2022 Congreso de Tecnología, Aprendizaje y Enseñanza de la Electrónica (XV Technologies Applied to Electronics Teaching Conference (TAEE) | 978-1-6654-2161-4/22/$31.00 ©2022 IEEE | DOI: 10.1109/TAEE54169.2022.9840644

LabPcb: graphical tools for learning PCB manufacturing, assembly and testing

Aurelio Vega
Universidad de Las Palmas de Gran Canaria (ULPGC)
Instituto Universitario de Microelectrónica Aplicada (IUMA)
Las Palmas de Gran Canaria, Spain
ORCID ID: 0000-0002-4154-8799

Carlos Vega
Universidad de Las Palmas de Gran Canaria (ULPGC)
Instituto Universitario de Microelectrónica Aplicada (IUMA)
Las Palmas de Gran Canaria, Spain
ORCID ID: 0000-0002-5629-1471

Juan Cerezo-Sánchez
Universidad de Las Palmas de Gran Canaria (ULPGC)
Instituto Universitario de Microelectrónica Aplicada (IUMA)
Las Palmas de Gran Canaria, Spain
ORCID ID: 0000-0002-2914-170X

***Abstract*—This work presents LabPcb, a graphical environment to facilitate the verification of PCB designs sent to the factory and their subsequent assembly and testing. LabPcb is the global name of the work environment made up of three applications. LPEditor to handle the creation and visualisation of PCBs by importing Gerber files, drill files, pick & place coordinates, test point coordinates or images of the PCB. LPMachine to simplify the control of machines working with printed circuits. LPController to handle multi-protocols communications between LabPcb and the compatible machines. The proposed solution allows the user to work with different workflows: a) manufacture of printed circuits, b) automatic assembly of SMD components, c) optical inspection, d) performance of electronic fault location tests, e) generation of assembly information and f) help to obtain diagrams through reverse engineering. It also allows the user to perform these workflows on physical or simulated machines, fulfilling assembly and teaching functions. This system has been tested on different drilling/milling and pick&place machines and with a large set of PCBs, demonstrating its educational capability.**

Keywords— prototyping, assembly components, electronic test, PCB design, pick & place machine.

I. Introduction

Although the current electronic design is based on SMD (Surface Mounted Device) components [1], we still use traditional breadboards with through-hole technology in our labs [2]. But it is also true that the low-cost modules, generally manufactured by Sparkfun, Arduino or Raspberry, are increasingly used in many practical applications. These modules allow you to assemble and program operational electronic systems quickly.

This work presents a graphical environment called LapPcb that makes it easy for students to prepare PCB designs for manufacturing and assembly using SMD components.

One of the main objectives is to standardize most supervision tasks required from the moment a printed circuit is designed until it is assembled and its operation can be verified. In addition, it is a didactic tool that allows us to quickly locate the electronic components on the design and facilitate the assembly of the components manually or through automatic pick&place machines.

It is not usual for the student to visualize simply the complete process of manufacturing, assembly and testing of printed circuits. For this reason, a tool has been developed that accompanies the student in the assembly and testing of circuits. The tool makes it easy for them to understand how they prepare for industrial manufacturing. LabPcb integrates into a single work environment a large number of processes that are visualized graphically and allow students to understand better the industrial manufacturing processes of the PCBs that they use in the teaching laboratory.

The optical, electrical and thermal inspection processes of printed circuits in the industrial environment require a high investment in specialized equipment. This type of equipment is usually developed to work offline or online, but in either case, they need highly specialized software for each of them. This equipment is not justified in teaching laboratories or most research group prototype manufacturing laboratories. Our software proposal allows the control of low-cost machines that will enable us to carry out these functions sufficiently efficiently.

II. LabPcb environment overview

LabPcb is the work environment's generic name, made up of three applications developed in C# .Net Framework in Visual Studio 2022. LPEditor is a printed circuit editor, LPMachine is a work machine management program, and LPController is a multi-protocol program that communicates with the specific machine (Fig. 1).

