Vol. 4, No. 1, 2022

https://doi.org/.....

UDK 004.9+621.3.049.75

K. Kolesnyk[1], A. Łukaszewicz[2], V. Dutka[1], D. Zahoruiko[1], B. Vasylyshyn[1]
[1] Lviv Polytechnic National University,
[2] Bialystok University of Technology

AUTOMATED DESIGN OF PRINTED CIRCUIT BOARDS MADE BY ELECTRONIC COMPUTER –AIDED DESIGN (CAD) WITH THE NEXT USING IN CNC- MACHINE

The article presents methods and means of 3D design of printed circuit boards in CAx. Automated placement of elements on the board is implemented by means of API SolidWorks using Visual Studio C#. The API application works by an algorithm that allows you to create a 3D layout of printed circuit boards. Each component of the library contains a conditional graphic notation of the element. With the help of the implemented algorithm, a comprehensive approach is provided, which consists in the fact that already at this stage the preparation of the strategy for tracing the conductors of the future printed circuit board is carried out, the classes of circuits are determined and the necessary technological parameters are set, as well as the data necessary for the preparation of design documentation is generated. After the completion of the work on the input of the scheme, a check is made for the presence of errors and compliance with the specified parameters, and if the test is successful, a list of circuits is generated for transfer to the tracing program. From this moment, any possibility of errors in the subsequent stages of design is excluded. CAD DipTrace was used to trace printed circuit boards and generate g-code. Printed circuit board processing is carried out on a CNC machine - CNC3018 using the Candle program. The printed circuit board tracks are created by forming a groove between the track and the metallized coating of the textolite. With the help of the formed height map, the uniform removal of the metallization layer over the entire area of the textolite is ensured.

In addition, holes are drilled for the output elements of the circuit, the printed circuit board is cut along the contour and covered with a layer of tin to prevent oxidation of its metallized coating. The considered CAD methods and tools made it possible to automate the design of the printed circuit board of the FM radio receiver control module. As a result of the performed work, means of automating the design of printed circuit boards were applied and a fully functional printed circuit board with a track width of 0.8 mm was obtained.

Key words: **CNC, DipTrace, CAD, FM, engraving, g-code, height map, Candle, SolidWorks.**

Introduction

At the time of computer technology, scientific and technological progress, the use of printed circuit boards for the manufacture of high-tech products (smart devices) has become widely used. Printed circuit board is a plate made of materials that do not conduct electric current well (textolite, getinax), on which or inside which at least one conductive pattern is formed. Electronic components are mounted on the printed circuit board, which is connected by the leads to the elements of the leading pattern by soldering or, much less often, welding, as a result of which an electronic circuit is formed module [1-4].

The system board is the most important component located in the case of any modern digital gadget. It is designed for powering, switching, controlling and coordinating the actions of all elements found in any computer equipment. Various SMD components (resistors, capacitors, fuses, diodes) and all other peripherals are connected to the system board using ports, outputs and connectors.

The most common reason that leads to rapid board failure is overheating. It can occur due to climate conditions, dust ingress, and a cheap cooling system. İt happens often that the cause of overheating is too much current applied to the board components. That is why it is extremely important to calculate such a current strength for each element in order to prevent breakage due to overheating in advance during the practical use of any board.

Design and manufacture of a printed circuit board in CAD

The PCB design process starts with creating a component library for the project. At this stage, the documentation for the components used is searched and studied. Each component of the library contains a conditional graphical designation of the component for the schema editor, and a seat for the topology editor. An integrated approach consists of the fact that has already appeared at this stage, a strategy for tracing the conductors of the future printed circuit board is prepared, circuit classes are determined and the necessary technological parameters are set, as well as data necessary for the preparation of design documentation is formed. At the same time, it is necessary at this stage to take into account the heating of components that are placed on the board for its further reliable operation, and it is advisable to conduct such thermal analysis in 3D modeling systems, for example, SolidWorks, Cadence, etc. [5-8]

After completing the circuit input work, it checks for errors and compliance with the specified parameters, and if the test is successfully passed, a list of circuits is generated for transmission to the trace program. From this point on, any possibility of errors occurring at further design stages is eliminated.

Automated placement of elements on the board should be carried out using the SolidWorks API. The API application is developed directly in Visual Studio, using program code generated when writing a macro in SolidWorks. In this case, the developed subsystem can work according to the following algorithm, as shown in fig. 1.

