

A Guide To Automating Layout Of Planar Magnetic Designs

by Alfonso Martínez, AutoPlanar, Madrid, Spain

Though planar magnetics have been around for years, the task of routing the windings for a planar magnetic component has largely remained a manual and tedious task. Reliance on manual routing of planar magnetics has a few disadvantages. In addition to being time consuming, it also entails a barrier to minor adjustments and improvements, as even minor changes imply a large or complete redesign of the PCB, which slows down product development and decreases final quality.

The reader may wonder, if this is such a tedious problem, why has it not been solved yet? Planar magnetics has been around for decades, as well as PCB design software. Well, a few attempts were made, first by Eagle CAD in the form a spiral-making script, and secondly by Nordcad, in the form an OrCAD plugin, already discontinued as far as the author knows.

As to why this task was not ultimately accomplished in the past, I don't know the real reason, but I can guess two possible causes. First of all, up to until recently, when KiCAD fully ported all their functionalities into Python, there was not an EDA/CAD software with scripting support that really allowed designers to work fully by code. So that was an obstacle.

And then based on my experience in power electronics, few engineers that have the knowledge to design planar magnetics (or power electronics, in general) have the knowledge or the inclination necessary to automate this process by software.

To overcome these problems, I developed a free online tool called AutoPlanar, which automates the process of laying out windings for a planar transformer or inductor (see the reference). This tool includes a wizard function to walk new users (and those less experienced in planar magnetics layout) through the process of routing a new magnetic design. Auto Planar also offers a design option for advanced users, which provides more freedom in editing the layer distribution.

In order to automate layout, this tool imposes a set of design rules with respect to placement of serial and parallel windings across single or multiple layers, and the use of layers and vias for interconnecting these windings, and adding I/O connectors. In this article, I explain the reasons for these design rules, and then walk the reader through the process of laying out a planar magnetic component using AutoPlanar's PCB Wizard. After doing this, I then explain how to create a new design or edit an existing one using the feature for advanced users, the PCB Advanced Creator.

Finally, in the latter sections of this article I explain the simple steps needed to process, visualize and download the complete magnetic design in the form of Gerber files or a KiCAD 6 project file. But before explaining AutoPlanar's design rules and how to use the tool, I'll explain the origins of AutoPlanar.

A Little Background And History

This story begins with a tale of two disappointments. The first one involved the EAGLE software. I was introduced to this software when it was still owned by CadSoft Computer. EAGLE came with a freemium perpetual license for hobbyists using small-medium boards, which happily was enough in my case.

But more importantly, my experience with the software taught me how intuitive it was (in contrast with OrCad—shame on your I-O keys for zooming) and how visually nice. So there I was, burning midnight oil after midnight oil manually routing boards for my wonderful and I-will-totally-come-back-to-them projects, loving every minute of EAGLE.

And then the first disappointment came: Autodesk bought EAGLE, and what was a huge DIY hobbyists' community suddenly became the victim of double-speak. What was "freemium" became exclusive and the great exodus away from EAGLE commenced, mostly to KiCAD.

In the meantime KiCAD grew from a collection of PCB tools to a proper software, most of the effort done by the open-source community, but with a huge support from CERN, who saw this for what it was: the best chance for hardware to achieve the openness and universality of software.

And so, I became infatuated with KiCAD. The learning curve of the graphical tool was not so steep, as EAGLE was somehow similar. But the feature that really captured my heart was the Python API. For those who are not familiar with the implication of this last sentence, it means that you can practically automate anything you could in the graphical tool using Python. And anything means anything, so yes, planar magnetics are in the game.

But I am getting ahead of myself. At some point in time the second disappointment came. I used to visit a certain Facebook community where there is a decent amount of conversation about power electronics, and one day a post made my productivity-sense tingle.

A poor guy was asking about a capacitance problem in his latest planar project, and felt that uploading a global view of the planar winding would help people help him. I didn't really have an answer for him, but his board was colorful and it attracted my attention. Fatally. I immediately focused on the monstrosity in Fig. 1.



Fig. 1. Oh, the humanity, this engineer had to route four too many tracks by hand.

After the first moment of dumbfoundedness, the question came: why? My guess is the guy was probably fed up with routing turns after who knows how many iterations trying to solve his parasitics problem; and he just didn't care enough (though I wonder if his client did). However, the real question was: why did he have to route it manually? That is an easily automated task (it's just geometry operations), so it is really nothing.

This second disappointment over the lack of automated routing for planars drove my KiCAD infatuation and provided me with a problem to solve. And solve it I did by developing the AutoPlanar tool.

Before we start into discussing AutoPlanar, I need to clarify something, as there is a question that tends to be asked and generates some kind of engineering turmoil: No, AutoPlanar does not use artificial intelligence of any kind for anything, just old-fashion geometry. The tool is based on the idea that any good planar magnetic can be pretty well defined by a list of layers, each with a specific number of turns belonging to a given winding.

At its origin, the tool was intended to accelerate the tedious first steps of laying out a planar winding by creating the spiral coils needed, giving the user the task of adding the vias and finalizing the board. But as development was done, it became clear that a whole tentative board design could be created, including the connection of the layers with vias and the placement of user-selected connectors.

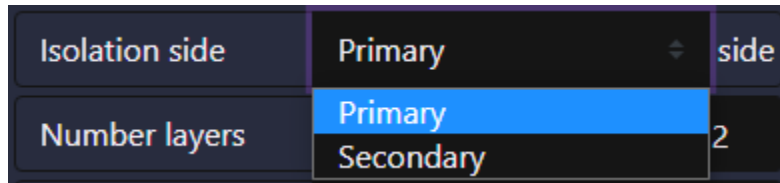
To fully automate the process, the user must introduce a sufficient number of input parameters to allow the algorithm to unequivocally select which turn goes into each layer and how they are connected. To accomplish this, certain constraints were added to reduce the number of input parameters while offering enough flexibility to cover all of the common cases. For the special cases out there the project file is offered as an output, so any necessary changes can be made, but without having to start from scratch.

Design Criteria For Automated Layout

Before getting into the explanation of the software itself, I would like to get into the constraints with which the tool has been designed and the reasons behind them. I could say that I have done and analyzed thousands of planar designs and that you should trust me when my software implements a certain option over another one, but my pride as a non-sales engineer forces me to share the reasons behind these decisions, so any customer can judge for themselves if they are worthy of their time.