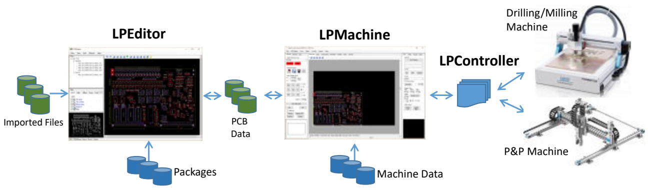


Fig. 1. LabPcb environment overview

LPEditor handles the creation and visualization of PCBs by importing Gerber files, drill files, pick & place coordinates, test point coordinates or images of the PCB. These files can be visualized combined at different rotation angles on both sides. It also includes new functionality for graphically viewing and editing the test points and the components in the design. LPMachine will be in charge of graphically representing the current machine with all the PCBs and elements placed. Allows the users to control the device manually or through automatic procedures with a natural approach. The interface is configured depending on the device's tools, simplifying its use in manufacturing and assembly tasks. Also, it includes software for automated imaging of the boards handy to keep and evaluate the results in the lab environment. Finally, the generic movement orders are sent to the LPController program, which is responsible for correctly translating them into the specific protocol of each machine.

978-1-6654-2161-4/22/$31.00 ©2022 IEEE

In summary, it can be commented that the workflows that are LabPcb's objectives are:

- Manufacture of printed circuits.
- Automatic assembly of SMD components.
- Optical inspection.
- Performance of electronic fault location tests.
- Generation of assembly information.
- Help to obtain diagrams through reverse engineering.

To support the different workflows, a set of software tools has been developed that implement functions as diverse as the following:

- Verification of Gerber and drills files.
- Validation of the pick&place files for the manual or automatic assembly of the PCB.
- Help in the manual assembly of components.
- Creation and export of pick&place files.
- Import or creation of test points.
- Import of RGB images obtained from AOI machines.
- Obtaining thermal images.
- PCB machining.
- Optical boards inspection.
- Carrying out electrical tests.
- Graphic simulation of pick&place machine operation.

III. LPEDITOR

The editor comprises a main menu at the top and three panels (Fig. 2). The left panel is divided into three vertically stacked zones. In the upper area, the type and number of files of the different loaded views of the printed circuit are displayed. The data loaded in each file is shown in the central area. In the lower area, a global view of the PCB is presented.

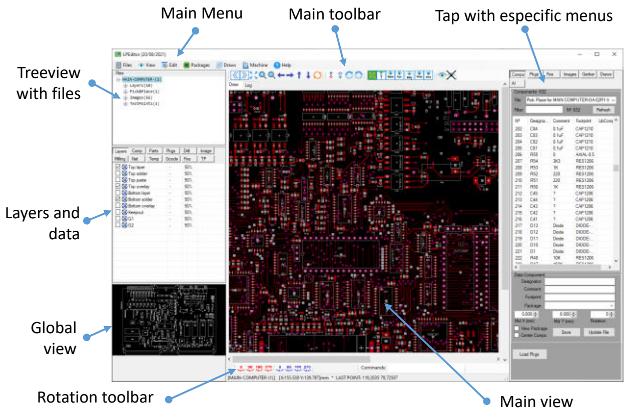


Fig. 2. LPEditor graphic interface

The central panel graphically displays the PCB with the visible layers. The central panel includes two toolbars, the main one located at the top and that allows us to change the view, zoom or view and hide different layers of the design. The toolbar in the lower area allows us to view the card from both sides and with rotations of 0º, 90º, 180º and 270º.

The right panel is made up of tabs that host different functions and occupy the entire right panel. Here we can find the component tabs that allow us to locate all the loaded components, edit their properties, and perform searches filtered by name.

To work with LPEditor, we must first create a printed circuit. We must define its dimensions and the number of layers, model, version, manufacturer, and comments we associate with the design. We can already visualize its contour in the central graphic panel when creating the PCB. Once the PCB is created, we can add new views. The importers found in the Files->Import menu will be used (Fig. 3). Within this menu, we will find the options of a) Gerber files, b) drill files, c) milling files, d) RGB images, e) pick&place files, f) temperature maps, g) drawing or h) test points.

