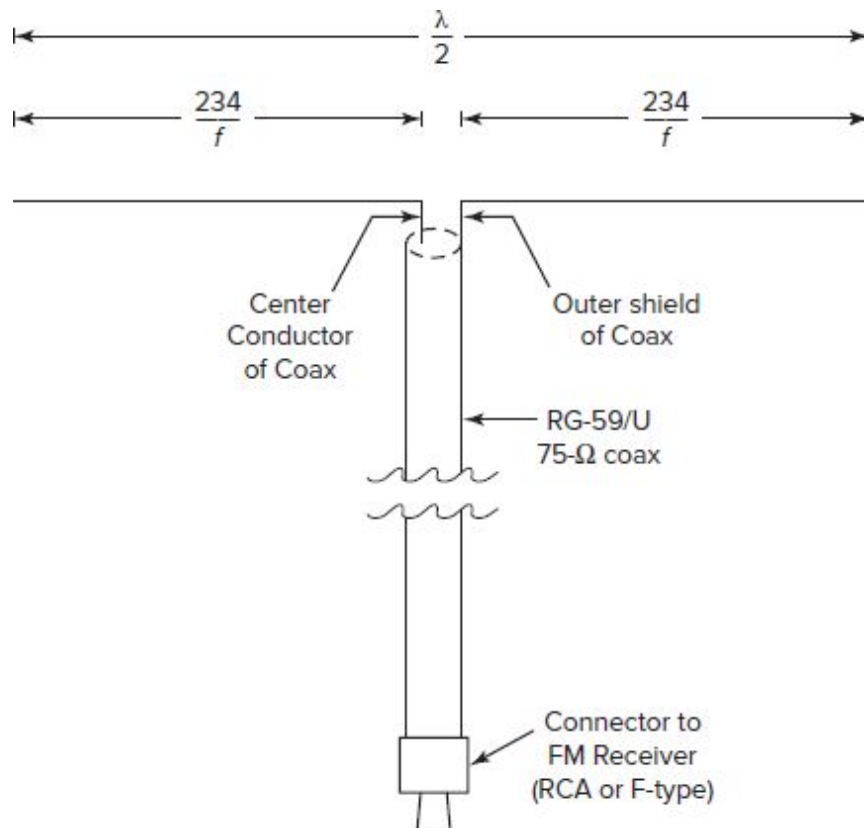


Fig. 16-2 Half-wave dipole with coax cable feed line.

5. Alternatively, you may also make the antenna of copper wire and coax cable feed line. Cut the #14 copper wire to the length specified by the formula in the text for a half-wave dipole, that is, $L = 468/f$, where f is in megahertz. Cut this wire into two equal pieces. Then attach the wire to the coax as shown in Fig. 16-2. Solder the connections to the coax cable. Be sure the coax cable has a connector that matches the one on the FM receiver. RCA and F-type coax connectors are the most common.
6. Connect the transmission line to antenna terminals on the FM receiver.
7. Turn on the receiver and tune it for your station. A weak station is best for this experiment.
8. Hold the antenna in your hands with your arms extended so the antenna is stretched out horizontally to its full length. Stand away from the receiver and any obstructions (wall, furniture, etc.) as far as possible. To determine the directional characteristic of the antenna, rotate your body slowly 360° while monitoring the signal

strength. Some FM receivers have a signal strength meter, which can be useful in determining receiver output. Note how the signal amplitude goes up and down as you turn. Predict the antenna directivity pattern from your observations.

(*Note:* If you do not get clear, pronounced variations in signal strength, try a different position in the room. Some obstruction may be interfering with normal operation. Also, try another but weaker station. Strong signals obscure the effect. Tune to the weakest signal you can find and then repeat this step.)

Can you determine the direction of the transmitting station from your prediction?

9. Repeat step 7 on another station at a different frequency. Does the antenna work well on this frequency?
10. Hold the antenna vertically and repeat the procedure described in step 7. What is the shape of the directivity pattern?

Questions

1. What is the ideal shape of a dipole antenna directivity curve?
2. What is the shape of a vertical dipole directivity curve?

page 127

Project 16-2 Plotting Directional Antenna Patterns

Objective

To construct several common antennas and to plot their radiation patterns.

Required

- 1 Radiometrix TX3A-914-64 UHF transmitter
- 1 Radiometrix RX3A-914-64 UHF receiver
- 1 RG-58A/U coax cable
- Breadboarding sockets, one each for transmitter and receiver
- 1 10-ft length of #14 bare copper wire
- 2 3-screw terminal blocks

- 1 power supply for transmitter (3 V), power supply or two AA cells in series
- 1 power supply for receiver (9 V battery) and battery terminal clip
- 1 digital multimeter
- Small wood pieces to form mounting platform for antennas (See text)
- Antenna radiation plot graph paper

Introduction

Despite its apparent mechanical simplicity, the antenna in a communications system is a critical link. It is the transducer that converts the transmitter voltage output into an electromagnetic wave and launches it into the air. At the receiver, the antenna converts the electromagnetic wave back into a voltage that can be amplified and otherwise processed to recover the transmitted information. This project is designed to give you an opportunity to build and test some common antennas such as the dipole, ground plane, and Yagi.

The transmitter and receiver to be used in this project are fully wired and tested modules made by U.K. company Radiometrix (www.radiometrix.com) and distributed in the United States by Lemos International (www.lemosint.com). These units are set to operate in the U.S. ISM band of 902–928 MHz, specifically 914.5 MHz. The modules are designed to transmit low-speed data up to 64 kbps by FSK. Only continuous wave signals will be used here to plot antenna patterns. Antenna lengths at this frequency are very short and are easy to construct from stiff copper wire.

The transmitter (TX) and receiver (RX) modules can plug into a small breadboarding socket. This socket can be mounted on a wooden structure as shown in Fig. 16-3. The vertical part of the structure will support the antenna. A three-screw terminal strip is used to attach the antenna elements, and coax cable is used to connect the antenna to the transmitter or receiver. Build one of these structures for the TX and another for the RX. DC for the RX should come from a 9-V battery so that the RX assembly can be rotated. The DC for the TX can come from two AA cells in series or a fixed power supply.

Once a link has been established, the receiver assembly will be rotated to plot the curve from readings taken on a digital multimeter from the receiver. The receiver has a built-in received signal strength indicator (RSSI) circuit that converts the received signal into a dc voltage proportional to the signal level. With no signal, the RSSI output is about 0.25 V and rises to a 1-V maximum value. The RSSI range is from 65 to 75 dB.

Before proceeding with the project, be sure to acquire and read the instruction manual for the RX3A and TX3A and have it handy for reference during the experiment. These units are designed to comply with FCC Part 15.249 regulations. **Be sure to read and follow these guidelines before you attempt this project.**

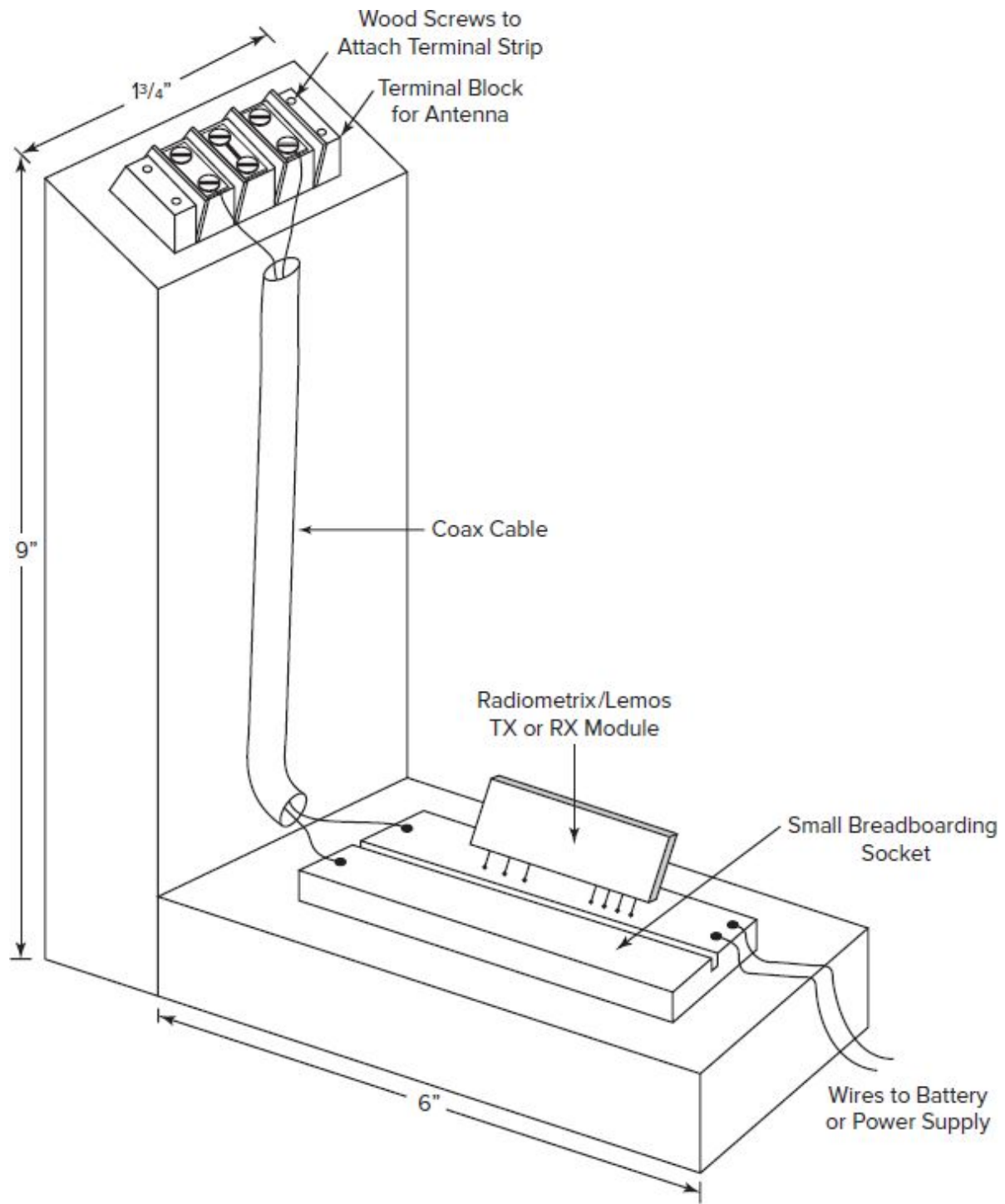


Fig. 16-3 Build two of these units.

