Enhancing the Manufacturability of Biodegradable PLA/Flax PCB Assemblies Through Design Optimization

Gergő Havellant and Attila Géczy
Department of Electronics Technology, Faculty of Electrical Engineering and Informatics,
Budapest University of Technology and Informatics,
Budapest, Hungary, H 1111,
havellantgergo@edu.bme.hu

Abstract— Electronic waste is a significant environmental problem today, which is why our research aims at developing sustainable, biodegradable materials. In previous research at the Department of Electronics Technology at the Budapest University of Technology and Economics, we successfully created a working Arduino Nano circuit on a biodegradable polylactic-acid (PLA)-based, flame-retarded, flax-reinforced substrate. The manufacturing process needs constant optimization, and the yield needs improvement for better manufacturability. This research presents an approach and interim results to improve the yield of production by reconsidering the design rules on an Arduino Uno re-design, based on "just enough technology" mindset.

Keywords— sustainable manufacturing, just enough technology, biodegradable printed circuit board, PCB, design for manufacturing

I. Introduction

The problem of electronic-waste requires new solutions for the electronics industry, starting from the material set and the design approaches. There are many papers and research paths revolving around changing the material set of classical electronics, the printed circuit board is usually the aim of the most radical changes [1, 2, 3], in the meanwhile, it is one of the least researched aspects of green electronics [1].

We previously introduced a PLA/Flax base material for PCBA (Printed Circuit Board Assembly), with biodegradability and flame retardancy. The production yield of the previously created Arduino Nano [4] on a biodegradable substrate could be improved – our research was presented previously at early technological readiness levels – practically an early demonstrator of the technology.

Several problems were encountered during the assembly processes. This experience was systematized, and we started to collect the aspects of design for manufacturability.

So, the paper summarizes the motivations for enhancements in the composite development for PCB application, where the data was gathered between 4-5 generations of the base substrate. In this paper we present a new demonstrator design based on the Arduino Uno. This circuit allows greater freedom in terms of size and requires only one-sided assembly. Biodegradable substrates are very similar to flexible substrates, so it is worth investigating rules that have been established there [5], and to approach the problem with the mindset of "just enough" design and technology, already presented in [6].

II. PCB MANUFACTURING AND MOTIVATION FOR ENHANCEMENT

A. *Drilling*

For the drilling we used the optimum parameters established for standard FR4 (Flame Retardant Class 4) materials and using standard solid carbide tools as a starting position. Unfortunately, for each of these substrates, the chip and burr removal were found to be inadequate, the substrate is likely to soften during drilling, become sticky and adhere to the tool. This is due to the fibers of the reinforcement material (flax textile) and the thermoplastic resin in the composite. The biodegradable substrate forms strings which only break after the hole is drilled and therefore wind up on the tool (Fig 1.). This will lead to tool breakage for small diameters (0.5 mm and less), and the roughness of the hole wall will not be optimal for the hole plating process.

Fig 1. Tool with strings/burrs from biodegradable composite after drilling.

When the stack was also fitted with 0.5 to 1 mm thick FR4 sheet on each of the 2 sides of the biodegradable substrate, we found that only the 0.5 mm tool caused chips to be deposited on the tool, but to a lesser extent than when drilling between non-FR4 sheets.

When the drill bit moved from the biodegradable substrate into the FR4 material, the thread-like chips/burrs wound onto the tool were removed by the epoxy glass fabric, supposedly tearing it, and thus removed by the burr exhaust. In small diameter tools, due to the smaller feed rate, the length of the thread-like burrs wound on the tool is longer, and this is probably not removed by the smaller amount of burrs produced by the FR4.

B. *Contour milling*

For contour milling, the optimum parameters established for standard FR4 materials were applied at first, and standard solid carbide tools were used, diamond-profiled solid carbide routers and spiral heads (this tool usually used for IMS (Insulated Metal Base) PCBs) were tested.