Fig. 3. Gerber import tools

One of the leading importers is the one that loads the coordinate files generated by CAD tools such as Altium Designer, Eagle, KiCad and Allegro. Coordinates are displayed graphically before being incorporated into the design. Later they can be edited or deleted just like the rest of the views. There is no limitation to the number of files that we can load from each view. In this way, we can load different coordinate files that correspond to different design configurations or different types of components.

IV. LPMACHINE AND LPCONTROLLER

LPMachine is the tool for the control of machines working with printed circuits. It is developed with a graphical interface similar to the one developed with LPEditor. It has three main panels (Fig. 4).

The left panel is divided vertically into three zones. The upper area allows to carry out the main tasks of connecting and disconnecting the machines with which work. In addition, it will enable manual movement of the machine head by clicking on the movement direction arrows. The displacement in millimetres of the head can be made in steps of 0.05, 0.1, 1, 10 and 100 mm. In this area, we also see the indication of the operational status of the machine that can be: "Not Connected", "Idle", "Run", "Hold", "Home"," Alarm", "Check" and "Door". Each state indicates whether the machine is OK or whether an error or warning event has occurred. In the lower area, there is an overview of the

978-1-6654-2161-4/22/$31.00 ©2022 IEEE

machine's work area with an indication of the zoom area visible in the central panel.

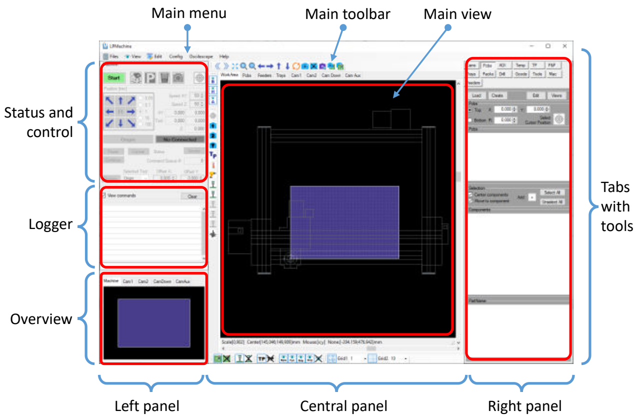


Fig. 4. LPMachine graphic interface

The central panel is occupied by the graphic display area of the machine and the PCBs that are loaded. Additionally, we have several toolbars that allow quick access to the work functions with the tool.

In the right panel, we find different tabs with the main work tools for each type of activity. The PCBs tap includes the main functions of loading and positioning the PCBs in the machine's work area with which we are working. This tab also allows us to view all the electronic components loaded on each of the PCBs and the associated partnames. The rest of the tabs include the functions for capturing images, temperature measurements, test points, drills, etc.

LPMachine is developed to work with the different machines that we have in our laboratories. In order to achieve this objective, it has been necessary to develop a communications driver structure like the one shown in Fig. 5 Communication between LPMachine and LPController can be carried out through serial or TCP-IP communications. The communication between LPController and the machines will be carried out through specific serial protocols of each manufacturer of the machines we want to connect. In the work team, the necessary drivers have been developed for the machines available in our laboratory; specifically, we have several LPKF, Bungard machines, and Creality 3D printers.

