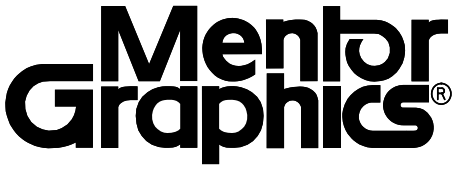
Board Station for New Users Training Series

For Mentor EN2002, Mentor EN2004



Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.

About This Training

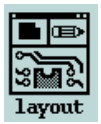
Welcome to the *Board Station for New Users Training Series*. This training will help you learn to use the following PCB tools:

Design Viewpoint Editor (DVE) is used to create and modify viewpoints for design evaluation by other applications.

Design Architect (Schematic and Symbol Editors) is used to create and modify schematics, component symbols, and electrical properties.

LIBRARIAN (PCB Graphics Editor) is used to create and modify PCB geometry data, catalog files, and mapping files.

PACKAGE (PCB Component Package Editor) is used to assign logical symbols to physical geometries.



LAYOUT (PCB Layout Editor) is used to place geometries and route traces on PCB designs.

FabLink (PCB Layout-to-Manufacturing Editor) is used to generate manufacturing data, drawings, and reports.

This training workbook contains conceptual information and lab exercises that use the PCB tools. Instructions for installing the necessary data for the lab exercises.

Introduction

Welcome to the Board Station for New Users Training Series. This training is designed to introduce you to the process of designing printed circuit boards using Mentor Graphics Board Station tools.

This module presents an overview of the training and a brief description of the process of circuit board design.

The lab material presents sequential tasks that demonstrate the conceptual topics. Some tasks include specific step-by-step instructions, while other more common tasks do not. Although most lab material consists of text, illustrations are included when discussing specific board design elements.

Course Prerequisites

In order to benefit sufficiently from the materials presented in this training series, you need experience in the art and technology of designing printed circuit boards. This training series does not cover any of the basic techniques and processes that are necessary to design printed circuit boards. This training series concentrates on the use of Design Architect and Board Station tools to improve the quality of your board designs.

Workshop Objectives

The *Board Station for New Users Training Series* is designed to introduce you to the basic processes and techniques of creating circuit boards with Board Station tools. This training is designed to give you a beginning experience and orientation to the Board Station design process.

Workshop Overview

The *Board Station for New Users Training Series* is not a course in printed circuit board design. Its intent is to develop your experience using Common User Interface elements and V8 Board Station tools and design processes. The knowledge you gain in this course increments your existing knowledge and experience in circuit board design.

To gain the most from this training, you should know how to perform the basic steps of designing a printed circuit board. You should have some knowledge of the basic circuit board technologies used at your company. You should also know the standard design rules that are required for your design and manufacturing processes.

The *Board Station for New Users Training Series* consists of seven training modules. If you are taking this training as a self-paced personal learning program, you can complete any module in any order because the information and lab training data is self-contained for every module. If you are taking this training in an instructor-led workshop, modules 1 through 4 are covered the first week, and modules 5 through 7 are covered in the second week of class. The training modules roughly follow the entire design process for designing and creating a PCB. The modules are:

- Module 1: Introduction to Board Station

  A brief overview of Board Station and design processes, the training materials, the user interface, the Design Manager, and the Bold Browser.

- Module 2: Preparing a Design for PCB

  An introduction to Design Architect with PCB personality module. You learn how to view and edit a design schematic, how to add symbols, wires, and design properties, and how to extract information about the design. You also learn how to create specialized design viewpoints for PCB and create back annotation objects.

- Module 3: Creating PCB Design Geometries

  This module covers parts libraries, creating and editing pin, via, component, board, and generic geometries, creating and editing catalogs, and mapping symbols to component geometries.

- Module 4: Packaging a Design for LAYOUT

  Packaging concepts and the input to the PACKAGE tool, control of flow of properties into PACKAGE, control how logic is packaged, interactive and automatic packaging, back annotation to the schematic, and PACKAGE output are covered in this module.

- Module 5: Placing Components on a Circuit Board

  In this module, you learn how to control and set up placement using properties. You learn how to both interactively and automatically place components on a board, and you learn how to evaluate and adjust placement.

- Module 6: Routing Traces on a Circuit Board

  This module covers design rules, interactive routing, grid-based automatic routing, routing with the Dynamic Editor, shape-based automatic routing, and area fills.

- Module 7: Creating Manufacturing Data

  Creating manufacturing data includes changing reference designators, creating an artwork order, creating and editing an aperture table, creating artwork and power fill areas for split power planes, creating thermal reliefs, simulating artwork data, creating test coupons, thieving patters, and panels, and creating drill tables, drill data, and milling paths. At the end of this module, you create both fabrication and assembly drawings. You also create aperture table and drill table reports, and a bill of materials.

http://www.mweda.com