CircuitStack: Supporting Rapid Prototyping and Evolution of Electronic Circuits

Chiuan Wang* **Hsuan-Ming Yeh*** **Bryan Wang*** **Te-Yen Wu*** **Hsin-Ruey Tsai***
Rong-Hao Liang* **Yi-Ping Hung†** **Mike Y. Chen†**
National Taiwan University
*{r03922001, b02902016, b02902096, r04922078, d01922006, rhliang}@ntu.edu.tw
†{hung, mikechen}@csie.ntu.edu.tw

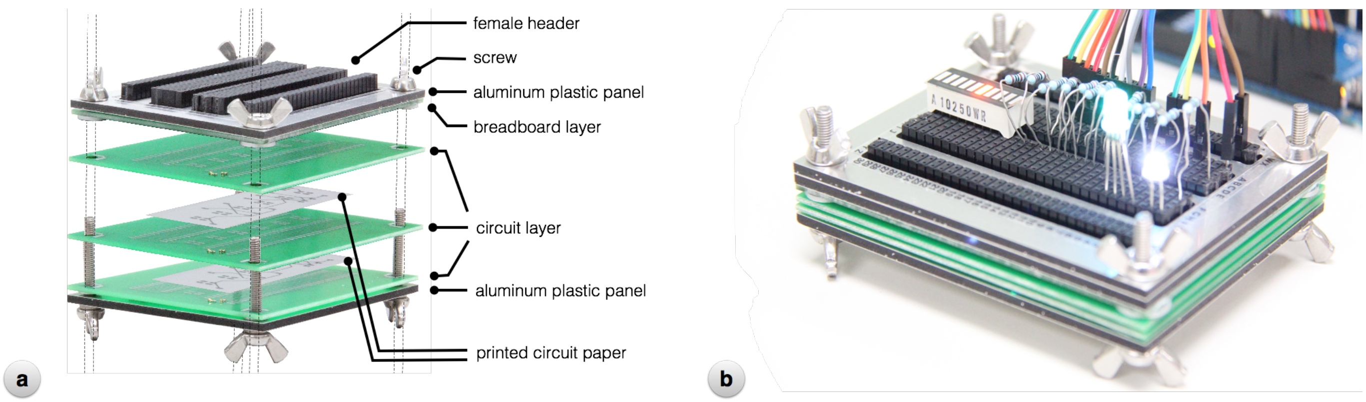

Figure 1. ***CircuitStack*** **is a hybrid system of breadboard and printed circuits which compose of customized PCBs in a stacked structure. (a) Overview. (b) Assembled state with placed components.**

ABSTRACT
For makers and developers, circuit prototyping is an integral part of building electronic projects. Currently, it is common to build circuits based on breadboard schematics that are available on various maker and DIY websites. Some breadboard schematics are used as is without modification, and some are modified and extended to fit specific needs. In such cases, diagrams and schematics merely serve as blueprints and visual instructions, but users still must physically wire the breadboard connections, which can be time-consuming and error-prone. We present CircuitStack, a system that combines the flexibility of breadboarding with the correctness of printed circuits, for enabling rapid and extensible circuit construction. This hybrid system enables circuit reconfigurability, component reusability, and high efficiency at the early stage of prototyping development.

Author Keywords
Circuit Prototyping; Hybrid System; Reconfigurability; Reusability; Breadboard; Printable Circuits; Conductive Ink.

UIST'16, October 16–19, 2016, Tokyo, Japan
Copyright © 2016 ACM. ISBN 978-1-4503-4189-9/16/10...$15.00
DOI: http://dx.doi.org/10.1145/2984511.2984527

ACM Classification Keywords
H.5.2. [Information Interfaces and Presentation]: Prototyping

INTRODUCTION
Circuit prototyping is an essential process in which makers and developers build, experiment on, and verify circuit designs for various types of electronic projects. Currently, circuit schematics, which indicate the wire connections between electrical components, and component datasheets are readily available on maker, DIY, and manufacturer websites. Recently, with the help of software tools, such as Fritzing[1], circuit designs in the form of breadboard schematics have been increasingly used because of their simplicity and ease of comprehension. Building a circuit on a plug-and-play breadboard with the aid of breadboard schematics has become simple and accessible to many more users.

Schematics and circuit diagrams are virtual files that serve as blueprints for the early stage of prototyping (e.g. breadboarding). Despite the convenience and easy guidance of breadboard schematics, they provide no further assistance to the process of physical circuit construction on a breadboard. Users typically must start from scratch and follow the schematics to plug component by component and wire by wire on breadboards. Increasing circuit complexity leads to tangled jumper wires that become fragile and error-prone.

[1]http://fritzing.org/

The ascending appeal and accessibility of digital fabrication systems have opened up a new field of unprecedented tools for expediting personal manufacturing by transforming virtual files to physical instances. Given 3D models and circuit schematics, 3D printers can print concrete 3D objects, and circuit printers can print functional circuits. For example, Instant Inkjet Circuit [8] proposed a technique allowing fast and low-cost circuit fabrication under laboratory conditions, and Argentum[2] generated printed circuit boards (PCBs) by using a wide variety of available materials. However, although off-the-shelf circuit printers ensure a rapid and correct outcome of conductive traces, manual component attachment is still required. In addition, design iterations require detachment from the conductive adhesives and re-printing of the circuits. Although the commercial product Squink[3] has shown the potential for automatically attaching components, fixated components on printed substrates are not reusable in the next design iteration.

This paper presents *CircuitStack* (Figure 1), a system that combines the flexibility of breadboarding with the speed and correctness of printed circuits. It uses a stacked design in which the top layer is a breadboard that provides a pluggable platform for component placement, supporting circuit reconfigurability as well as component reusability. The bottom layer sandwiches and interfaces with printed circuit paper (PCP) in the middle layers to provide the physical wiring to the breadboard layer.

