

Electronic Test Engineering and PCB Design Programme

FA-0799 · 20 days across five modules

Embedded Systems and Hardware

Introduction

Develop electronic test-engineering workflows through project management, requirements gathering, technical documentation, PCB design and mechanical modelling. Five modules combine guided engineering tasks with Word, Excel, Outlook, Altium Designer, OrCAD/Allegro and Autodesk Fusion.

The allocations total 20 days: project management and lean (2), test-engineering fundamentals (5), Altium PCB design (3), OrCAD/Allegro (5) and Fusion (5). Advanced board examples use prepared designs and selected exercises; the programme does not promise independent completion of multiple production-ready boards.

Learning outcomes

- Gather engineering requirements and prepare electrical/mechanical design specifications.
- Plan project work, communicate progress and document issues and handoffs.
- Prepare board design, fabrication and assembly documentation and analyse engineering data.
- Create and review schematics, footprints, placement, layer stacks, routing and constraints.
- Use guided MCU and more complex PCB examples to practise design and troubleshooting.
- Build mechanical models and assemblies, review selected simulation scenarios and prepare visualisations and shared documentation.

Prerequisites

For electronic test and interface-hardware engineers. Basic electronics and PCB concepts are useful; the advanced Altium track assumes familiarity with schematic capture and PCB layout. Use prepared, appropriately licensed versions of the selected tools and reference designs.

Training outline

1. Project management and lean – 2 days

Day 1

Introduction to project management

- What is project management?
- The project management process
- The different types of project management roles

Lean methodology

- What is lean methodology?
- The principles of lean methodology
- The tools and techniques of lean methodology
- Case study: applying lean methods to an electrical-engineering project

Day 2

Project planning

- How to create a project plan
- How to estimate project costs and resources
- How to manage project risks

Project execution

- How to communicate with project stakeholders
- How to resolve project conflicts
- How to measure project progress

Project closeout

- How to document project lessons learned
- How to hand off project deliverables

2. Test-engineering fundamentals – 5 days

Day 1: Understanding Engineering Duties and Responsibilities

Session 1: Understanding the Role of a Test Engineer

- Introduction to test engineering duties
- Gathering and understanding design requirements

Session 2: Creating Design Specs with MS Word

- How to use MS Word to create and format design specs
- Incorporating electrical and mechanical design guidelines into specifications

Day 2: Documentation for Design, Fabrication, and Assembly

Session 1: Documentation Fundamentals

- The importance of proper documentation in engineering processes
- Identifying key elements in engineering documentation

Session 2: Advanced Documentation Techniques with MS Word

- Using advanced MS Word features for effective documentation
- Hands-on: Developing a sample board design document

Day 3: Advanced Data Handling and Reporting with Excel

Session 1: Leveraging Excel in Test Engineering

- Using Excel for data analysis and reporting in test engineering
- Basic and advanced formulas, functions, and data visualization techniques

Session 2: Hands-On Excel for Test Engineers

- Real-world exercises on using Excel for test engineering tasks
- Creating custom Excel templates for regular engineering tasks

Day 4: Effective Customer Engagement and Communication

Session 1: Principles of Effective Customer Engagement

- Understanding customer needs and expectations
- Effective communication and feedback handling

Session 2: Outlook and supported email/calendar services

- Email and calendar workflows with Outlook and a supported Exchange or Microsoft 365 environment
- Advanced features for task management, scheduling, and team communication

Day 5: Putting It All Together: Integrated Project

- Putting it all together

- Assessment

3. Altium Designer PCB examples – 3 days

Day 1: Introduction and Simple MCU-based PCB Design

Session 1: Review of Altium Designer Basics

- Quick overview of Altium Designer features
- Review of Schematic Capture and PCB layout in Altium

Session 2: Designing a simple MCU-based PCB using a selected reference design

- Understanding the schematic and board layout of the MCU
- Creating the schematic in Altium
- Layout and routing of the PCB

Session 3: Practical

- Begin design of the MCU-based PCB
- Guidance on schematic creation and layout

Day 2: Complex PCB Design 1

Session 1: Introduction to Complex PCB Design

- Challenges in complex PCB design
- Techniques for managing complex designs in Altium