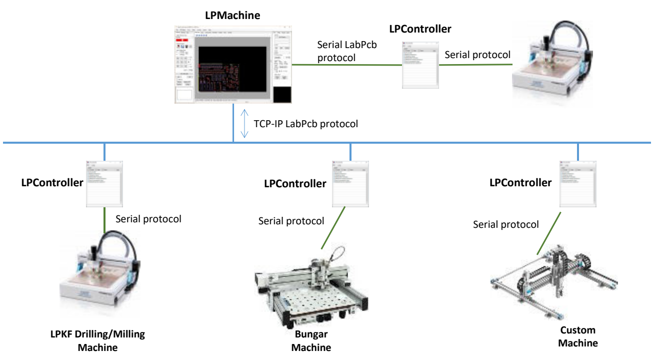


Fig. 5. Example of work with different machines

Each of the machines must configure LPMachine properly within the configuration forms. Two of the forms used are presented in Fig. 6. The form in Fig. 6.a allows us to configure up to two simultaneous connections with two controllers. In Fig. 6.b, we see the form to define the available tools of each machine. The tools that can be defined are the following: 1 Drill/Mill, 1 Laser, 1 Test Point, 2 Cameras Top, 1 Camera Down, 1 Camera Temp, 5 Pick&Place, 1 Temp Spot. In the future, top and bottom welding heaters may also be activated. For each machine, only the tools that are available to them will be activated. For each tool, we will have to define the offset of their positions concerning the position considered as the origin.

The configurations can be saved in files and later loaded when necessary. If a tool is not defined for a machine, the corresponding functions will be disabled.

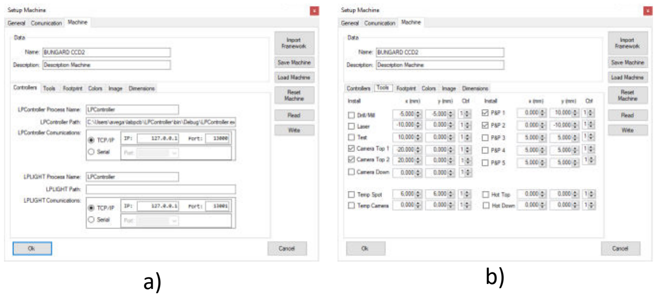


Fig. 6. Machine setup forms

V. Obtaining composite images of printed circuits.

One of the outstanding features of LPEditor is its ability to incorporate scaled images of printed circuits. The import can be done manually, but the maximum potential is obtained by importing images obtained on AOI machines. The OMRON RNS II-pt optical inspection equipment [3] has been used. This compact electronic circuit inspection equipment is designed for industrial use in low-volume or high-mix production environments. The equipment has a camera with a movement system in the XY axes. The camera has a tiny field of view, so to produce a global image of the PCB, the partial images of the different areas must be combined. The kit's software provides the functionality to capture the combined image of the PCB but does not allow the image to be exported at full resolution. The software has been developed to combine the partial images and generate a global image with the maximum resolution.

The image generation process is represented in Fig. 7, as well as a section of the file that describes the partial coordinates of each of the captured images. The equipment manufacturer's software, Ez-Image Teaching, was used to capture the partial images. The equipment generates 512x512 pixel images in BMP format and sweeps the entire area of the card from left to right.

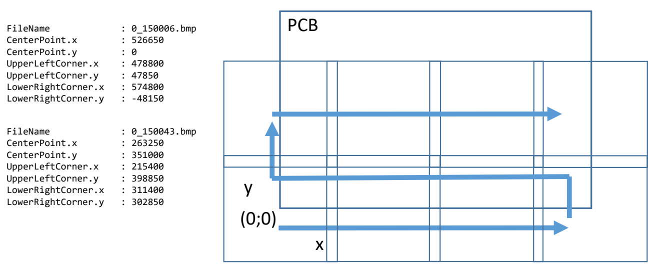


Fig. 7. Image acquisition sequence

For a PCB in double-Europe format (233x160mm), a total of 567 images are generated. LPEditor will import the partial images and build a high-resolution global image of the PCB (Fig. 8). Uploading of individual images or a single composite

978-1-6654-2161-4/22/$31.00 ©2022 IEEE

image is allowed. In either case, the uploaded images can be moved, adjusted, and scaled if necessary. In addition, we can choose to make it visible or not visible at any time.

