

## Library Part Creation in the Xpedition Flow

Education Duration :4 Days

Education Language : English - Italian

Trainer : Pietro Vergine

Prerequisite : Experience in schema and PCB design, familiarity with Windows operating system, participation in "xPCB Layout Introduction" training and participation in "xDX Designer" training

Target Audience : Library managers for the xPCB Layout environment, schema and layout designers who will organize the library, design managers

Content :

Well-maintained and consistent libraries are the keys to efficient design work in xPCB Layout®. The Library Part Creation in the Xpedition Flow course will give you the skills necessary to create, protect, add to, and change the different data types in your Central Library. The lecture modules discuss the Central Library philosophy as well as how to use the xDM Library Tools™ utilities, and how to best interface library objects into your design process. Hands-on lab exercises reinforce the lecture topics as you create a small library of symbols, parts, and cells from scratch under the guidance of our industry expert instructors.

You Will Learn How To

- Start a new Central Library and copy elements between libraries
- Create partitions and set up partition search order schemes
- Control default text properties and create custom properties
- Create symbols for use in xDX Designer schematics
- Create padstacks for use in xPCB Layout and the Cell Editor
- Create cells to represent the part footprint and physical characteristics in xPCB Layout
- Integrate symbols and cells together to define parts (PDBs) for xPCB Layout
- Maintain xPCB Layout layout templates
- Configure xDX Databook™ and use it to support xDX Designer and xPCB Layout

Hands-on Labs

Throughout this course, extensive hands-on lab exercises provide you with practical experience using xDM Library Tools software. Hands-on lab topics include:

- Creating a Central Library
- Library Manager and Library Services
- Padstack, Basic Cell, and Basic Symbol Editors

- Advanced symbols
- Cell Editor graphic commands
- Creating Package Cells, Mechanical Cells and Drawing Cells
- PDB Editor basic usage
- Advanced PDB topics
- Expedition PCB Template creation
- xDX DataBook configuration
- xDX Designer to xPCB Layout data flow

## **Schematic Entry in the Xpedition® Flow**

Education Duration : 3 Days

Education Language : English - Italian

Trainer : Pietro Vergine

Prerequisite : -

Target Audience : Schema and layout designers who will use XDxD Designer

Content :

The Schematic Entry in the Xpedition Flow course will help you to improve your knowledge and skills with Design Definition solutions. Using the xDX Designer tools suite, you will gain proficiency in project management, schematic capture with xDX Designer, part selection using xDX DataBook™, and much more. You will also learn how to prepare your final schematic for interfacing with the Mentor Graphics xPCB Layout tool.

Detailed lab exercises help reinforce lecture topics and provide you with extensive tool usage experience under the guidance of our industry expert instructors.

You Will Learn How To

- Create flat and hierarchical schematic designs using xDX Designer proficiently
- Place wires and buses in the design to create net connections
- Search, select, and verify parts used in your design using xDX DataBook
- Create symbols for your schematics through a variety of methods, including the Symbol Wizard
- Effectively build intelligence into your design using properties
- Compile, package and export your schematic to xPCB Layout for place and route

## Hands-on Labs

Throughout this course, extensive hands-on lab exercises provide you with practical experience using xDX Designer and xDX DataBook software. You will also be introduced to the xPCB Layout and xDM Library tool. Hands-on lab topics include:

- Creating a project
- Configuring xDX Designer
- Managing projects
- Adding parts, nets, and buses to a schematic
- Creating schematics for flat and hierarchical designs
- Creating symbol graphics
- Adding properties to symbols and schematics
- Verifying the integrity of your design
- Cross-refereing using net labels
- Interfacing to the xPCB Layout tool
- Reusing design components
- Documenting a project
- Archiving a project

What are the idiosyncrasies of this course?

