The Art of Electronics

Laboratory Curriculum

A Comprehensive Project-Based Course

Companion to Horowitz & Hill's Classic Text

with Supplementary Material from *The X Chapters*

AoE

Course Philosophy:

"Learn by building real instruments you'll keep on your bench."

Each phase anchors to specific themes in *The Art of Electronics* and culminates in practical, professional-quality projects.

Total Duration: 24–36 weeks (self-paced)

Prerequisites: Basic physics, algebra, introductory calculus

Outcome: Mixed-signal capstone instrument

Course Materials and Licensing

This curriculum is designed as a companion to:
The Art of Electronics (3rd Edition)
by Paul Horowitz and Winfield Hill
Cambridge University Press, 2015

and

The Art of Electronics: The X Chapters
by Paul Horowitz and Winfield Hill
Cambridge University Press, 2020

Document Version: 1.0
Last Updated: December 1, 2025

Recommended Citation:
Electronics Laboratory Curriculum, Project-Based Companion to
The Art of Electronics, Version 1.0, 2025.

Contents

Preface **9**

1 Orientation & Laboratory Setup **11**
1.1 Phase Overview and Goals . . . 11
1.2 Laboratory Setup Requirements . . . 12
1.2.1 Workbench Organization . . . 12
1.2.2 Test Equipment . . . 13
1.2.3 Prototyping Materials . . . 13
1.2.4 Software Requirements . . . 13
1.3 Safety Fundamentals . . . 14
1.3.1 Electrostatic Discharge (ESD) Protection . . . 14
1.3.2 Power Supply Safety . . . 14
1.4 Mini-Project 0.1: Lab Power & Measurement Check . . . 14
1.4.1 Background . . . 15
1.4.2 Circuit Description . . . 15
1.4.3 Procedure . . . 15
1.4.4 Analysis and Comparison with AoE . . . 16
1.5 Mini-Project 0.2: Signal Visualization Workshop . . . 17
1.5.1 Exercises . . . 17
1.6 Laboratory Notebook Best Practices . . . 18
1.6.1 Notebook Format . . . 18
1.7 Phase 0 Checklist . . . 19

2 Analog Foundations & Building Blocks **21**
2.1 Phase Overview and Goals . . . 21
2.2 Theoretical Foundation . . . 22
2.2.1 The Op-Amp as a Universal Building Block . . . 22
2.2.2 Transistor Fundamentals . . . 22
2.2.3 Filter Fundamentals . . . 23
2.3 Core Project: Analog Building-Block Toolkit Board . . . 23
2.3.1 System Requirements . . . 24
2.3.2 Subsystem 1: Configurable Op-Amp Stage . . . 24
2.3.3 Subsystem 2: Second-Order Active Low-Pass Filter . . . 25
2.3.4 Subsystem 3: Transistor Switch . . . 25
2.3.5 Subsystem 4: Emitter Follower Buffer . . . 26
2.3.6 System Integration . . . 27
2.4 Troubleshooting Guide . . . 27

2.4.1 Op-Amp Circuits 27
2.4.2 Transistor Circuits 28
2.5 Phase 1 Checklist 28

3 Precision & Low-Noise Instrumentation **29**
3.1 Phase Overview and Goals 29
3.2 Noise Fundamentals 30
3.2.1 Types of Noise 30
3.2.2 Noise Calculations 30
3.3 Precision Amplifier Topologies 31
3.3.1 The Instrumentation Amplifier 31
3.3.2 The Transimpedance Amplifier (TIA) 31
3.4 Core Project: Sensor Front-End & Data Logger 31
3.4.1 System Architecture 32
3.4.2 Option A: Photodiode Light Meter 32
3.4.3 Option B: Strain Gauge / Load Cell 33
3.4.4 Option C: Thermocouple / RTD 33
3.4.5 Noise Budget Template 34
3.4.6 Grounding and Shielding Practices 34
3.5 Phase 2 Checklist 34

4 Power Electronics & Protection **37**
4.1 Phase Overview and Goals 37
4.2 Linear vs. Switching Regulators 38
4.2.1 Linear Regulators 38
4.2.2 Switching Regulators 38
4.2.3 When to Use Each 39
4.3 Core Project: Multi-Rail Laboratory Power Supply 39
4.3.1 Specifications 39
4.3.2 Protection Features 39
4.3.3 Architecture Options 40
4.3.4 Adjustable Regulator Design 40
4.3.5 Testing Procedures 41
4.4 Phase 3 Checklist 42

5 Digital Logic, Interfaces, and Mixed-Signal Integration **43**
5.1 Phase Overview and Goals 43
5.2 Logic Families and Interfacing 44
5.2.1 Common Logic Families 44
5.2.2 Level Shifting 44
5.3 ADC and DAC Fundamentals 44
5.3.1 ADC Selection Criteria 45
5.3.2 Effective Number of Bits (ENOB) 45
5.4 Core Project: Mixed-Signal Measurement & Control Unit 45
5.4.1 System Requirements 45
5.4.2 Microcontroller Selection 45
5.4.3 Interface Design Requirements 46
5.4.4 PCB Layout Guidelines 46

5.5 Phase 4 Checklist . . . 47

6 High-Speed, RF-Lite, and Signal Integrity 49
6.1 Phase Overview and Goals . . . 49
6.2 When Does High-Speed Matter? . . . 50
6.3 Core Project: High-Speed Buffer / Line Driver . . . 50
6.3.1 Specifications . . . 50
6.3.2 Experiments . . . 51

7 Capstone: Integrated Instrument or System 53
7.1 Phase Overview and Goals . . . 53
7.2 Capstone Project Options . . . 53
7.2.1 Option A: Modular Data Acquisition System . . . 53
7.2.2 Option B: Precision Bench Instrument . . . 54
7.2.3 Option C: Smart Power Platform . . . 54
7.3 Capstone Requirements . . . 54
7.3.1 System Architecture Document . . . 54
7.3.2 Complete Schematics and Layout . . . 55
7.3.3 Test and Characterization Plan . . . 55
7.3.4 Test Results and Analysis . . . 55
7.3.5 Reflection Document . . . 55

A How to Use *The Art of Electronics* Along the Way 57
A.1 Pre-Reading Strategy . . . 57
A.2 Design Bookmarking . . . 57
A.3 Post-Mortem Reading . . . 57

B Equipment Recommendations 59
B.1 Budget-Conscious Setup . . . 59
B.2 Professional Setup . . . 59

C Component Sources 61
C.1 Major Distributors . . . 61
C.2 Budget Options . . . 61
C.3 Specialty Suppliers . . . 61

Preface

This curriculum represents a comprehensive, hands-on approach to learning electronics alongside the seminal text *The Art of Electronics* by Horowitz and Hill. Rather than treating electronics as a purely theoretical discipline, this course embraces the philosophy that true understanding comes from building, measuring, debugging, and iterating on real circuits.

Course Philosophy

The approach taken here differs from traditional electronics courses in several key ways:

Project-Centered Learning: Each phase culminates in one or more substantial projects—not exercises or problem sets, but real instruments and tools you would be proud to keep on your workbench.

Iterative Understanding: You will encounter concepts multiple times at increasing levels of sophistication. A transistor encountered as a simple switch in Phase 1 becomes a precision amplifier stage in Phase 2 and a high-speed buffer in Phase 5.

Measurement-Driven Design: Every design is verified through careful measurement. The difference between calculated and measured performance is not a source of frustration but the most valuable learning opportunity.

Integration Over Isolation: Real instruments are mixed-signal systems combining analog, digital, and power electronics. This curriculum builds toward that integration from the beginning.

How to Use This Curriculum

This document can serve multiple purposes:

- **Self-Study Guide:** Work through the phases at your own pace, spending more time on areas that challenge you.
- **Formal Course Spine:** Instructors can use this as a laboratory component for a one-year electronics sequence.
- **Project Reference:** Experienced practitioners can pick individual projects that address specific learning goals.

Prerequisites

This course assumes:

- Basic physics (electricity and magnetism fundamentals)

- Algebra and introductory calculus
- Willingness to learn circuit simulation software
- Access to basic laboratory equipment (detailed in Phase 0)

No prior electronics experience is required, though students with some background will progress more quickly through the early phases.

