PCB Design & Layout

For DIY Etching

A Project-Based Tutorial for

Printed Circuit Board Designs

Using Ultiboard & DIY Etching

By

A. B. Lawal

PCB Design & Layout for DIY Etching

ISBN: 9791220277075

Published in the United States

Ultiboard and *Multism* are a suite of circuit design programs owned by *NI* (https://www.ni.com*)*, formerly called *National Instruments* Corporation.

The author or the publisher of this book is in no way affiliated with NI (*National Instruments*) Corporation.

Table of Contents

About the Author

How this Book can Help You

How to Use the Project Files and DIY Demo Videos for Etching PCBs

1. Getting Started
    1.1. Installation Prerequisites
    1.2. How to Install the Full Version of Multism & Ultiboard Design Suite
2. Ultiboard Interface at a Glance
    2.1 Brief Introduction to Ultiboard 14 User Interface
    2.2 Quick and Helpful Ultiboard Shortcuts
3. How to Open a Design
    3.1. How to Transfer Existing Designs from Multism to Ultiboard
    3.2. How to Open a Sample Design in Ultiboard
        3.2.1. What are EWPRJ and EWNET Extensions?
4. How to Place Parts in the Workspace
    4.1. How to Use the Design Toolbox
    4.2 How to Use the Select Toolbar to Enable/Disable Parts
    4.3 How to Change the Orientation of Parts & Labels (Rotation)
    4.4. What is PCB Silkscreen?
    4.5. How to Move & Increase Diameters of THT pads
    4.6 How to View our Board in 3D
5. How to Trace/Route Our Board
    5.1 How to Trace Manually (Routing)
    5.2 How to Use Autoroute (Start/Resume Autorouter)
    5.3 How to Resize the Workspace and Spreadsheet View Window
    5.4. How to Check & Adjust the Width of Our Traces
    5.5. DRC Errors & How to Correct Them
    5.6. How to Move a Trace Segment
6. How to Place PCB Mounting Holes

7. How to Place a Power Plane on PCB design
    7.1. Uses of Power Planes
8. How to Place Graphics on PCB design
    8.1. How to Change the Color of Graphics on PCB design
    8.2. How to Use “Copper Delete » Copper Island” to Remove Copper from PCB
    8.3. How to Use “Keep-in/Keep-out area” to Remove Copper from PCB
9. How to Place Text on PCB Design
10. How to Print out Your PCB Layout
11. How to Manually Etch Your Printed Circuit Board
    11.1. A Special Note About the DIY Etching Process
    11.2. Materials You Need
    11.3 Another DIY Etching Technique
12. How to Get Further Help
13. More Helpful Resources

About the Author

I am a Control Systems Engineer, Systems Integrator and a Content Creator. I have worked with over a thousand clients across business sectors, mostly the PCB manufacturing and PLC automation industry. I have written numerous books, articles, and leadership classes for higher education institutions.

I have over 15 years of experience in Control Systems Engineering and PCB manufacturing industry. I have had the opportunity to work within world class organizations such as Kraft Heinz, Procter & Gamble, and Post Holdings.

I believe in excellence and I'm highly driven by successful people. I am dedicated to seeing my clients succeed and achieve their goals. I love to create PLC programs and help manufacturing companies grow. I've successfully coached over a thousand business owners and leaders.

I'm proud to boast of extensive experience and a successful company which has been in business for over 15 years.

How this Book can Help You

In this tutorial you will learn step by step how to use *Ultiboard* to route and make a single-layer Printed Circuit Board layout that you can print out on paper. Finally, you will learn with demo videos a very inexpensive DIY method for transferring your layout to a Copper Clad board that you can **etch and solder manually**. No heat transfer is involved.

After reading and completing the simple demo projects in this book, you will learn many features of Ultiboard very fast and very effectively **without getting overwhelmed**. You will not need to export any files or send gerbers to a PCB manufacturer/fabricator.

We will be using the *National Instruments*' **Ultiboard** and **Multism** PCB Design suite, which I found to be the best among several others I have used. Any of the versions 12, 13 and 14 of this suite works perfectly well. There is a link in this book for you to download a **hassle-free trial version** of the suite that you can use for many days to learn and practice many projects of your own.

Merely having the Ultiboard user manual, or referring to its help contents, is far from sufficient in becoming a skillful PCB designer. Therefore, this book is extremely useful for building PCB design skills very fast.

First, it will give you a big head start if you have never designed a PCB layout before. Then it will teach you more advanced techniques you need to learn, design and build anything from simple to complex PCB layouts using mostly Ultiboard.

