

# BoardLab: PCB as an Interface to EDA Software

**Pragun Goyal**

MIT Media Lab  
75 Amherst St,  
E14-548

Cambridge, MA 02139  
pragun@media.mit.edu

**Harshit Agrawal**

MIT Media Lab  
75 Amherst St,  
E14-445

Cambridge, MA 02139  
harshit@media.mit.edu

**Joseph A. Paradiso**

MIT Media Lab  
75 Amherst St,  
E14-548P

Cambridge, MA 02139  
joep@media.mit.edu

**Pattie Maes**

MIT Media Lab  
75 Amherst St,  
E14-548G

Cambridge, MA 02139  
pattie@media.mit.edu

## ABSTRACT

The tools used to work with Printed Circuit Boards (PCBs), for example soldering iron, multi-meter and oscilloscope involve working directly with the board and the board components. However, the Electronic Design Automation (EDA) software used to query a PCB's design data requires using a keyboard and a mouse. These different interfaces make it difficult to connect both kinds of operations in a workflow. Further, the measurements made by tools like a multi-meter have to be understood in the context of the schematics of the board manually. We propose a solution to reduce the cognitive load of this disconnect by introducing a handheld probe that allows for direct interactions with the PCB for just-in-time information on board schematics, component datasheets and source code. The probe also doubles up as a voltmeter and annotates the schematics of the board with voltage measurements.

## Author Keywords

Electronic design automation.

## ACM Classification Keywords

H.5.m. Information interfaces and presentation (e.g., HCI): Miscellaneous.



**Figure 1. BoardLab probe**

## INTRODUCTION

Use of Electronic Design Automation (EDA) software has become quite popular in the design of Printed Circuit Boards (PCBs). A very common paradigm is to design the

Permission to make digital or hard copies of part or all of this work for personal or classroom use is granted without fee provided that copies are not made or distributed for profit or commercial advantage and that copies bear this notice and the full citation on the first page. Copyrights for third-party components of this work must be honored. For all other uses, contact the Owner/Author. Copyright is held by the owner/author(s).

UIST'13-Adjunct, Oct 08-11 2013, St Andrews, United Kingdom

ACM 978-1-4503-2406-9/13/10.

<http://dx.doi.org/10.1145/2508468.2514936>

schematics first and then convert them to a PCB layout. The resulting PCB layout can then be fabricated and assembled with electronic components.

Access to the schematic design data of a PCB is quite helpful while testing and assembling a PCB as it gives the user a good idea of how the components are connected. This becomes even more important while debugging a PCB or while diagnosing a PCB for a fault.

While most modern EDA software offer an easy to use WYSIWYG interface, the keyboard and mouse interface required to query and annotate design data on EDA software creates a disconnect in the workflow of assembling or testing a PCB. This is because most operations and tools used to assemble or test a PCB involve working directly with the circuit board and its components. However, in order to locate a component in the schematic or in the PCB layout the user either has to try to find a visually similar pattern in the PCB layout file or key in the identifier for the component from the PCB silkscreen annotation. Both of these are cumbersome and lead to unnecessary cognitive load. Small component sizes and similar looking components further magnify this problem.

Experienced electronics designers work their way around this by annotating the board with design data, for example descriptors, part-numbers, component values, etc. However, the board space limits the amount of design information that can be embedded in the annotations.

The BoardLab system consists of a hand-held probe that allows for direct interaction with the board and board components for just in time information on board schematics, component datasheets and source code. It can also take voltage measurements, which are then annotated on the board schematics. The emphasis is on providing the same kind of interaction with the PCB as is afforded by other commonly used tools, for example soldering iron, multi-meter, oscilloscope, etc.

## RELATED WORK

Previous work to provide just-in-time information about electronic circuits involves the use of tagged components to track their positions in the circuit [1]. However, this technique requires specially constructed components.

Our work is similar in spirit to FreeD [2], where a position orientation tracked hand tool is used to provide for

interaction with the physical work piece and the design data at the same time.