Acknowledgments

This curriculum owes its existence to the extraordinary work of Paul Horowitz and Winfield Hill. Their text has educated generations of electronics practitioners, and this course attempts to create a practical companion to their theoretical exposition.

Build well. Measure carefully. Learn continuously.

Chapter 1

Orientation & Laboratory Setup

Duration: 1–2 weeks **Difficulty:** Beginner

AoE Reference

Primary Reading:

- Preface and Introduction (philosophy of practical electronics)
- Appendix A: Oscilloscope Primer
- Appendix B: Power Supplies
- Appendix N: Laboratory Practice

Supplementary from *The X Chapters*:

- Chapter 1x: Real-World Passive Components

Before diving into circuit design and construction, you must establish a safe, organized laboratory environment and become comfortable with your measurement tools. This phase, while seemingly administrative, establishes habits that will pay dividends throughout the course.

1.1 Phase Overview and Goals

The primary goal of this phase is to prepare your workspace and establish proficiency with fundamental tools before tackling substantive projects. A well-organized lab with properly understood equipment eliminates countless hours of frustration later.

Learning Objectives

By the end of this phase, you will be able to:

1. Set up a safe, organized electronics workbench with proper ESD protection
2. Use a digital multimeter (DMM) for voltage, current, and resistance measurements
3. Operate an oscilloscope including triggering, timebase adjustment, and cursor measurements
4. Generate and verify test signals using a function generator

5. Perform basic circuit simulation using SPICE software
6. Document measurements in a laboratory notebook following professional conventions
7. Apply AoE's rules-of-thumb to predict and verify circuit behavior

1.2 Laboratory Setup Requirements

1.2.1 Workbench Organization

Your workbench is the foundation of all your work. A poorly organized bench leads to mistakes, safety hazards, and wasted time.

Equipment Required

Essential Workbench Equipment:

- ESD-safe work mat (minimum 24" × 36")
- Grounding wrist strap and grounding point
- Adequate lighting (adjustable desk lamp with magnification preferred)
- Power strip with surge protection (minimum 6 outlets)
- Storage bins for components, organized by type
- Heat-resistant surface or silicone mat for soldering
- Fume extractor or adequate ventilation for soldering

Essential Hand Tools:

- Diagonal cutters (flush-cut preferred)
- Long-nose pliers (smooth and serrated jaws)
- Wire strippers (adjustable or multi-gauge)
- Precision screwdriver set (Phillips, flathead, Torx)
- Tweezers (fine-point, ESD-safe)
- Soldering iron (40 W–60 W, temperature-controlled)
- Solder (60/40 or lead-free, 0.031" diameter)
- Solder wick and/or solder sucker for desoldering
- Third-hand tool or PCB holder

1.2.2 Test Equipment

Quality test equipment is essential for meaningful measurements. While professional-grade equipment is ideal, entry-level instruments are sufficient for this course.

Table 1.1: Recommended Test Equipment Specifications

Instrument	Minimum Specifications	Recommended Specifications
Digital Multimeter	3.5 digit, DC/AC voltage and current, resistance, continuity	4.5 digit, true RMS, capacitance, frequency, temperature
Oscilloscope	2 channel, 50 MHz bandwidth, 500 MS/s	4 channel, 100 MHz+, 1 GS/s, FFT function
Function Generator	Sine/square/triangle, 1 MHz, amplitude control	DDS-based, 10 MHz+, arbitrary waveform capability
Power Supply	Dual-rail, 0–30 V, 0–3 A, current limiting	Triple output, tracking mode, digital display
LCR Meter	Basic capacitance and inductance	Multi-frequency, Q-factor measurement

1.2.3 Prototyping Materials

- **Breadboards:** Minimum 830 tie-points, multiple boards recommended
- **Jumper Wire Kit:** Solid-core 22 AWG in multiple lengths and colors
- **Perfboard/Stripboard:** For semi-permanent prototypes
- **Component Assortments:**
  - Resistors: 1/4W, 1% metal film, E24/E96 series (10Ω to 1 MΩ)
  - Capacitors: Ceramic (10pF to 1 µF), electrolytic (1 µF to 1000 µF)
  - Diodes: 1N4148 (signal), 1N400x (rectifier), various Zeners
  - Transistors: 2N3904/2N3906 (BJT), 2N7000/BS170 (MOSFET)
  - Op-amps: LM358, TL072, OPA2134 (assortment of general-purpose to precision)
  - Voltage regulators: 78xx/79xx series, LM317

1.2.4 Software Requirements

- **SPICE Simulator:** LTspice (free, recommended), ngspice, or TINA-TI
- **Schematic Capture:** KiCad (free, full-featured) or your simulator's built-in editor
- **Documentation:** Laboratory notebook (physical or digital), spreadsheet software
- **Data Analysis:** Python with NumPy/Matplotlib, MATLAB, or similar

1.3 Safety Fundamentals

Safety Warning

Electronics work involves electrical hazards, chemical exposure (solder flux, cleaning agents), and burn risks. Always:

- Work on de-energized circuits whenever possible
- Use the one-hand rule when measuring live high-voltage circuits
- Allow components (especially voltage regulators, power resistors) to cool before handling
- Work in a ventilated area when soldering
- Wear safety glasses when cutting leads or working with batteries
- Never work when fatigued or distracted

1.3.1 Electrostatic Discharge (ESD) Protection

ESD can destroy sensitive components (CMOS ICs, MOSFETs, precision op-amps) without any visible damage. Establish these habits:

1. Always use a grounded wrist strap when handling sensitive components
2. Store components in ESD-safe bags or conductive foam
3. Touch a grounded metal surface before handling components if no wrist strap is available
4. Use ESD-safe tools (marked with the ESD symbol)
5. Avoid synthetic clothing and carpeted floors in your work area

1.3.2 Power Supply Safety

- Always start with current limiting enabled and set appropriately
- Verify power supply polarity before connecting to circuits
- Use fuses in permanent installations
- Capacitors can retain charge—discharge them before working on circuits
- Never exceed component voltage or power ratings

1.4 Mini-Project 0.1: Lab Power & Measurement Check

Project Description

Project Title: Linear Voltage Regulator Characterization
Objective: Build a simple linear regulator circuit and use it to practice measurement techniques while verifying AoE's rules-of-thumb about regulator behavior.
Time Estimate: 3–4 hours

1.4.1 Background

Three-terminal voltage regulators (78xx series) are ubiquitous in electronics. Despite their simplicity, they illustrate fundamental concepts: voltage regulation, thermal management, and the trade-offs between linear and switching approaches.

AoE discusses these regulators extensively, providing rules-of-thumb for dropout voltage, ripple rejection, and thermal behavior. This project verifies those predictions experimentally.

1.4.2 Circuit Description

Build a 12 V to 5 V linear regulator using a 7805 regulator with proper input and output capacitors.

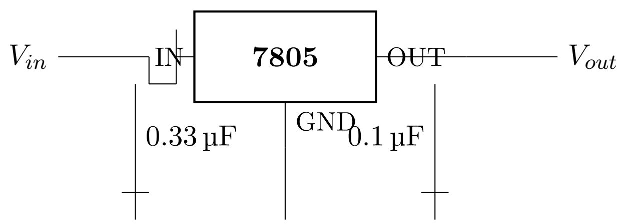

Figure 1.1: Basic 7805 linear regulator circuit

1.4.3 Procedure

Step 1: Build the Circuit

1. Identify the 7805 pinout (check datasheet—variants exist!)
2. Install input capacitor (0.33 µF ceramic) close to the regulator
3. Install output capacitor (0.1 µF ceramic) close to the regulator
4. Add a load resistor (start with 100 Ω, 250 mW dissipation at 5 V)

Step 2: Line Regulation Measurement

Line regulation measures output voltage change as input voltage varies.

1. Set load to 100 Ω (fixed)
2. Vary V_{in} from 7 V to 15 V in 0.5 V steps
3. Record V_{out} at each step
4. Calculate line regulation: $\frac{\Delta V_{out}}{\Delta V_{in}} \times 100\%$

Step 3: Load Regulation Measurement

Load regulation measures output voltage change as load current varies.