Finally, if you have questions or need further help, I urge you to use the support link I provided in the last Chapter of this book. I will get back to you very quickly.

How to Use the Project Files and DIY Demo Videos for Etching PCBs

You will find at the end of chapter 11 the link to download all the project files, printouts of my completed projects (in pdf format) and demo videos that demonstrates a very inexpensive do-it-yourself method for transferring PCB layouts to a Copper Clad board that you can **etch manually**.

For effective learning and to get good practice, I strongly advise you attempt the projects and challenges in this book on your own. Build your own layouts after you have learned how to do it in this book and watched the accompanying videos. Then later, you can cross check your layouts with mine. Also feel free to use or modify any of the designs you downloaded as you wish.

1. Getting Started

1.1. Installation Prerequisites

Before you begin to follow this project-based tutorial, and to be able to complete the accompanying exercises in this book, make sure you set up your computer correctly, and that you've downloaded the accompanying assets (files) and videos. So, ensure you install both Multism and Ultiboard as explained in this book. Only **one installation** is needed for both.

You are expected to have a general knowledge of your OS (operating system), such as Windows or Mac. In addition, you should know how to use a mouse, standard commands and menus of your OS, and how to open, save, and close files.

1.2. How to Install the Full Version of Multism & Ultiboard Design Suite

If you have not already installed this design suite, here's how to install it. A step by step video that shows how to download and install the full version (14.2) is already included in your download. I highly recommend you watch this video first. Alternatively, you can just read how to install it below.

After downloading the zipped folder containing your suite to your preferred location, right-click on it and select "Extract files". A new folder is created where all the installation files are saved. If you are asked to enter a **password** to continue your extraction, just enter **123**.

Once the extraction process is completed, open the new folder and locate an exe file named **Autorun** (1) as shown in Fig. 1.1. Double click on this file to begin the installation.

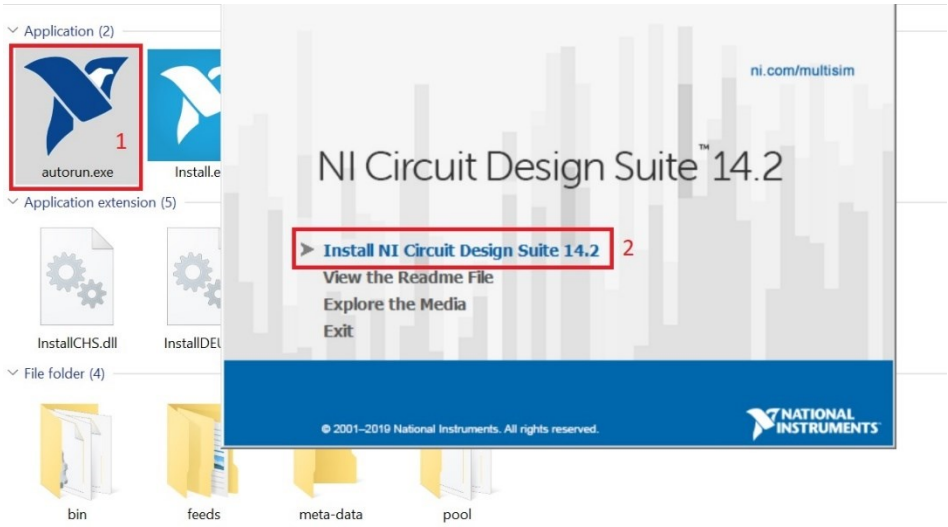

Fig. 1.1: How to install Multism and Ultiboard

Next, click on "**Install NI Circuit Design Suite 14.2**" (2) to continue the installation which will take a few minutes. When the installation is complete, please watch the video to learn how to activate your software correctly. If you made mistakes during your installation, you can uninstall it and reinstall again.

2. Ultiboard Interface at a Glance

2.1 Brief Introduction to Ultiboard 14 User Interface

Fig. 2.1A is a screenshot of the most essential panels of the Ultiboard 14.2 user interface you need. The ten most important and frequently used elements in the interface are numbered **1** to **10** as shown in Fig. 2.1.