Isolation Sides

Although the software supports up to ten windings, it only supports two isolation sides, primary (left) and secondary (right), as these two groups are used to place the connectors at either side of the magnetic core (Fig. 2). This was done to facilitate the placing of the connectors and avoid possible collision with the magnetic core, as the connectors are placed on the outermost points in the width dimension of the board, while the magnetic core is placed along the height dimension.



Isolation side	Primary	side
Number layers	Primary	2
	Secondary	

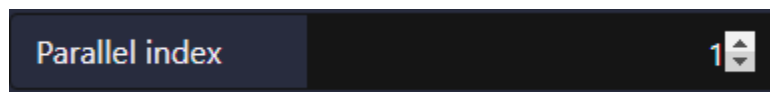
Fig. 2. Screen capture from the AutoPlanar web platform, where the isolation side is picked for a given winding.

One Parallel Per Layer

The first of these design criteria is that only one parallel turn may be present in a layer. We can have as many serial turns as we desire, but two parallels must be placed in at least two layers (Fig. 3). The inverse is not true. One parallel may be distributed in as many layers as it has turns, in any combination.

The reason behind that restriction is that if more parallel turns are desired it is because we want to decrease the current density in the tracks. And while paralleling windings is a common trick in wirewound magnetics, having two parallels would be really inefficient here in a planar device, as we would be losing the track-to-track distances for conducting, as opposed to just widening the existing track.

Another reason would be that we would be increasing the number of faces where the H field would be perpendicular. This would increase the losses due to proximity effect without producing any appreciable improvement in the skin effect distribution.



Parallel index	1
----------------	---

Fig. 3. Screen capture from the web platform, where the parallel index is picked for a given layer. The choices here are limited to just one or two parallel turns per layer.

Because of these reasons, the software limits the user by only allowing the selection of the parallel index that will be routed in the layer, which in turn provides freedom in distributing the serial turns in any combination.

Via Placement

When a serial turn does not start and end in the same layer (which is only true if you had only one turn in your winding), vias are needed to continue the turn in multiple layers (Fig. 4). To automate this, the two types of winding were identified: those with multiple turns in a layer, and those with only one turn per layer in all their layers.

Why is this important? Because the windings with multiple turns per layer will need a connection layer in which the internal end of the track connects with the terminal connector. And in the automatic designs done by the tool, in the case of multilayered serial turns (where one parallel turn is distributed over several layers) this layer is also used to connect the outermost end of the spiral in one layer, to the innermost end of the continuing spiral in the next layer.

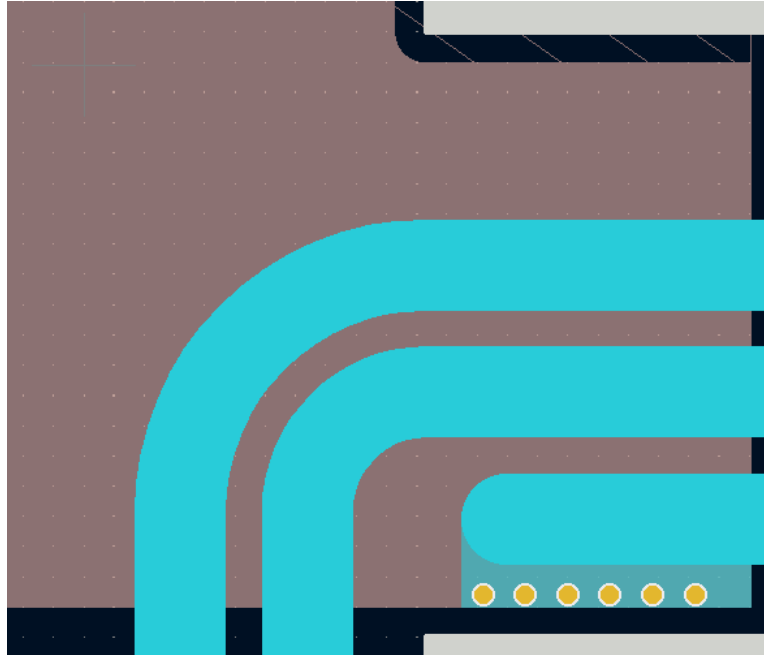


Fig. 4. Connection of innermost end of a multiple turn spiral (blue) to the side using a connection layer (beige).

A connection layer was used in order to simplify all the different cases that would have implied having alternating inward and outward spirals. The way the automatic design is done, all turns in a layer start (or end) at the center, and end (or start) at the outermost radius.

But returning to the topic of where the vias are placed, and totally independent of the reason for the connection layer, we have the condition that multiple-turns-per-layer windings need two vias (or lines of vias, which will be explained later), one at the center innermost radius of the spiral and one at the outermost radius (Fig. 5). Meanwhile, single-turn-per-layer windings need only the innermost vias to continue to the next layer, not blocking the connection to the terminal connector and thus not needing a dedicated connection layer.

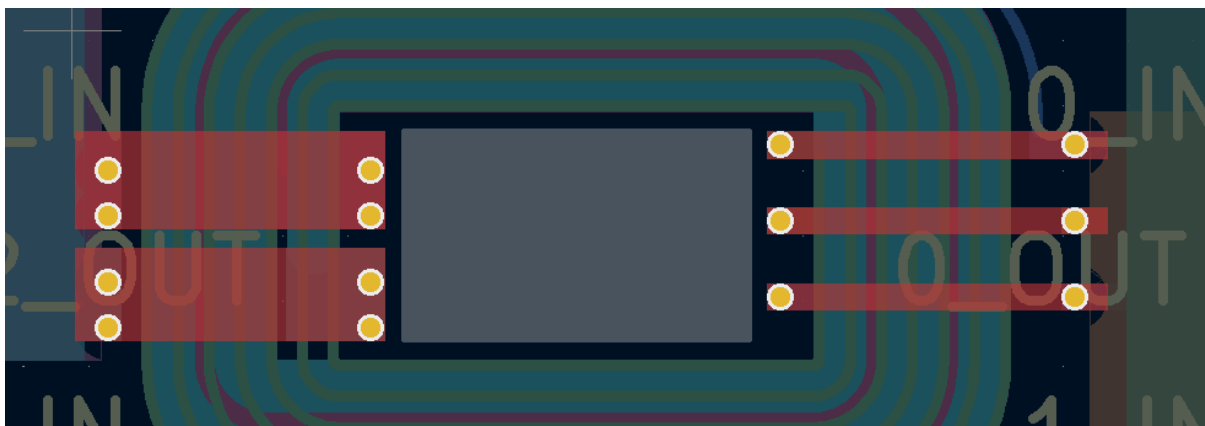


Fig. 5. Connection layer (in red) for different windings, taking the center of the spiral back to the exterior in order to continue the turns.