Procedure

Constructing the Circuits

1. Mount the TX3A transmitter on the breadboarding socket and wire it as shown in Fig. 16-4. With a 3-V dc supply, the TX output is about 1 mW. Pin 4 is the enable pin. Connect it to +3 V to enable the transmitter. Ground pin 7, the data input, to transmit a continuous binary 0 frequency.
2. Construct the receiver unit by plugging the RX3A module into the breadboarding socket of the other assembly. Wire the circuit as shown in Fig. 16-5. Use a 9-V battery for the supply. A digital multimeter will be used to measure the RSSI output.
3. Construct the transmit antenna, using the #14 wire and coax. Figure 16-6 shows the details of a standard half-wave dipole. The antenna length is determined with the familiar formula $L = 468/f_{\text{MHz}}$, where L is the length in feet and f is the operating frequency in MHz. Multiply your value by 12 to get the dimension in inches. Cut the wire to that length plus about 1 inch and cut that in half. The extra 1 inch is divided between the two pieces and provides sufficient length to attach to the screw terminals. The antenna elements are screwed to the two outer terminals of the strip as shown. This also provides a convenient way to connect the coax to the antenna. The coax should be kept as short as practical, no more than about 1 ft. Connect the coax to the breadboarding socket with short pieces of #22 solid hookup wire that can be soldered to the coax leads.

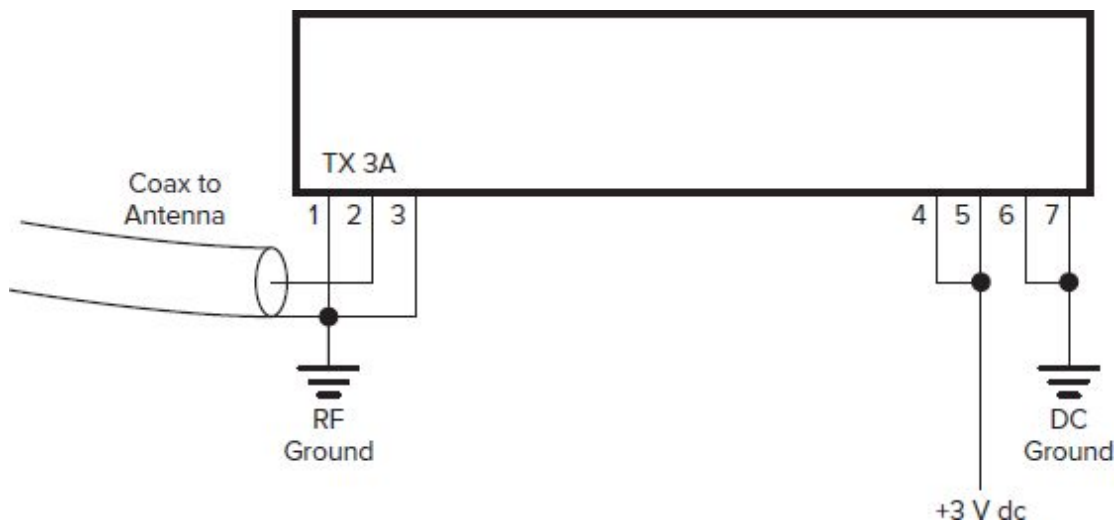


Fig. 16-4 Transmitter wiring.

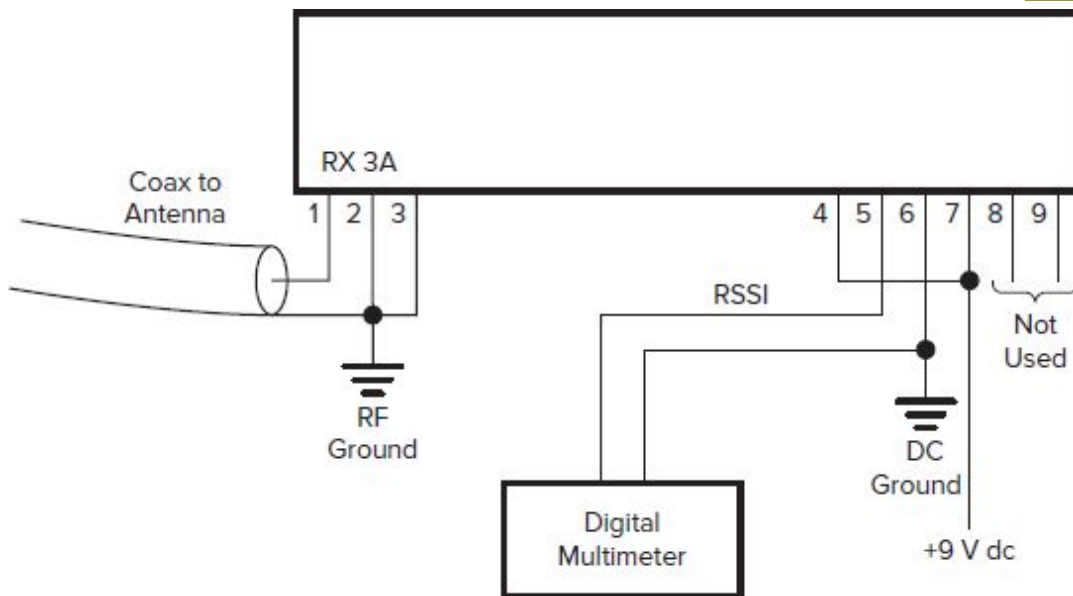


Fig. 16-5 Receiver wiring.

4. Construct the receiving antenna just as previously described in step 3 and Fig. 16-6.

Testing the Circuits

5. Place the receiver assembly well away from the transmitter. The absolute minimum distance is about 10 ft to be sure you will be using the far-field signal. Farther apart is better. The maximum range of the TX-RX combination is about 100 ft indoors. The transmitting and receiving antennas should be parallel to one another.
6. Turn on the transmitter power. Turn on the receiver power. Note the dc reading on the DMM. You should see a voltage of 0.25 to 1 V. Adjust the position of the receiver for a 1-V reading. Be sure the antennas stay parallel. The power level on the transmitter may also be adjusted to achieve this level by varying the transmitter dc supply voltage from no less than 2.7 V to up to 12 V if necessary.

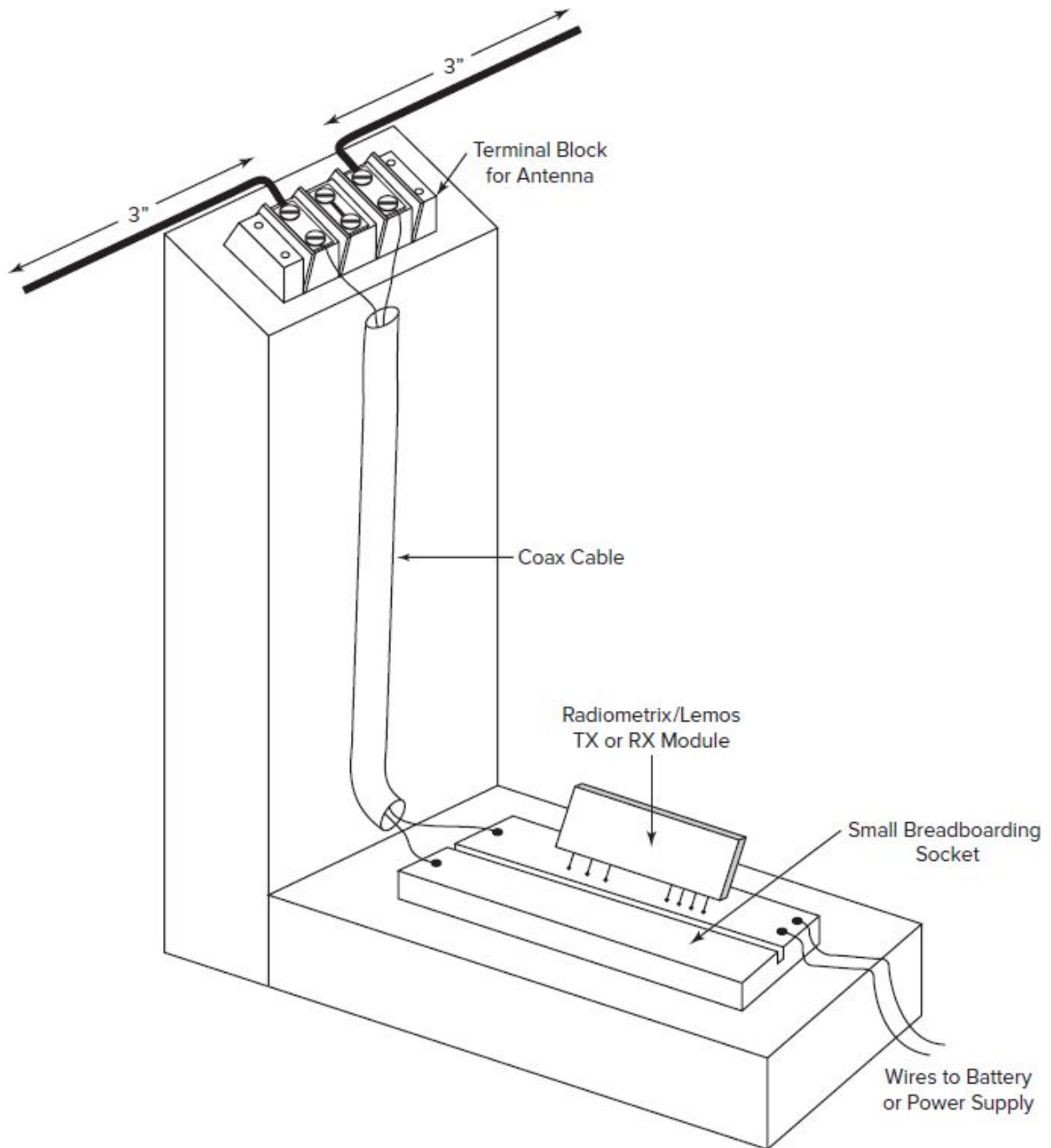


Fig. 16-6 Dipole antenna for RX and TX.

Important Notice

Because of the nature of RF and especially UHF signals, this project is subject to a variety of factors that may make operation difficult. The primary factor that may limit or restrict operation is the

lab environment with its many potential variations that may cause reflections and signal absorption. Human presence and motion can also affect results. For best results, the experimental area should be as free of clutter and objects as possible and human presence and motion should be held to a minimum. It may also be necessary to experiment with factors such as distance between transmitter and receiver and transmitter power. Repositioning the units is often all that is necessary to make the project operational.

Plotting Antenna Patterns

7. Plot the radiation curve of the receiver dipole by rotating the receiver in place for 360 degrees in 10-degree increments. Note the DMM voltage reading at each angle. Make a table of voltages for each angle.
8. Plot your voltage readings on the polar antenna graph designated earlier in the required materials section. Does the plot resemble the classical dipole response? Explain any differences.