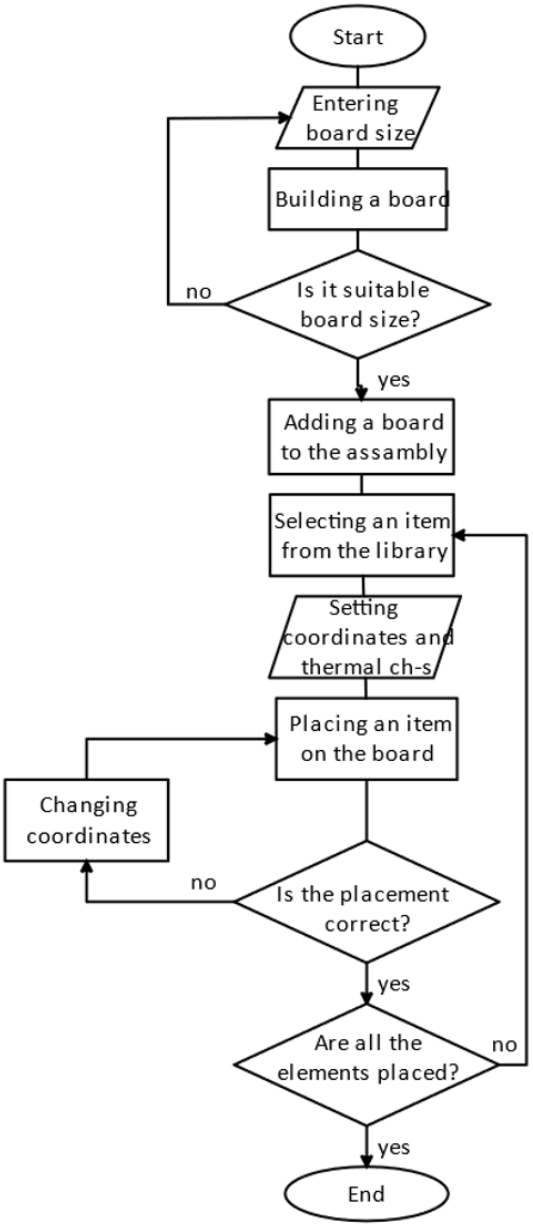

Fig. 1. Algorithm of operation of the subsystem with 3D design of printed circuit boards

The followed production of boards uses CAD Dip Trace needed for forming g-code. It is strongly advised to follow the steps of editing printed boards [9-15].

Demonstration of CNC-Machine manufactured possibilities is shown fort he controls module of the FM radio Dimensions of the circuit boards is 46x46 mm. The circuit board is shown at the Fig 2.

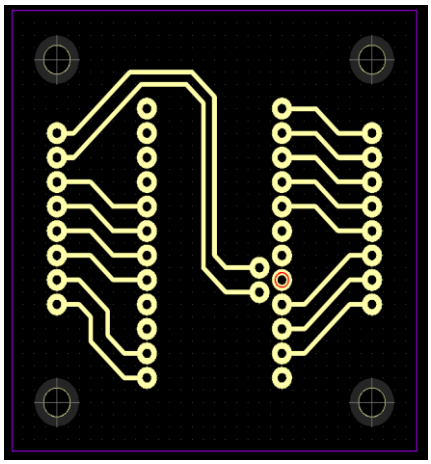

Fig. 2. Printed circuit board for FM radio control module in CAD DipTrace

Fig. 3. CNC Machine - CNC3018

Steps and process of printing CNC-machine _CNC3018 is shown at the Fig 3.

The first step and the followed steps use Candle program that is interfaced and used hand to hand with CNC-Machine (Fig 4).

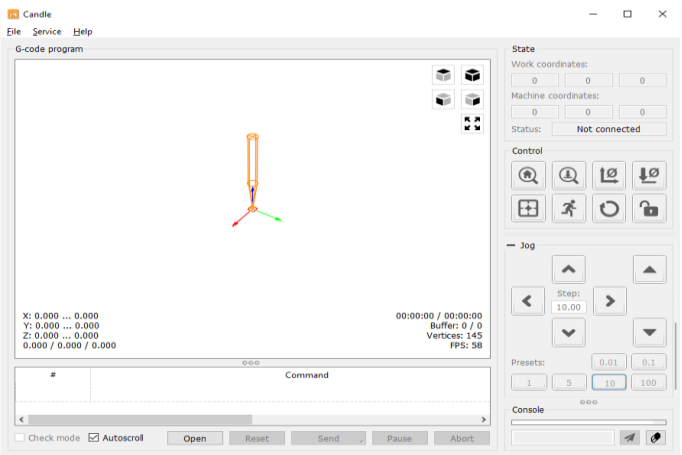

Fig. 5. The program for the first stage of the formation of printed circuit board tracks

The tracks of the printed circuit board are created by forming a groove between the track and the metallized coating of the textolite, which is shown in Fig. 6.