Independently of needing a connection layer or not, the vias for the windings in the primary isolation layer are placed on the left face of the central column, for square shapes (see Fig. 5 again); and on the left 90° arc in case of a round central column (Fig. 6). Similarly, all the vias from the windings on the secondary isolation side are placed on the right side, following the same pattern.

The space allocated for the vias of each winding is proportional to the thickness of that winding's trace in comparison with the rest: A winding with a thicker track width will need more vias to conduct the current from one layer to another than a winding with a thinner track width. The only exception would be that there is a minimum via width that must always be maintained.

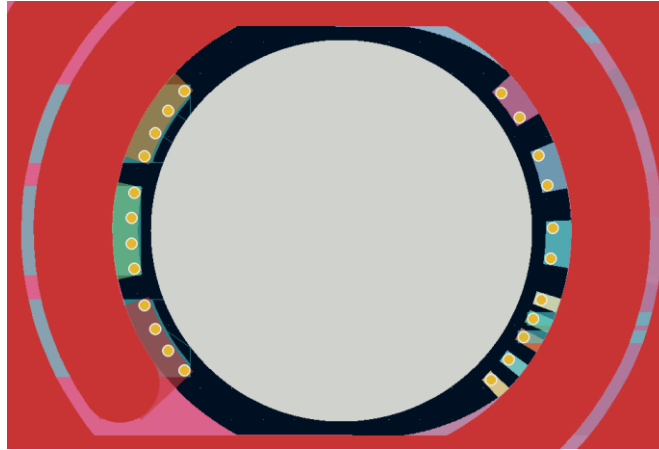


Fig. 6. Vias for primary and secondary side in a round central column. Thicker tracks receive a larger angle for their vias.

Lines Of Vias

As has been explained, the vias for each side are divided in sectors of 90°. These sectors are then projected at a radius equivalent to half width of the central column plus the specified clearance to the core, producing a total arc length. This length would also depend on the shape of the column.

This arc length is then shared among all the turns that need to be connected to different layers, and once the length is available, the total number of vias that each connection will have are calculated, taking into account the specified via diameters and clearances. These lines of vias will be the paths where the current will pass from one layer to another (Fig. 7).

Ideally the diameter of these vias must be as low as possible, while the number of them must be as high as possible in order to maximize the area of copper that the current has available when crossing through the vias. In reality, this number is constrained by the PCB manufacturer's capabilities, and it is the one reason why these parameters are inputs to the design process of an automatic planar transformer.

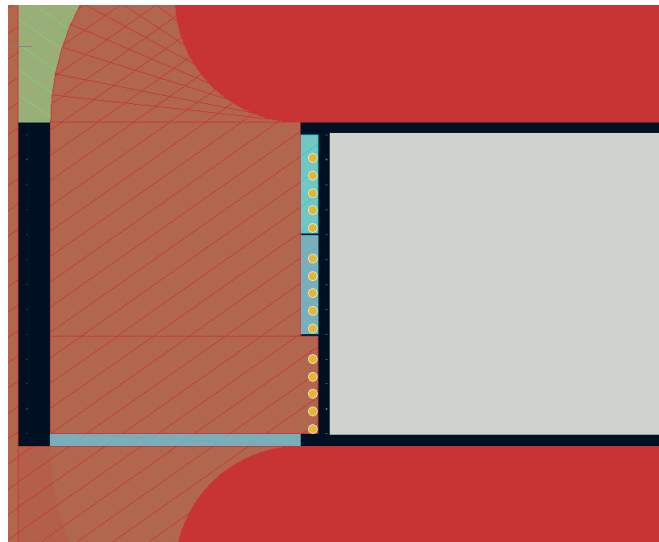


Fig. 7. Line of vias for multiple windings, keeping the clearance from the core, other vias and tracks.

Right now only one line of vias is supported in the tool, but adding a second one placed orthocyclic (at a 60° angle) to the first one is on the roadmap of the tool. More than that would be inefficient, as it would imply increasing the total turn length just to have a line of vias with a resistance several times higher than the others. That's because not only the current must go through a longer path, but also a narrower one due to the small space left between the orthocyclic vias in the first line.

Connectors

In the current version of the tool four types of connectors are implemented. However, the way the software architecture was designed took into account any possible connector, so additional types can be added to the library with minimal effort. And as mentioned before, the user has the final design available to be altered in any desired way.

If we review these four connectors in order of complexity, the first one is the classic and timeless banana connector, where there is no customize option (Fig. 8a). The second connector available to the users of AutoPlanar is the pin, with pin diameters that range from a pitch of 0.7 mm to 1.4 mm (Fig. 8b).