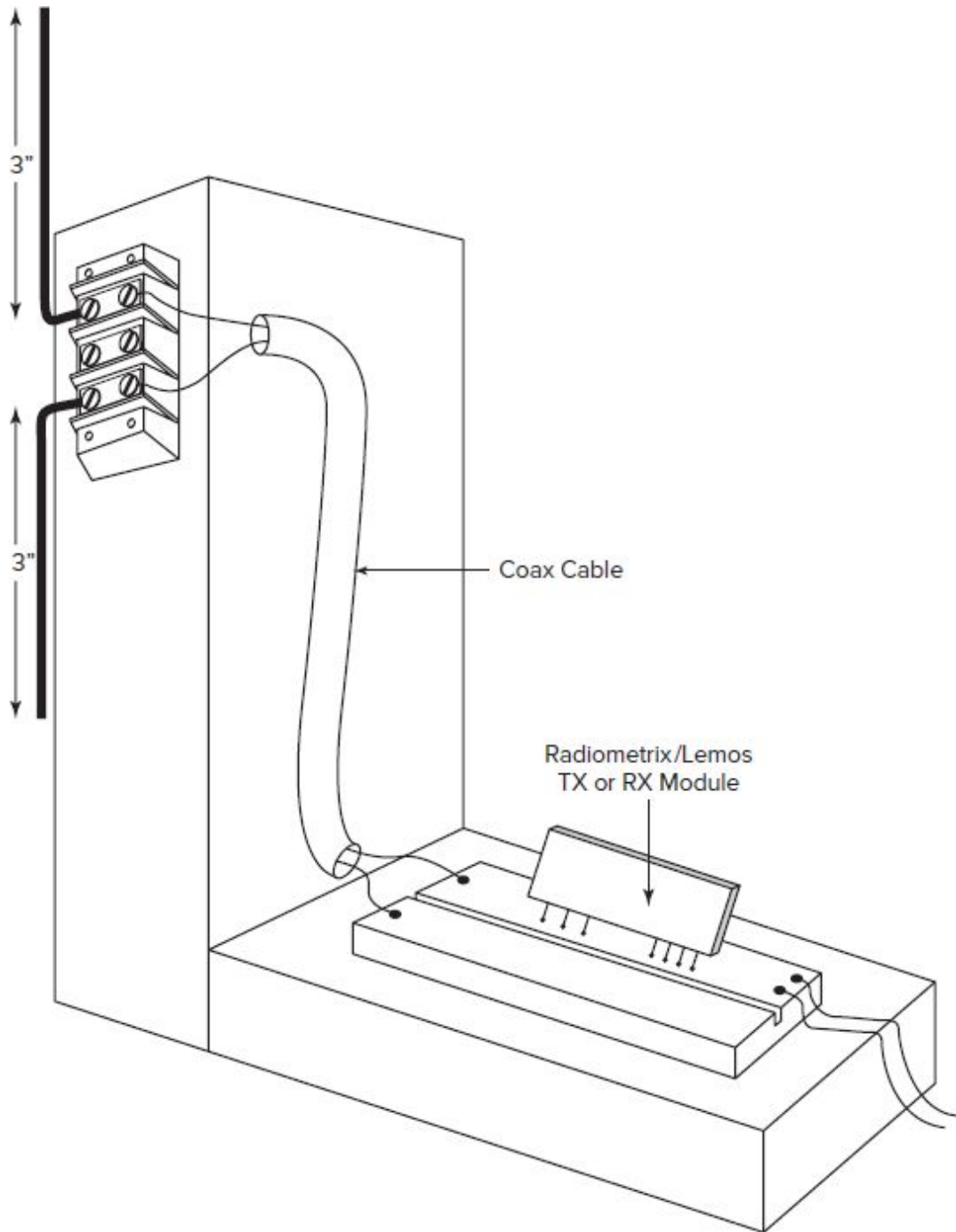


Fig. 16-7 Mounting the RX antenna vertically.

9. Reposition the receiver so its antenna is parallel to the transmitter antenna. Again adjust the DMM voltage reading for a value of 1 V as described earlier.

10. Unscrew the three-screw terminal strip from the top of the wooden assembly and rotate it 90 degrees to create a vertical dipole. Reattach the terminal strip to the vertical wooden piece as shown in Fig. 16-7.
11. Note the RSSI output voltage. Explain your results.
12. Turn off the power on both receiver and transmitter.

Building a Ground Plane Antenna

13. Return the terminal strip to the top of the wooden assembly of the receiver and disconnect the dipole antenna. Leave the coax in place.
14. Construct the ground plane antenna on the receiver assembly as shown in Fig. 16-8. The four radials are made by crossing two lengths of wire and screwing them to the three-screw strip as shown. Bend the radials down slightly, as this will provide a better impedance match to the coax. The vertical element is one-quarter wavelength long and computed with the formula $L = 234/f_{\text{MHZ}}$. Convert L to inches by multiplying your value by 12. You can actually use one of the halves of the dipole for the vertical element.

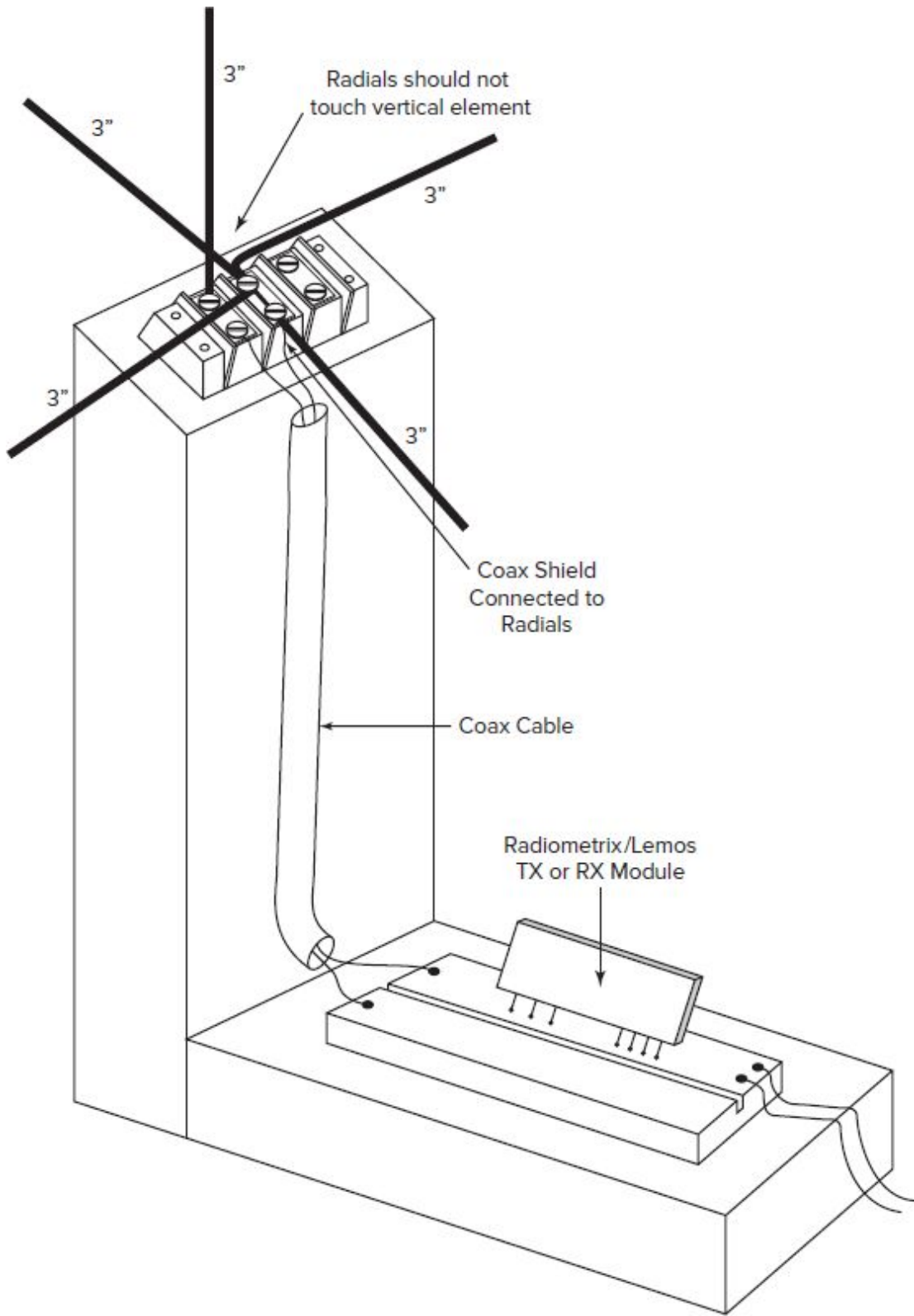


Fig. 16-8 Ground plane vertical antenna on the RX.

15. Apply power to the transmitter and receiver. Set the RSSI output to 1 V as previously.
16. Plot the radiation curve of the receiver dipole by rotating the receiver in place for 360 degrees in 10-degree increments. Note the DMM voltage reading at each angle. Make a table of voltages for each angle.
17. Plot your voltage readings on the polar antenna graph paper designated earlier. Does the plot resemble the classical ground plane response? Explain any differences.
18. Turn off the power to both units.

Constructing a Yagi Antenna

19. Disconnect the ground plane antenna from the terminal block. Replace the two segments of the dipole so they are connected to the coax as before in Fig. 16-6.