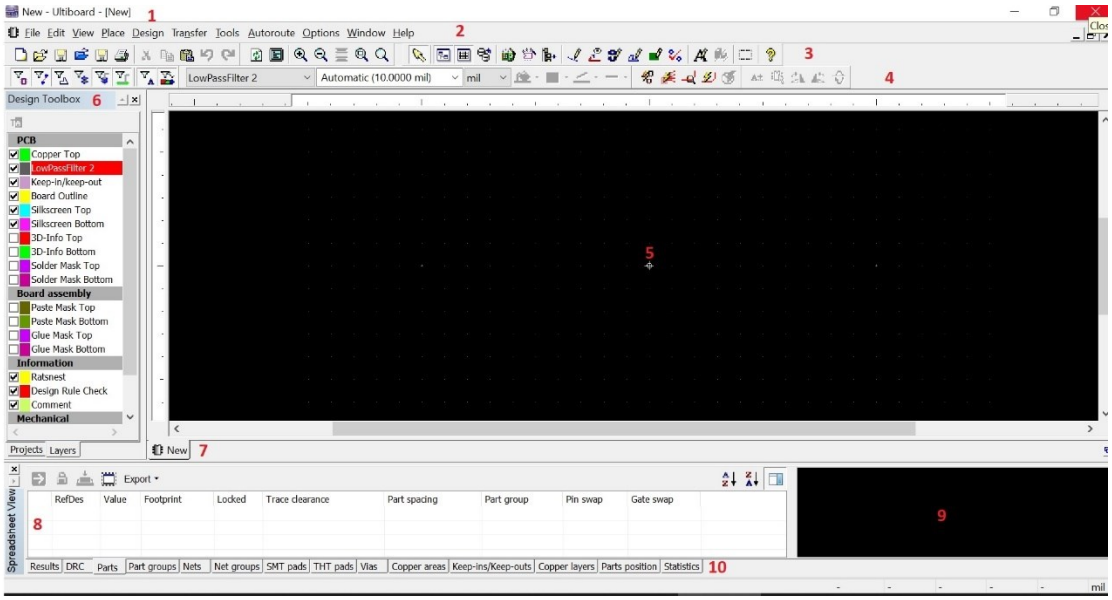

Fig. 2.1: Ultiboard 14.2 User Interface

Each element and its function(s) are explained briefly as follows:

1. **Project Title Bar**: The title of the project (or part of the project) that is currently open or that you are currently working on is displayed here.

2. **Standard Toolbar/Menu Bar**: This bar displays buttons used for commonly performed functions such as Undo, Redo, Save, Cut, Copy, Paste, Print, etc. This bar also has the *Autoroute* button that you can use to automatically place parts or route them.

3. **View Toolbar**: This bar contains the buttons used for modifying the way the workspace is displayed. It also displays the *folders* for opening your own projects and Ultiboard's *sample projects*. There is also an important *View 3D* button for displaying your PCB in 3D format. We also have the *Zoom in* and *Zoom out* buttons here.

4. **Select Toolbar**: Here you find buttons used to control selections, that is, for selecting only the parts or traces you want to use in your project. When you have many parts and traces on your workspace, such as in complex PCB designs, use this toolbar to select only those you want to use.

5. **Workspace**: This is the space where your PCB design project is laid out. There are two scrollbars below and on the right side of it which you can use to move your layout up/down or left/right. It can be resized easily as explained later in this book.

6. **Design Toolbox**: This is a panel you can use to select or navigate through your project files. You can also use it to show, hide (by unchecking the boxes) or dim different elements on your design. You can also use it to change the color of graphics on your layout.

7. **Project Tab Selection Toolbar**: This is used to select or open the project or part of a project you want to work on or display. You will find this toolbar very helpful if your project has to or more parts.

8. **Spreadsheet View**: This is used for fast advanced viewing and editing of important parameters or details of parts such as shapes, trace widths/heights, attributes, reference designators and design constraints (such as locking layers), and so on.

9. **Spreadsheet Preview Window**: Located on the right side of the **Spreadsheet View**, **t**his window is used to preview the work you do or the parts you select in the **Spreadsheet View** window.

10. **Spreadsheet View Tabs**: These tabs are a special part of the **Spreadsheet View** window and are located at the bottom of it for easy access. They are used to select or open an important parameter you want to view or work on within the **Spreadsheet View** window. Some of the important Tabs are Results, Nets, Copper Areas, DRC, THT pads and SMT pads. These tabs will be explained later under different sections of this book

2.2 Quick and Helpful Ultiboard Shortcuts

I highly recommend you use the following shortcuts to save time when working in Ultiboard:

1. Inside the workspace, you can use your **mouse scroll wheel** to quickly **zoom in** (scroll up) or **zoom out** (scroll down). You will find this very helpful if you need to select or see more/less detail of a part, trace, via (hole), etc. in your design.

2. Press **CTRL + Z** on your Keyboard one or more times to quickly **undo/reverse** your last action or more than one action.

3. Press **CTRL + S** on your Keyboard to **save your work** quickly.

4. Press **CTRL + N** on your keyboard to **start a new design**.

5. Press **CTRL + O** on your keyboard to **open an existing design** from your computer.