Our key contribution is the design and implementation of the physical structure of CircuitStack composed of customized PCBs. First, we divide existing circuit prototyping tools into categories from which we explain the attributes and practices that concern the prototyping process. We then describe the design and composition of both the hardware and software of CircuitStack. We demonstrate how the system of CircuitStack can accomplish circuit reconfigurability, component reusability, and high efficiency with direct breadboard schematic virtual-to-physical conversion at the early stage of prototyping development; furthermore, the initial feedback gathered from a workshop revealed the usefulness of this tool. We conclude with application examples and discussions to emphasize several unique properties, features, and advantages of our system.

RELATED WORK

Because circuit prototyping has been an inevitable process in hardware development, the manufacture of new prototyping systems continues aiming to solve efficiency problems and lower the entry barriers and learning curves. Recently, Fritzing and 123D Circuits[4] have been two favored circuit prototyping software tools used in hobby electronics, DIY, and education because of their easy-to-read breadboard schematics and easy-to-use drag-and-drop user interface. Both Fritzing [9] and 123D Circuits present an easy virtual-to-physical transition from breadboard schematics to Arduino[5] PCB shields, for creating a robust and lasting version of a prototype. However, the conversion from virtual breadboard schematics to physical breadboard circuits is not well explored. Hence, We explore all hardware solutions and divide them into three categories: pluggable boards, circuit fabrication tools, and modularized electronics.

Pluggable Boards

A breadboard is one of the most frequently used tools for electronic circuit prototyping, because of its ease of circuit modification and compatibility with conventionally available dual in-line package (DIP) components. The pluggability of a breadboard enables circuit modification because the connections can be easily reconfigured by attaching or detaching the wires and components from a breadboard. Reconfigurability is the ability to repeatedly change and rearrange the components of a circuit cost-effectively, thereby suiting the needs of repetitive trial-and-error approaches. However, building a circuit on a breadboard requires manual wiring. Complicated circuits involving integrated circuit (IC) chips built on a breadboard often result in complicated wiring when jumpers are used. This markedly slows down the process of prototyping and increases the difficulty of circuit modification and the placement of new components. Visible Breadboard [14] is a pluggable and jumper-wire-free prototyping platform that entails using a capacitive touch layer for cable connections that saves an average of approximately 55% of time compared with that of the same circuit built on a breadboard. Nevertheless, this method is not compatible with most standardized components because of its dimensions.

Fabrication Tools for Circuit Prototyping

Circuit prototyping can also be achieved by crafting and sketching with conductive foil tape [15] or a conductive ink pen [10, 12]. To minimize wiring efforts, circuit traces can be created using digital computer-aided design (CAD) tools and inkjet printers. Instant Inkjet Circuits [8] demonstrated the use of inkjet printable silver on a paper substrate for creating printable circuits. Commercial products such as AgIC[6], Argentum and Squink already support this features, and Squink further supports automatic component attachment. Jung et al. [7] addressed the feasibility of inkjet printable circuits in industrial electronic production. Circuit Stickers [6] extended the use of [8] and presented a method for attaching components by using stickers to create reliable circuits by expanding contact areas. In addition, PaperPulse [16] enables designers without a technical background to produce layered electronic circuit designs by using inkjet printers. Furthermore, Printem [3] enabled users to easily fabricate high-conductive circuit prototyping in copper traces with a standard office printer. Such approaches require users to assemble components manually using soldering techniques, conductive silver adhesives, or Z-Tape[7]. Edwards proposed a method for manufacturing custom PCBs on a substrate with predefined circuits, which enable users to apply a customize printed circuit on it [5]. In order to support circuit modification, Circuit Eraser [13]

[2]http://cartesianco.com/
[3]http://www.botfactory.co/
[4]http://123d.circuits.io/
[5]http://www.arduino.cc/
[6]http://agic.cc/
[7]http://solutions.3m.com/

methods		component reusability	circuit reconfigurability: extensibility	circuit reconfigurability: modifiability	wiring effort saving	supporting package
non-modularized	breadboard	✓	✓	✓	-	DIP
non-modularized	circuit printing [3, 5, 6, 7, 8, 9, 17]	-	-	-	✓	DIP / SMD / customized
non-modularized	circuit drawing / erasing [10, 11, 12, 13, 14, 16]	-	△	△	-	DIP / SMD / customized
non-modularized	**CircuitStack**	**✓**	**✓**	**✓**	**✓**	**DIP**
modularized	modularized electronics [2, 4]	✓	△*	△*	✓	customized

✓ Full support △ Partial support * Depend on module design
DIP: Dual In-line Package SMD: Surface-Mount Device

Figure 2. A summary of prototyping methods and their supporting features.

enabled users to *delete* existing conductive patterns using a melamine sponge soaked in benzene. However, problems regarding component reusability and a complete circuit reconfigurability remain unresolved in this category, mainly because of the permanent adhesives and difficult-to-remove components.

Modularized Electronics

The major advantage of modularized electronics, the plug-and-play mechanism, provides developers with a convenient means of producing circuit prototypes. Some commercial modularized electronic products are designed for educational purposes. For instance, LittleBits [2] and LightUp [4] presented a magnetic-snapping mechanism for prototyping without the need for wiring or electrical expertise. Alternative modularized prototyping toolkits such as Arduino Grove[8] and Cubit[9] offer advanced users a wide variety of modules by adopting standardized socket connectors, and Inter-Integrated Circuit (I2C) and Serial Peripheral Interface (SPI) interfaces. Furthermore, SAM[10] introduces remote-control capabilities through a wireless connection. Nevertheless, circuit reconfigurability of these work is limited by the variety of available modules and their physical designs.

Summary

Figure 2 presents a summary of features supported by different tools and methods in circuit prototyping. CircuitStack provides component reusability and circuit reconfigurability, which other nonmodularized hardware techniques only partially support. In addition, CircuitStack supports conventional DIP electronic components, thus providing more freedom on circuit reconfiguration than modularized prototyping tools do. We explain how CircuitStack supports the features listed in Figure 2 in the following sections.