The pattern of the diamond-profiled tool was covered by the decomposing substrate material, thus the chip/burr removal from the milling slots was insufficient, the tool was significantly heated by friction after a very short milling distance, and the cutting was insufficient. The risk of carbide tool breakage was increased due to the cutting inadequacy caused by the material adhering to the tool (Fig 2.). The material deposited and stuck on the tool is difficult to remove.

Fig 2. Tool with the stuck material from the biodegradable substrate

For milling, spiral profiled tools for IMS substrates were also tried, and the spiral head was slightly better for burr removal, but in this case the tool was stuck with a quantity of the composite that adversely affected the phenomenon.

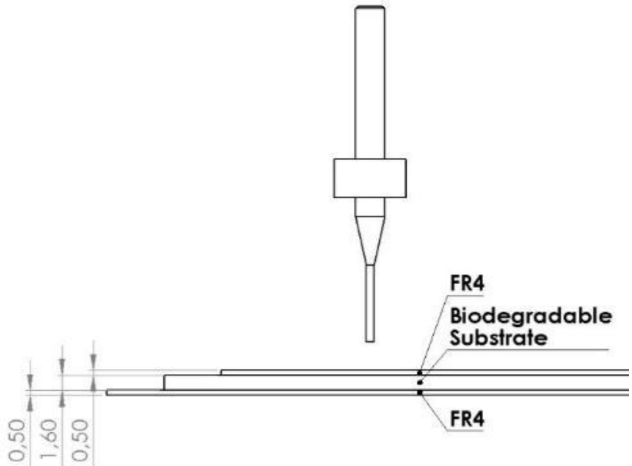

Fig 3. - Sketch of the stack for countour milling

If 0.5mm thick FR4 board is placed under and on top of the biodegradable substrate in a stack, the spiral (helix) profiled mill tools adhered considerably less material (Fig 3.), and the milling surface thus obtained was not too far from optimal.

C. Photomasking

The substrate is under continuous development, with trials of different impregnators. The leak of them can cause poor adhesion to the photoresist mask. This can lead to short circuits and tears during production. The picture (Fig 4.) shows that after the mask layer is applied, the non-plated holes in the substrate material leak impregnators which inhibit the removal of the copper.

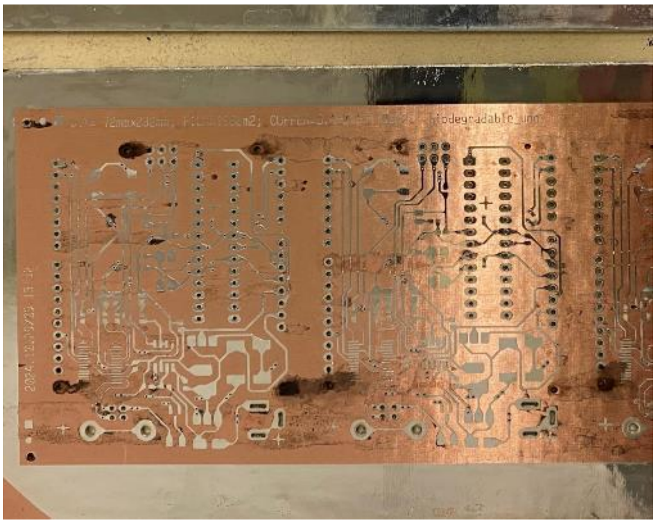

Fig 4. The leaking contamination originated from impregnators

D. *Soldermask burning*

In the experiments, we used SunChemical XV501T type solder masks, typical for FR4-based PCBs. The original burn-in of 150 °C/1 hour is non-acceptable for bio-substrates, as long exposure can lead to deformation, shrinkage, warping. Therefore, the burn in is done at 110 degrees, which is gentler on the substrate, but not perfect for coatings. The anti-solder mask does not cure perfectly, so it can blend with the silk layer applied. (Fig 5.)

Fig 5. The uneven contour is clearly visible

III. PCB ASSEMBLY: MOTIVATION FOR ENHANCEMENTS

A. *PCB issues*

If the surface is uneven, it can cause mask adhesion problems, which leads to short circuits and opens in the circuit. Examples of these can be seen in the pictures. (Fig 6. and 7.)