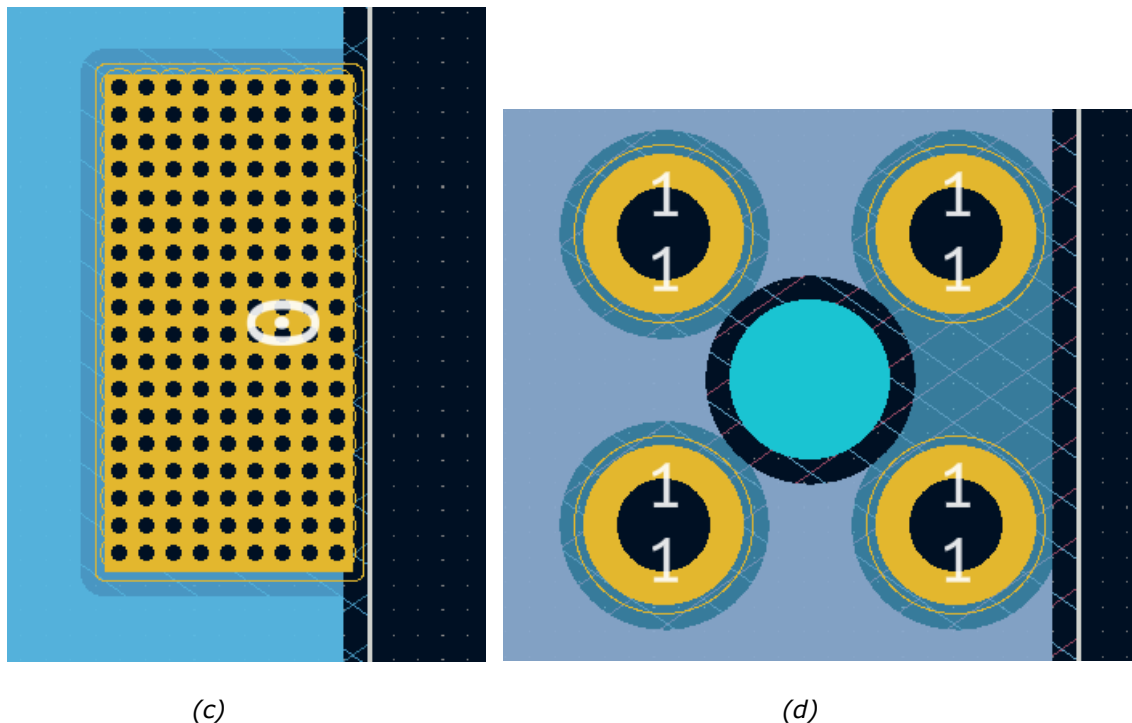
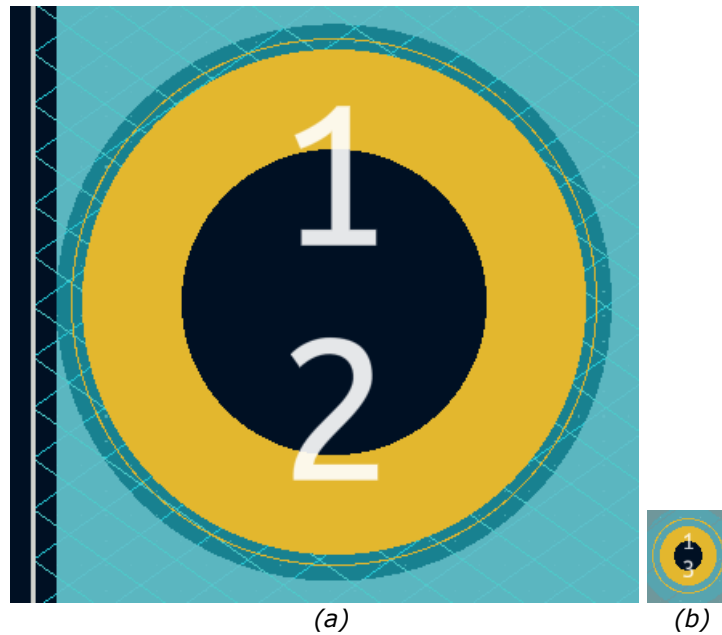


Fig. 8. The four connector types offered in Auto Planar. Images for the connectors are shown to scale relative to each other. Depicted here are the banana connector (a), the pin (b), the pad connector (5 by 10 mm in this case) (c) and the Wurth connector (of metric 3) (d).

The next connector is an open-square pad where anything can be soldered, and that is defined by its width and height (Fig. 8c). The pad is additionally populated with as many vias as fit in the area, in order to maximize the current transfer to all layers.

The fourth and last connector is the whole Würth Redcube series (Fig. 8c). For this connector we have three configurable parameters: the metric of the screw, the choice of screw connection (male or female), and finally the choice of screw socket (blind or through hole).

Designing With AutoPlanar

We have gone over the criteria used when designing this tool, but we haven't seen it yet! Fear not, this section will show you the two different ways you can create your automatic planar.

First Method: The PCB Wizard

We begin with the simplest method, designed as an onboarding process for those not so familiar with the tool. It consists of a series of guided steps that take the user through the definition of the board and windings, and suggest a layer structure based on maximizing the interleaving of layers. The suggested structure can be edited later using the second method.

The first time we access autoplanar.com, it will ask us if we want to use the wizard to create our first magnetic (Fig. 9). If we accept, we will be taken to the PCB Wizard (if not we can always go again by clicking on the namesake button on the top menu).

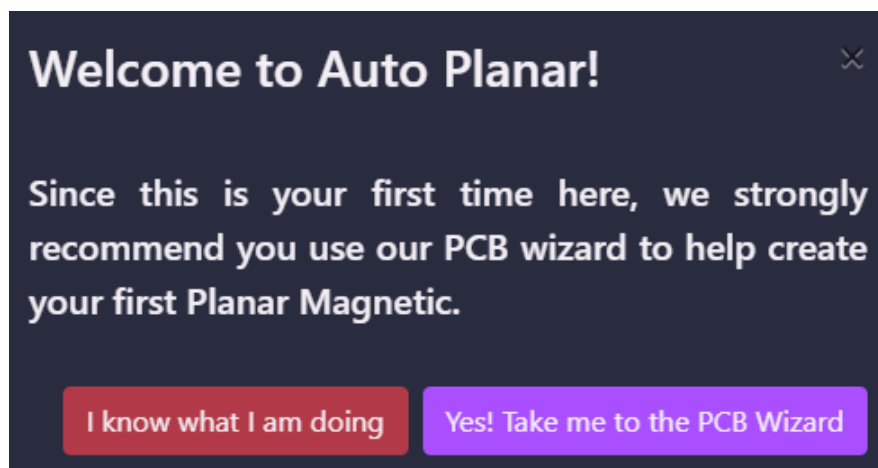


Fig. 9. AutoPlanar's welcome screen gives users a choice of starting a design on their own or using the PCB Wizard.

The process consists of eight steps, plus two extra per added secondary. A progress bar will inform us at all times of which step we are on, and how much we have left to introduce. At any point we can go further and back in the process with the buttons at the bottom of the page (Fig. 10).