1. Set $V_{in} = 12\,\text{V}$ (fixed)
2. Vary load from 0 mA to 500 mA

3. Record V_{out} at each load current
4. Calculate load regulation: $\frac{\Delta V_{out}}{\Delta I_{load}} \times 100\%$

Step 4: Ripple Rejection Measurement

1. Add a 1 V peak-to-peak, 120 Hz sine wave to the DC input (simulating rectifier ripple)
2. Measure input and output ripple with oscilloscope (AC coupling)
3. Calculate ripple rejection ratio in dB

Step 5: Thermal Measurement

1. Apply 500 mA load at $V_{in} = 12\,\text{V}$
2. Calculate power dissipation: $P = (V_{in} - V_{out}) \times I_{load}$
3. Monitor regulator temperature over 10 minutes
4. Calculate thermal resistance: $\theta_{JA} = \frac{T_J - T_A}{P}$

Practical Tip

Use a thermocouple or IR thermometer to measure the regulator tab temperature. Be careful—the regulator can get hot enough to cause burns at high power dissipation!

1.4.4 Analysis and Comparison with AoE

After completing measurements, compare your results with the predictions from *The Art of Electronics*:

Table 1.2: Comparison of Measured vs. Predicted Performance

Parameter	AoE Typical	Datasheet	Measured
Output Voltage	5.0 V ± 4%		
Line Regulation	3 mV/V typ.		
Load Regulation	15 mV typ.		
Ripple Rejection	70 dB typ. @120Hz		
Dropout Voltage	2 V typ.		
Thermal Resistance	∼50 °C/W (TO-220)		

Deliverables

- Completed measurement table comparing AoE predictions to actual results
- Line regulation plot (V_{out} vs V_{in})
- Load regulation plot (V_{out} vs I_{load})
- Oscilloscope screenshot showing input and output ripple

- Thermal measurement data and calculation
- Written analysis: "What I expected vs. what I measured, and why they might differ"

1.5 Mini-Project 0.2: Signal Visualization Workshop

Project Description

Project Title: Oscilloscope and Function Generator Proficiency
Objective: Develop fluency with oscilloscope operation through systematic exploration of waveform generation, measurement, and documentation.
Time Estimate: 2–3 hours

1.5.1 Exercises

Exercise A: Basic Waveform Generation and Measurement

1. Generate a 1 kHz sine wave, 2 V peak-to-peak
2. Display on oscilloscope with proper scaling
3. Measure and record: frequency, period, peak-to-peak amplitude, RMS amplitude
4. Repeat for square and triangle waves at the same frequency

Exercise B: Triggering Mastery

1. Generate a pulse train (variable duty cycle)
2. Practice triggering: rising edge, falling edge, level adjustment
3. Generate a complex waveform (AM modulated or burst mode)
4. Achieve stable triggering on the modulation envelope

Exercise C: Rise Time and Bandwidth

1. Generate a fast square wave (1 MHz or higher)
2. Measure rise time using oscilloscope cursors
3. Calculate the approximate bandwidth of your signal source
4. Understand the relationship: $BW \approx \frac{0.35}{t_r}$

Exercise D: SPICE Simulation Correlation

1. Simulate a simple RC low-pass filter in SPICE
2. Build the same filter on breadboard
3. Apply a square wave input and compare:
   - Simulated output waveform
   - Measured output waveform
4. Document any differences and hypothesize causes

Deliverables

- Annotated oscilloscope screenshots for each waveform type
- Rise time measurement with calculation showing bandwidth
- SPICE simulation output and corresponding oscilloscope capture
- Brief written comparison of simulation vs. measurement

1.6 Laboratory Notebook Best Practices

Throughout this course, maintain a laboratory notebook documenting all work. Good documentation habits are essential for:

- Debugging circuits that "worked yesterday"
- Building on previous work without re-learning
- Professional engineering practice

1.6.1 Notebook Format

Each entry should include:

1. **Date and Title:** Clear identification of the work session
2. **Objective:** What you intend to accomplish
3. **Schematic:** Hand-drawn or printed, with component values
4. **Procedure:** Steps taken, in enough detail to reproduce
5. **Data:** Raw measurements in tables or plots
6. **Observations:** Unexpected behavior, anomalies, questions
7. **Conclusions:** What worked, what didn't, next steps

Practical Tip

Photograph your breadboard layouts and paste them into your notebook. When debugging later, you'll appreciate having a record of exactly how things were connected.

1.7 Phase 0 Checklist

Before proceeding to Phase 1, verify:

- ☐ Laboratory workspace is organized with proper ESD protection
- ☐ All test equipment is functional and calibrated
- ☐ SPICE software is installed and you can run a simple simulation
- ☐ Mini-Project 0.1 (regulator characterization) completed with documentation
- ☐ Mini-Project 0.2 (oscilloscope proficiency) completed with documentation
- ☐ Laboratory notebook established with consistent formatting

Chapter 2

Analog Foundations & Building Blocks

Duration: 3–4 weeks **Difficulty:** Beginner to Intermediate

AoE Reference

Primary Reading:
- Chapter 1: Foundations (1.1–1.7)
- Chapter 2: Bipolar Transistors (2.1–2.3)
- Chapter 3: Field-Effect Transistors (3.1–3.2)
- Chapter 4: Operational Amplifiers (4.1–4.4)
- Chapter 6: Filters (6.1–6.2)

Supplementary from *The X Chapters*:
- Chapter 4x: Advanced Topics in Op-Amps

This phase establishes the analog foundation upon which all subsequent work builds. You will design, build, and characterize the fundamental building blocks that appear in virtually every analog system: amplifiers, buffers, filters, and switches.

2.1 Phase Overview and Goals

The goal of this phase is to develop intuition for analog circuit behavior through hands-on construction and measurement. By the end of this phase, you should be able to look at a new analog circuit and predict its approximate behavior before simulation or measurement.

Learning Objectives

By the end of this phase, you will be able to:

1. Design and analyze inverting, non-inverting, and buffer op-amp configurations
2. Calculate gain, bandwidth, input impedance, and output impedance for basic amplifier stages
3. Design active low-pass and high-pass filters with specified cutoff frequencies

4. Configure transistors as switches for driving LEDs, relays, and other loads
5. Implement emitter/source followers as buffers and understand their loading effects
6. Use SPICE simulation to predict circuit behavior and compare with measurements
7. Recognize and diagnose common problems: oscillation, clipping, loading, offset

2.2 Theoretical Foundation

2.2.1 The Op-Amp as a Universal Building Block

The operational amplifier is perhaps the most versatile component in analog design. Understanding its behavior under negative feedback is essential.

Golden Rules of Op-Amp Analysis

When an op-amp operates with negative feedback:

1. **The output adjusts to make the voltage difference between the inputs zero.**
   $V_+ \approx V_-$ (virtual short)
2. **The inputs draw no current.**
   $I_+ \approx I_- \approx 0$ (for most practical purposes)

These "golden rules" enable quick analysis of most op-amp circuits.

Gain-Bandwidth Product

Real op-amps have finite gain that rolls off with frequency. The gain-bandwidth product (GBW or GBP) is approximately constant:

$$GBW = A_{OL} \times f_{OL} = A_{CL} \times f_{CL} \tag{2.1}$$

where A_{OL} is open-loop gain, f_{OL} is open-loop bandwidth, A_{CL} is closed-loop gain, and f_{CL} is closed-loop bandwidth.

Practical Tip

An op-amp with 1 MHz GBW configured for gain of 10 will have a bandwidth of approximately 100 kHz. This trade-off between gain and bandwidth is fundamental.

2.2.2 Transistor Fundamentals

BJT as a Switch

For switching applications, operate the transistor in saturation (fully on) or cutoff (fully off):

- **Cutoff:** $V_{BE} < 0.6\,\mathrm{V}$ — transistor is off, $I_C \approx 0$

- **Saturation:** $I_B > \frac{I_C}{\beta_{forced}}$ — transistor is fully on, $V_{CE(sat)} \approx 0.1\,\text{V}–0.3\,\text{V}$

  The base resistor is chosen to ensure saturation with margin:

$$R_B = \frac{V_{drive} - V_{BE}}{I_B} = \frac{V_{drive} - V_{BE}}{\frac{I_C}{\beta/10}} \tag{2.2}$$

where the factor of 10 ensures hard saturation.

Emitter Follower (Common Collector)

The emitter follower provides:

- Voltage gain ≈ 1 (unity)
- High input impedance ($\approx \beta \times R_E$)
- Low output impedance ($\approx R_S/\beta$)

This makes it ideal as a buffer between high-impedance sources and low-impedance loads.