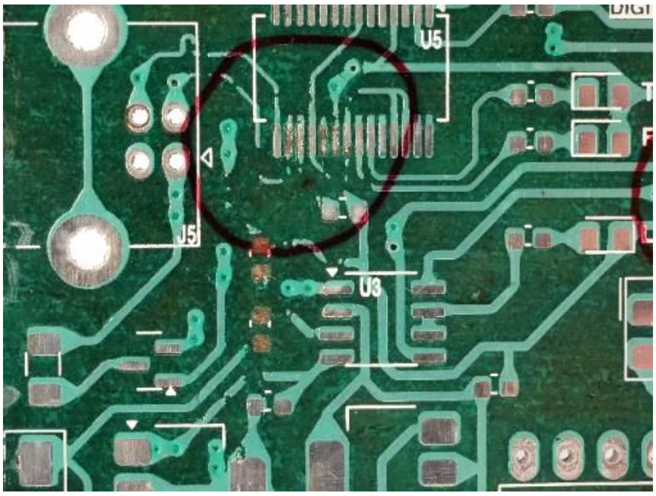

Fig 6. Opens in the wiring due to insufficient masking

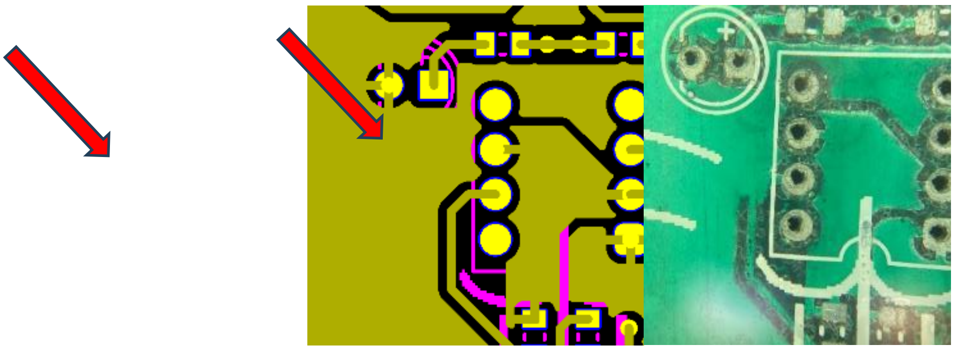

Fig 7. Short circuits in the wiring due to insufficient masking

Two problems may occur due to warpage and shrinkage. Milling and drilling can be imprecise, and the hole plating cannot be properly formed or is damaged, so the two sides are no longer connected and the circuit is inoperable. This is shown in Figure 8 for an in-between experimental generation of the material.

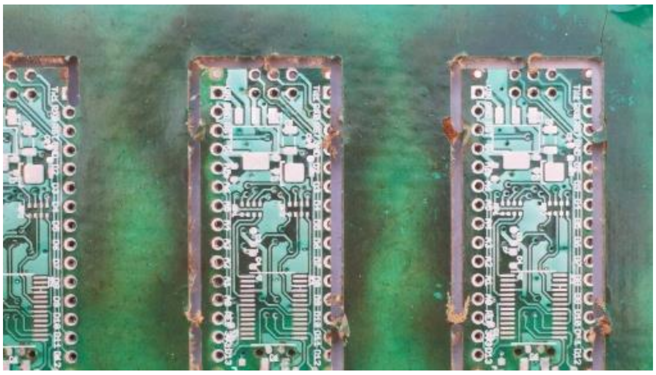

Fig 8. Routing problem, because of the shrinkage

B. Located issues during the assembly

Traditional solder (SAC305) with high melting point alloys is not suitable for biodegradables, so careful soldering optimizations are needed.

The solution is to use tin-bismuth (Sn-Bi) alloy with a melting point of 138 °C. Another problem is that the Arduino Nano design is double-sided and when soldering the first side, if the profiles are not optimized, the substrate may warp, making it difficult to stencil print the second side.