SYSTEM DESIGN AND IMPLEMENTATION

Hardware

Wire routes on printable circuits can be instantly created, and components are easily attached onto and detached from a breadboard. CircuitStack is a hybrid of a breadboard and printable circuits that involve using a stacked structure in which the breadboard layer inherits the exact same layout as a breadboard, and pieces of PCP are sandwiched on the bottom layers (Figure 1). Therein, components and wires are handled and placed separately on different layers. On the basis of previous results [8], we used a Brother DCP-J105 printer and materials from Mitsubishi Paper Mill to create printable circuits.

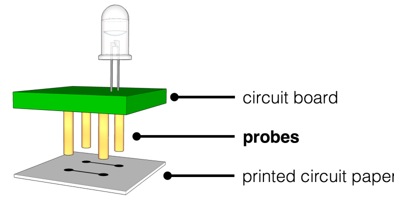

Figure 3. A conceptual illustration explains the technique of contact probes simultaneously reaching PCP conductive traces. The probes in our hardware system are actually contact springs for tolerating dimensional errors and ensuring connection reliability.

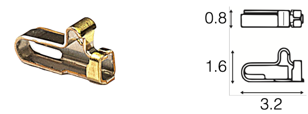

Figure 4. The contact spring (part number OG-320816) is used in our hardware system. The original height is 1.6 mm, and the maximum compression range is 0.5 mm. Units in the figure are in millimeters.

Key Component: Contact Spring

Conventionally, conductive adhesives such as glues and tapes are used to create connections with PCP conductive traces. However, to apply circuit reconfigurability and component reusability to prototyping, we replaced conductive adhesives with another contact method — *probing*. Because there are numerous distinct signals and traces on PCP, several probes must come into contact with multiple spots on the PCP. Figure 3 depicts four probes simultaneously in contact with four contact points on the PCP. Because PCP is a flat surface, the probes must be in equal height stretching from the bottom side of the circuit board to the PCP surface. Guaranteeing equal heights is often impossible, because of the dimensional errors caused during production and soldering. Thus, we applied *compressible probes*, the OG-320816 contact springs[11] (Figure 4) in our physical design, so that the dimensional errors can be tolerated. Another advantage of using compressible probes is that they ensure a reliable connection, because the springs push even harder against traces on PCP when compressed. With this nonadhesive, contact-based method, the connection pattern for circuit board can be simply modified by replacing with a new piece of PCP, and thus enabling circuit reconfigurability.

Signal Flow Overview

Because the main objective of the physical design of CircuitStack is to pass electrical signals to the PCP layers under the breadboard, the customized breadboard layer has a breadboard layout with additional contact springs on its bottom

[8] http://www.seeed.cc/grove/
[9] http://cubit.cc/
[10] http://samlabs.com/info
[11] http://kgs-ind.com/

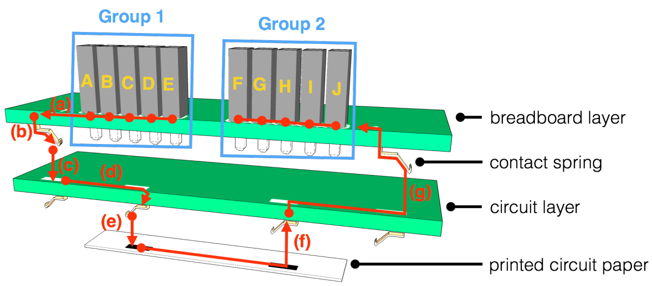

Figure 5. A simplified illustration of the CircuitStack mechanism shows the flow of electrical signals passing from (a) to (g) through routes and contact springs on PCBs, for establishing connections between the female headers of columns A to E and columns F to J.

side (Figure 1a). Figure 5 shows a simplified illustration containing only the first row of the breadboard layer and its relevant portion of the circuit layer. Because a breadboard layout is adopted, the signals on columns A to E (Group 1) and columns F to J (Group 2) are distinctly electrically shorted. For connecting the signals of Group 1 and Group 2, the signal flow is explained as follows:

1. The signal is passed through the route on the breadboard layer (Figure 5a) and thereafter to the contact spring (Figure 5b).
2. A contact point on the circuit layer then receives the signal (Figure 5c) and delivers it to another contact spring through a route on the same layer (Figure 5d).
3. Next, the contact spring sends the signal to the PCP (Figure 5e), and the conductive route on the PCP carries the signal to the next contact spring (Figure 5f).
4. Finally, the signal is passed to the Group 2 female headers on the breadboard layer through the routes and contact springs (Figure 5g).

This mechanism enables the PCP to determine the connectivity between Group 1 and Group 2 according to its layout.

Breadboard Layer of CircuitStack

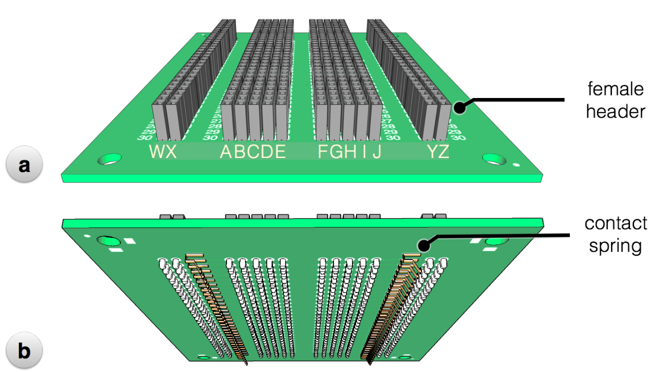

Figure 6. Breadboard Layer. (a) The top side is composed of 14×30 female headers in which the columns are denoted as A to J and W to Z. (b) The bottom side is composed of two columns of contact springs, each with 32 rows, to deliver electrical signals across PCBs.