2.2.3 Filter Fundamentals

First-Order Filters

A first-order filter has a single pole and provides 20 dB/decade (or 6 dB/octave) rolloff.

Low-pass RC filter:

$$f_c = \frac{1}{2\pi RC} \tag{2.3}$$

High-pass RC filter: Same equation, but the capacitor and resistor positions are swapped.

Second-Order Active Filters

Active filters using op-amps can provide:

- Steeper rolloff (40 dB/decade for second-order)
- Controlled damping (Q factor)
- Gain in the passband
- High input impedance

The Sallen-Key topology is the most common for second-order filters.

2.3 Core Project: Analog Building-Block Toolkit Board

Project Description

Project Title: Modular Analog Toolkit
Objective: Create a collection of characterized analog building blocks that can be reconfigured for various applications. This board becomes your “analog Lego set” for future

projects.
Time Estimate: 8–12 hours (design, build, characterize)

2.3.1 System Requirements

Your toolkit board should include the following subsystems:

Table 2.1: Analog Toolkit Subsystem Requirements

Subsystem	**Specifications**	**Quantity**
General-Purpose Op-Amp Stage	Configurable as inverting, non-inverting, or buffer; selectable gain (1, 10, 100); bandwidth $> 100\,\mathrm{kHz}$ at unity gain	2
Active Low-Pass Filter	Second-order Sallen-Key; f_c selectable (1 kHz or 10 kHz); Butterworth response	1
Active High-Pass Filter	Second-order Sallen-Key; $f_c = 100\,\mathrm{Hz}$; Butterworth response	1
Transistor Switch	NPN, capable of 500 mA; LED indicator; flyback protection for inductive loads	1
Emitter Follower Buffer	Input impedance $> 100\,\mathrm{k\Omega}$; output capable of driving $50\,\Omega$	1

2.3.2 Subsystem 1: Configurable Op-Amp Stage

Design

Use a quality general-purpose op-amp such as the TL072 or OPA2134. The configuration should be selectable via jumpers.

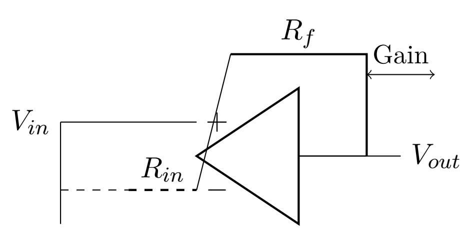

Figure 2.1: Configurable op-amp stage concept (simplified)

Gain settings should be implemented using precision resistor networks and DIP switches or jumpers:

- Unity gain (buffer): $R_f = 0$, $R_{in} = \infty$
- Gain of 10: $R_f = 10\,\mathrm{k\Omega}$, $R_{in} = 1\,\mathrm{k\Omega}$ (inverting)
- Gain of 100: $R_f = 100\,\mathrm{k\Omega}$, $R_{in} = 1\,\mathrm{k\Omega}$ (inverting)

Characterization Requirements

For each gain setting, measure and document:

1. DC gain (apply DC input, measure DC output)
2. -3 dB bandwidth (frequency at which gain drops by 3 dB)
3. Input offset voltage
4. Output swing limits (at the load you'll typically use)
5. Slew rate (using square wave input)

2.3.3 Subsystem 2: Second-Order Active Low-Pass Filter

Design

Implement a Sallen-Key low-pass filter with Butterworth response (Q = 0.707).

For a Sallen-Key low-pass filter with equal resistors ($R_1 = R_2 = R$) and a damping factor for Butterworth response:

$$f_c = \frac{1}{2\pi R\sqrt{C_1 C_2}} \tag{2.4}$$

For Butterworth response with unity gain: $C_2 = 2C_1$.

Design example for $f_c = 1\,\text{kHz}$:

Choose $R = 10\,\text{k}\Omega$, then:

$$C_1 = \frac{1}{2\pi \times 10^4 \times 10^3 \times \sqrt{2}} \approx 11.2\,\text{nF} \tag{2.5}$$

$$C_2 = 2C_1 \approx 22.4\,\text{nF} \tag{2.6}$$

Use standard values: $C_1 = 10\,\text{nF}$, $C_2 = 22\,\text{nF}$.

Characterization Requirements

1. Measure frequency response from 10 Hz to 100 kHz
2. Plot Bode diagram (magnitude and phase)
3. Verify -3 dB point matches design
4. Verify rolloff rate (40 dB/decade beyond f_c)
5. Check for any peaking near f_c (indicates Q too high)

2.3.4 Subsystem 3: Transistor Switch

Design

Use a general-purpose NPN transistor (2N3904 or 2N2222) to switch loads up to 500 mA.

Key design considerations:

- Calculate base resistor for hard saturation at maximum load

- Include flyback diode (1N4148) for inductive loads
- Add LED indicator to show switch state

Base resistor calculation:
For $I_C = 500\,\mathrm{mA}$, $\beta_{min} = 50$, and a forced β of 10 for hard saturation:

$$I_B = \frac{I_C}{10} = 50\,\mathrm{mA} \tag{2.7}$$

For a 5 V drive signal:

$$R_B = \frac{V_{drive} - V_{BE}}{I_B} = \frac{5 - 0.7}{0.05} = 86\,\Omega \tag{2.8}$$

Use $R_B = 82\,\Omega$ (standard value).

Characterization Requirements

1. Measure $V_{CE(sat)}$ at various collector currents
2. Measure switching speed (rise and fall times)
3. Verify proper operation with inductive load (observe flyback diode clamping)
4. Measure power dissipation and temperature rise at maximum current

2.3.5 Subsystem 4: Emitter Follower Buffer

Design

The emitter follower provides current gain while maintaining approximately unity voltage gain.
Key specifications:

- Input impedance $> 100\,\mathrm{k\Omega}$
- Output impedance $< 100\,\Omega$
- Capable of driving $50\,\Omega$ loads

For high input impedance, consider using a Darlington pair or a JFET input stage.

Characterization Requirements

1. Measure voltage gain at various frequencies
2. Measure input impedance using the voltage divider method
3. Measure output impedance by loading with known resistors
4. Observe distortion (crossover, clipping) at various output levels

2.3.6 System Integration

Once individual subsystems are characterized, connect them in series to verify proper operation:

1. Signal source → Buffer → Low-pass filter → Amplifier → Load
2. Verify that loading effects match predictions
3. Document the overall frequency response

Deliverables

Phase 1 Deliverables:

- Complete schematic of toolkit board with component values and AoE references
- SPICE simulation files for each subsystem
- Measured Bode plots for filter and amplifier stages
- Comparison table: "Simulated vs. Measured Performance"
- Photographs of completed board with labeled subsystems
- Written reflection: "What I predicted vs. what I observed, and lessons learned"

2.4 Troubleshooting Guide

2.4.1 Op-Amp Circuits

Table 2.2: Common Op-Amp Problems and Solutions

Symptom	**Likely Cause**	**Solution**
Output stuck at rail	Open feedback path or positive feedback	Check feedback resistor connections; verify inverting input
Oscillation at high frequency	Insufficient phase margin	Add compensation capacitor; reduce gain at high frequencies
Large DC offset at output	Input offset voltage amplified by gain	Use precision op-amp; add offset trim circuit
Distorted output	Output hitting rail; slew rate limiting	Reduce input amplitude; use faster op-amp
Reduced bandwidth	Op-amp not specified for high frequency	Check GBW; use appropriate op-amp for application

2.4.2 Transistor Circuits

Table 2.3: Common Transistor Problems and Solutions

Symptom	Likely Cause	Solution
Transistor always off	Insufficient base current; wrong polarity	Check base resistor value; verify NPN vs. PNP
Transistor always on	Base resistor too small; collector-emitter short	Increase base resistor; replace transistor
High V_{CE} when "on"	Not fully saturated	Decrease base resistor to increase I_B
Transistor overheating	Operating in linear region or over-current	Ensure hard saturation; verify load current
Voltage spikes at turnoff	Inductive load kickback	Add flyback diode across load

2.5 Phase 1 Checklist

Before proceeding to Phase 2, verify:

- ☐ All toolkit subsystems are built and functional
- ☐ Characterization data collected for each subsystem
- ☐ SPICE simulations completed and compared with measurements
- ☐ Bode plots generated for filter and amplifier stages
- ☐ Documentation complete with schematics and photographs
- ☐ AoE sections read and rules-of-thumb verified experimentally

Chapter 3

Precision & Low-Noise Instrumentation

Duration: 4–5 weeks **Difficulty:** Intermediate

AoE Reference

Primary Reading:
- Chapter 5: Precision Circuits (5.1–5.12)
- Chapter 8: Low-Noise Techniques (8.1–8.11)
- Chapter 4: Op-Amps IV (4.5–4.6, Instrumentation Amplifiers)

Supplementary from *The X Chapters*:
- Chapter 5x: Precision Techniques Expanded
- Chapter 8x: Noise in Depth

This phase transitions from "making circuits work" to "making circuits work precisely and quietly." You will learn to deal with microvolt-level signals, understand noise sources, and apply techniques that distinguish professional instrumentation from hobbyist projects.