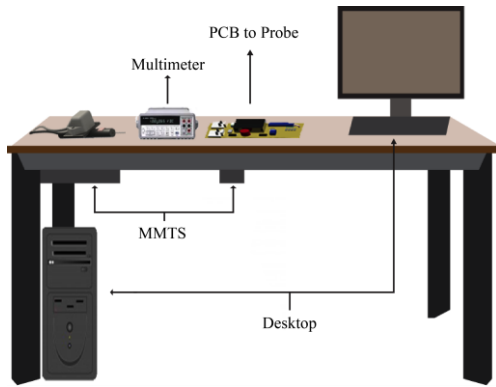


Figure 2: System overview

**SYSTEM OVERVIEW**

Our system has several components, including an external MMTS (magnetic motion tracker system) and an external multi-meter (Figure 2). A *Polhemus FASTRAK* system (an AC 6D magnetic tracking system) is used to estimate the 3D position and 3D orientation of the probe using a sensor mounted on the probe. A HP-34401A desktop multi-meter is used to provide voltage measurements to annotate the schematics. The probe houses a conducting metal tip, which is connected to the positive terminal of the multi-meter (Figure 1). The negative terminal of the multi-meter is separately connected to a convenient reference point such as the ground on the PCB. The multi-meter is programmatically accessed using the onboard RS232 serial port. The probe also has a small push-button which has to be clicked to make a selection or a measurement.

The software was written in python 2.7 and runs on a PC with 4GB of RAM and a quad-core Intel processor running on Ubuntu 12.04. The software parses the design of the PCB and the associated schematic from Eagle 6.0 .brd and .sch files respectively.

The board file is parsed to create a layout map of the all the features of the PCB. The position and orientation of the probe is used to compute the position of the tip. The position of the tip is then overlaid on the PCB map to identify the PCB feature being pointed to by the probe. Each component in the board and schematic files has a unique identifier; this identifier is used to identify, for each physical component on the PCB, the associated schematic symbol.

Three modes are supported by the system as described below. The modes can be cycled through by double clicking the button on the probe.

**a) Component Select Mode**

In the component select mode, the probe allows the user to select a single component. In this mode, the view highlights

the selected component in the schematic. The motivation is to help the user understand the component’s connections with other components. If the component has a datasheet attribute, the view also loads the datasheet for the component.

**b) Voltage annotation mode**

Over the course of testing a PCB, a user might make many measurements of voltage across different nodes on the circuit. Conventionally, the user has had to manually understand what a measurement made on the PCB means in the context of the schematic. The voltage annotation mode allows for automatic annotation of the schematic with the voltage measured using the probe (Figure 3).

**c) Code highlight mode**

Microcontrollers are quite commonly used in modern electronic circuits. Most microcontrollers are programmed in languages like assembly, C, C++, Java, etc. The code highlight mode allows the user to introspect the code related to a net or a trace on the PCB by highlighting the lines of code that refer to the pin of the microcontroller connected to the net. In our current implementation, this feature supports standard Arduino boards running Arduino sketches.

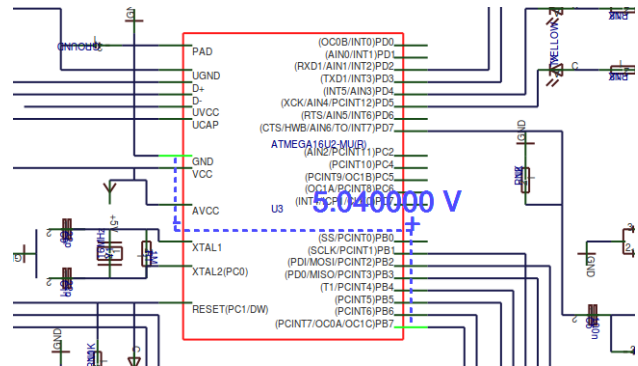


Figure 3. Schematic Voltage Annotation

**CONCLUSIONS AND FUTURE WORK**

In this work we’ve proposed a handheld probe based interface to query and annotate EDA design data for a PCB. In future work, we will investigate the possibilities of integration with circuit simulation tools like SPICE. Annotating the measured value against the expected (simulated) voltage might aid the user in testing. We also wish to study the possibilities of using multiple handheld probes. This will allow for measurements like impedance to be annotated on the schematics.

**REFERENCES**

1. Asgar, Z., Chan, J., Liu, C. & Blikstein, P. LightUp: a low-cost, multi-age toolkit for learning and prototyping electronics. *In Proc. IDC 2011* 225-226
2. Zoran, Amit & Joseph A. Paradiso. FreeD – A Freehand Digital Sculpting Tool. *In Proc. CHI 2013*: 2613-2616