Class Overview Palomar College
DT-226 Basic PCB Design Course

Module	Subject	Synopsis
Module 1:	**Course Introduction and Altium**	Introduction to the DT-226 course Walking through the course structure, policies, and responsibilities
Module 2:	**The Amazing World of PCB Design**	Launch of the Course with first looks at PCB Design. What are PCBs, and a brief history. Cover the PCB Design Process of the Library, Schematic, PCB Documentation. Create our first PCB Project, understand the Project and Schematic environment
Module 3:	**Components and your Library**	Understand the Importance and structure of the Component Library. The Five Pillars of your Library. What is required, and how to create components. Look in detail at the Manufacturing Part Search (Secondary Library)
Module 4:	**Finding and Component Placement**	Schematic Best practices and Grid Systems. How to search and place components into your schematic. Tools available to Clean-up the schematic (Aligning & Moving), Working with Generic Components.
Module 5:	**Schematic Connections**	Walk through all various types and methods to wire up the schematic of Net Labels, Power Ports, Buss, Off Page Ports, Buss' and including advanced connection methods of SIgnal Harness' Creating Circuit Connectivity Special Schematic Wiring Tricks. Including Annotation.
Module 6:	**Schematic ERC Verification**	Setup, overview, and run the ERC and Violation Categories and checks. Verify Circuit Continuity with the Connection Matrix. How to locate and clear Error messages. Running other Schematic Verification of Item Manager, Detailed look at ActiveBOM.
Module 7:	**Welcome to the PCB**	Push our design into the PCB Stage. Navigating the PCB. PCB Design Flow Module-A014 PCB Design Environment
Module 8:	**MIDTERMS**	Review the First half of the course Looking at the Library and Schematic Stages of the Design. Special attention to Material and Quizzes. Take the Course **Mid-Terms**
Module 9:	**PCB Structure Set up**	Required steps to begin the PCB Design of Mechanical Layer Assignments, Detailed look at the Layer Stackup Type and Factors you must consider when building your Stackup. PCB Materials, Via Stack Manager. The three Methods to create and adjust the PCB Shape,
Module 10:	**Design Rule Checks (DRC)**	Setting up and working with Classes and Rooms. Fundamentals, definition, and objectives. of the DRC Rule System. Look at the biggest mistakes made with DRCs. Running and identifying PCB Errors. Special PCB Design Rule Tools

	Component Placement	The rules for good Component Placement. Use the Mechanical layer Courtyard on the component to help placement—Special Alignment tools for repositioning components. FInally running, Finding, and clearing the Component placement DRCs.
	Routing of the PCB Design	We must first consider why we don't just AutoRoute our PCB. Examine the preliminary routing necessities, following the Golden Rules for Routing. A detailed look at the Routing Process Flows and Methods to Route the PCB, including a look at the Special Tools and tricks of ActiveRoute.
	Polygons, Pours & Planes	Regarding the Pros and Cons of Polygons, Pours, and Planes, look at the 8 Rules of PCB Grounding. Basics and concerns of EMC in the PCB. Set up and work with Split Planes and Polygons. Using the Polygon Pour Manager
	PCB Verification and Cleanup	Review and run the DRC and Violation Categories and checks. Run ALL other design verification tools for continuity, Design Consistency and Correctness. How to locate and clear Error messages. Final Preparations of the design to prepare for the Documentation Stage.
	Documentation	Creating the Fabrications and Assembly drawings. Including Views, Fab & Assy Notes, and Dimensions The setup and running using Output Job Files and the Project Releaser the Design Outputs. For the Fabrication and Assembly Data Packages. How to Publish and Release for Manufacturing.
	FINALS	A detailed review of the First half of the course Looking at the Library and Schematic Stages of the Design. Special attention to Material and Quizzes. Take the Course **FINAL**