3.1 Phase Overview and Goals

Precision instrumentation is where electronics becomes truly challenging—and rewarding. A circuit that functions perfectly at the millivolt level may fail catastrophically when you need microvolt resolution. Understanding and controlling noise, offset, drift, and grounding are essential skills.

Learning Objectives

By the end of this phase, you will be able to:

1. Design instrumentation amplifiers for differential measurements
2. Design transimpedance amplifiers for photodiode and other current-output sensors

3. Calculate noise budgets including thermal, shot, and 1/f noise contributions
4. Apply proper grounding, shielding, and layout techniques for low-noise circuits
5. Select precision references and understand their specifications
6. Implement signal conditioning chains with defined accuracy and resolution
7. Debug noise problems systematically using spectral analysis

3.2 Noise Fundamentals

3.2.1 Types of Noise

Understanding noise sources is essential for low-noise design:

Thermal Noise (Johnson-Nyquist Noise)

Every resistor generates noise due to thermal agitation of electrons:

$$e_n = \sqrt{4k_B T R \Delta f} \tag{3.1}$$

where $k_B = 1.38 \times 10^{-23}\,\mathrm{J/K}$, T is absolute temperature (K), R is resistance (Ω), and Δf is bandwidth (Hz).

At room temperature (300 K):

$$e_n \approx 4\,\mathrm{nV}/\sqrt{\mathrm{Hz}} \times \sqrt{R_{k\Omega}} \tag{3.2}$$

Example: A $10\,\mathrm{k\Omega}$ resistor has approximately $4\,\mathrm{nV}/\sqrt{\mathrm{Hz}} \times \sqrt{10} \approx 12.6\,\mathrm{nV}/\sqrt{\mathrm{Hz}}$.

Shot Noise

Current flowing through a junction exhibits shot noise:

$$i_n = \sqrt{2qI\Delta f} \tag{3.3}$$

where $q = 1.6 \times 10^{-19}\,\mathrm{C}$ is the electron charge and I is DC current.

1/f Noise (Flicker Noise)

Noise that increases at low frequencies, typically specified as noise density at some reference frequency (often 1 Hz or 10 Hz).

3.2.2 Noise Calculations

When combining multiple noise sources, they add in quadrature (RMS sum):

$$e_{total} = \sqrt{e_1^2 + e_2^2 + e_3^2 + \ldots} \tag{3.4}$$

Practical Tip

A source contributing less than 1/3 of the dominant noise source adds less than 5% to the total noise. Focus your efforts on the largest contributors.

3.3 Precision Amplifier Topologies

3.3.1 The Instrumentation Amplifier

The instrumentation amplifier (in-amp) provides:

- Very high input impedance on both inputs
- High common-mode rejection ratio (CMRR)
- Gain set by a single resistor
- Low drift and offset

The classic three-op-amp instrumentation amplifier uses a differential amplifier preceded by two non-inverting buffers with shared feedback.

Gain equation:

$$G = \left(1 + \frac{2R_1}{R_G}\right) \times \frac{R_3}{R_2} \tag{3.5}$$

For matched resistors and the standard configuration:

$$G = 1 + \frac{2R}{R_G} \tag{3.6}$$

3.3.2 The Transimpedance Amplifier (TIA)

For current-output sensors (photodiodes, PMTs), the transimpedance amplifier converts current to voltage:

$$V_{out} = -I_{in} \times R_f \tag{3.7}$$

Key design considerations:

- Feedback capacitor (C_f) for stability (sensor capacitance causes phase shift)
- Op-amp input bias current adds to sensor current
- Op-amp current noise appears at output multiplied by feedback resistance

The feedback capacitor should satisfy:

$$C_f \geq \sqrt{\frac{C_{in}}{2\pi R_f GBW}} \tag{3.8}$$

where C_{in} is the total input capacitance and GBW is the op-amp gain-bandwidth product.

3.4 Core Project: Sensor Front-End & Data Logger

Project Description

Project Title: Precision Sensor Acquisition System
Objective: Design and build a complete signal chain from sensor to digital output, applying precision techniques to achieve a defined accuracy and resolution.
Sensor Options:

- **Option A:** Photodiode light meter (transimpedance amplifier approach)
- **Option B:** Strain gauge / load cell (instrumentation amplifier approach)
- **Option C:** Thermocouple / RTD (precision low-level measurement)

Time Estimate: 15–20 hours

3.4.1 System Architecture

Regardless of sensor choice, your system should include:

1. **Sensor Interface:** Appropriate front-end for your sensor type
2. **Analog Signal Conditioning:** Amplification, filtering, level shifting as needed
3. **Voltage Reference:** Stable reference for ratiometric measurement or ADC reference
4. **Analog-to-Digital Conversion:** Resolution appropriate to measurement needs
5. **Data Logging:** Serial output or SD card storage

3.4.2 Option A: Photodiode Light Meter

Specifications

Table 3.1: Photodiode Light Meter Specifications

Parameter	Target Value
Measurement Range	1 nA to 1 mA (6 decades)
Resolution	12 bits within each range
Bandwidth	DC to 100 Hz
Noise Floor	$<$ 1 pA RMS in low-current range

Design Approach

1. **TIA Stage:** Use a low-bias-current op-amp (FET input) with switchable feedback resistors for range selection
2. **Gain Stage:** Additional amplification to fill ADC input range
3. **Anti-Alias Filter:** Low-pass filter with $f_c \approx 100\,\text{Hz}$
4. **ADC:** 16-bit sigma-delta ADC for best low-frequency resolution

Noise Budget Example:
For the lowest range (highest sensitivity), with $R_f = 10\,\text{M}\Omega$:

$$e_n(R_f) = 4\,\text{nV}/\sqrt{\text{Hz}} \times \sqrt{10000} = 400\,\text{nV}/\sqrt{\text{Hz}} \tag{3.9}$$

$$\text{In } 100\,\text{Hz BW:} \quad e_n = 400 \times \sqrt{100} = 4\,\mu\text{V} \tag{3.10}$$

$$\text{Referred to input:} \quad i_n = \frac{4\,\mu\text{V}}{10\,\text{M}\Omega} = 0.4\,\text{pA} \tag{3.11}$$

This meets the 1 pA noise floor target.