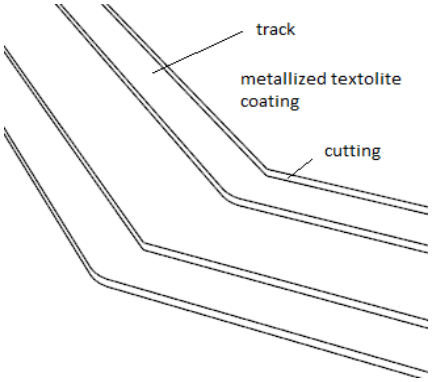

Fig. 6. Forming a groove between the track and the metallized textolite coating

Since the metallized coating is no more than 0.1 mm and the textolite itself on which it is applied is not perfectly even, we need to make a height map of the working surface, the process is shown in Fig. 7.

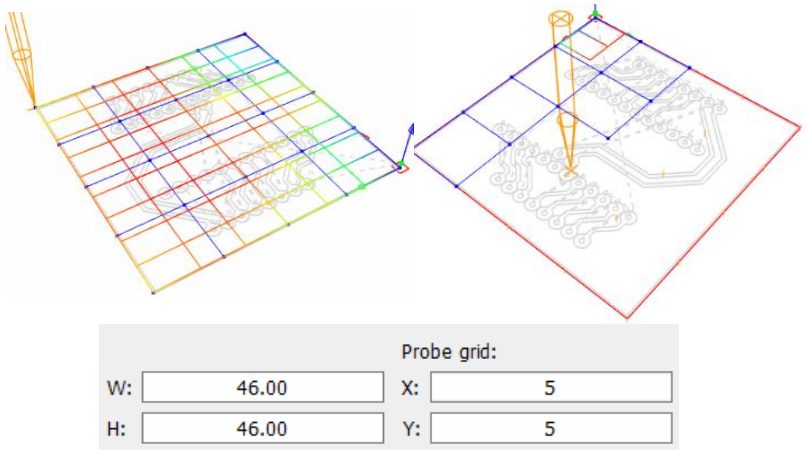

Figure 7. Measurement of the height map

The height map is created as follows: the CNC machine has a so-called Z-probe, which, when closed, sends a signal to stop the machine along the Z axis (tool axis). The textolite before processing is an excellent conductor throughout its metallization surface, therefore it ensures the closure of the Z-probe. The program indicates the size of the future printed circuit board and the grid (number of points) of measurement, in our case it is 46x46mm and 5x5=25 measurement points. As a result, the CNC machine measures 25 points and plots them on the height map, as well as immediately displays the color scheme of the curvature of the textolite in the program interface. Thus, with the help of a height map, the metallization layer is uniformly removed over the entire area of the textolite. In our case, the distortion is minimal, and when working on a textolite of a larger size, the distortion will be more significant, so it can be called a height map measurement tool, the main tool when working with printed circuit boards on a CNC Machine, and in the absence of such a tool in the arsenal of a CNC Machine, it makes it impossible to work with printed circuit boards in principle. Using the W0.2x3.175x30º milling cutter, we start the first stage of cutting the printed circuit board (Fig. 8).

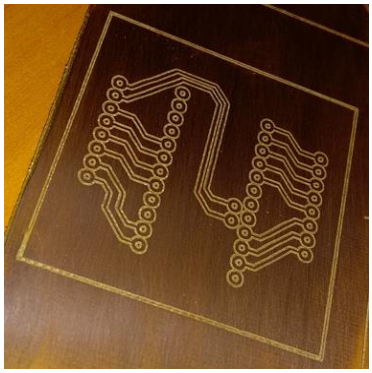

Fig. 8. Printed circuit board after the first stage of cutting

The next stage is drilling holes for the output elements of the circuit and cutting the printed circuit board according to the contour. Since these two stages do not require the accuracy of cutting along the Z axis (Axis of the tool), we cannot use the height map when working, but since the stages require cutting through the textolite, for this, it is necessary to put a substrate under the textolite, so that in the case of cutting the textolite, it will not be damaged machine table The printed circuit board after two stages of cutting is shown in Fig. 9.