The breadboard layer (Figure 6) consists of the repeated row-wise pattern in Figure 5 with four additional columns, W, X, Y, and Z (Figure 6a). The top side of the breadboard layer consists of 14×30 through-hole female headers, exhibiting the appearance, the pluggable mechanism, and the layout of a breadboard (Figure 6a). The bottom side of the breadboard layer consists of two columns of contact springs (Figure 6b), each of which is electrically shorted with its closest row of female headers on the top side through a wire route on the PCB. Because there are 30 rows × 2 inner columns (A-to-E and F-to-J) + 4 outer columns (W-to-Z) = 64 distinct electrical signals on a breadboard layout, 64 contact springs are placed on the bottom side (Figure 6b).

Circuit Layer of CircuitStack

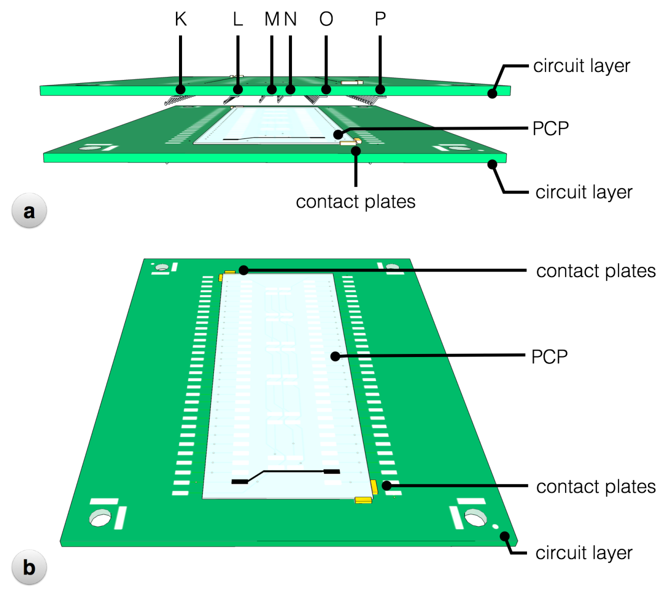

Figure 7. Circuit Layer. (a) The circuit layer consists of 6 columns (K to P) of contact springs on the bottom side. PCP is placed on the center of circuit layer. (b) The top side of circuit layer includes contact plates serving as physical constraints to aid PCP alignment. The translucency of PCP and marks on the top side help alignment as well.

The circuit layer consists of multiple layers of PCBs. Each of the PCBs is in exactly the same design. For example, Figure 7a shows a sandwiched structure in which a piece of PCP is placed between two PCBs. Figure 7b shows the top side of the circuit layer. Contact spring columns K, L, O, and P in Figure 7a are repeated row-wise pattern of that on the circuit layer in Figure 5. Whereas columns K and P are responsible for passing signals downward to a circuit layer, columns L and O come into contact with PCP.

Signals on columns W to Z on a breadboard such as that shown in Figure 6a are usually the power source signal and ground signal; thus, making interconnections between columns A to J (Figure 6a) and columns W to Z is a common practice. Hence, a few additional springs are sparsely added to columns M and N (Figure 7a) used as shortcuts for PCP routing to connect signals to column W to Z when assembled. Without columns M and N, long routes on PCP for power and ground signals may block other routes causing

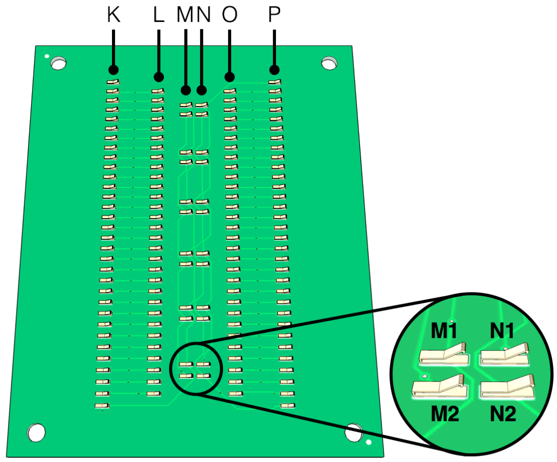

Figure 8. The bottom side of the circuit layer consists of six columns of spring contacts whose labels match that in Figure 7a. M1, M2, N1, and N2 are electrically shorted with columns W, X, Y, and Z (Figure 6a), respectively, when assembled. The pattern of M1, M2, N1, and N2 is repeated across columns M and N in this figure.

unsuccessful routing on a CAD software. Figure 8 shows a complete illustration of the bottom side of the PCB. The contact springs on the bottom side of the PCB enable the signals to pass downward to another piece of PCP placed underneath it. The deployment of contact springs further enables the possibilities of multi-layer circuit design, as shown in Figure 1a.

Aluminum Plastic Panel

After the layers are stacked, tightening the screws on the corners to force the contact springs to push against the sandwiched PCP is the final step of assembly. However, because the PCBs are composed of flexible plastics, the assembly method results in uneven force distribution on the outermost PCBs causing the outermost PCBs to flex outwards by approximately 1 mm. Such a phenomenon might not only prevent contact springs in the center of PCB from coming into contact with PCP but also cause the springs on the corners of the PCB to be destroyed by the excessive forces. To mitigate this phenomenon, we successfully used hard plastic backboards, composed of aluminum plastic panels (Figure 1), to evenly distribute the force across the PCBs and minimize the flex. Alternative choices include using a wider compression range of compressible probes such as pogo pins[12].

Software

In novel fabrication systems, digital files are transformed into physical instances. Our software is a two-step circuitry generation process: 1) extracting endpoint positions of wires in the Fritzing file to generate an unrouted CircuitStack EAGLE[13] file, and 2) autorouting in EAGLE to complete the final circuitry. (We explain in detail in the following paragraphs.) However, the entire process from breadboard schematics to PCP is simple and generally automatic for users. Clicking two buttons in Figure 9a and Figure 9c enables users to view the following details as a black box.