3.4.3 Option B: Strain Gauge / Load Cell

Specifications

Table 3.2: Load Cell Measurement Specifications

Parameter	**Target Value**
Full-Scale Input	10 mV (typical load cell output at full load)
Resolution	1 part in 10,000 (approximately 14 bits)
Accuracy	±0.1% of full scale after calibration
CMRR	>100 dB
Bandwidth	DC to 10 Hz

Design Approach

1. **Bridge Excitation:** Stable voltage reference for bridge excitation
2. **Instrumentation Amplifier:** High-CMRR in-amp (INA128, AD620, or similar)
3. **Gain Calculation:** For 10 mV input to 2.5 V output: $G = 250$
4. **ADC:** 16-bit or 24-bit ADC with differential input

3.4.4 Option C: Thermocouple / RTD

Design Approach

For thermocouples:

- Signal level: 40 µV/°C (Type K)
- Cold junction compensation required
- Typically use integrated solutions (MAX31855, AD849x) or precision front-ends

For RTDs:

- Resistance measurement with minimal self-heating
- 3-wire or 4-wire configuration for lead resistance cancellation
- Current source excitation with precision measurement

3.4.5 Noise Budget Template

Complete this table for your chosen sensor:

Table 3.3: Noise Budget Template

Noise Source	**Spectral Density**	**Equivalent BW**	**RMS Contribution**
Op-amp voltage noise	$\text{nV}/\sqrt{\text{Hz}}$	Hz	nV
Op-amp current noise	$\text{pA}/\sqrt{\text{Hz}}$	Hz	pA
Feedback resistor thermal	$\text{nV}/\sqrt{\text{Hz}}$	Hz	nV
Source resistance thermal	$\text{nV}/\sqrt{\text{Hz}}$	Hz	nV
ADC quantization	—	—	LSB
Total (RSS)	—	—	—

3.4.6 Grounding and Shielding Practices

Safety Warning

Ground loops are the most common source of problems in precision instrumentation. Follow these rules religiously:

- Use star grounding—all ground returns meet at a single point
- Keep signal ground and power ground separate until the star point
- Shield cables and connect the shield at one end only
- Use differential measurements whenever possible
- Route sensitive signals away from high-current or switching circuits

Deliverables

Phase 2 Deliverables:

- System block diagram showing all analog and digital subsystems
- Complete noise budget spreadsheet with calculated and measured values
- Schematic with detailed grounding and shielding notes
- Calibration data: measured vs. known stimulus at multiple points
- Noise floor measurement: output with input shorted/terminated
- Time-series data log demonstrating successful measurement
- Written analysis: "Where the noise came from and how I reduced it"

3.5 Phase 2 Checklist

Before proceeding to Phase 3, verify:

- ☐ Complete signal chain designed and simulated
- ☐ Noise budget calculated with all significant contributors
- ☐ Circuit built with proper grounding and shielding
- ☐ Noise floor measured and compared with budget
- ☐ System calibrated against known reference
- ☐ Data logging functional and producing meaningful output
- ☐ Documentation complete with noise analysis and calibration data

Chapter 4

Power Electronics & Protection

Duration: 4–5 weeks **Difficulty:** Intermediate to Advanced

AoE Reference

Primary Reading:

- Chapter 9: Voltage Regulation and Power Conversion (9.1–9.13)
- Chapter 1: Foundations (diodes, thermal behavior)
- Chapter 3: Field-Effect Transistors (power MOSFETs)

Supplementary from *The X Chapters*:

- Chapter 9x: Switching Regulators and DC-DC Converters

Power electronics is where theory meets thermal reality. This phase covers the design of power supplies that deliver clean, stable power under varying load conditions while surviving the abuse of the real world.

4.1 Phase Overview and Goals

Power supply design integrates nearly every concept from previous phases: feedback, thermal management, transient response, and protection. A good power supply is the foundation of every reliable electronic system.

Learning Objectives

By the end of this phase, you will be able to:

1. Design linear regulators with proper heat sinking and current limiting
2. Understand the operating principles of buck, boost, and buck-boost converters
3. Size inductors, capacitors, and switching devices for specified ripple and load
4. Implement protection circuits: overcurrent, overvoltage, reverse polarity
5. Analyze transient response and stability of power supply feedback loops

6. Measure efficiency, line regulation, load regulation, and transient response
7. Make informed trade-offs between linear and switching approaches

4.2 Linear vs. Switching Regulators

4.2.1 Linear Regulators

Advantages:

- Low output noise (no switching harmonics)
- Simple design, few components
- Fast transient response
- No EMI concerns

Disadvantages:

- Low efficiency: $\eta = \frac{V_{out}}{V_{in}}$ (always less than 1)
- Power dissipation: $P_{loss} = (V_{in} - V_{out}) \times I_{load}$
- Requires heat sinking at higher powers
- Cannot boost voltage (output must be less than input)

4.2.2 Switching Regulators

Advantages:

- High efficiency (80%–95% typical)
- Can boost, buck, or invert voltage
- Lower heat dissipation for a given power level

Disadvantages:

- Output ripple and switching noise
- EMI generation and susceptibility
- More complex design (inductor, control loop)
- Potentially slower transient response

4.2.3 When to Use Each

Table 4.1: Regulator Selection Guidelines

Consideration	Favor Linear	Favor Switching
Power level	$< 5\,\mathrm{W}$	$> 5\,\mathrm{W}$
Efficiency requirement	Not critical	Critical (battery, high power)
Noise sensitivity	Very high (audio, precision)	Moderate to low
Space constraints	Space available for heatsink	Need compact solution
Voltage ratio	$V_{in}/V_{out} < 2$	$V_{in}/V_{out} > 2$

4.3 Core Project: Multi-Rail Laboratory Power Supply

Project Description

Project Title: Bench Power Supply with Multiple Outputs
Objective: Design and build a professional-quality power supply with fixed and adjustable outputs, suitable for powering projects developed in this course.
Time Estimate: 20–30 hours

4.3.1 Specifications

Table 4.2: Multi-Rail Power Supply Specifications

Output	Specification	Type
+5V Rail	$5\,\mathrm{V}$ ±2%, $1\,\mathrm{A}$ maximum	Fixed (Linear)
+3.3V Rail	$3.3\,\mathrm{V}$ ±2%, $1\,\mathrm{A}$ maximum	Fixed (Linear)
Adjustable Rail	0–24 V, 0–3 A	Adjustable (Pre-reg + Linear)
Common Specifications		
Line Regulation	$< 0.1\%$ for 10% input change	
Load Regulation	$< 0.5\%$ from no-load to full load	
Ripple	$< 5\,\mathrm{mV}$ peak-to-peak	

4.3.2 Protection Features

- **Overcurrent Protection:** Foldback current limiting on adjustable rail
- **Overvoltage Protection:** Crowbar circuit on fixed rails
- **Reverse Polarity Protection:** Input diode or P-channel MOSFET protection
- **Thermal Shutdown:** Protection against excessive pass transistor temperature

4.3.3 Architecture Options

Option A: Pure Linear (Simpler, Lower Efficiency)

- Transformer: Multiple secondary windings or single winding with multiple taps
- Rectification: Bridge rectifiers with filter capacitors
- Regulation: Three-terminal regulators for fixed rails; LM317/LM350 or discrete pass transistor for adjustable rail

Option B: Pre-Regulated (Higher Efficiency)

- Input: DC input (wall adapter or AC-DC module)
- Pre-Regulation: Buck converter to track output voltage + dropout margin
- Final Regulation: Linear post-regulators for low noise

4.3.4 Adjustable Regulator Design

For the adjustable 0–24 V output, a discrete design provides better insight than integrated solutions:

Pass Transistor Selection

The pass device must handle:

- Maximum voltage: $V_{in(max)} - V_{out(min)} = 30 - 0 = 30\,\text{V}$
- Maximum current: 3 A
- Maximum power dissipation (worst case): $P = (V_{in} - V_{out}) \times I = 30 \times 3 = 90\,\text{W}$

⚠ Safety Warning

The 90 W worst-case dissipation is extreme. Practical designs either:
- Use pre-regulation to reduce the voltage across the pass device
- Implement foldback limiting to reduce current at low output voltages
- Accept that full current is only available at higher output voltages

Foldback Current Limiting

Foldback limiting reduces the short-circuit current below the maximum load current, protecting the pass transistor during fault conditions:

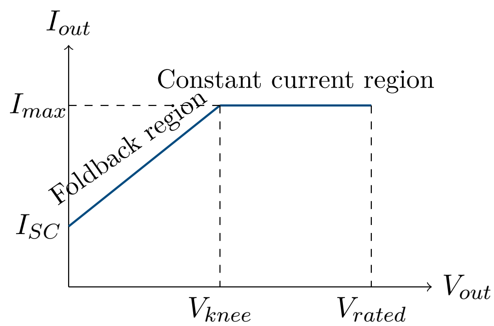

Figure 4.1: Foldback current limiting characteristic

4.3.5 Testing Procedures

Efficiency Measurement

$$\eta = \frac{P_{out}}{P_{in}} = \frac{V_{out} \times I_{out}}{V_{in} \times I_{in}} \times 100\% \tag{4.1}$$

Measure at multiple load levels and output voltage settings.