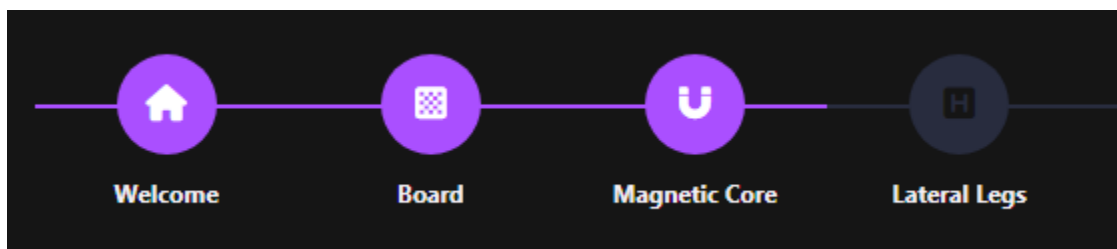


Fig. 10. Buttons at the bottom of the page allow users to jump between steps in the design process.

Step 1. Welcome: This one works as an introduction and asks for the reference name that will be used to identify the design (Fig. 11).

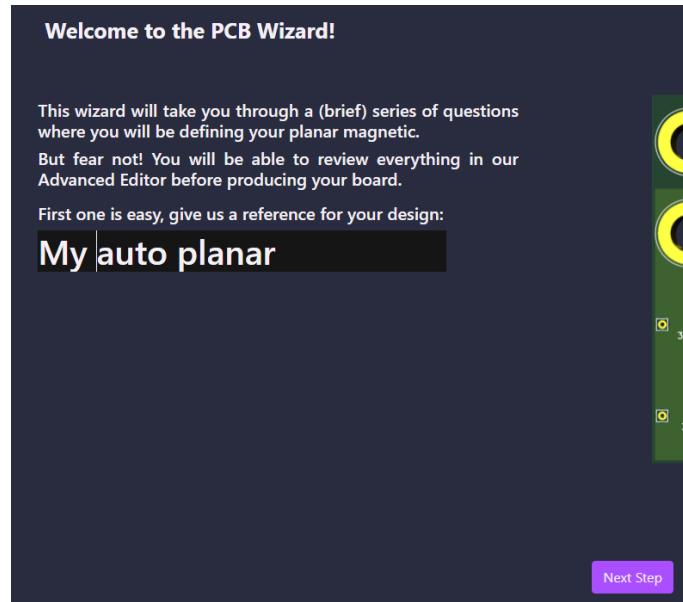


Fig. 11. Naming the design.

Step 2. Board dimensions: In this step we will define the dimensions of the board. At the moment only rectangular boards are implemented (Fig. 12).

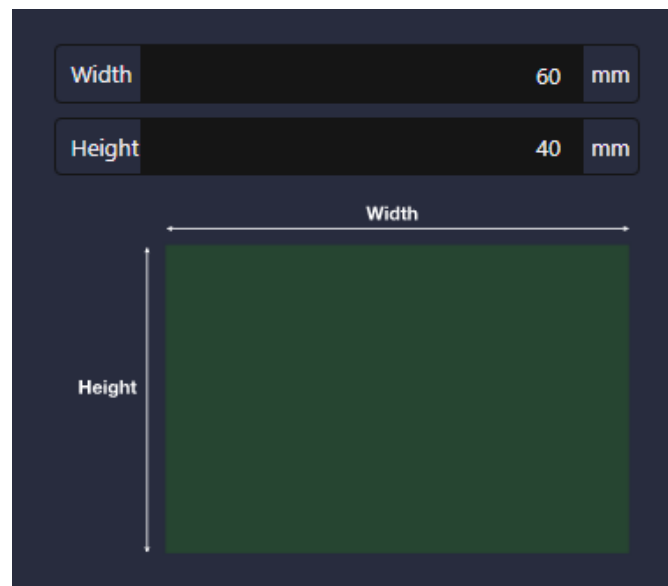


Fig. 12. Specifying board dimensions.

Step 3. Central column dimensions: In this step we will choose between a magnetic core with a round central column or a square one. We also specify its dimensions. This shape will define the future layout of the spirals that comprise our planar (Fig. 13).

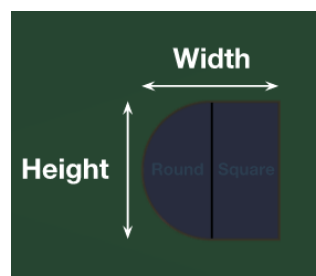


Fig. 13. Specifying core shape and dimensions.

Step 4. Lateral columns: All magnetic cores have one or two columns that are used to facilitate the flux path around our windings, in this way minimizing the leakage inductance and constraining the H field inside the winding window. This step lets the user enable or disable the holes for the lateral legs. In case they are enabled, the user must introduce their dimensions, assuming they are always square, plus the distance between central and lateral columns, which represents the width that is available for turns (Fig. 14).



Fig. 14. Specifying dimensions and position of the lateral columns.

Step 5. Clearances and via diameters: This step is used to introduce the future PCB manufacturer's capabilities regarding the vias, defining the diameters of the drills. It is recommended to have them as small as allowed, because the smaller they are, the more vias will be able to fit the software for connecting different layers, which will improve the current density in them.

In this step, we will also specify the clearances that will mark the distances left between different elements of the PCB. Again, it is recommended to have the minimum value that will comply with the isolation requirements, as overconstraining can produce really inefficient designs (Fig. 15).

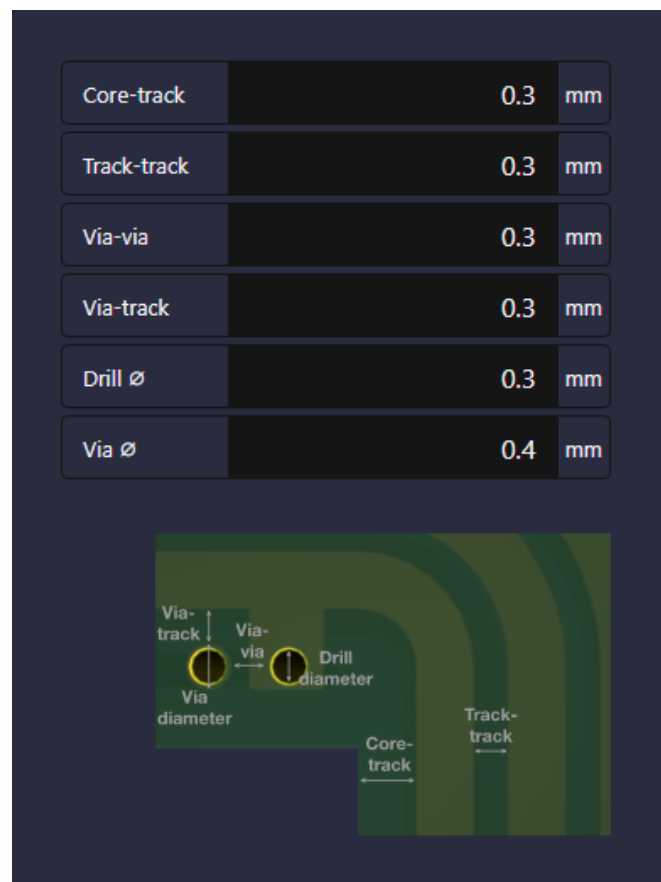


Fig. 15. Specifying clearances and via diameters.