The leakage of impregnators might also be an issue during soldering. With rapid heating, specific types may leak to the surface. (This aspect is currently researched with alternate impregnators). If the design is defined as non-solder mask defined, the cutout of the solder mask is larger than the solder pad, leaving a gap, which can be an issue in these situations. The contamination is shown in Figure 10.

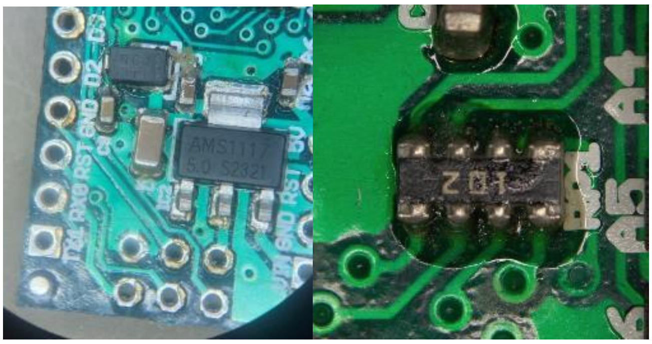

Fig 10. Contaminations of impregnators (due to insufficient dessication of impregnators and rapid soldering).

IV. Assisting production by design

The issues described earlier are originated from the experimental substrate, and while the composite of the substrate is actively researched and improved generation to generation, the design and manufacturing may also follow the requirements in-between.

A. Short-circuits and breaks, vias

To reduce short circuits and breaks, the width of the trace and the clearance between the trace and the insulation should be increased. In the design of the presented Arduino Uno design, we rerouted traces with 0.5 mm trace widht, and the clearance was set to be 0.4 mm. The only deviation from these values is for the SSOP package where it is necessary to choose smaller values. For this part, the pad width is 0.4 mm and the clearance is 0.25 mm. Open cirucits caused by rarely found faulty vias can be reduced with redundant vias (Fig. 11) on all vias.

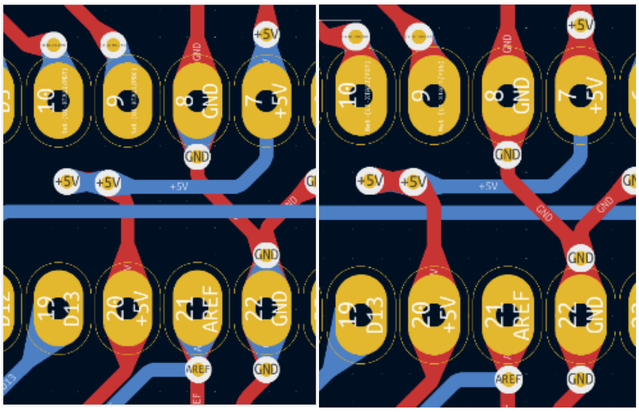

Fig. 11. The double vias are connected on both sides; Red - Top side, Blue - Bottom side

B. Impregnator leakage

Figure 4 showed that during the masking process the linseed oil contaminates the surface. The source of this is holes that do not have hole plating, leaving an open surface for the oil to pass through. The solution is straightforward: metallized holes should be used in the designs. To solve contamination leakage is to use a solder mask defined design [5]. In this case, the cutout on the solder mask is smaller than the copper pad underneath, eliminating the gap.

