

Board Station for New Users Training Series

For Mentor EN2002, Mentor EN2004



Copyright © 2001-2004 Mentor Graphics Corporation. All rights reserved.
Confidential. May be photocopied by licensed customers of
Mentor Graphics for internal business purposes only.

The software programs described in this document are confidential and proprietary products of Mentor Graphics Corporation (Mentor Graphics) or its licensors. No part of this document may be photocopied, reproduced or translated, or transferred, disclosed or otherwise provided to third parties, without the prior written consent of Mentor Graphics.

The document is for informational and instructional purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in the written contracts between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

RESTRICTED RIGHTS LEGEND Use, duplication, or disclosure by the Government is subject to restrictions as set forth in the subdivision (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013.

A complete list of trademark names appears in a separate "Trademark Information" document.

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

This is an unpublished work of Mentor Graphics Corporation.

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 patterns, 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>