Step 6. Number of windings. This one is really easy, just indicate how many windings are needed in your design. The remaining two steps will be repeated for each winding that you introduce (Fig. 16).

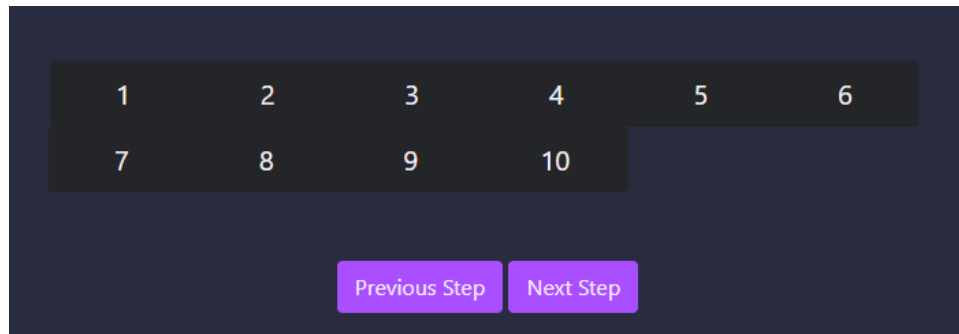


Fig. 16. Specifying the number of windings.

Step 7. Winding definition: In this seventh step we will define the corresponding winding (starting with the primary) by introducing the width and height of the conducting track. Here, height is expressed in microns (μm), not in copper ounces, as in common in PCBs, to allow for the creation of cut strips out of copper foils. In this step, we will also specify the number of turns, the number of parallels and the isolation side (as explained previously) (Fig. 17).

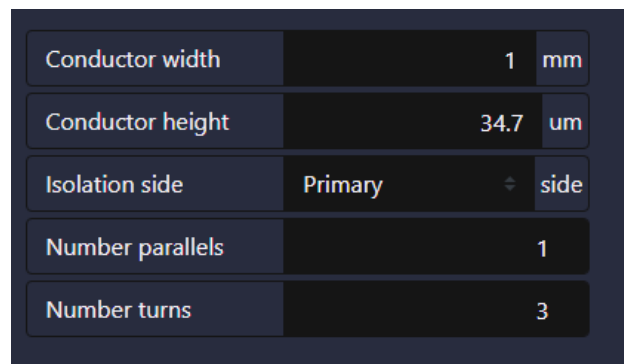


Fig. 17. Defining a winding.

Step 8. Connector definition: In this last (unless you have more windings to define) step, we can choose between the four types of connectors explained in the previous section: banana, pin, pad, and redcube. Depending on the chosen connector the definition will vary.

After finishing all the necessary steps, we can either go back and review any of them, or click on the Submit button, which will request the API for a layer distribution and save the resulting design in the user's dashboard, where he can later edit or process it.

This automatic layering functionality aims to help the newcomers with a simple suggestion of how many layers are needed for their PCB, trying to fit as many turns of the same parallel as possible in each layer, together with a tentative distribution based on interleaving—as much as possible—the layers.

Once the process is finished (which happens really fast, so no processing page will be shown), the users will be taken to their dashboard, where they will see their new design ready to be processed.

Second Method: The PCB Advanced Creator

If the previous method was designed to be introductory—or helpful for people in need of a planar magnetic, but lacking the expertise to decide every little detail themselves—this second method is designed for the advanced users, and allows them to edit the layer distribution in any fashion.

This section can be reached from two places: to start with an empty design the user can click on the button "PCB Advanced Creator" on the top toolbar (Fig. 18).

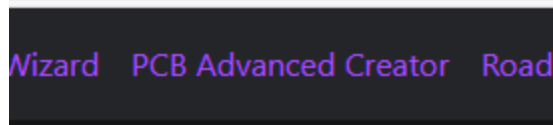


Fig. 18. Button to create a new design—selecting the PCB Advanced Creator via the top toolbar.

The second way would be by editing an existing design from the user's dashboard. In this case the section will be configured with the current design that is being edited (Fig. 19).

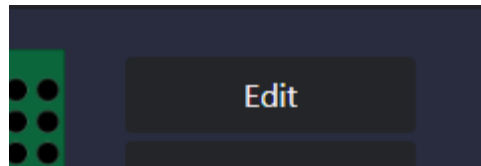


Fig. 19. Button to edit an existing design.

The view that appears on the next screen (Fig. 20) is divided into three columns, with the first two being a compilation of the same information appearing on the PCB Wizard from the previous section (Figs. 10 to 17), the PCB and Windings configuration, the only difference being that they now can be modified directly here, instead of going step by step. And a little detail in each winding: there is a field in this view for the number of layers associated with it.

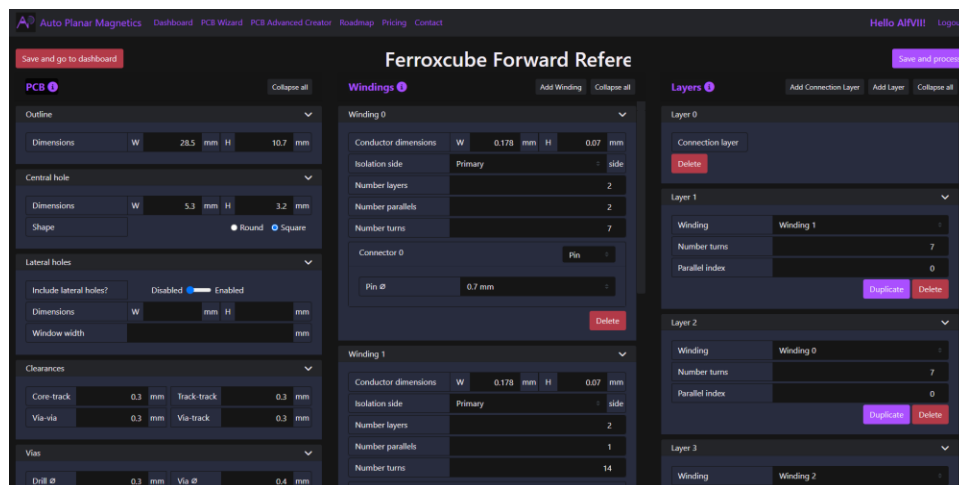


Fig. 20. Whole Advanced PCB Editor. The middle column is shown close-up in Fig. 21.