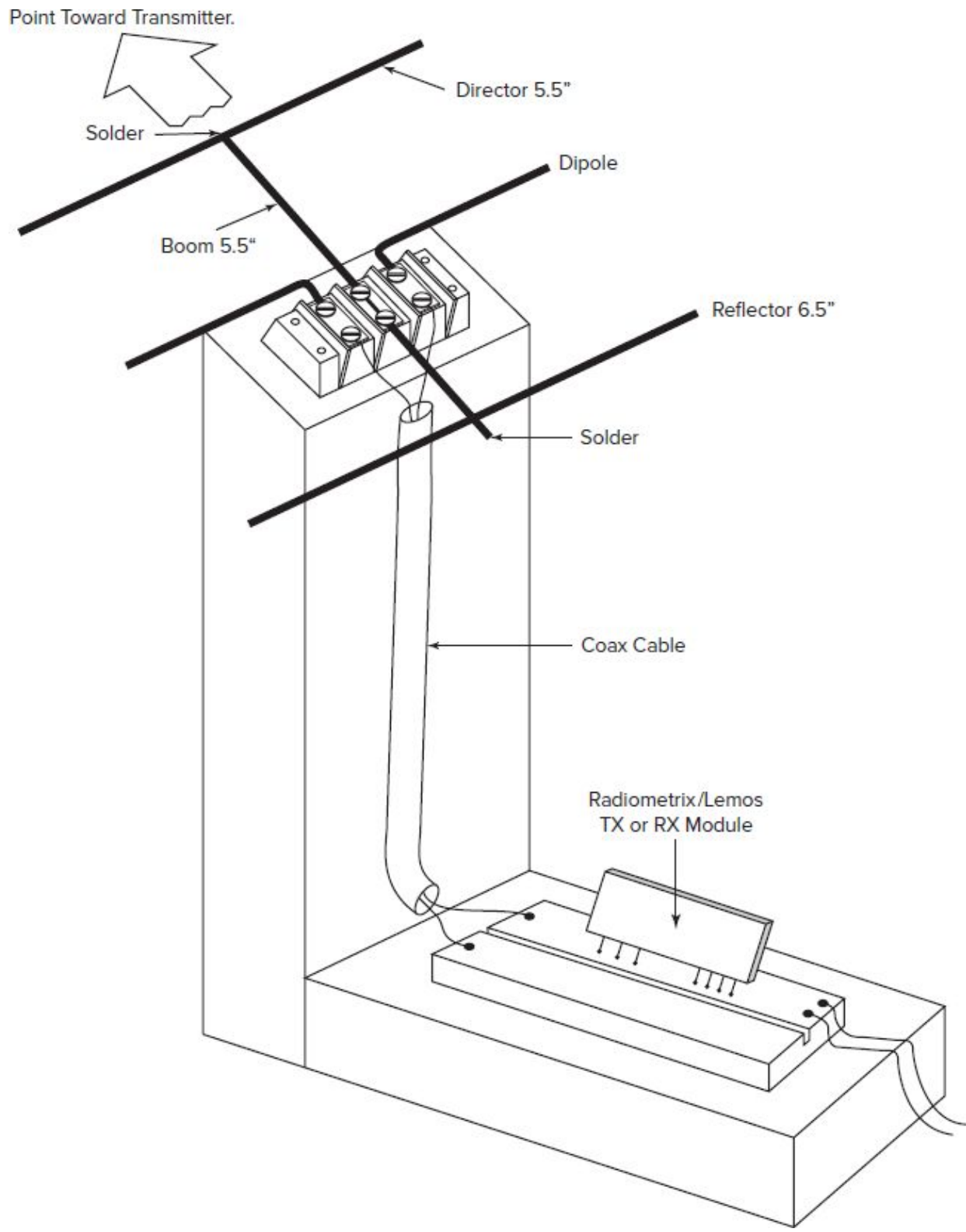


Fig. 16-9 Building the Yagi antenna.

20. Refer to Fig. 16-9. This shows a three-element Yagi with dipole driven element plus a reflector and a director parasitic elements. Build the remainder of the antenna by cutting the wire for the boom, reflector, and director to the dimensions as shown. Solder the director and reflector to the ends of the boom. Connect the boom to the center screws on the terminal block as shown. The dipole should be centered between the director and reflector. Be sure to maintain the dimensions as closely as possible.
21. Point the Yagi director at the transmitting antenna.
22. Apply power to both units. Set the RSSI output to 1 V as described previously by moving the receiver farther away or reducing the transmitter dc voltage. Plot the radiation curve of the receiver Yagi by rotating the receiver in place for 360 degrees in 10-degree increments. Note the DMM voltage reading at each angle. Make a table of voltages for each angle.
23. Plot your voltage readings on the polar antenna graph page 134 paper. Does the plot resemble the classical Yagi response? Explain any differences.
24. With the Yagi pointed at the transmitter dipole, readjust the RSSI output to 1 V. Unscrew the boom assembly, removing the director and reflector from the terminal strip and set it aside. Read the RSSI value. Explain your results.
25. Turn off the power to both units.

Questions

1. What may account for any variations in the plotted antenna patterns from the ideal patterns given in the text?
2. In step 24, when you removed the director and reflector, you returned the antenna to a simple dipole. From your readings, estimate what the Yagi gain is in dB.

NAME

DAT

E _____



Internet Technologies

Project 17-1 Review of Internet Protocol

Objective

To refresh your knowledge of the Internet protocol.

Introduction

The protocols TCP/IP, or Transmission Control Protocol/Internet Protocol, are at the heart of all Internet transmissions. Because they are so ubiquitous, it is essential that you understand them in detail. The text provides an introduction, but there is much more involved. This project will give you the opportunity to explore TCP/IP in more detail.

Required

PC with Internet access.

Procedure

1. For information, do a Google, Yahoo, or Bing search on TCP/IP tutorial. You should also look at the Wikipedia material. For additional details, search on IPv4 and IPv6.

2. After you have reviewed the material, answer the following questions.

Questions

1. Explain the main differences between IPv4 and IPv6. Explain the need for the change from v4 to v6.
2. Who invented TCP/IP? When?
3. How does TCP/IP compare to the OSI model? The same? Different? How?
4. Provide a detailed description of how Ethernet and TCP/IP combine in most applications to deliver Internet access to most PCs. Draw a flow diagram based on the OSI model.

page 136

Project 17-2 Review of OTN

Objective

To expand your knowledge of OTN.

Introduction

OTN is the optical transport network. This new standard and protocol is replacing SONET and other optical technologies in the Internet. It offers some major benefits over SONET and will eventually be the primary backbone of the Internet. This project lets you dig deeper into OTN.

Required

PC with Internet access.

Procedure

1. Search on the terms OTN and optical transport network. Look for tutorials and short summaries.
2. Answer the questions below.

Questions

1. What standards organization develops and maintains OTN?
2. What are the primary data rates of OTN? The maximum?
3. What FEC is used in OTN?
4. State two main benefits of OTN over SONET.
5. What networking product or equipment terminates OTN?

NAME

DAT

E _____

A graphic for Chapter 18 featuring the word "Chapter" in a small, white, sans-serif font above the large number "18" in a bold, white, sans-serif font. The text is centered within a dark, irregular, cloud-like shape with a gradient from black to grey.

Chapter 18

Microwave and Millimeter-Wave Communication

Project 18-1 Exploring Millimeter Waves

Objective

To learn more about millimeter waves.

Introduction

As more and more communications services are developed and put into service, less spectrum is available to implement new services. While spectrum is scarce, the existing spectrum is poorly utilized. Such inefficiency reduces future wireless possibilities. One potential area of opportunity is millimeter waves, or the spectrum above 30 GHz. More and more, new applications are using this space as technology provides components, circuits, and equipment that can work in those outer limits. Millimeter waves are expected to be the home of the forthcoming fifth-generation (5G) cell phone service. This project allows you to examine millimeter waves to see their potential.

Required

PC with Internet access.

Procedure

1. Do an Internet search on the term “millimeter waves.” Print out as much relevant information as possible, especially looking for information on background, components, circuits, equipment, and applications.
2. Perform any additional search needed to answer the questions below.

Questions

1. Define the frequency range of millimeter waves. What is the current upper frequency limit?
2. List the current uses of millimeter waves and at least two potential future applications.
3. Name two ways that millimeter wave signals are generated.
4. List the primary benefits and disadvantages of millimeter waves.
5. Name two roadblocks to the future use of millimeter waves.
6. Name two companies that make millimeter-wave components, circuits, or equipment.

NAME

DAT

E _____



Satellite Communication

Project 19-1 Exploring GPS

Objective

Review the operation and application of GPS and its alternatives.

Introduction

The Global Positioning System (GPS) is the premier satellite navigation system in the world. Not only is it used for military navigation, but also it has become a major commercial and personal benefit as new applications have been found and low-cost GPS radios have become available. Today, GPS is so widely used that it plays an active role in our everyday lives. This project provides a way to learn more about it.

Required

PC with Internet access.

Procedure

1. Perform an Internet search on GPS tutorial. Print out and read any material you find. Don't forget to look at Wikipedia at

www.wikipedia.com.

2. Go to the websites of the major GPS receiver manufacturers such as Garmin, Trimble, Magellan, TomTom, and others and explore the material available such as white papers and application notes. Also look at some of the specific products available and their applications.
3. Answer the questions below.

Questions

1. Name two other current and future global navigation satellite systems (GNSS).
2. List 10 specific applications for GPS.
3. Is GPS used in cell phones? Explain.
4. Many GPS users fear that by having a GPS receiver, others can track them. Is that true? Explain.
5. What is geocaching?
6. What is assisted GPS?

page 140

Project 19-2 GPS Experience

Objective

To experience using GPS firsthand with a GPS receiver.

Introductions

GPS radios are very common today. They come as a built-in option in most new cars, and there are dozens of portable handheld units available for personal use. Most are available with U.S. maps and a large color liquid-crystal display (LCD). The purpose of this project is to allow you to experience GPS.

Required

Modern GPS receiver or smartphone with GPS and maps.

Procedure

1. Purchase, rent, or borrow one of the common consumer GPS radios. Get one made by Garmin, Magellan, or TomTom. Other units are also suitable. Be sure to get one that has built-in maps of your area. Use your smartphone if it has GPS.
2. Read the instruction manual for the receiver.
3. Take the GPS receiver outside and get an initial fix. Identify your location on the screen. Time how long it takes for you to achieve a link to the satellites.
4. Take the receiver indoors; wait 5 minutes. Explain what happens and why.
5. Use the receiver to find your way to a destination some distance from you. Follow any specific directions that the unit gives you.
6. Does your receiver have voice output? Was it helpful or distracting?
7. Will your GPS receiver work in Europe? Explain how.

NAME

DATE

Chapter

20

Optical Communications

Project 20-1 Optical Fiber Data Transmission

Objective

To demonstrate the transmission of binary data over a simple fiber-optic link.

Required

- 1 oscilloscope
- 1 function generator (TTL square wave)
- 1 fiber-optic cable (3 ft), 1000 micrometer (1 mm) diameter
- 1 IR LED, with housing and connector (Industrial Fiber Optics IF-E91A-R)
- 1 IR phototransistor, with housing and connector (Industrial Fiber Optics IF-D92)
- 2 NPN transistors (2N3904, 2N4401, or similar)
- 1 100- Ω resistor
- 1 1-k Ω resistor
- 1 3.9-k Ω resistor
- 1 33-k Ω resistor
- 1 220-k Ω resistor

Note: The fiber-optic cable, IR LED (IF-E91A-R) and the IR phototransistor (IF-D92) are available as a kit (IF-E10) from *Industrial Fiber Optics*.