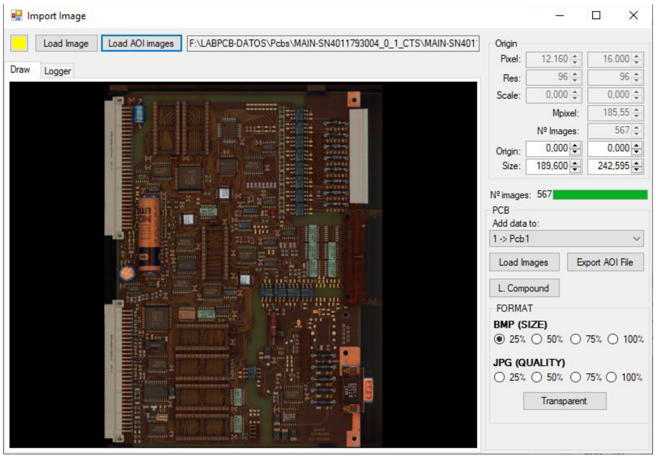


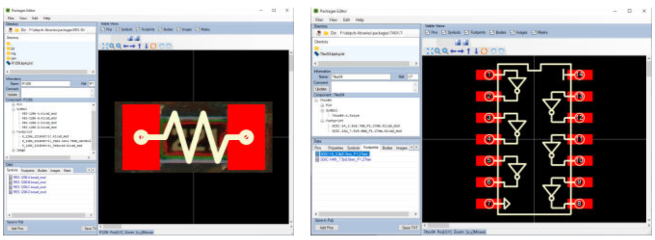
Fig. 8. Image importer

VI. WORK WITH PACKAGES.

In the design of printed circuits, it is common to work with the footprint of the components. Different footprints and a physical model representing the case can be associated with the same component (Fig. 9). In LPEditor, you can work with packages that have four different views, which are the following:

- *Footprint*. We import the KiCad footprints.
- *Body*. The body provides information on the physical dimensions of the components.
- *Symbol*. Graphical representation of the package scaled concerning physical dimensions. This view helps recognize the real images of printed circuits.
- *Mask*. The masks delimit the area occupied by a specific component within the images. Mask editing allows our editor to be used as an image labelling tool for training AI algorithms.

We can use packages to represent electronic components from their coordinates on printed circuit boards in the manufacturing workflow. This view allows us to check the correct coordinates assignment before the pick&place process.



a) b)

Fig. 9. Package editor

Our tool is designed to facilitate the reverse process of identifying electronic components on the accurate scaled view of the board on which we are working. The direct applications are the following:

- Generate the coordinate file for a printed circuit from its Gerber files or its real scaled composite image.
- Generation of operating diagrams of assembled circuits using reverse engineering techniques. From real circuits, the generation of technical documentation is facilitated, which allows the understanding of its functionality.
- Printed circuit board image labelling function for training Deep Learning algorithms.

Fig. 10 shows a section of a wind turbine control system card that has been captured in order to identify the components and generate the pick&place coordinate files.

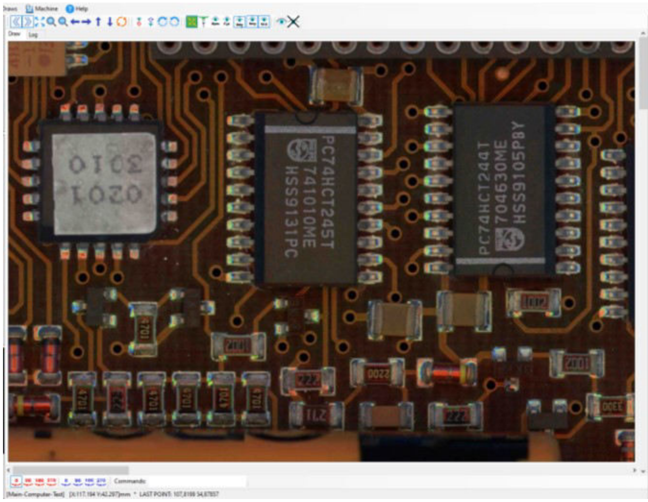


Fig. 10. PCB section without packages

Fig. 11 shows the same section of the circuit on which the packages (R1206, C1206, SOIC20,...) corresponding to the components displayed in the section have been manually placed. You can see that in the displayed view the footprints and the symbols associated with the packages are active. LPEditor will allow us to export the coordinate files of the components in CSV format. The exported coordinates can be imported into other printed circuit design tools to serve as a template for the design of a PCB that replicates the one shown.