Session 2: Designing Complex PCB 1

- Introduce the first prepared complex PCB example
- Discuss unique challenges and approaches to the design
- Begin schematic creation and layout design in Altium

Session 3: Practical

- Continue design of the first complex PCB
- Guidance and troubleshooting

Day 3: Complex PCB Design 2 & 3

Session 1: Designing Complex PCB 2

- Introduce the second prepared complex PCB example
- Discuss unique challenges and approaches to the design
- Begin schematic creation and layout design in Altium

Session 2: Practical

- Continue design of the second complex PCB
- Guidance and troubleshooting

Session 3: Designing Complex PCB 3

- Introduce the third prepared complex PCB example
- Discuss unique challenges and approaches to the design
- Begin schematic creation and layout design in Altium

Session 4: Practical

- Continue design of the third complex PCB
- Guidance and troubleshooting

4. OrCAD/Allegro PCB design – 5 days

Day 1: Introduction to OrCAD/Allegro and IEEE Standard Symbol Creation

Session 1: Introduction to OrCAD/Allegro

- Overview of Cadence software
- Introduction to OrCAD/Allegro interface and features
- Navigation and tool utilization

Session 2: Electronic symbol libraries and applicable symbol conventions

- Understanding applicable schematic-symbol standards and conventions
- Creating and managing symbol libraries in OrCAD/Allegro

Day 2: Schematic Creation and Footprint Creation

Session 1: Introduction to Schematic Creation: OrCAD Capture

- Basics of schematic design in OrCAD Capture
- Building and editing schematics

Session 2: Footprint Creation of Electronic Components

- Understanding footprints in PCB design
- Creating and managing footprint libraries in OrCAD/Allegro

Day 3: Transition to PCB Design and Board Constraints

Session 1: Switching to Printed Circuit Board Design (PCB Design)

- Transitioning from schematics to PCB layout
- Overview of PCB design workflow in OrCAD/Allegro

Session 2: Board Shapes and Board Restrictions

- Defining board shapes in OrCAD/Allegro
- Setting and managing board restrictions

Day 4: Component Placement and Layer Stack Up

Session 1: Placements on PCB

- Principles of component placement
- Tools and techniques for component placement in OrCAD/Allegro

Session 2: PCB Layer Stack Up Setup and Calculations

- Understanding layer stack-up in PCB design
- Setting up and calculating layer stack-up in OrCAD/Allegro

Day 5: Routing, VIAs, and Practical Assignments

Session 1: Constraint Manager Settings and Introduction to Routing on PCB

- Setting constraints for routing
- Basics of routing in OrCAD/Allegro

Session 2: VIAs in Printed Circuit Board

- Understanding the role of VIAs in PCB design
- Creating and managing VIAs in OrCAD/Allegro

Session 3: 2 Layer Board Design Assignments

- Introduction to the assignment
- Guided practice and completion of a 2-layer board design

5. Autodesk Fusion design, simulation and documentation – 5 days

Day 1: Getting Started with Fusion 360

- Session 1: Introduction to Fusion 360
- Session 2: Understanding the Basic User Interface
- Session 3: Creating and Editing Sketches
- Session 4: Building 3D Models
- Session 5: Importing and Exporting Files

Day 2: Diving Deeper into Design and Assembly

- Session 1: Exploring Parametric Modeling
- Session 2: Applying features and constraints
- Session 3: Understanding Design Intent
- Session 4: Working with Sheet Metal Design
- Session 5: Assembling Components

Day 3: Simulating Real-world Conditions

- Session 1: Getting Started with Simulation
- Session 2: Performing Stress Analysis
- Session 3: Conducting Thermal Analysis
- Session 4: Fluid-flow analysis concepts and prepared demonstration with a compatible CFD tool; Fusion model preparation is not a built-in flow solver
- Session 5: Motion/joint studies and their limits as physical analysis

Day 4: Bringing Designs to Life with Rendering

- Session 1: Introduction to Rendering
- Session 2: Applying Materials and Textures
- Session 3: Managing Lighting and Shadows
- Session 4: Positioning the Camera
- Session 5: Creating Animations

Day 5: Collaborating and Documenting Designs

- Session 1: Collaborating in Fusion 360
- Session 2: Sharing Files
- Session 3: Working with Teams
- Session 4: Documenting Designs
- Session 5: Publishing Designs