Introduction

More and more data communication is being carried out over optical fiber cables rather than via twisted-pair, coax, or other wire conductor cables. This includes large communications systems such as the telephone network as well as smaller systems such as local area networks. High speed, high capacity, wide bandwidth, low noise, high security, and other characteristics give fiber-optic equipment major advantages over older and more conventional data communications techniques. If you plan to work in the electronics communications field, you will most surely encounter fiber-optic equipment. The purpose of this experiment is to familiarize you with the basic elements of all fiber-optic systems.

First you will examine the fiber-optic cable itself and note its construction and characteristics. You will learn how to cut the end of a cable and mount it in connectors. Next you will construct an LED transmitter and send light pulses down the cable. Finally, you will build a receiver with a photo transistor and demonstrate the reception of the light pulses. This experiment will give you a good feel for the hardware and operation of typical fiber-optic equipment.

page 142

Procedure

1. Locate the fiber-optic cable. Use a wire stripper to remove about $\frac{1}{8}$ in. of outer insulation to expose the fiber element itself. Note the clear plastic fiber. Look at it under a magnifying glass if one is available.
2. Point one end of the cable at a strong light source—a window (daylight, of course), a lamp, an overhead light fixture, a flashlight, or whatever is convenient. Observe the other end of the cable, and you will see the light being transmitted. Put your finger over the end of the cable pointed at the light and note how the light at the other end disappears. Point the end of the cable at several light sources with different intensities and see how the light at the other end varies. Experiment further as desired.

3. Construct the transmitter circuit shown in Fig. 20-1. The physical connections to the transistor and LED also are given. Do *not* connect the cable to the LED yet.

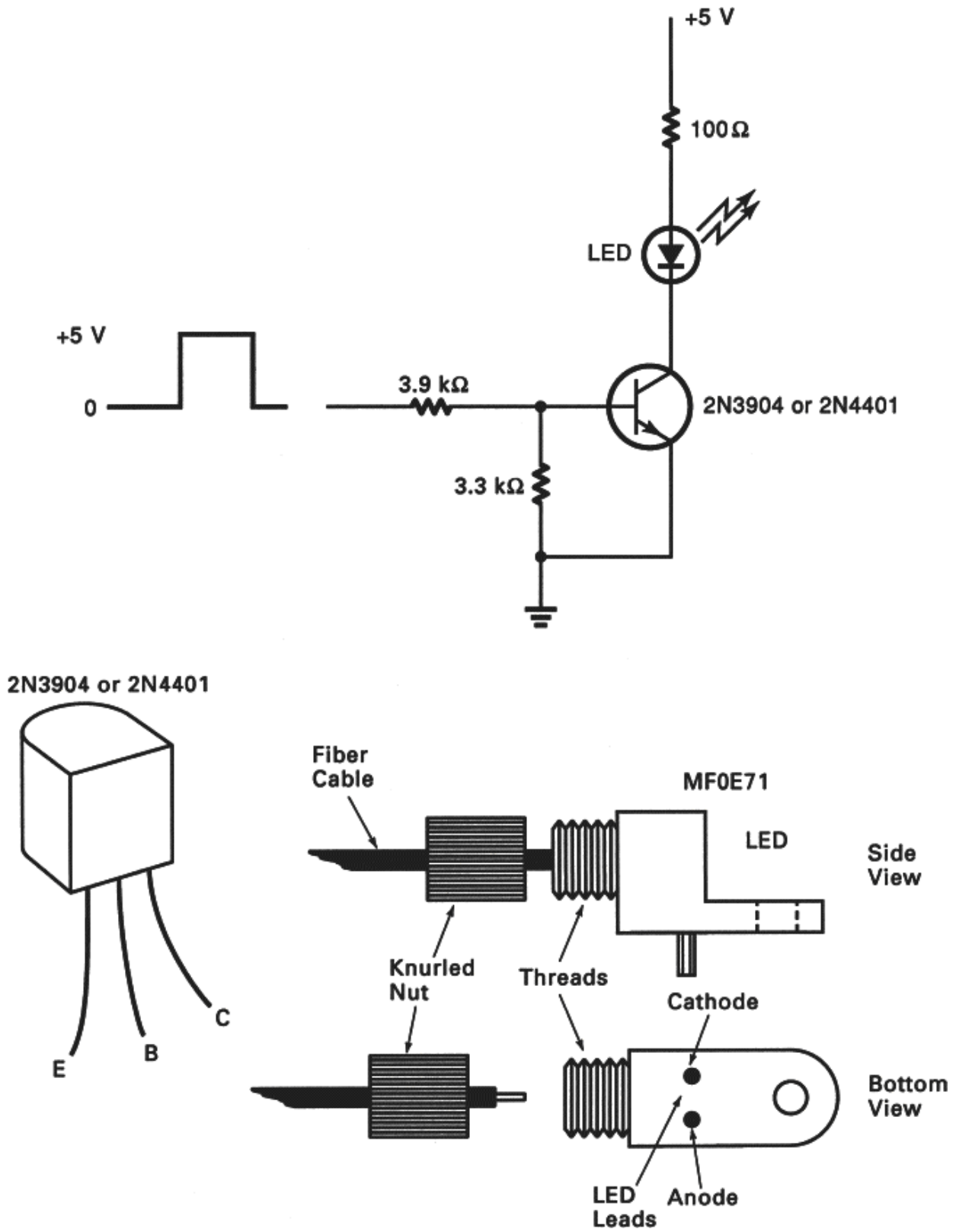


Fig. 20-1 LED transmitter.

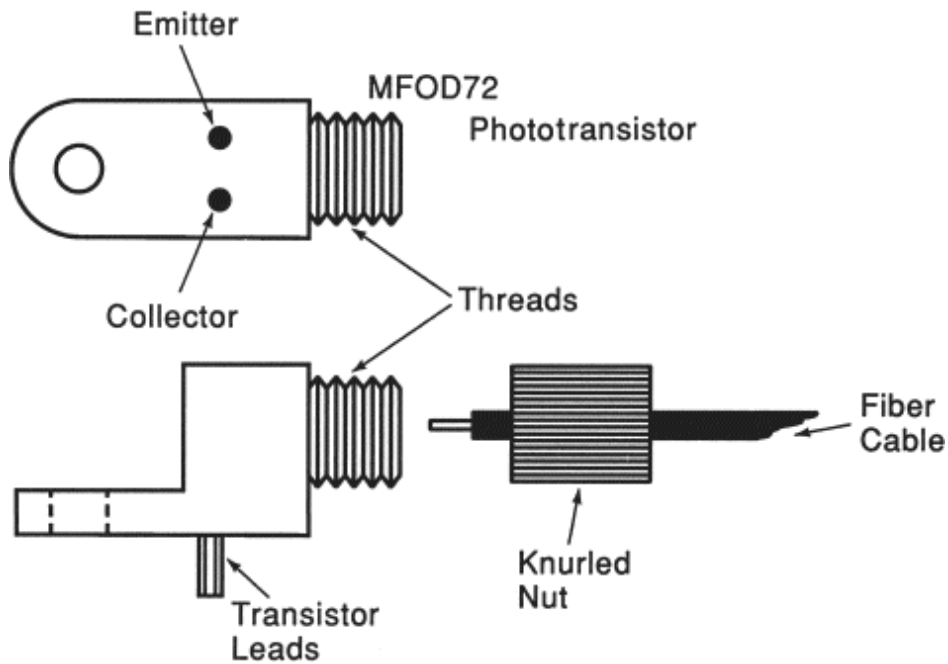
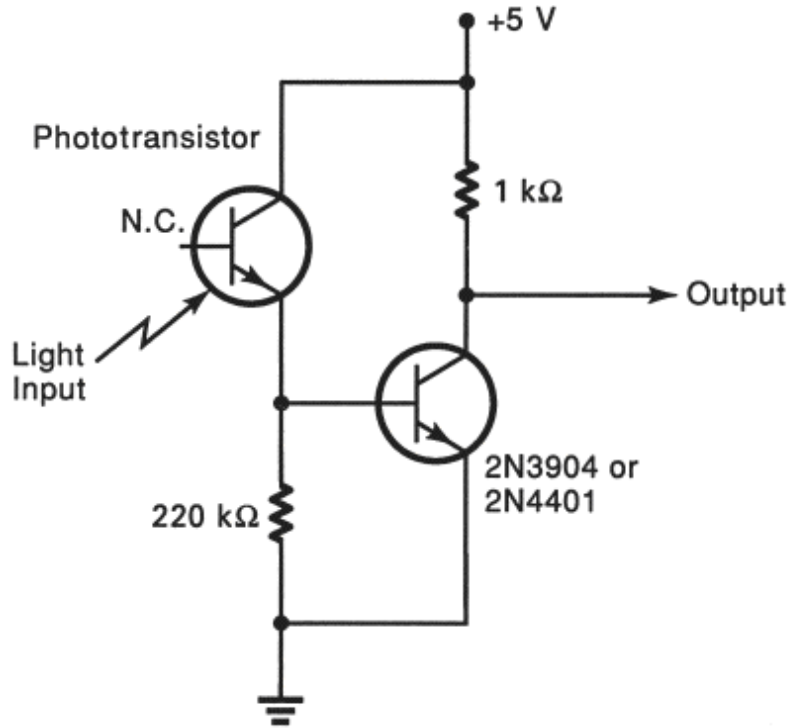


Fig. 20-2 Receiver circuit.

4. Connect a TTL square wave from a function generator to the input of the transmitter and set the transmitter frequency to about 1 Hz. Apply power to the transmitter circuit and look inside the

hole where the fiber cable goes. You should see the faint red glow of the LED switching off and on at a 1-Hz rate.

5. Turn off the power and wire the receiver circuit as in Fig. 20-2.
6. Prepare the fiber-optic cable by stripping about $\frac{1}{8}$ in. of outer sheath from each end. Use a sharp razor blade or X-Acto knife to cut the end of the fiber clean and square. Rotate the cable as you cut so that you get a flat, even end surface.
7. Remove the knurled nut from the LED housing and slip it over one end of the fiber-optic cable. Then insert the fiber into the LED housing as Fig. 20-1 shows. Push the fiber cable all the way in and then tighten the nut.
8. Repeat the above procedure for the other end of the cable and install the cable in the photo transistor assembly.
9. Set the function generator to supply a square wave with a frequency of 1 kHz. Assuming that input, predict the output from the receiver by studying the transmitter and receiver circuits.
10. Apply power to the circuit. Connect an oscilloscope to the receiver output. What signal do you see? Does it conform to your prediction?
11. What is the receiver output when the transmitter input is 0 (gnd)? What is it when the receiver input is 1 (+5 V)? Record both results.
12. While observing the receiver output, begin increasing the function generator frequency slowly beyond 1 kHz. Describe what happens. Explain why. What is the maximum frequency before the output begins to decrease in amplitude?
13. Verify that it is the light over the fiber-optic cable that is operating the receiver. Disconnect the cable from the transmitter end while observing the pulse output at the receiver. Describe what you see. How close does the cable have to be to the LED to see an output?
14. Can analog signals be transmitted over a fiber-optic link?

page 144

page 145

Project 20-2 Infrared Remote Control

Objective

To demonstrate infrared (IR) remote control.