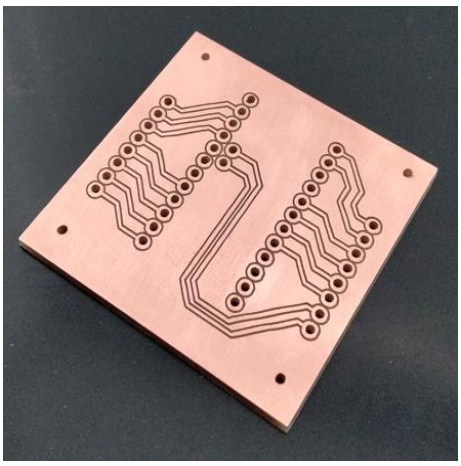

Fig. 9. Printed circuit board after two stages of cutting

To prevent oxidation of the metallized coating of the printed circuit board, it should be covered with a layer of tin, for an even application, you can use a desoldering braid, although it is intended for removal, it leaves a minimal layer on the copper metallization, which was to be achieved. The completed printed circuit board is shown in Fig. 10.

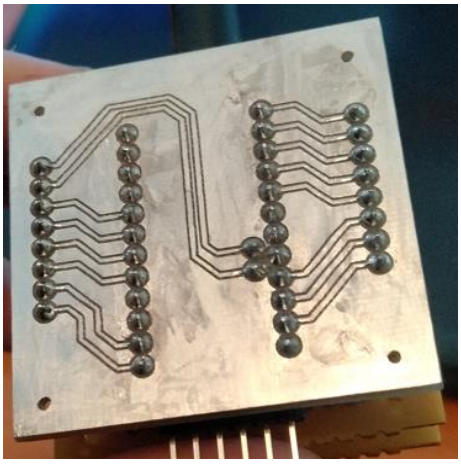

Fig. 10. Completed printed circuit board

Conclusion

As a result of the work performed, we used tools for automating the design of printed circuit boards and received a fully functional printed circuit board with a track width of 0.8 mm. In the future, it is planned to increase the accuracy of the CNC Machine and obtain a printed circuit board with a track width of 0.4 mm. It is also planned to apply a soldering mask. If necessary, it is also possible to completely remove the metallization, leaving only the tracks.

To consider CAD methods and tools, it was possible to provide an automated design of a printed circuit board of a control module for an FM radio receiver. This approach to the design of printed circuit boards is advisable to use in the design of printed circuit boards for their use as part of transceivers for data transmission.

References

1. FlorensWasser, NormanHendrich, DanielAhlers, JianweiZhang Topology-aware routing of 3D-printed circuits Additive Manufacturing Volume 36, December 2020, 101523 https://doi.org/10.1016/j.addma.2020.101523

2. A.H. Espera, J.R.C. Dizon, Q. Chen, R.C. Advincula, 3D-printing and advanced manufacturing for electronics, Prog. Addit. Manuf. (2019) 245–267, https://doi.org/10.1007/s40964-019-00077-7.

3. Nevliudov, I., Razumov-Fryzyuk, E., Nikitin, D., Bliznyuk, D., Strelets, R. (2021), "Technology for creating the topology of printed circuit boards using polymer 3D masks", Innovative Technologies and Scientific Solutions for Industries, No. 1 (15),P. 120–131. DOI: https://doi.org/10.30837/ITSSI.2021.15.120

4. Trochimczuk R., A. Łukaszewicz, M. Melnyk, A. Kernytskyy Design of mechatronics systems using CAx environment /. Methods and tools in CAD. – Białystok: Publishing House of Białystok University of Technology, 2021. 7-14. DOI:10.24427/978-83-66391-87-1_01

5. A. Łukaszewicz, G. Skorulski, R. Szczebiot, The main aspects of training in the field of computer-aided techniques (CAx) in mechanical engineering, Proceedings of 17th International Scientific Conference on Engineering for Rural Development, May 23-25, 2018, Jelgava, Latvia, 865-870. DOI: 10.22616/ERDev2018.17.N493

6. Kusiak A. Smart manufacturing. Int J of Prod Res 2018; 56(1-2): 508–517. https://doi.org/10.1080/00207543.2017.1351644

7.N. Evstatieva and B. Evstatiev, "Modelling the Temperature Conditions of a Printed Circuit Board," 2021 12th International Symposium on Advanced Topics in Electrical Engineering (ATEE), 2021, pp. 1-4, doi: 10.1109/ATEE52255.2021.9425281.