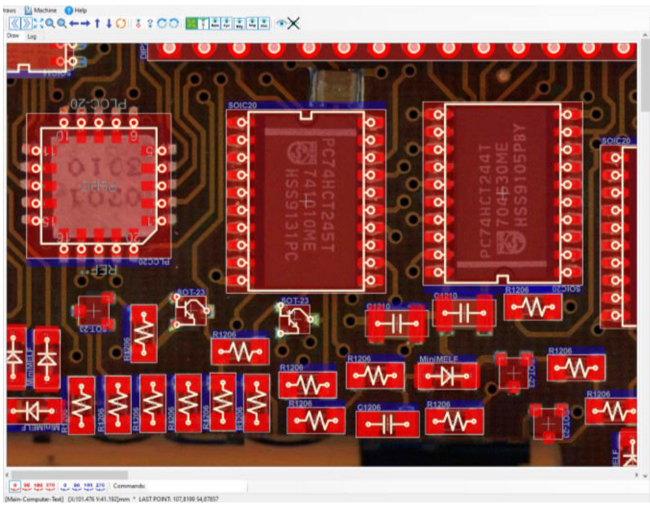


Fig. 11. PCB section with footprint and symbols visible

In Fig. 12, the section is shown with only the symbols of the packages. It is the working group's objective to be able to

978-1-6654-2161-4/22/$31.00 ©2022 IEEE

signalize the electrical connections through interconnection lines such as those indicated in order to later be able to export the netlist of the design.

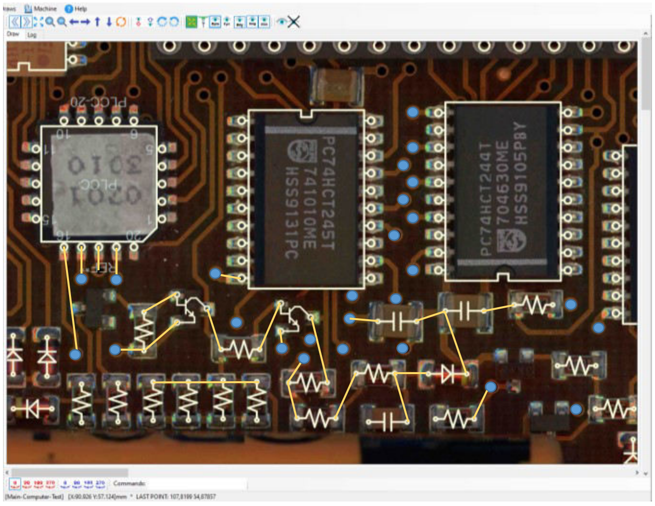


Fig. 12. PCB section with the representation of electrical connections

VII. LABELLING OF IMAGES.

The image labelling process consists of marking the positions of the electronic components that compose it on the image of the printed circuit. In addition, we must indicate the rotation and the area occupied by the component. Optionally we can indicate the type of component and its partname. With this information, training can be carried out later using Deep Learning techniques of electronic component identification algorithms on the scaled image of the circuit.

LPEditor makes it easy to work with electronic circuits in reverse flow, starting from a manufactured PCB, carrying out the reverse engineering process graphically and simply. Our application will allow us to carry out the entire process of importing, verifying, and generating internal files graphically. In Fig. 13, we see the manual labelling process of an electronic component. We use the appropriate packages for each type of component and describe their properties for the subsequent processing of the images.