Required

- 1 oscilloscope
- 2 power supplies, +12 V and -12 V
- 1 remote control unit from a TV set, VCR, etc.
- 1 IR diode (940-nm wavelength), IRD500 or equivalent
- 1 op amp, general-purpose (741, LF411, AD548, etc.)
- 2 0.1 μ F ceramic disk capacitors
- 1 10-M Ω
- Batteries for the remote control unit (usually 2 AAA type)

Introduction

IR (infrared) remote control units are used on virtually all TV sets today. They are also used in many other applications, including remote controls for radios, stereo systems, CD players, ceiling fans, VCRs, and cable boxes. IR is also used in computer systems for wireless local area networks (LANs) and as a wireless interface to printers and personal digital assistants (PDAs). IR is cheap and easy to use.

The digital data to be serially transmitted is applied to a modulator that transmits the binary 1s and 0s as bursts of high-frequency pulses. These pulses then turn an infrared LED off and on. Different types of binary codes have been developed to represent numbers and control functions (volume, channel change, etc.). As the button on a remote is pressed, the transmitter sends the binary code for that digit or function again and again.

In this experiment, you will build an IR receiver and observe the digital codes produced by a typical TV remote control.

Procedure

1. Construct the circuit shown in Fig. 20-3. The IR sensitive diode operates with reverse bias. The IR diode looks like a clear LED. The short lead is the cathode, and the cathode is the lead nearest the flat side of the housing. When infrared light strikes

the diode, it conducts and a small current flows. This current is amplified by the op amp and produces an output. You will monitor the op amp output on the oscilloscope.

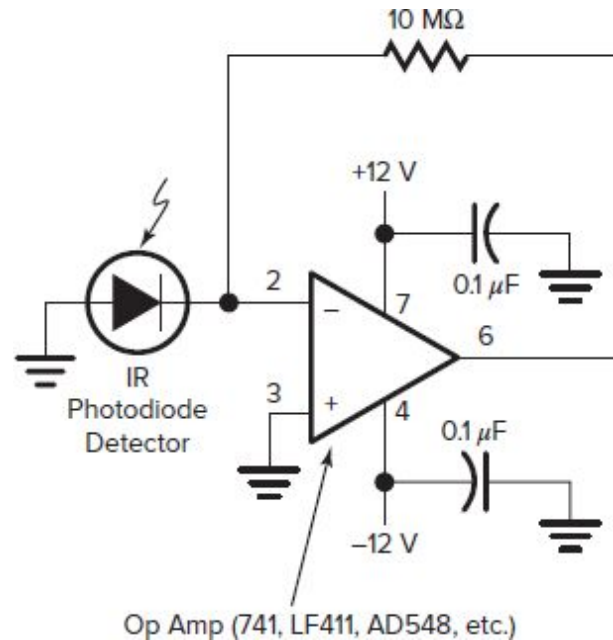


Fig. 20-3 IR receiver.

2. Apply power to the circuit. Connect the scope to the op page 146
amp output.
3. Point the remote control at the IR diode and press any key. Be sure that the remote is less than 1 ft away from the diode. Hold the key down continuously while you adjust the oscilloscope for a stable display. Experiment with the position of the remote with respect to the diode. Adjust the position for maximum output voltage on the scope. What do you see?
4. While continuously pressing one of the buttons on the remote control unit, adjust the scope sweep to display the serial binary code. You will have to play around with the sweep setting and triggering control to get a stable display.
5. Measure the pulse spacing and pulse width of the pulses you observe.
6. Explain how the pulses differ with different buttons depressed.
Note: To observe the signal on the scope, you will need to keep one of the buttons on the remote unit pressed continuously.

This will considerably drain the batteries in the remote. If the signal gets weak or the unit “dies,” you can assume that the batteries must be replaced.

7. Press the 1 through 9 buttons and try to discern the binary code.
8. While pointing the remote at the receiver, back away and increase the distance between you with the remote and the receiver. Continue increasing the distance until the signal is no longer shown on the scope. You may need a partner to observe the scope and tell you when the signal disappears. Measure the maximum distance before the signal drops out.
9. While pointing the remote directly at the IR diode with about 5 ft separating the two, begin moving horizontally to the left in a circular arc while still keeping the diode in the same plane. See Fig. 20-4. At some point, the signal will disappear. At this point, estimate the angle between the centerline and the line where the signal dropped out. Repeat this procedure to the right. What is the approximate total view angle of the IR diode?

Discussion Topics

1. What do you think are the three most commonly pressed buttons on a remote control unit?
2. Explain the results you got in step 8 of the Procedure.
3. If another remote control from a different product is available, investigate its performance by repeating steps 2 through 6. Describe your results. Are the pulses and codes the same?

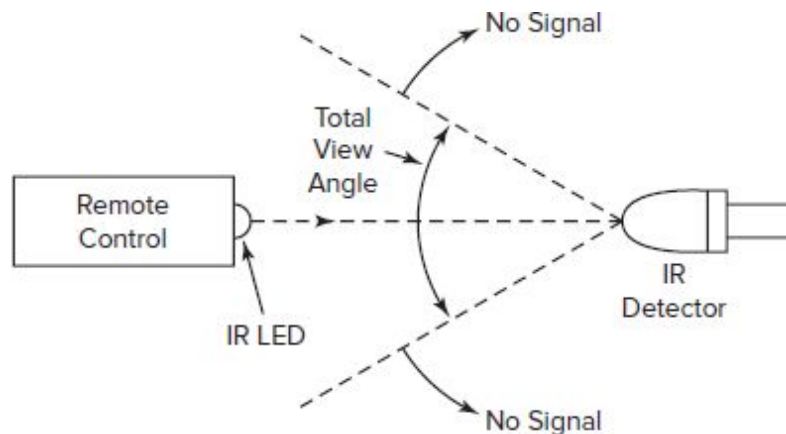


Fig. 20-4 View angle of the IR diode.

Objective

To further explore fiber-optic communications systems and dense wavelength division multiplexing.

Introduction

The fastest communications and networking techniques use fiber-optic methods. High-speed lasers transmit the data over glass or plastic fiber cable at speeds to 100 Gbps and higher. Further bandwidth increases are coming from a frequency division multiplexing method called dense wavelength division multiplexing (DWDM). It uses different wavelength, frequencies, or “colors,” of light to transmit multiple high-speed streams on the same fiber. This project gives you a review of the basic fiber-optic communications methods and further details on DWDM.

Required

PC with Internet access.

Procedure

1. Perform a Web search on the terms fiber optic technology, dense wavelength division multiplexing (DWDM), polarization mode and chromatic dispersion, and similar terms. Seek out tutorial material.
2. Answer the questions below.

Questions

1. Why doesn't fiber replace twisted pair in the local loop?
2. What is the term describing the connection for the consumer's home by fiber-optic cable?
3. What is the name of the short-range fiber network that is beginning to provide fiber to consumers? Name two standards associated with them.

4. What are the primary factors that restrict the data rate and distance on fiber-optic cable? How is it overcome?
5. Name two standards used for 100 Gbps fiber systems.

NAME

DAT

E _____

A graphic for Chapter 21 featuring the text 'Chapter 21' in white. The text is centered within a dark, irregular, cloud-like shape that has a gradient from black to grey. The number '21' is significantly larger than the word 'Chapter'.

21

Cell Phone Technologies

Project 21-1 Cell Phone Technology in Depth

Objective

To explore the operation of the cell phone network in more detail.

Introduction

The text provides an introduction to and overview of the various cell phone technologies. This project gives you a chance to dig deeper into the specific details of cell phone systems and operation.

Required

PC with Internet access.

Procedure

1. Perform an Internet search on the terms 3G and 4G cell phones and LTE.
2. Answer the questions below.

Questions

1. Name the two most common forms of third-generation cell phone technologies.
2. Name the four largest and most widespread cell phone carriers. What 2G and 3G technologies do each use?
3. Define EV-DO and EV-DO Rev A and Rev B. What company developed this technology?
4. Define HSDPA, HSUPA, and HSPA+.
5. What is LTE?
6. What organization defines, develops, and maintains 3G/4G cell phone standards?

Project 21-2 Long-Term Evolution and OFDM

Objective

To explore the details of the fourth-generation Long Term Evolution technology.

Introduction

The fourth-generation Long-Term Evolution (LTE) cell phone technology has been adopted around the world as the standard for cell phones. Most carriers have already built LTE networks and are expanding them as they gradually phase out the older 2G and 3G systems. Because LTE is so pervasive, you should be familiar with it. This project will help you identify the key facts about this technology and its base modulation methods using OFDM.

Required

PC with Internet access.

Procedure

1. Do an Internet search on the terms LTE, LTE Advanced, and OFDM. Seek out tutorial materials.
2. Answer the questions below.

Questions

1. Has LTE totally replaced the older 2G and 3G technologies? Explain.
2. What are the channel bandwidths allowed for LTE OFDM?
3. What modulation methods are used with OFDM?
4. What is the subcarrier bandwidth in LTE OFDM?
5. What are the MIMO configurations allowed in LTE?
6. What is the maximum possible downlink data rate for LTE under ideal conditions?
7. What are the main features of LTE Advanced?
8. How are higher data rates achieved in LTE-A?
9. Are voice calls handled over LTE? Explain.
10. What is the multiuser access mode used in LTE?
11. What is the major concern that most cellular operators share about the future of LTE?
12. What may be the technology of 5G cellular systems?

Project 21-3 Small Cell Networks

Objective

To investigate the small cells movement for future cellular expansion.

Introduction

Small cells are miniature cellular base stations that are designed to expand existing cellular telephone systems. They are referred to by the term *heterogeneous networks* or HetNets. They offer several important benefits over the larger standard macro base stations. This project will help you understand the basics of small cells.

Required

PC with Internet access.

Procedure

1. Search on the terms small cells, heterogeneous networks, or similar terms. Seek tutorial materials.

2. Answer the questions below.

Questions

1. List the four major sizes or types of small cells and their basic caller capacities.
2. What is the main rationale for small cells? What problems do they solve, and what are the benefits?
3. Do small cells replace macro base stations?
4. What is the radio technology of small cells?
5. How are small cells powered?
6. What is the backhaul for small cells?
7. Is a Wi-Fi access point a small cell?
8. What is SON?