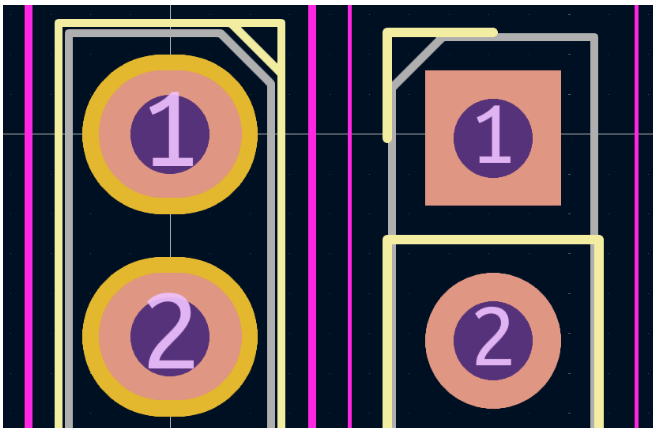

Fig 12. New THD footprint

The left part of Figure 12 shows the newly created footprint for THD parts, the right part the original footprint. The purple colour represents the solder mask, the yellow is the copper layer. The net soldering area is the same in both cases with the copper layer increased underneath. In this case there is 0.2 mm undercut in each direction, and so the clearance between the two pads is 0.54 mm.

When redesigning the Uno, the goal was to create rules applicable up to at least size 0603. So, we started by converting parts with this size code first. Here the design was not so straightforward. The goal was to find a middle point of two values: to have as much copper as possible hanging under the mask, but to leave enough isolation distance between the two pads. So, the undercut was 0.1 mm, and the insulation distance was 0.45mm.

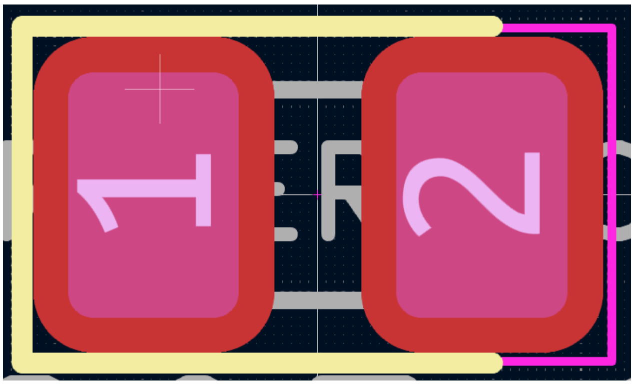

Fig 13. 0805 size LED; purple – solder mask, red - copper

The 0805 size now allowed to return to 0.2 mm undercut and also gave an insulation gap of 0.5mm. This is virtually identical to that used for THD components. From size 0805 (Fig. 13) upwards these rules can be easily adhered to, and in the case of 0603 it was found during manufacture that the values given were adequate. In the future, smaller components might also be implemented as one main goal is to improve surface quality, improving tracing and footprint dimensions.

The SSOP-28 packaged USB to serial converter (FT232R) presented a challenge. (Shown in Figure 14.)

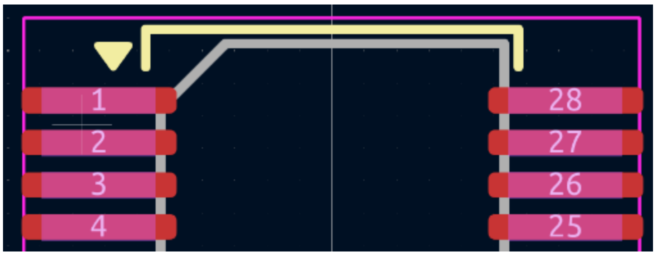

Fig 14. SSOP package; purple – solder mask, red – copper

For this component, the distance between the pins is 0.65 mm, the pad width in the original footprint is 0.45 mm. Thus, the clearance distance is 0.2 mm. While the distance is small, the pad size should be increased further along the SMD directions to hang below the mask, which is not feasible. For the implementation we reduced the width of the pad to 0.4 mm. This increased the insulation distance to 0.25 mm. This is a component which shows, that by the physical implementation, the dimensions do not allow for more design optimization. I have also increased the length of the longer side of the pad under the mask, so that adhesion of the component would also improve.

V. Production based on the new design

Based on the new design, three panelized lots were produced (Fig 15). For the assembly, we used reflow soldering, which requires stencil printing (Fig. 16) with the previously mentioned tin-bismuth alloy type.