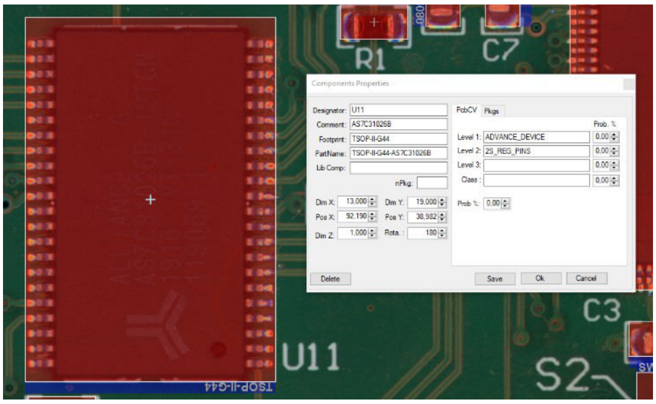


Fig. 13. Manual labelling of electronic component

Once the labelling process is finished, all the information necessary for the treatment of the printed circuit will be available. In Fig. 14, we see a processed board, and it can be seen that in the right panel of the editor, we have the list of components generated during the process. This information can be exported to files in *.CSV format for later processing by the image dataset generation tools.

The ultimate goal is to have an algorithm for classifying and identifying electronic components associated with our LabPcb work environment. Currently, a first identification tool has already been successfully carried out.

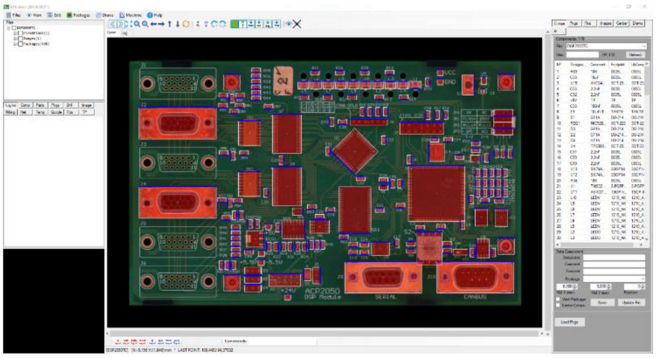


Fig. 14. Overview of a labelled PCB

VIII. OPERATION IN PICK&PLACE MODE

To work in pick&place operation mode, the machine must have a suitable pick&place head. The data of the positions of the electronic components will be loaded into the PCBs placed in the machine's work area. The definitions, feeders, and component trays will have to be defined and loaded with the machine. LPMachine has a configuration menu for feeders and trays.

Once the feeders and trays have been defined, they will be assigned a position in the work area. The trays may be in any place where the pick&place heads have access. Feeders will typically be located at the front or rear of the machine, as shown in Fig. 15.

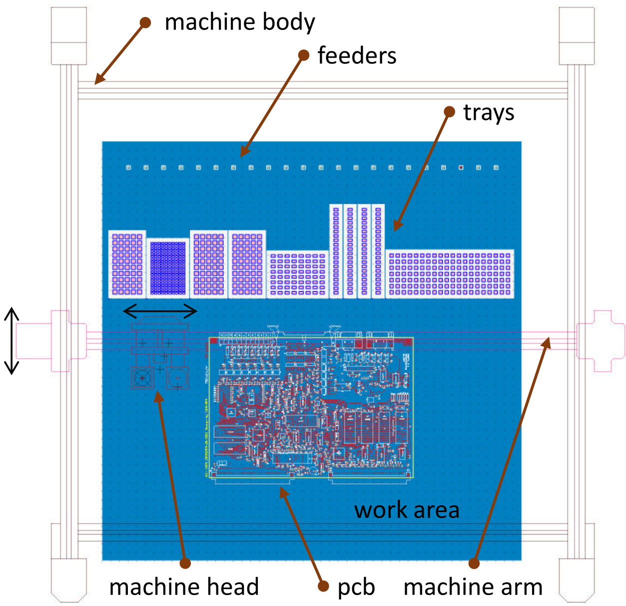


Fig. 15. Positioning of feeders and trays.