NAME

DAT

E _____

A graphic for Chapter 22 featuring the text 'Chapter 22' in white. The text is centered within a dark, irregular, cloud-like shape that has a gradient from black to grey. The number '22' is significantly larger than the word 'Chapter'.

22

Wireless Technologies

Project 22-1 Comparing Short-Range Wireless Technologies

Objective

To examine the most widely used short-range wireless technologies in more detail.

Introduction

There are literally dozens of different wireless technologies in use today. The most popular are Wi-Fi, Bluetooth, ZigBee, NFC, RFID, and ISM band. Each seems to have targeted a specific application niche. And each has unique characteristics. In this project, the object is to dig deeper to get a more thorough understanding of the various technologies available.

Required

PC with Internet access.

Procedure

1. Using Internet searches and any other available resources, make a table listing the six technologies listed in the introduction and give the following information for each:
 - a. Operating frequency range
 - b. Maximum data rate
 - c. Standard designation and standard organization, if applicable
 - d. Modulation method(s)
 - e. Maximum range
2. For each of the six wireless technologies you identified above, list its primary application.
3. Search for any new wireless standard not listed in Chapter 22 of the text or in this project. List at least two.
4. What FCC regulation generally applies to these technologies?
5. What short-range wireless technologies are usually found in smartphones?

Project 22-2 Wireless Monitoring

Objective

To demonstrate how physical conditions can be monitored wirelessly.

Introduction

There are many applications that require only simple monitoring of physical conditions. Some examples are the monitoring of open windows or doors or the position of an object. A single switch can be used as a sensor. A roller lever switch could be used to sense an open garage door. A magnetic reed switch would make a good sensor on a window. Special limit switches are available for a wide range of indications. If the switch is open, it signals that the door or window is closed. However, if the switch closes, this is an indication that the door or window is open. The closed switch can then send a signal to indicate this fact.

The wireless transmitter and receiver modules used in Project 16-2 can be used to make a wireless monitoring system. Figure 22-1 shows the transmitter diagram, and Fig. 22-2 shows the receiver. If

the transmitter switch is open, no power is applied to the transmitter circuits, so no signal is sent. Closing the switch applies power to the circuit. A 555 timer IC supplies a 1.5 Hz rectangular wave to the TX3A transmitter. This is indicated by a flashing LED. The transmitter is frequency modulated by the 555 rectangular pulses and sends a 914 MHz signal to the receiver. The receiver picks up the signal, demodulates it, and supplies the recovered rectangular wave to a transistor that flashes an LED indicating an open condition.

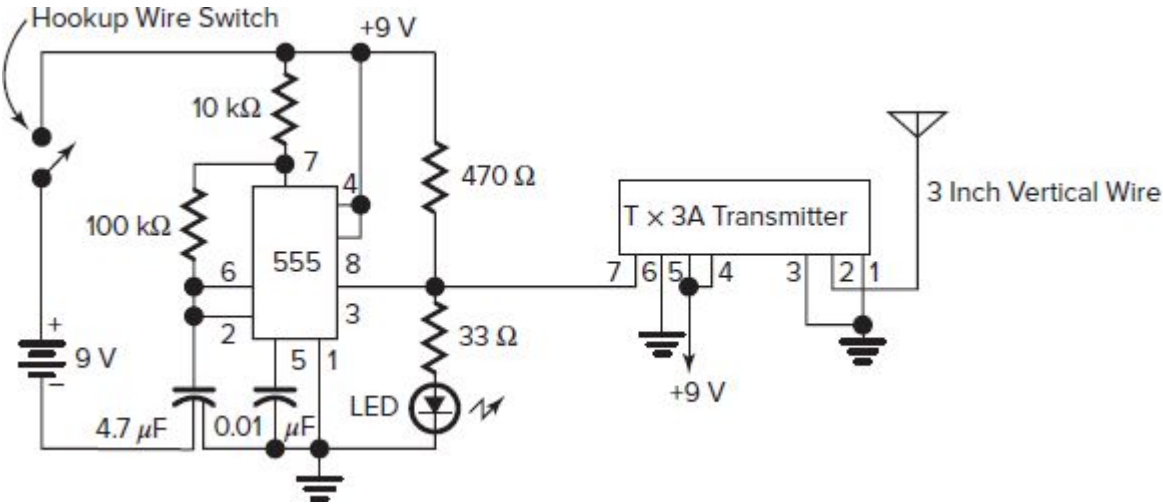


Fig. 22-1 Transmitter circuit.

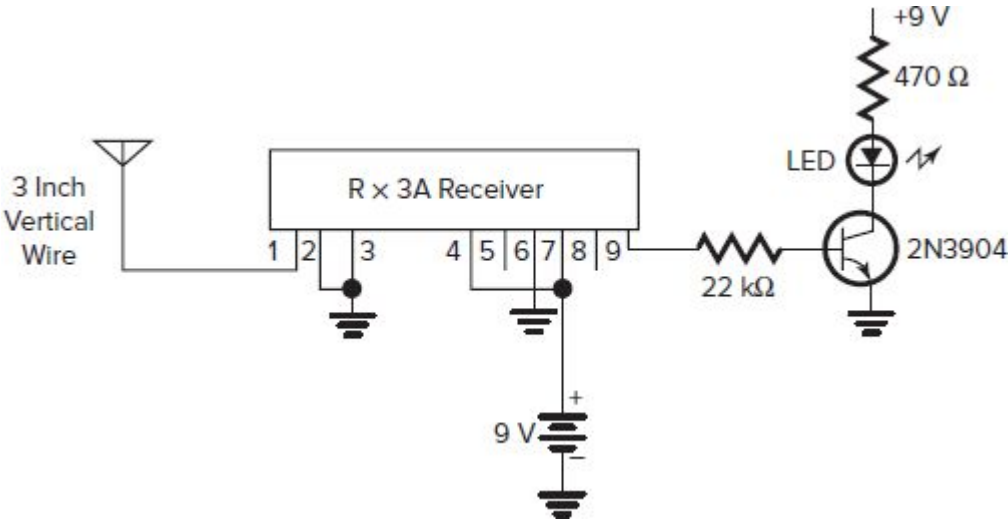


Fig. 22-2 Receiver circuit.

Required

- 1 Radiometrix TX3A-914-64 UHF transmitter
- 1 Radiometrix RX3A-914-64 UHF receiver
- 2 breadboarding sockets, one each for transmitter and receiver
- 2 power supplies for transmitter and receiver (9V), battery with battery terminal clips
- 1 555 timer IC
- 2 LEDs
- 1 2N3904 NPN transistor or equivalent
- 1 100-k Ω resistor
- 1 10-k Ω resistor
- 2 470 Ω resistor
- 1 33 Ω resistor
- 1 4.7 μ F electrolytic or tantalum capacitor
- 1 0.01 μ F ceramic capacitor

Several feet of #22 hookup wire to use as the switch contacts and as the antennas

Procedure

1. Construct the transmitter circuit on one of the breadboarding sockets. The antenna is simply a 3-in. length of hookup wire plugged vertically into one of the breadboarding connections that connect to pin 2 of the TX3A. A pair of hookup wires will serve as the switch. Just touch the two bare wires together to close the switch and apply power.
2. Test the transmitter by connecting the hookup wires together. The LED should flash. If it does, disconnect power.
3. Construct the receiver on the other breadboarding socket. Again, use a 3-in. piece of vertical hookup wire for the antenna on pin 1 of the receiver.
4. Test the receiver by applying power. Then turn on the transmitter nearby by closing the hookup wire switch. The receiver LED should flash. If all is OK, disconnect power.
5. A key test of the circuit is to separate the transmitter and receiver to see the maximum possible range.

Questions

1. What was the maximum range of the system? What limits it?
2. What FCC rules (which Part?) govern wireless on this frequency?

NAME

DAT

E _____

A graphic for Chapter 23 featuring the text 'Chapter 23' in white. The text is centered within a dark, irregular, cloud-like shape with a gradient from black to grey.

Chapter

23

Communication Tests and Measurements

Project 23-1 RF Test Equipment Familiarization

Objective

To explore the types of test equipment used in RF testing.

Introduction

Testing RF equipment is one of the most challenging parts of working with wireless systems. The reasons for this stem largely from the very high frequencies involved as well as from the complexity of most of the widely used systems. Furthermore, the equipment must conform to the rules and regulations of the FCC and of one or more standards organizations. The test instruments used for these tests and measurements are among the most sophisticated and expensive available. The purpose of the project is to become familiar with the key instruments used in such work.

Required

PC with Internet access.

Procedure

1. Using Internet access, go to the websites below.

www.aeroflex.com
www.anritsu.com
www.keysight.com
www.lecroy.com
www.rhode-schwarz.com
www.tektronix.com

Examine the types of test instruments available for RF and wireless.

2. Using the information you gained from the sites listed above, and any necessary Internet search, answer the questions below.

Questions

1. Name the three primary test instruments used in RF test.
2. Which instrument, an oscilloscope or a spectrum analyzer, gives the most useful information about a complex modulated signal?
3. What arbitrary waveform generator (AWG) products have the highest output frequency? What is it? Can AWG signals include modulation?
4. What test instrument is used to produce an eye diagram?
5. What test instruments would you use to measure jitter? What is the unit of measurement for jitter?
6. What is a vector signal analyzer? What is it used to test?
7. What is the best instrument to be used to find EMI?

Appendix

A

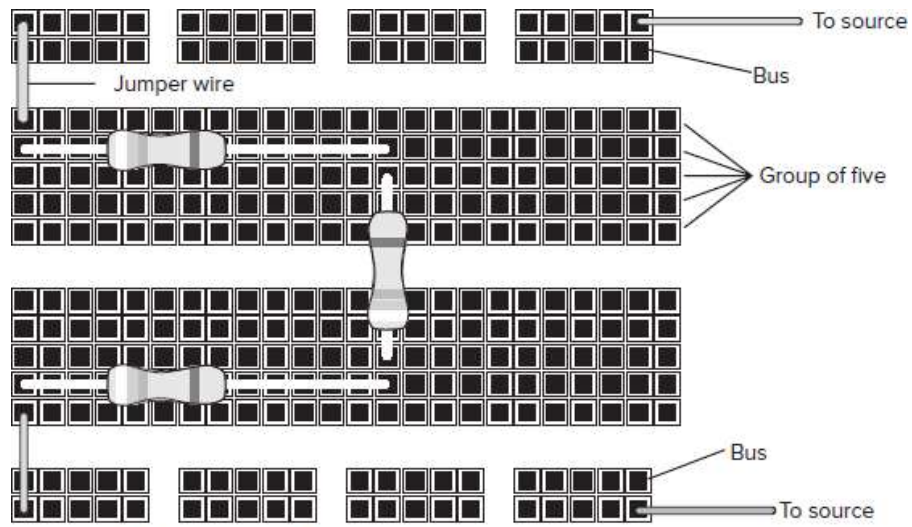
Breadboarding

CAUTION Breadboarding is generally limited to low-power circuits. High currents would cause high temperatures, and the breadboard would be damaged. High voltage could cause arcing and would be very dangerous. If there is any doubt, consult your instructor before breadboarding a circuit.