[12]http://www.ysdzsz.com/
[13]http://www.cadsoftusa.com/

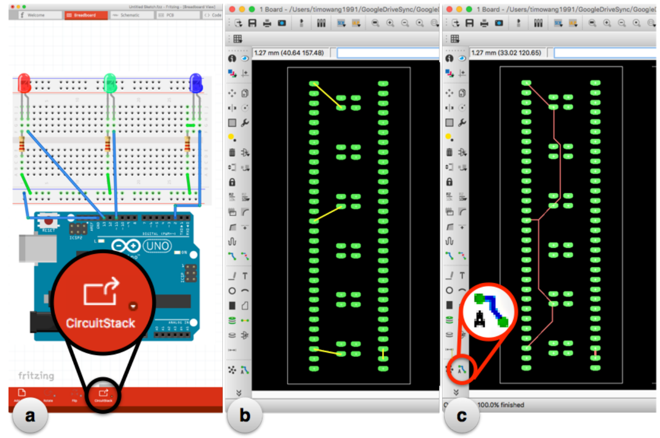

Figure 9. A breadboard schematic shown in (a) is displayed by the open source software Fritzing. A *CircuitStack* button is added to export the current breadboard schematic to an EAGLE file shown in (b) whose layout matches columns L to O in Figure 8. Clicking the enlarged bottom left button, *Autorouter*, instantly completes the wire routing in (c).

Wire endpoint extraction is the first step of circuitry generation. Fritzing project files are in the extensible markup language (XML) format which contain both the locations of elements and wires. We extracted only the wire locations whose endpoints were both plugged into the Fritzing breadboard (e.g. green wires in Figure 9a) from the XML file. Information on the blue wires shown in Figure 9a is discarded because the wires are external connections to other devices such as Arduino. The extracted wire information is then exported to an EAGLE file that creates the PCP design shown in Figure 7b by clicking the *CircuitStack* button in Figure 9a.

The routing algorithm of EAGLE software is used to create conductive traces on the 31.78 mm $\times$ 81.66 mm PCP, using 10-mil (0.254 mm) trace width and 12-mil (0.305-mm) trace-to-trace clearance. The shortest path and longest possible detour path connecting M1 to N1 (Figure 8) on PCP are 3.81 mm and 191.20 mm, respectively. The exported EAGLE XML file from Fritzing contains only single-pin elements as green ellipses for contact springs and unrouted connections as yellow lines (Figure 9b). The yellow lines are resulted from the extraction of green wire positions (Figure 9a) in Fritzing. Finally, clicking the *Autorouter* button completes and prints the PCP routing (Figure 9c). Note that the traces do not cross over with irrelevant spring contacts; thus, the viability of the printed circuit is warranted.

SYSTEM FLOW AND WALKTHROUGH

This section describes some properties and the entire procedure of using CircuitStack by demonstrating the process of building a countdown timer project consisting of a 4-digit seven segment LED display (common anode), a shift register (74HC595), a real-time clock (RTC) IC chip (DS1307), a push button, and an Arduino Mega. We also elaborate on some probable scenarios encountered in the prototyping process.

Direct Use of Internet Resources: Typically, the first step for makers and hardware developers using CircuitStack is to surf

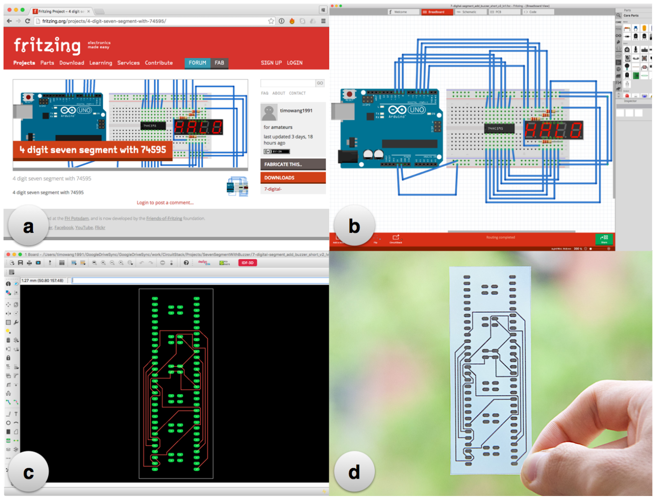

Figure 10. Converting a breadboard schematic into PCP. (a) Download a breadboard schematic from the Fritzing Project website. (b) Open the breadboard schematic with Fritzing software. (c) An EAGLE PCB file is created by first clicking on the *CircuitStack* button in Fritzing and (d) then on the *Autorouter* button in EAGLE to create the PCP result.

the web for schematics and datasheets. A .fzz file can be downloaded from the Fritzing Project webpage[14] and be directly converted into PCP (Figure 10) for immediate examination without completely starting from scratch. A step-by-step assembly process is illustrated in Figure 11.

Revisions to Internet Resources: An existing breadboard schematic may not fit the needs of a project. Hence, a user revises the schematic by adding a few components and wires such as an RTC chip and a push button (Figure 12a). Finally, the user removes the old PCP and slides in new PCP (Figure 12b). The procedure of circuit revision enables using components not only with reusability but also with fixed location, thereby facilitating circuit modification without the need for component reattachments.

Hybrid Use of Internet Resources: A user may be uncertain about some wire connections of a component, such as the RTC chip, and prefers to swap wires around the component. Therefore, plugging jumper wires on the breadboard is also an option for testing a circuit (Figure 13a). Thus, CircuitStack presents the hybrid usage of both a conventional pluggable mechanism and a novel printing approach.

Instant Component Replacement and Removal: Occasionally, components may malfunction and are replaced with a new one. For example, a 4-digit seven segment LED display may malfunction or the user need to change it from a common anode to a common cathode. For another instance, a push button can also be replaced by another button with a different part number (Figure 13b). With the detachability of CircuitStack, replacing or removing a component and circuit reconfiguration is instantly achieved.