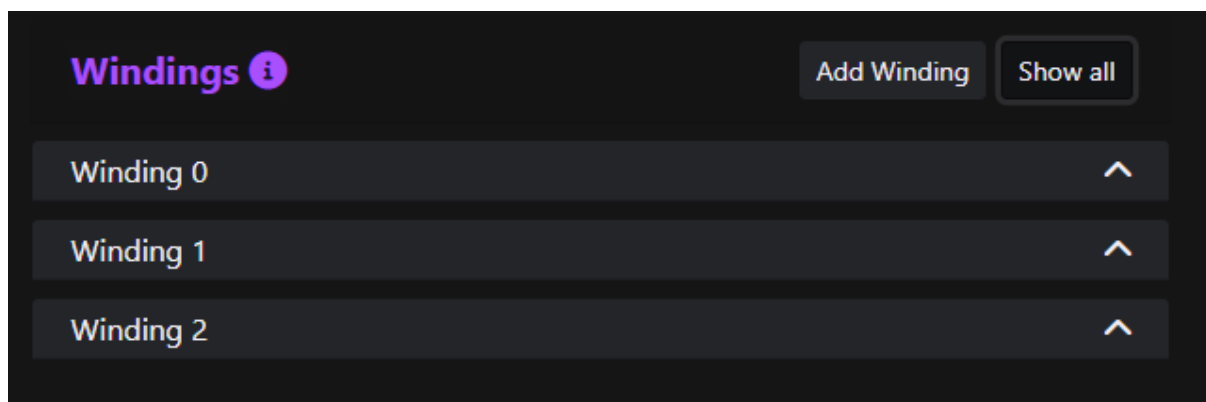


Fig. 21. Winding column for a three-winding PCB with the elements collapsed for better viewing.

The third and new column is the layers. This column consists of a list of elements ordered from top to bottom (Fig. 22). Each one of these elements represents one layer of our soon-to-be-done PCB. To add a new layer, we can create a new one using the provided buttons, or duplicating an existing one; in either case the new layer will be placed at the bottom of the stack.

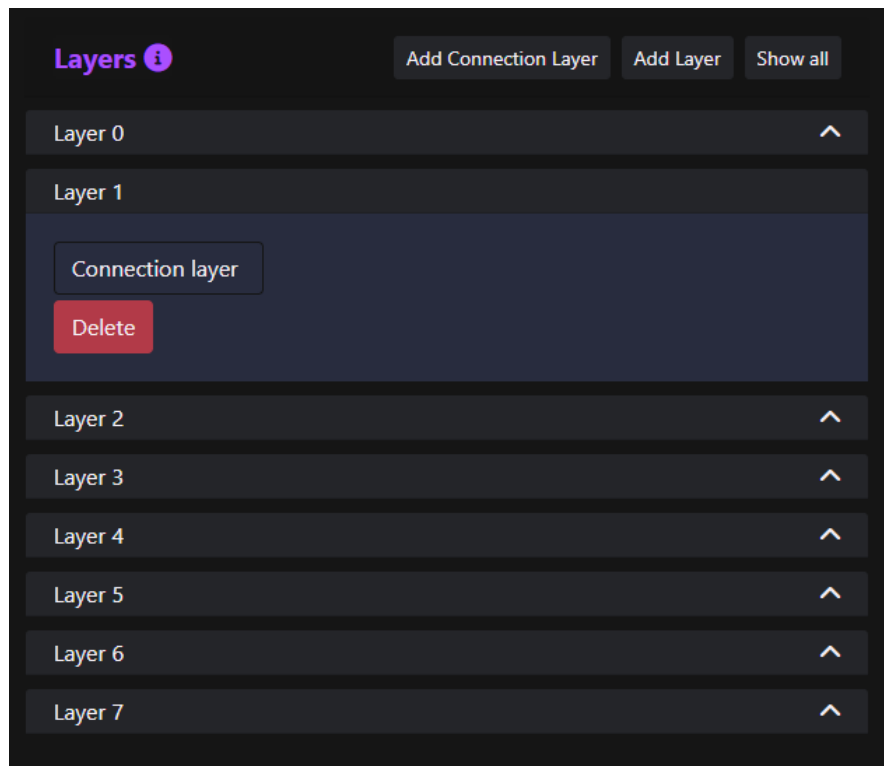


Fig. 22. Layers column for an eight-layer PCB with the elements collapsed for better viewing.

As was mentioned before, the layers can be of two types: winding layers or connections layers (Fig. 23). A winding layer must be associated with one unique winding, and the user must specify the number of turns that specific winding will contain, and to which parallel those turns belong.

Connections layers are used for routing connections between the beginning and ending of all the windings that have more than one turn per layer. As these layers will be used for the parallels, it is possible to define multiple of them in order to keep the current density under control. For more information about the reasons behind these decisions, revisit the Design Criteria section.

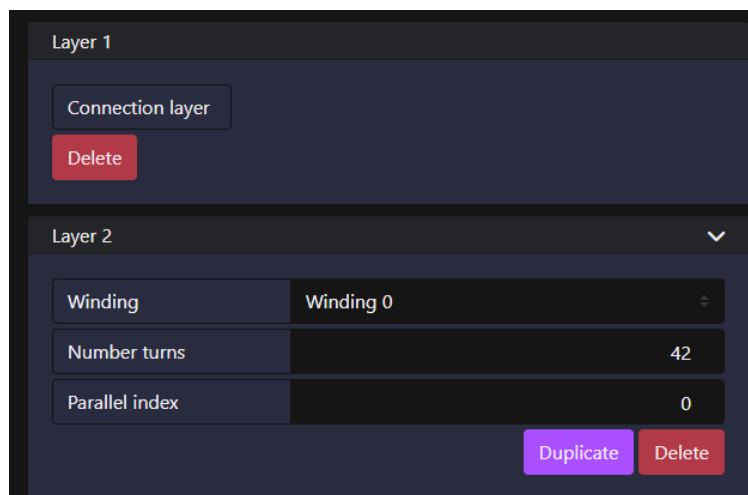


Fig 23. The two types of layers—connections layer (top) and winding layer (bottom).