Both feeders and trays will have to be assigned a component partname so that the system can know from which feeder the component will have to be taken. Once the system is configured, the placement process can be launched. The alignment of the components can be carried out employing the classic process of centring through the lower camera or by centring the collection point through one of the top cameras of the equipment. The software will select specific components

978-1-6654-2161-4/22/$31.00 ©2022 IEEE

to place or indicate that all those defined in the file loaded in memory are placed.

IX. RESULTS OBTAINED AND PROBLEMS FOUND

The developed software allows working with physical or simulated machines to fulfil assembly and teaching functions. The system has been tested with different drilling/milling and pick&place machines and many PCBs, demonstrating its educational capability.

This work is based on the previous experience of the working group in the development of aid tools in the production of printed circuits [4], [5]. The group has a manufacturing line for printed circuit prototypes and an automatic pick&place assembly line. Experience in the design, manufacture and testing of complex electronic systems with a high level of integration has allowed us to work on developing the LabPcb environment to facilitate the integration of the different phases of the manufacture and assembly of printed circuits.

The development of LabPcb has been carried out with the collaboration of various students of the Final Degree Project of the degrees of Telecommunication Technologies and the Degree of Industrial Electronic and Automatic Engineering. And of Master's Thesis of the Master's Degree in Electronics and Applied Telecommunications of the IUMA. All degrees from the University of Las Palmas de Gran Canaria (ULPGC).

These tools are in the debugging phase, and we hope to have a first version available for students shortly. The group is also working on image identification algorithms with satisfactory initial results. Due to the wide variety of packages on the market, a high number of correctly labelled printed circuits is required to have a sufficiently varied component dataset for algorithm training.

Our system is also prepared to work as an automatic pick&place machine. Internally, we have already developed the primary management functions for feeders and component trays. These new features are already being tested and released in the next workbench revision. Therefore, the objective is to tackle the maximum number of usual functions in a manufacturing and assembly laboratory that we can carry out under a standard interface instead of using multiple applications from each one of the manufacturers of each machine.

By way of example, we can indicate that the software tools presented allow the reuse of milling machines, 3D printing machines, or any Gantry XYZ-type machine in an optical inspection system, thermal imaging or automatic electrical test machine. The only necessary thing is to have or develop the required interconnection driver for each specific protocol. Additional functions are implemented through the Lighting Kit and thermal and electrical measurements.

At present, different pick&place machines are being worked on. One of them is a single-head machine with which the positioning functions are debugged.

It is working on three lines. On the one hand, in the debugging and improvement of the tools that make up the LabPcb environment. On the other hand, in the development of low-cost polyvalent equipment for the automatic capture of composite images and the improvement of the component classification algorithms from the images captured by our system.

The aim is to develop and offer a cloud service to identify electronic components. The labelling process of the images obtained can be carried out remotely and automatically on our servers or hosted on servers in the cloud.

REFERENCES

[1] G. R. Blackwell, "Surface Mount Technology," *Electronic Systems Maintenance Handbook*, pp. 12–1, Dec. 2017, doi: 10.1201/9781420036855-12.

[2] D. R. Wilcox, G. W. Wstenkhler, and H. Harz, "A new pedagogy for the electronics laboratory," *ASEE Annual Conference and Exposition, Conference Proceedings*, 2011, doi: 10.18260/1-2--17361.

[3] "VT-RNSII ptH Desktop Portable PCB Inspection System/Features | OMRON Industrial Automation Australia." https://www.omron.com.au/products/family/2982/feature.html (accessed Mar. 04, 2022).

[4] A. Vega, S. Felipe, J. Monagas, and A. Déniz, "Tool to aid production of electronic prototypes with SMD technology," Aug. 2016. doi: 10.1109/TAEE.2016.7528358.

[5] A. Vega, S. Felipe, and A. Déniz, "Control software for pick & place machines," Aug. 2016. doi: 10.1109/TAEE.2016.7528357.

978-1-6654-2161-4/22/$31.00 ©2022 IEEE