The hybrid system of CircuitStack combines the benefits of breadboarding and printed circuit techniques. The rapidity of printable circuits assists the user to leap forward in the process of physical circuit construction. The pluggable mechanism of female headers imitating breadboard practices enables flexible circuit modification.

[14]http://fritzing.org/projects/

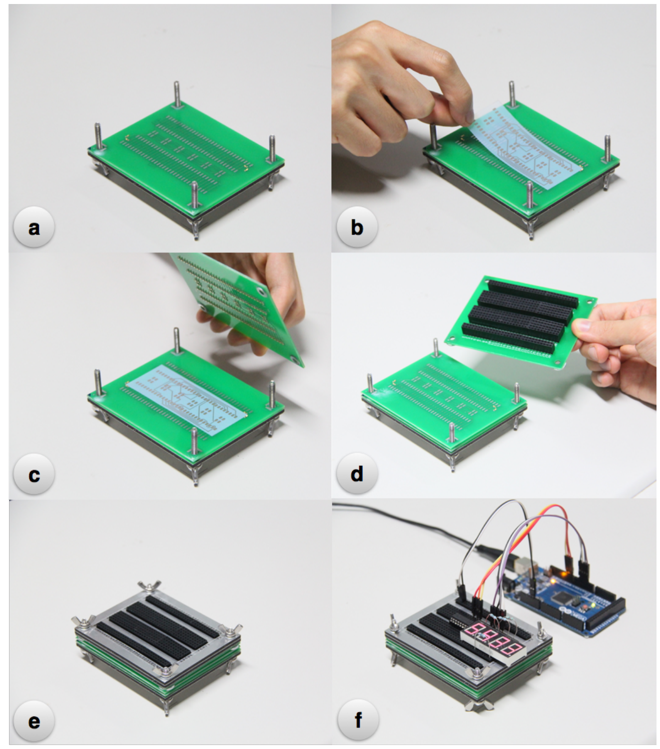

Figure 11. Step-by-step assembly process of using CircuitStack. (a) The assembly process begins with a circuit layer with four screws on the edges. (b) A piece of PCP is placed on the circuit layer. (c) Another circuit layer is placed on top of the PCP. (d) The breadboard layer is stacked upward. (e) An aluminum plastic panel support is placed on the top, and four nuts fastened the entire stack structure. (f) Electronic components and wires from Arduino are plugged and functioning.

Workshop

To validate the process proposed in Walkthrough, we conducted a preliminary workshop to understand the breadboard prototyping experiences with and without using CircuitStack. Sixteen participants (mean age = 22.9; eleven males and five females) were recruited. The three tasks they performed were 1) building a circuit upon a breadboard schematic, 2) minor revision by adding a push button, and 3) major revision by adding a timer IC, sequentially. All of them were provided with required materials and taught how to use our software and hardware tools.

From the observations and feedback, we derived three findings: 1) 11 out of 16 indicated that they encountered difficulty with muddled wires on a breadboard without CircuitStack, whereas 8 out of 16 reported that prototyping on CircuitStack with markedly fewer wires was a relief. 2) Nine participants suggested using improved color schemes of female headers for locating component placement more effectively. One of them proposed alternating every five rows in black and white. 3) A user mentioned that CircuitStack was

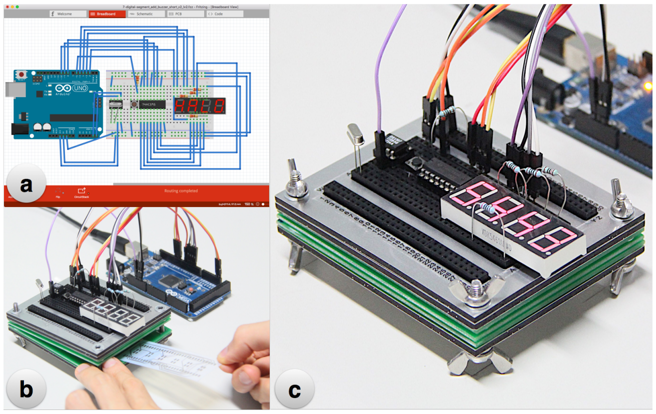

Figure 12. Revising the circuit. (a) Revising the breadboard schematic in Figure 10 by adding an RTC chip and a push button. (b) Loosening the four nuts and sliding new PCP into the circuit layer. (c) Results.

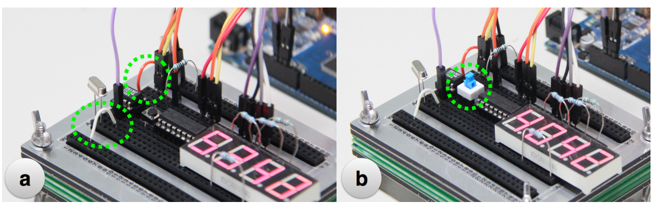

Figure 13. Using CircuitStack with manual wiring on the breadboard layer. (a) Swapping wires. (b) Replacing a push button.

particularly easy to use when working with digital ICs such as multiplexers, shift registers, and components with complicated pin layouts. Another common response from four users described that a larger-size CircuitStack would be expected in future versions because they often leave some space between components when working on a regular breadboard. These qualitative results will help improve our system in the future and provide some design guides for future developers.