Fig 15. Panelized PCB frames based on the new design

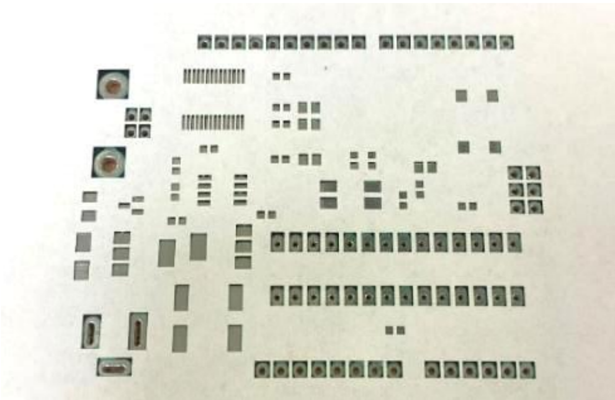

Fig. 16. Stencil printing with proper alignment.

The reflow was carried out in a vapour phase reflow machine (ASSCON QUICKY 450 [7] unit). The heat transfer medium in this was set by a low boiling point D02TS (165 °C) [8] Galden PFPE (perfluorpolyether). During vapour phase soldering, the vapour of the fluid fills the bottom of the tank. It is into this vapour space that the substrate is immersed, where the condensation from the vapour space will perform the heat transfer. This results in a highly controlled soldering process, as no point on the panel can be heated above boiling (as opposed to infra soldering, where colour and surface properties determine heat transfer). Since we want to solder the biodegradable substrate as gently as possible, vapour phase soldering is a perfect choice. Soldering a TH (through hole) component with solder paste can be complicated. A solution is that paste can be applied with the end of the tweezers (manually) and then melted with a soldering iron. Not all of the paste will be applied to the meniscus that is formed, so there will be solder balls left on the panel afterwards, which is usually not acceptable.

Another solution is to use a low-temperature or conventional solder wire (SAC305) with its melting point of 217 °C. In this case, experience has shown that you need to set the soldering iron to a minimum of 50-80 °C above melting point, not favourable with biodegradable substrates. In this case the joint has to be made very quickly, there is no guarantee that a hole filling will be formed at all. This is also suggesting that redundant vias are needed for through-hole components.

To avoid these complications, we applied Pin-in-Paste (PiP) technology [9, 10]. This way, paste is also inserted into the holes of the THD parts during printing and component placement. An added advantage is that the substrate is not subjected to extra heat. It has to be noted, that not all parts were acquired for PiP technology so the easiest way to ensure that they could be soldered even with reduced vapour phase reflow temperatures was doing a test. The chosen DC power connector can no longer withstand this temperature, so it was soldered eventually with hand-soldering method. The TH components are shown below in Figure 17.

Results

The assembled circuits were immediately functioning, meaning a 100% yield. We prepared a test circuit (Fig. 18) on which the circuits can be placed to determine the correct operation of all outputs. The test program switches the LEDs one by one as a running light. If all LEDs light up correctly, the circuit is correct. If any LED fail then we have an open or if more than one LED lights up at the same time then we have a short circuit between two pins. All the outputs are working perfectly for all four UNOs, so the new design rules are proving effective. The soldering was found to be clean and adequate, no leakage of impregnators was found, soldering of the hole-mounted parts with pin-in-paste technology is proved to be efficient.

Fig. 18. Simple test circuit for the Biodegradable Uno

VI. Conclusion

We have to conclude that PCB manufacturing and assembly is non straightforward task on novel materials, such as biodegradable substrates. As for flexible substrates, biodegradables will need new design aspects and manufacturing considerations. In this paper we systematized our findings and showed solutions by design and proper process choices.

We suggested a new setup for drilling and contour milling, we suggested improvement on via redundancy and footprint design. We suggested reflow soldering with VPS, along with Pin-and-Paste technology. Summing up, we created new design rules for manufacturability along the experiences from manufacturing.

Finally, we achieved 100% yield. Figure 20 shows one of four fully functional boards on PLA/Flax substrates where smallest copper feature was 0.4 mm wide, and the insulation (the track distance) was 0.25 mm.

Figures 19-21 present the final results.