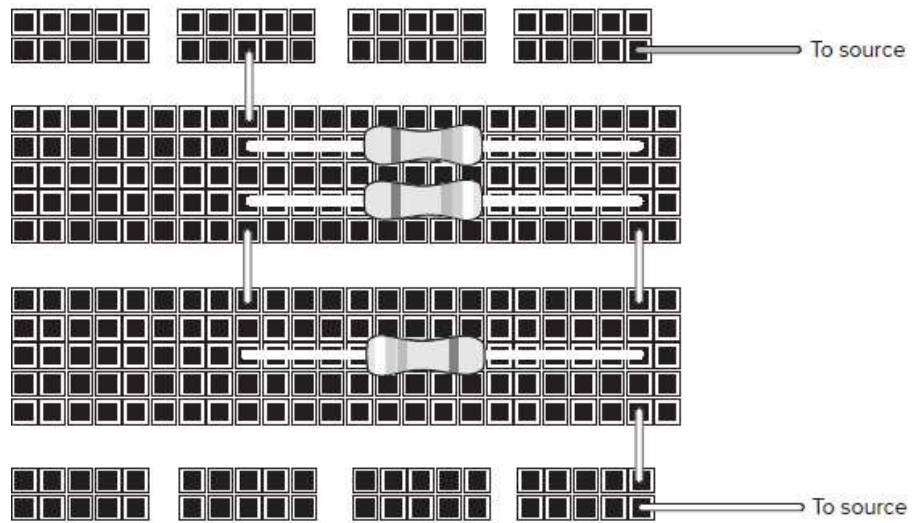
The name *breadboarding* has its origin in the early years of electronics, when circuits were sometimes fashioned on a flat piece of wood that looked like a breadboard (or actually was a breadboard). In case you didn't know, breadboards are used in some kitchens for working with dough or cutting bread. Today, wood breadboards are seldom used, and circuits are prototyped and tested using electronic breadboards, which are molded plastic with holes and internal metal contact strips. A circuit can be easily and quickly built by shoving component leads and jumper wires into the holes. Electronic breadboards are also called *prototyping boards*.

Figure A-1(a) shows how a series circuit appears on a typical electronic breadboard. Each group of five holes is electrically

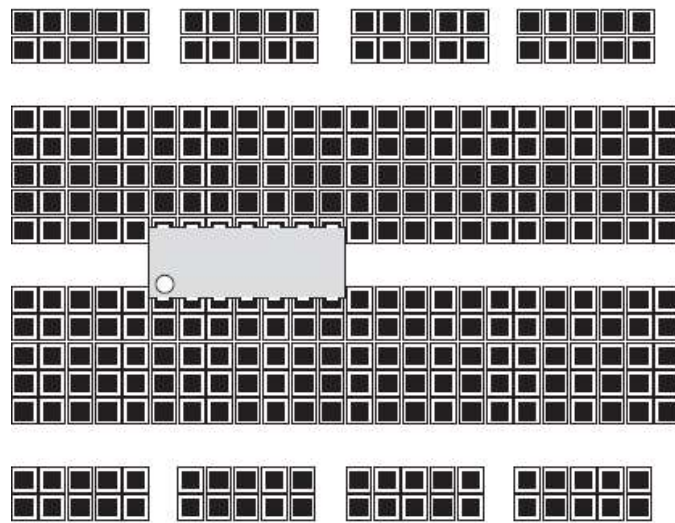
connected. Or, put another way, there is electrical continuity from any one contact to the other four contacts in that group. So, the top horizontal resistor is connected to the vertical resistor because each has a lead that is inserted into one group of five in Fig. A-1(a). Note that the vertical resistor and the bottom horizontal resistor are connected using another group of five.



(a) Series circuit on a breadboard



(b) Parallel circuit on a breadboard



(c) Integrated circuit on a breadboard

Fig. A-1 Electronic breadboard.

There are also buses on breadboards. They are most often used for power, ground, or any signal requiring more than five common contact points. Figure A-1(a) shows two buses at the top and two at the bottom. Every contact along a bus is electrically connected to every other contact on that bus. Thus, the top source wire in Fig. A-1(a) is electrically connected to the top horizontal resistor via the top bus and jumper wire. Likewise, the bottom source wire is connected to the bottom horizontal resistor via the bottom bus and jumper wire. Note that the four buses in Fig. A-1(a) are electrically isolated from each other. So, one could use them as follows:

- +12-V bus
- -12-V bus
- +5-V bus
- ground bus

Figure A-1(b) shows a parallel circuit. Notice how jumper page 161 wires are used to connect the top two resistors to the bottom resistor. There is a problem in Fig. A-1(b) in that both source leads are not connected to the resistors. Can you see why? How can the problem be corrected? Two possible answers are:

- Move the bottom source lead to the third bus down in Fig. A-1(b)
- Add a jumper wire from the third bus down to the bottom bus in Fig. A-1(b)

Figure A-1(c) shows that integrated circuits in a dual-inline package conform to the contact arrangement of the breadboard. The IC pins and the board contacts have the same center-to-center spacing of 0.1 in. Each IC pin in Fig. A-1(c) has four contact points available for interconnections. In cases when this is not enough, a jumper wire can connect an IC pin to an unused group of five, or two jumper wires to two groups, and so on.

Jumper wires are usually made from no. 22 or no. 24 solid, tinned, insulated wire. About $\frac{1}{4}$ in. of insulation is removed from each end.

After a wire is used many times, the end might become bent and fragile. When this happens, cut off the damaged end. It is a good idea to use different wire colors, for example, red for positive, black for ground, green for negative, and white for signal paths. This becomes more of an issue when large, complicated circuits are breadboarded.

In classroom or lab settings, the component leads are usually left full-length, unless the ends are damaged as mentioned before. This is because the components are used over and over. This means that components might stand above the breadboard by an inch or more. This is usually acceptable, but be sure that they don't flop around and touch each other and cause short circuits.

Some component wires might be too big to fit comfortably into the contact openings. Do not force them. One solution is to solder a short length of no. 22 wire to the end of any wire too big in diameter for the breadboard. Some components cannot be soldered without ruining them. Fuses and lamps are examples. In these cases, fuse holders or lamp sockets will be required for soldering to the no. 22 wires. Consult your instructor in these cases.

Rules and Tips for Breadboarding

- High-power circuits should *not* be breadboarded.
- High-frequency circuits are usually not breadboarded. The higher the frequency, the more this is true.
- High-gain circuits might not perform well. There are often problems with noise and oscillation.
- Keep all component leads and wires as short as possible.
- Surface-mount components cannot be used without soldering leads to them or using special adapters (these are available for some surface-mount ICs).
- Arrange the placement of physical parts similar to the placement in the schematic drawing. This reduces errors and allows for easier checking. However, in the case of high-gain circuits, the output should be located away from the input regardless of how the schematic is drawn.

- Inventory all the parts before breadboarding. This reduces errors of omission and makes it less likely that an incorrect part will be used.
- Have your partner or a third party check your work for errors. He or she will often notice things that you will not.
- Work methodically. Wire all the power connections, page 162 then the grounds, and then the signal paths. Another method is to do resistors first, then capacitors, then ICs, and so on. Some people place a pencil check on the schematic for each interconnection and part as they are wired.
- Bypass the power bus or buses to ground with a 15- or 25- μ F electrolytic or tantalum capacitor. Be sure to observe polarity and to use capacitors with an adequate voltage rating. Power-supply bypass capacitors will often correct strange behavior arising from noise or unwanted oscillation. The bypass capacitors should be located near any high-gain or high-speed logic devices that could cause a problem. For TTL and CMOS logic ICs, a 0.1- μ F ceramic capacitor is recommended across the power to ground pins.

Appendix

B

GROL Review/Final Exam

As you learned in Project 1-1, the General Radiotelephone Operators License (GROL) is a credential issued by the FCC to those persons who want to work on selected marine and aircraft radio and other electronic equipment. To receive this license, you must pass an exam consisting of questions from Elements 1 and 3. Element 1 covers rules, regulations, and operating procedures. Element 3 covers electronic fundamentals, communications theory and circuits, and related equipment. The exam must be taken with one of several Commercial Operators License Examination Managers (COLEMs). You can find the details about the GROL and COLEMs at the FCC website:

wireless.fcc.gov/commoperators/index.htm?job=home

Look for the section Commercial Operator Home. You can review the types of licenses available and their requirements.

To help you prepare for the exam, the FCC provides a pool of sample questions for Elements 1 and 3. Element 3 also provides a good review of all the basic principles covered in the text related to this manual (*Principles of Electronic Communication Systems, 5th edition*) as well as a good summary of the related electronic fundamentals. Element 3 or parts thereof also make a good final examination for a course using this text.

The Element 3 question pool contains hundreds of questions divided up into sections. These are:

- Subelement A Principles
- Subelement B Electrical Math
- Subelement C Components
- Subelement D Circuits
- Subelement E Digital Logic
- Subelement F Receiver Theory
- Subelement G Transmitters
- Subelement H Modulation
- Subelement I Power Sources
- Subelement J Antennas
- Subelement 3K Aircraft
- Subelement 3L Installation, Maintenance, & Repair
- Subelement 3M Communications Technology
- Subelement 3N Marine
- Subelement 3O Radar
- Subelement 3P Satellite
- Subelement 3Q Safety

You should print out a copy of the Element 3 question page 164 pool. It is a Microsoft Word file. There is a group of questions for each of the subelements. Answers are given at the bottom of each page.

Subelements A through E cover fundamentals that are the prerequisite to a communications course. These are a good overview of the basics every communications engineer or technician should know.

Subelements F through J plus 3L and 3M cover material in the accompanying textbook.

The remaining subelements cover topics unique to the GROL license requirements.

Instructors

You can use these questions for chapter quizzes or on a final exam.

Students

You can use the questions as chapter reviews. You can also use the questions as a learning exercise. Give yourself an exam using the questions to see how you do. Review any topics you miss. Look up the answers to the questions you do not know in the accompanying text or another text, or do an Internet search.

Finally, plan to study for the FCC GROL exam. It is within reach with a little study. It is an excellent hiring credential as the license is widely recognized throughout industry.



501-1111