DISCUSSION

Multi-layer PCP

When a circuit becomes complex, the routing algorithm fails to produce complete traces on a single-layer PCP because wire crossing is not allowed. Similar to conventional PCB manufacturing processes, multi-layer PCP design does provide a solution to wire crossing. For multi-layer PCP, the Vertical Interconnect Access (VIA) design should be applied on the circuit layer to allow electrical signals to flow across one or more adjacent PCP layers. In contrast to previous studies that have directly achieved VIA by opening holes with a felting needle [17], or connecting both sides of PCP by a VIA copper rivet [1] or two polymer sheets [10], our design easily brings VIA to circuit papers through the use of contact springs. In the hardware of CircuitStack, multi-layer PCP can be realized by stacking more circuit layers and PCP to repeat the mechanism shown in Figure 5. In the software, a user just simply turns on the multilayer option in EAGLE and places the pieces of PCP on different circuit layers as shown in Figure 1a. Compared with previous methods, our contact-spring

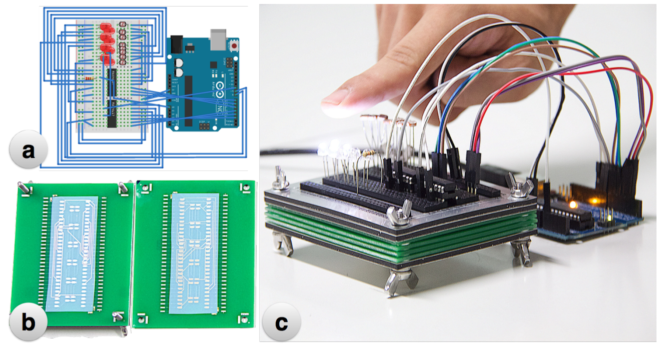

Figure 14. A demonstration for multi-layer PCP as a circuit becomes complex, incorporating eight pairs of a proximity sensor (photoresistor) and an LED controlled by two analog switches (MAX4617). The LED lights up as its paired photoresistor detects a nearby object. (a) Schematic. (b) Two pieces of generated PCP mounted on two circuit layers. (c) Result. Note that four PCBs are on the stack.

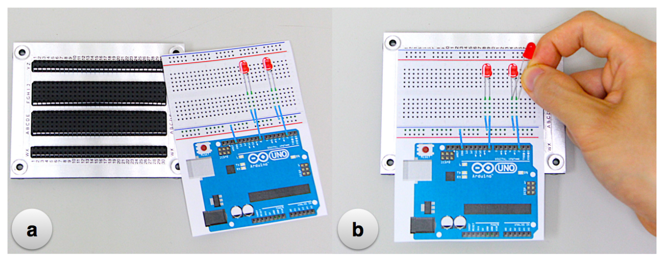

Figure 15. Printed breadboard schematic aids users to directly plug components onto the breadboard layer of CircuitStack.

method neither suffers from alignment problems nor requires drilling handiwork or manual modification of VIA positions on the PCP; therefore, it is easily generalizable to circuit designs with two or more layers, helping developers to build circuits that are more complex in an iterative fashion, as the example shown in Figure 14.

Visual Feedback

Although jumper wires being hidden in-between circuit layers improves the prototyping experience, it results in a lack of visual feedback of the interconnection information on the board. Users constantly must refer to the on-screen breadboard schematics to acquire full connection information and plug the components into the correct positions. There is a creative solution for such circumstances known as a paper template[15] which entails placing a printed breadboard schematic view on top of a breadboard (Figure 15). Thus, users can plug components directly onto a breadboard without constantly switching back and forth between on-screen schematics and CircuitStack.

Resistivity and Alternative Conductive Materials for PCP

Resistivity is a common problem for conductive inks. Although many existing ink products are claimed to have low resistance, the difference in conductivity between ink and

[15] http://blog.fritzing.org/2009/12/02/paper-templates-for-your-breadboard-prototypes/

copper material is still too substantial to be neglected. For instance, as explained in the implementation section on software, the shortest and longest paths result in a 1-Ω and a 104-Ω traces, respectively. Conductive traces on the PCP layer of CircuitStack can be replaced with low resistance materials to resolve resistivity issue such as Printem [3] film which is also created using regular CAD software and standard printers. Another implication is that CircuitStack can reduce the number of resistor plugged on the breadboard layer by using conductive traces with different widths and lengths, which not only can reduce the cable clutters on the breadboard, but can also further simplify the prototyping process.

Component Compatibility

Applying the breadboard layout to the breadboard layer of CircuitStack enables the components used on regular breadboards to remain compatible, and conventional breadboard practices are also preserved. In contrast to other approaches [6, 8, 12, 14, 16] that require additional component customization effort, CircuitStack is compatible with through-hole components and DIP IC chips. Note that the clip springs in female headers are slightly larger than those in regular breadboards, leading to weak connection reliability for DIP IC chips. Mounting DIP IC chips on off-the-shelf IC sockets[16] completely solves this problem. Otherwise, no additional modification of components is necessary.

LIMITATIONS AND FUTURE WORK

Software: Our software implementation that bridges Fritzing to EAGLE may not be a fluent operation. We will build upon the open source of Fritzing to provide not only just a smooth, simple flow of procedure for PCP creation but also additional features such as component placement optimization and support for multiple CircuitStacks operating concurrently for more complex circuits.

Modularized Stack: Although we reduced the effort of manual wiring, the only jumper wires left on CircuitStack in Walkthrough are used to connect electrical signals to the Arduino board. For eliminating jumper wires entirely, we refer to the design of Arduino shields. Shields extend the capability and are easily mounted on an Arduino board in order to save time and space from additional wiring and circuit building. We plan to redesign the layout of the bottom layers and the structure of CircuitStack into an Arduino-compatible shield, to deliver wireless prototyping experiences.

Alternative Assembly Approach: A more effective assembly approach may improve efficiency for joining the multiple layers of CircuitStack together. As shown in Figure 16, each 3D printed case is designed to contain one of each layers. Magnets are added to the slots around the edges. Assembling the CircuitStack layers merely requires close proximity to another layer with magnetic snapping. A sufficient number of magnets must be placed in the slots to overcome the resistance forces of the contact springs. However, excessive magnetic forces may cause difficulty in separating the assembled layers manually. Thus, a high-fidelity design and striking a balance between forces will be part of future work.