8. N. Vakrilov, A. Stoynova and B. Bonev, "3D Thermal Modelling and Verification of Power Electronic Modules", 42nd International Spring Seminar on Electronics Technology (ISSE), 2019 DOI:10.1109/ISSE.2019.8810307

9.Umesh Shinde, Rahul Somalwar, Namesh A. Kale, Ashish J Nandeshwa, Antariksh V. Mendh. Short paper on CNC based PCB milling machine considering human safety. Journal of Research in Engineering and Applied Sciences. July 2020, 5(3):104-107 DOI:10.46565/jreas.2020.v05i03.005

10.Shilpa, V. Jean, and S. H. Mahmood. "Design and Implementation of Three-Axis Cost Efficient CNC PCB Milling Machine." International Conference on Recent Trends in Electrical, Control and Communication (RTECC). IEEE, 2018 DOI:10.1109/RTECC.2018.8625647

11. Syed Usama BUKHARI, Ioan BONDREA, Remus BRAD Automated PCB Inspection System. TEM Journal. Volume 6, Issue 2, May 2017 Pages 380-390, ISSN 2217-8309, DOI: 10.18421/TEM62-25.

12. Kamble, P., Khoje, S., &Lele, J. Implementation of paper PCB using conductive ink 2D plotter. International Conference on Computing Communication Control and Automation (ICCUBEA), 2018 DOI:10.1109/ICCUBEA.2018.8697781

13. Abdalla Milad Faraj, Hisham Fathi Ali, Abdussalam Ali Ahmed, Mohammed Khaled Akel. CNC machine for image and PCB layout drawing Global Journal of Engineering and Technology Advances, 2022, 11(02), 013–024. https://doi.org/10.30574/gjeta.2022.11.2.0078

14.Allaparthi, M., Rajik Khan, M., Teja, B., 2018. Three-dimensional finite element dynamic analysis for micro-drilling of multi-layered printed circuit board. Materials Today: Proceedings 5.2, 7019-702 https://doi.org/10.1016/j.matpr.2017.11.365

15. Lin, D. T., Wang, C. Y., Fu, L. Y., 2019. Cryogenic auxiliary drilling of printed circuit boards. Circuit World 45.4, 279-286. https://doi.org/10.1016/j.procir.2020.10.007

К. Колесник[1], А. Лукашевич[2], В. Дутка[1], Д. Загоруйко[1], Б. Василишин[1]
[1]Національний університет "Львівська політехніка",
[2]Білостоцький технологічний університет,

АВТОМАТИЗОВАНЕ ПРОЕКТУВАННЯ ДРУКОВАНИХ ПЛАТ У САПР ТА ЇХ ПОДАЛЬШЕ ВИГОТОВЛЕННЯ ЗА ДОПОМОГОЮ CNC-MACHINE

В статті представлено методи та засоби 3D проектування друкованих плат в CAx. Автоматизоване розміщення елементів на платі реалізовано засобами API Solidworks за допомогою Visual Studio C#. API-додаток працює за алгоритмом, що дозволяє забезпечувати 3D компоновку друкованих плат. Кожен компонент бібліотеки містить умовно-графічне позначення елемента. За допомогою реалізованого алгоритму забезпечено комплексний підхід, який полягає в тому, що вже на цьому етапі здійснюється підготовка стратегії трасування провідників майбутньої друкованої плати, визначають-

ся класи ланцюгів і задаються необхідні технологічні параметри, а також формуються дані необхідні для підготовки конструкторської документації. Після завершення робіт з введення схеми здійснюється перевірка на наявність помилок і відповідність заданим параметрам, і в разі успішного проходження тесту генерується список ланцюгів для передачі в програму трасування. З цього моменту виключається будь-яка ймовірність виникнення помилок на подальших етапах проектування. Для трасування друкованих плат та формування g-коду застосовано САПР DipTrace. Обробка друкованої плати проводилася на CNC machine - CNC3018 за допомогою програми Candle. Доріжки друкованої плати створювалися шляхом формування канавки між доріжкою та металізованим покриттям текстоліту. За допомогою сформованої карти висот забезпечено рівномірне зняття шару металізації по всій площі текстоліту.

Крім того, реалізовано висвердлювання отворів для вивідних елементів схеми, обріз друкованої плати за контуром та покрито шаром олова для запобігання окислення її металізованого покриття. Розглянуті методи та засоби САПР дозволили забезпечити автоматизоване проектування друкованої плати модуля керування для FM радіоприймача. В результаті виконаної роботи застосовано засоби автоматизації проектування друкованих плат та отримано повністю функціональну друковану плату, із шириною доріжки 0.8 мм.

Ключові слова: **CNC, DipTrace, САПР, FM, гравіювання, G-Code, карта висот, Candle, SolidWorks.**