Transient Response Testing

1. Apply a step load change (e.g., 10% to 90% of rated current)
2. Capture output voltage on oscilloscope
3. Measure: voltage dip/spike magnitude, recovery time

Protection Testing

1. **Current Limit:** Gradually increase load until limiting engages
2. **Short Circuit:** Apply brief short and verify safe behavior
3. **Thermal:** Operate at high power and verify thermal shutdown (if implemented)

Deliverables

Phase 3 Deliverables:

- Complete schematic with component values and thermal calculations
- Bill of materials with cost breakdown
- Efficiency measurements at various load and output voltage conditions
- Line and load regulation measurements
- Ripple waveforms at each output under load
- Transient response oscilloscope captures
- Protection testing documentation with photographs
- Thermal measurements at maximum load conditions

4.4 Phase 3 Checklist

Before proceeding to Phase 4, verify:

- ☐ All specified output rails functional and within tolerance
- ☐ Protection features tested and documented
- ☐ Efficiency measured and optimized where possible
- ☐ Thermal design adequate for sustained operation
- ☐ Documentation complete with test data and photographs

Chapter 5

Digital Logic, Interfaces, and Mixed-Signal Integration

Duration: 4–5 weeks **Difficulty:** Intermediate

AoE Reference

Primary Reading:

- Chapter 10: Digital Logic (10.1–10.8)
- Chapter 11: Programmable Logic Devices (11.1–11.4)
- Chapter 12: Logic Interfacing (12.1–12.8)
- Chapter 13: Digital Meets Analog (13.1–13.9)

Supplementary from *The X Chapters*:

- Chapter 10x: Logic Pathology
- Chapter 13x: ADC and DAC Advanced Topics

This phase bridges the analog and digital worlds. You will learn how to interface microcontrollers with analog circuits, convert signals between domains, and manage the unique challenges of mixed-signal systems.

5.1 Phase Overview and Goals

Modern instruments combine analog signal conditioning with digital processing, control, and communication. Success requires understanding both domains and their interaction—particularly how digital noise couples into sensitive analog circuits.

Learning Objectives

By the end of this phase, you will be able to:

1. Interface logic families (CMOS, LVTTL, etc.) with appropriate level shifting
2. Design robust digital interfaces with proper termination and protection

3. Select and interface ADCs and DACs for specified resolution and speed
4. Partition mixed-signal PCB layouts to minimize noise coupling
5. Implement basic microcontroller interfaces for measurement and control
6. Debug timing, metastability, and signal integrity issues
7. Apply decoupling and filtering strategies for mixed-signal systems

5.2 Logic Families and Interfacing

5.2.1 Common Logic Families

Table 5.1: Logic Family Characteristics

Family	**Supply**	V_{OH} **(min)**	V_{OL} **(max)**	**Notes**
5V CMOS (HC)	5 V	4.4 V	0.5 V	Standard 5V logic
3.3V LVCMOS	3.3 V	2.9 V	0.4 V	Common for modern MCUs
LVTTL	3.3 V	2.4 V	0.4 V	TTL-compatible thresholds
1.8V LVCMOS	1.8 V	1.45 V	0.35 V	Low-power digital

5.2.2 Level Shifting

When interfacing between different logic levels:

- **5V to 3.3V:** Resistor divider (simple, slow) or dedicated level shifter (TXB0108, etc.)
- **3.3V to 5V:** Many 5V CMOS inputs accept 3.3V directly; otherwise use buffers or MOSFETs
- **Bidirectional:** Dedicated bidirectional level shifters or MOSFET-based circuits

5.3 ADC and DAC Fundamentals

5.3.1 ADC Selection Criteria

Table 5.2: ADC Architecture Comparison

Architecture	Speed	Resolution	Best For
Flash	$> 100\,\mathrm{MS/s}$	6–8 bits	Video, high-speed sampling
SAR	$1\,\mathrm{kS/s}$–$10\,\mathrm{MS/s}$	10–18 bits	General-purpose, MCU-integrated
Sigma-Delta	$10\,\mathrm{S/s}$–$1\,\mathrm{MS/s}$	16–24 bits	Precision, low-frequency
Pipelined	$10\,\mathrm{MS/s}$–$500\,\mathrm{MS/s}$	8–14 bits	High-speed with moderate resolution

5.3.2 Effective Number of Bits (ENOB)

Real ADC performance is limited by noise and distortion:

$$ENOB = \frac{SINAD - 1.76}{6.02} \tag{5.1}$$

where SINAD is signal-to-noise-and-distortion ratio in dB.

A 16-bit ADC may have an ENOB of only 12–14 bits in a practical system.

5.4 Core Project: Mixed-Signal Measurement & Control Unit

Project Description

Project Title: Digital Instrument Controller
Objective: Build a microcontroller-based system that interfaces with analog circuits from previous phases, demonstrating proper mixed-signal design practices.
Time Estimate: 15–20 hours

5.4.1 System Requirements

- Read at least one analog signal (from precision AFE or power supply feedback)
- Provide a user interface (buttons/encoder for input, display for output)
- Control at least one output (relay, MOSFET, or DAC-driven setpoint)
- Demonstrate clean partitioning of analog and digital domains

5.4.2 Microcontroller Selection

For this project, select a microcontroller platform based on your experience:

- **Beginner:** Arduino (ATmega328P, ESP32)
- **Intermediate:** STM32 (Nucleo boards), Teensy

- **Advanced:** Custom design with bare MCU or FPGA + soft-core

The focus is on the electronics interfacing, not firmware complexity.

5.4.3 Interface Design Requirements

Analog Input Interface

- Input protection (clamping diodes, series resistors)
- Anti-alias filtering ($f_c < f_s/2$)
- Voltage scaling to match ADC input range
- Isolation from digital noise (physical separation, filtering)

User Interface

- Debounced pushbuttons (hardware RC or software debouncing)
- Rotary encoder with proper pull-ups and filtering
- Display interface (I2C OLED, SPI LCD, or parallel character LCD)

Output Control

- Relay driver with flyback protection
- MOSFET switch with gate drive considerations
- Optoisolator for ground isolation if needed

5.4.4 PCB Layout Guidelines

Practical Tip

Mixed-signal layout rules:
1. Separate analog and digital ground planes; connect at a single point near the ADC
2. Route analog signals away from digital signals, especially clocks
3. Place decoupling capacitors as close as possible to IC power pins
4. Use ground plane(s) under all signal traces
5. Keep analog components on opposite side of board from digital
6. Use star power distribution from the main filter capacitors

Deliverables

Phase 4 Deliverables:

- Complete schematic showing analog, digital, and power partitioning
- PCB layout (if fabricated) or annotated breadboard photos
- Timing diagrams for key interfaces (hand-drawn acceptable)

- Noise measurement: analog signal with digital circuitry running vs. stopped
- Demonstration of complete system operation
- Written “EMI/noise lessons learned” document

5.5 Phase 4 Checklist

Before proceeding to Phase 5, verify:

- ☐ Microcontroller interface functional with analog front-end
- ☐ User interface (display, buttons) operating correctly
- ☐ Output control (relay/MOSFET) functioning as designed
- ☐ Noise coupling documented and minimized
- ☐ Documentation complete with timing analysis

Chapter 6

High-Speed, RF-Lite, and Signal Integrity

Duration: 4–6 weeks (Optional, Advanced) **Difficulty:** Advanced

AoE Reference

Primary Reading:
- Appendix H: Transmission Lines
- Chapter 7: Oscillators (7.1–7.3)
- Chapter 13: Digital Meets Analog (high-speed sections)

Supplementary from *The X Chapters*:
- Chapter 1x: Real-World Passive Components (parasitics at high frequency)
- Chapter 4x: High-Speed Amplifiers

This optional phase pushes into territory where wires become transmission lines, capacitors become inductors, and your intuition from low-frequency work may mislead you.

6.1 Phase Overview and Goals

Learning Objectives

By the end of this phase, you will be able to:

1. Identify when transmission line effects become significant
2. Design proper termination for various signal types
3. Recognize and diagnose reflections, ringing, and crosstalk
4. Select and apply high-speed op-amps correctly
5. Implement basic clock generation and distribution
6. Use time-domain reflectometry concepts for diagnosis

6.2 When Does High-Speed Matter?

The rule of thumb: treat a trace as a transmission line when:

$$l > \frac{\lambda}{10} = \frac{v_p}{10 \times f} \tag{6.1}$$

where l is trace length, λ is wavelength, v_p is propagation velocity ($\approx 0.5c$ for typical PCB), and f is the highest significant frequency.