In the following we plan to test systematic designs on different widths of traces and clearances on the next generation of improved substrates. Further optimizations in drilling are also in our focus.

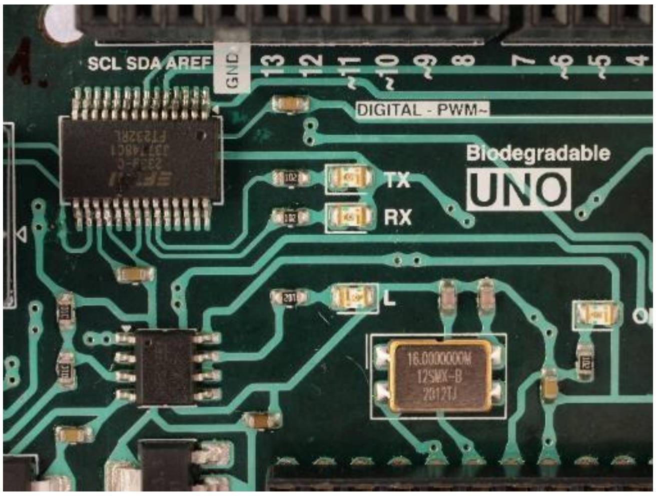

Fig. 19. A finished assembly – focus on soldering and positioning

Fig. 20. A finished assembly

ACKNOWLEDGEMENT

Funded by the Research Fellowship Programme (Code: EKÖP-24-2-BME-215) of Ministry of Culture and Innovation of Hungary from the National Fund for Research, Development and Innovation. This research is funded by the DESIRE4EU HORIZON-EIC-2023-PATHFINDERCHALLENGES-01-04 Project No. 101161251.

REFERENCES

[1] Soon, C.F., Yee, S.K., Nordin, A.N. et al. Advancements in Biodegradable Printed Circuit Boards: Review of Material Properties, Fabrication Methods, Applications and Challenges. Int. J. Precis. Eng. Manuf. 25, 1925–1954 (2024). https://doi.org/10.1007/s12541-024-01027-2

[2] K. Dušek, D. Koc, P. Veselý, D. Froš, A. Géczy, Adv. Sustain. Syst. 2024, 2400518.

[3] A. Géczy, C. Farkas, R. Kovács, D. Froš, P. Veselý and A. Bonyár, "Biodegradable and Nanocomposite Materials as Printed Circuit Substrates: A Mini-Review," in IEEE Open Journal of Nanotechnology, vol. 3, pp. 182-190, 2022, doi: 10.1109/OJNANO.2022.3221273

[4] Attila Géczy et al 2024 Nanotechnology 35 435201: Implementation of microcontroller board on a sustainable and degradable PLA/flax composite substrate: a case study

[5] Joseph Fjelstad: (2006) Flexible Circuit Technology, Third Edition

[6] D. J. C. Ruiz, A. Géczy, V. Grennerat and P. Xavier, "Tiny, Distributed, and Eco-Optimized: Proposal of Design Guidelines for Environmentally Friendly ML Devices," 2024 IEEE/ACM Symposium on Edge Computing (SEC), Rome, Italy, 2024, pp. 443-448, doi: 10.1109/SEC62691.2024.00050.

[7] Asscon Quicky 450 VP datasheet: https://www.pmtech.co.uk/pdfs/TDS/Equipment/asscon/q450.pdf (date of access: 2025.05.01.)

[8] D02TS type Galden liquid: https://www.thermalfluidshub.com/product/galden-d02-ts-testing-fluid-5kg-container-d02ts (date of access: 2025.05.01.)

[9] Martinek, P. and Krammer, O. (2018), "Optimising pin-in-paste technology using gradient boosted decision trees", Soldering & Surface Mount Technology, Vol. 30 No. 3, pp. 164-170.

[10] Krammer, O., Varga, B. and Dušek, K. (2017), "New method for determining correction factors for pin-in-paste solder volumes", Soldering & Surface Mount Technology, Vol. 29 No. 1, pp. 2-9.