[16]http://www.mouser.com/Connectors/IC-Component-Sockets/

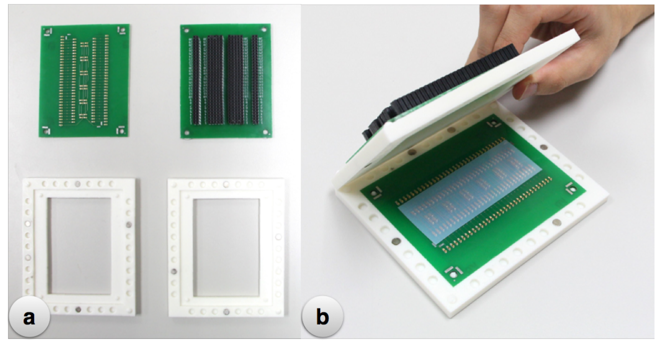

Figure 16. Magnetically-assembled CircuitStack. (a) Each 3D printed case fits one PCB layer. Magnets are on the edges of 3D printed case. (b) The conceptual assembled state of our magnetic-snapping approach.

CONCLUSION

CircuitStack is an enabling technology for virtual-to-physical transition from breadboard schematics to circuits. We discuss some common attributes, practices, and challenges that concern the prototyping process. To achieve superior prototyping experiences, we developed a hybrid system of a breadboard and pieces of printed circuit paper, which are sandwiched between customized PCBs in a stacked structure. The system not only provides makers and developers with circuit reconfigurabilityand component reusability at the early stage of prototyping development, but also engages novices in personal electronic fabrication based on existing breadboard schematics [11]. We detail the implementations of both hardware and software, and summarize with several application examples and discussions to explain the unique properties, features, advantages, and limitations of our system. We sincerely hope that the CircuitStack system can help users accelerate their circuit prototyping processes.

ACKNOWLEDGEMENTS

We sincerely acknowledge the helpful comments of the Associate Chairs and the anonymous reviewers. This work was supported in part by the Ministry of Science and Technology, National Taiwan University, and Intel Corporation under Grants MOST 105-2633-E-002-001, MOST 103-2221-E-002-178-MY2, MOST 104-3115-E-002-002, NTU-ERP-105R890851, and NTU-ICRP-105R104045.

REFERENCES

1. Andersson, H. A., Manuilskiy, A., Haller, S., Hummelgrd, M., Sidn, J., Hummelgrd, C., Olin, H., and Nilsson, H.-E. Assembling surface mounted components on ink-jet printed double sided paper circuit board. *Nanotechnology 25*, 9 (2014), 094002.
2. Bdeir, A., and Rothman, P. Electronics as material: Littlebits. In *Proc. TEI '12* (2012), 371–374.
3. C, V. P., and Wigdor, D. Printem: Instant printed circuit boards with standard office printers and inks. In *Proc. ACM UIST '15* (2015), 243–251.
4. Chan, J., Pondicherry, T., and Blikstein, P. Lightup: An augmented, learning platform for electronics. In *Proc. IDC '13* (2013), 491–494.

5. Edwards, C. System and process for manufacturing custom electronics by combining traditional electronics with printable electronics, Aug. 24 2006. US Patent App. 11/331,191.
6. Hodges, S., Villar, N., Chen, N., Chugh, T., Qi, J., Nowacka, D., and Kawahara, Y. Circuit stickers: Peel-and-stick construction of interactive electronic prototypes. In *Proc. ACM CHI '14* (2014), 1743–1746.
7. Jung, H. C., Cho, S.-H., Joung, J. W., and Oh, Y.-S. Studies on inkjet-printed conducting lines for electronic devices. *Journal of Electronic Materials 36*, 9 (2007), 1211–1218.
8. Kawahara, Y., Hodges, S., Cook, B. S., Zhang, C., and Abowd, G. D. Instant inkjet circuits: Lab-based inkjet printing to support rapid prototyping of ubicomp devices. In *Proc. ACM UbiComp '13* (2013), 363–372.
9. Knörig, A., Wettach, R., and Cohen, J. Fritzing: A tool for advancing electronic prototyping for designers. In *Proc. TEI '09* (2009), 351–358.
10. Lo, J., and Paulos, E. Shrinkycircuits: Sketching, shrinking, and formgiving for electronic circuits. In *Proc. ACM UIST '14* (2014), 291–299.
11. Mellis, D. A., Buechley, L., Resnick, M., and Hartmann, B. Engaging amateurs in the design, fabrication, and assembly of electronic devices. In *Proc. ACM DIS '16* (2016), 1270–1281.
12. Mellis, D. A., Jacoby, S., Buechley, L., Perner-Wilson, H., and Qi, J. Microcontrollers as material: Crafting circuits with paper, conductive ink, electronic components, and an ”untoolkit”. In *Proc. ACM TEI '13* (2013), 83–90.
13. Narumi, K., Shi, X., Hodges, S., Kawahara, Y., Shimizu, S., and Asami, T. Circuit eraser: A tool for iterative design with conductive ink. In *Proc. ACM CHI '15* (2015), 2307–2312.
14. Ochiai, Y. Visible breadboard: System for dynamic, programmable, and tangible circuit prototyping with visible electricity. In *Virtual, Augmented and Mixed Reality. Applications of Virtual and Augmented Reality*, vol. 8526 of *Lecture Notes in Computer Science*. 2014, 73–84.
15. Qi, J., and Buechley, L. Sketching in circuits: Designing and building electronics on paper. In *Proc. ACM CHI '14* (2014), 1713–1722.
16. Ramakers, R., Todi, K., and Luyten, K. Paperpulse: An integrated approach for embedding electronics in paper designs. In *Proc. ACM CHI '15* (2015), 2457–2466.
17. Ta, T., Fukumoto, M., Narumi, K., Shino, S., Kawahara, Y., and Asami, T. Interconnection and double layer for flexible electronic circuit with instant inkjet circuits. In *Proc. ACM UbiComp '15* (2015), 181–190.