For edge rate considerations, use the rise/fall time:

$$f_{knee} \approx \frac{0.35}{t_r} \tag{6.2}$$

Example: A signal with 1 ns rise time has frequency content up to approximately 350 MHz. At this frequency, any trace longer than about 4 cm is a transmission line.

6.3 Core Project: High-Speed Buffer / Line Driver

Project Description

Project Title: 50Ω Line Driver with Transmission Line Experiments
Objective: Design a fast buffer capable of driving a matched 50 Ω load through coaxial cable, and experimentally observe transmission line effects.
Time Estimate: 10–15 hours

6.3.1 Specifications

- Output: 50 Ω matched, capable of ±2 V into 50 Ω
- Bandwidth: DC to 50 MHz (minimum)
- Rise time: < 5 ns
- Square wave: Clean edges at 10 MHz

6.3.2 Experiments

Experiment A: Termination Effects

Using a length of coaxial cable (e.g., 3 m of RG-58):

1. Measure with proper 50 Ω termination
2. Measure with open circuit (no termination)
3. Measure with short circuit
4. Measure with 100 Ω and 25 Ω (mismatch cases)

Document the reflections, overshoot, and ringing in each case.

Experiment B: Cable Length Effects

With proper termination, compare:

1. 0.5 m cable
2. 3 m cable
3. 10 m cable

Observe propagation delay and any attenuation.

Deliverables

Phase 5 Deliverables:

- Schematic of high-speed buffer with layout considerations documented
- Oscilloscope captures showing reflections under various termination conditions
- Annotated waveforms correlating observations with transmission line theory
- Written summary translating AoE theory into observed phenomena

Chapter 7

Capstone: Integrated Instrument or System

Duration: 6–8+ weeks **Difficulty:** Advanced

The capstone project integrates knowledge and skills from all previous phases into a single, substantial piece of equipment worthy of a permanent place on your bench.

7.1 Phase Overview and Goals

The capstone is intentionally open-ended. You will define requirements, make design trade-offs, and execute a complete engineering project from concept to characterized prototype.

Learning Objectives

By the end of this phase, you will have:

1. Defined and documented a complete system specification
2. Made and justified architectural decisions
3. Integrated analog, digital, and power subsystems
4. Characterized performance against specifications
5. Identified limitations and potential improvements
6. Produced professional-quality documentation

7.2 Capstone Project Options

Choose one of the following, or propose your own with instructor approval:

7.2.1 Option A: Modular Data Acquisition System

A multi-channel measurement system with:

- 4+ analog input channels with configurable gain and filtering
- 16-bit or better resolution
- Software-configurable sample rate (1 S/s to 10 kS/s)
- Data logging to SD card or USB
- Display of real-time measurements
- Alarm thresholds with audible/visual indicators

7.2.2 Option B: Precision Bench Instrument

A high-accuracy voltmeter or current meter featuring:

- 5.5-digit resolution (minimum)
- Multiple ranges with auto-ranging capability
- True RMS AC measurement
- Data hold and min/max capture
- RS-232 or USB data output

7.2.3 Option C: Smart Power Platform

An advanced bench power supply with:

- Digital control of voltage and current limits
- Real-time monitoring of output power
- Sequencing capability for multi-rail applications
- Battery charging profiles
- USB interface for computer control
- Extensive protection with fault logging

7.3 Capstone Requirements

Regardless of project choice, you must deliver:

7.3.1 System Architecture Document

- Block diagram showing all subsystems and their interconnections
- Justification for key design choices with AoE references
- Trade-off analyses (e.g., linear vs. switching, resolution vs. speed)
- Risk assessment and mitigation strategies

7.3.2 Complete Schematics and Layout

- Professional-quality schematics with component values
- Clear labeling of functional blocks
- Layout showing analog/digital partitioning
- Grounding, shielding, and decoupling strategies annotated

7.3.3 Test and Characterization Plan

- Definition of key performance metrics
- Test procedures for each metric
- Required equipment and calibration references
- Pass/fail criteria based on specifications

7.3.4 Test Results and Analysis

- Complete test data in tabular and graphical form
- Comparison of measured vs. specified performance
- Analysis of any discrepancies
- Recommendations for improvement

7.3.5 Reflection Document

- What AoE concepts proved most valuable
- What only became clear through building and measuring
- Biggest challenges and how you overcame them
- What you would do differently in a second iteration

Deliverables

Capstone Deliverables Summary:

- System Architecture Document (5–10 pages)
- Complete Schematic Package (all sheets, BOM)
- Layout Files or Annotated Assembly Photographs
- Test and Characterization Report (10–20 pages)
- Working Prototype (demonstrated)
- Reflection Document (2–3 pages)
- 15-minute Presentation (optional, for formal courses)

Appendix A

How to Use *The Art of Electronics* Along the Way

This appendix provides structured guidance for integrating *The Art of Electronics* reading with hands-on project work.

A.1 Pre-Reading Strategy

Before each phase:

1. Skim the relevant AoE sections listed in the phase introduction
2. Write down 3–5 rules-of-thumb or "red flags" you'll watch for
3. Note any unfamiliar terms or concepts for deeper study
4. Review the figures and schematics—they often convey more than the text

A.2 Design Bookmarking

As you design your circuits:

1. Annotate schematics with AoE references:
   - "Compensation network inspired by AoE §4.9.2"
   - "Snubber per AoE's rectifier surge suppression guidance"
2. Record your predictions based on AoE rules-of-thumb
3. Note any deviations from AoE's recommended approaches and why

A.3 Post-Mortem Reading

After measuring your circuits:

1. Return to AoE and compare your issues with their troubleshooting advice

2. Use their oscilloscope screenshots as reference for diagnosing odd behavior
3. Update your notes with what you learned from the discrepancies
4. Consider: Would following AoE more closely have avoided the problem?

Appendix B

Equipment Recommendations

B.1 Budget-Conscious Setup

For students or hobbyists on a budget, these minimum specifications will suffice:

- **Oscilloscope:** Rigol DS1054Z or similar (4-channel, 50 MHz, upgradeable)
- **Function Generator:** FY6900 or similar DDS generator
- **Power Supply:** Basic dual-rail linear supply or repurposed ATX supply
- **Multimeter:** Any 4.5-digit handheld with true RMS
- **Soldering:** Temperature-controlled station (Hakko FX-888D or similar)

B.2 Professional Setup

For serious laboratories or teaching environments:

- **Oscilloscope:** 200 MHz+ bandwidth, 1 GS/s+, MSO capability
- **Function Generator:** 25 MHz+ arbitrary waveform generator
- **Power Supply:** Programmable with sequencing and data logging
- **Multimeter:** 6.5-digit bench multimeter with GPIB/USB
- **Additional:** Spectrum analyzer, LCR meter, curve tracer

Appendix C

Component Sources

C.1 Major Distributors

- **Digi-Key** (digikey.com): Comprehensive selection, excellent parametric search
- **Mouser** (mouser.com): Similar to Digi-Key, strong on passives
- **Newark/Farnell** (newark.com): Good for industrial and European parts
- **Arrow** (arrow.com): Competitive pricing on volume orders

C.2 Budget Options

- **LCSC** (lcsc.com): Low-cost components from China, good for basic parts
- **AliExpress**: Inexpensive component kits and modules (verify authenticity)
- **eBay**: Useful for vintage or hard-to-find parts (beware counterfeits)

C.3 Specialty Suppliers

- **Texas Instruments** (ti.com): Free samples for students
- **Analog Devices** (analog.com): Sample programs available
- **Mini-Circuits** (minicircuits.com): RF components and test equipment

Final Notes

This curriculum provides a framework, not a prescription. Adapt the projects to your interests, available equipment, and learning goals. The most important thing is to build, measure, and iterate.

The Art of Electronics is a dense text, and you will not absorb everything on first reading. That's intentional. Return to it throughout your electronics career, and you will find new insights each time.

The electronics laboratory is a place of continuous learning. Every circuit that doesn't work as expected is an opportunity to deepen your understanding. Embrace the debugging process—it's where real expertise is built.

Build well. Measure carefully. Learn continuously.