- There are at least three if not four versions of Schematic Entry using xDX Designer that are available for a student to take. It is very important that the student attend to correct class.
  - If the student is using VX release tools and the Xpedition Flow – Then this is the correct class.
  - If the student is using VX release software and PADS Layout in and integrated flow – then they should take the xDX Designer for PADS Layout Integrated Flow class.
  - If they are using older software (EE or PADS versions of software) or trying to use the software in a netlist flow environment, please contact a Instructor to help you guide the student to the correct class for the student particular needs.
- This class takes a student all the way from concept to exporting a design into xPCB Layout. (Should be a big selling feature!)
- If a shortened version of the class is required you should be able to work with an instructor to pick and choose among the topics to give the customer exactly what they need. Even though this is a flow class, all the labs have a starting point file, so labs can be skipped. All hierarchy information has been moved to the last day of the class. If the customer never using hierarchy in the design, it would make it easy to prune that topic from the current material.

## **Xpedition xPCB Layout Introduction**

Education Duration : 3 Days

Education Language : English - Italian

Trainer : Nigel Woolaway

Prerequisite : Experience in schema design, familiarity with Windows / Linux operating system and basic knowledge of PCB technology

Target Audience : PCB Design beginners, design managers or system managers who want to learn the process

Content :

From fundamental library concepts, to the PCB editor environment and the PCB layout process, you will gain hands-on experience in integrating a source schematic, placing and routing the board, and outputting the fabrication data.

You Will Learn How To

- Create Padstacks, cell (footprint) and part (PDB) definitions, and build intelligence into components
- Create a layout template
- Start a PCB layout by integrating with a schematic design
- Define board geometry
- Place parts, swap pins & gates and back annotate
- Verify the Layout with on-line and batch Design Rule Checking
- Utilize Constraint Manager to define layout and routing constraints
- Route, using both manual/interactive and automatic tools
- Add test points
- Generate plane data
- Finalize the silkscreen
- Generate Gerber data

Hands-on Labs

- Creating padstacks, cells, parts, and layout templates



- PCB editor overview
- Starting a PCB layout
- Defining board geometry
- Component explorer
- Creating planning groups
- Placing parts and planning groups
- Defining constraints in Constraint Manager
- Verifying the layout
- Net explorer
- Interactive routing
- Sketch and hug routing
- Tuning
- Adding test points
- Generating planes
- Finalizing the silkscreen
- Generating Gerber data
- Generating Drill data
- Dimensioning the design
- Creating drawings

## **Xpedition xPCB Layout Advanced**

Education Duration : 3 Days

Education Language : English - Italian

Trainer : Pietro Vergine

Prerequisite : Xpedition xPCB Layout Introduction "training, familiarity with Windows / Linux operating system and basic knowledge of PCB technology

Target Audience : Participants in "Xpedition xPCB Layout Introduction" training and who want advanced product training

Content :

Detailed lab exercises help reinforce what is discussed during the lectures, and provide you with extensive tool usage experience under the guidance of our industry expert instructors.



You will learn how to

- Use advanced placement commands to place, copy and move circuits
- Use Constraint Manager Net Class definitions to solve routing problems
- More efficiently use interactive routing commands including sketch router
- Efficiently work with via and pad definitions and changes
- Define and use Multi Via Objects (MVOs)
- Define MicroVias and route using blind and buried definitions
- Assign and route differential pairs and check for hazards
- Set maximum and minimum net length constraints and pin-to-pin delay formulas
- Define Parallelism rules and set Crosstalk constraints
- Use the Report Writer to document your designs
- Create and Autoroute bus paths, shielded buses, and match length buses

Hands-on Labs

Throughout this course, extensive hands-on lab exercises provide you with practical experience using xPCB Layout software. Hands-on lab topics include:

- Placement using array placement and radial move utilities, and Copy to Clipboard command
- Routing using Sketch Router in multiple ways
- Adding Hard and Soft fences
- Create breakout traces, teardrops and curved traces
- Using Constraint Manager Net Classes to solve routing problems
- Fanout and padstack processing
- Defining and routing Signal Topology
- Defining and routing Differential Pairs
- Defining and routing signal length constraints
- Defining and resolving crosstalk and parallelism
- Definitions and routing using MicroVias
- Defining report documentation using the Report Writer
- Using the Topology Planning and Routing
- Tools