In order to help the user keep track of the current number of layers, the number of layers that are assigned to a specific winding in the right-most column (Layers) must match the number of layers configured to that winding in its corresponding section in the central column (Windings). Although this seems like an unnecessary burden on the user, it is designed to force a conscious awareness of the layer distribution and avoid any possible error layer, working as a kind of Redundancy Check.

To help in the process of reviewing and verifying a design, an error console is also provided at the bottom of the page. In this console an error will appear for each inconsistency between the defined windings and the configured layers (Fig. 24).

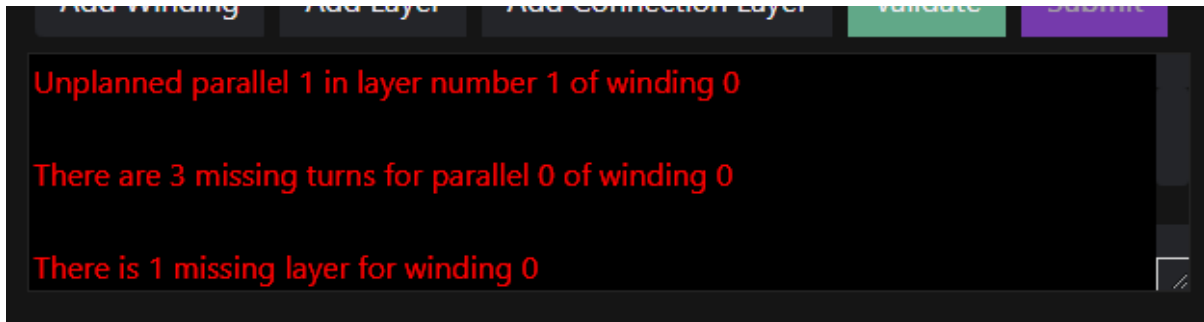


Fig. 24. Error console informing us of several error in our design.

Right now we have gone over all the elements in our PCB Advanced Creator, but not on how to create or delete them. The creation part is controlled by a collection of buttons in the bottom part of the left-most column, on top of the error console. As pictured in Fig. 25, we have four buttons:

- Add Winding: Used for adding a new winding to our design, with the default values.
- Add Layer: Used for adding a winding layer at the bottom of the stack.
- Add Connection Layer: Used for adding a connection layer at the bottom of the stack.
- Update/Create: Used to finish the current design and save it. If the design was new this button would create a new entry in our dashboard. If we were editing an existing design, it would update the design and remove any processed files we would have had associated with it. In both cases the user will be taken to their dashboard. In case of having errors in the console, this button will be disabled.

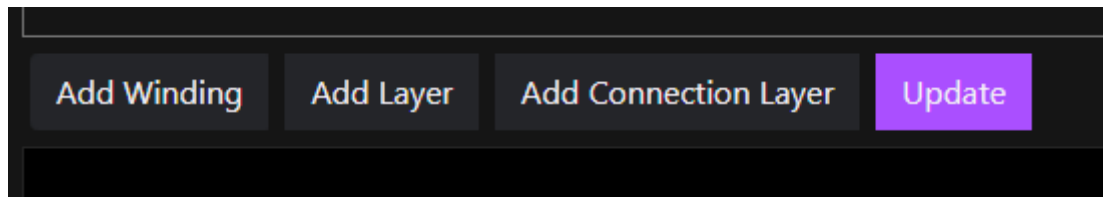


Fig. 25. Available buttons.

Some of the previous buttons are also repeated on top of their corresponding column (Windings or Layers) in order to facilitate the user's experience.

Finally, in case the user has any doubt about the meaning of any of the fields in this section, there are three help buttons placed at the beginning of each column, accessible by clicking on the icon with an "i" inside a circle (Fig. 26).

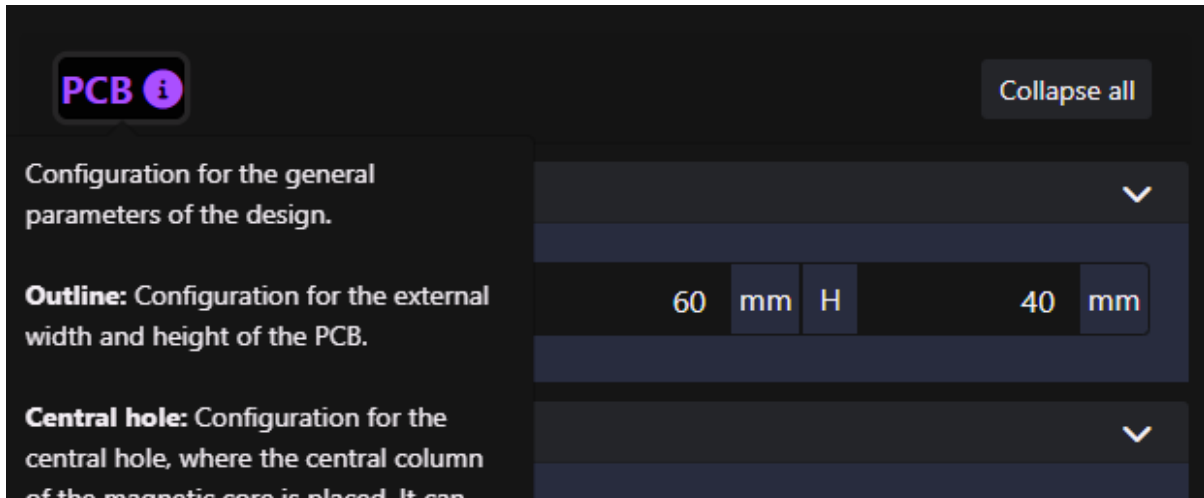


Fig. 26. Unraveled helpbox for the PCB column.

Processing And Getting Your Planar Magnetic

We have covered all the different ways of creating and describing your planar magnetic, and the reason why they are like they are. In this section we will review how to obtain our processed PCB, visualize it and download it.

The Dashboard

The greeting page of any logged user will be their dashboard, containing two example designs by default, one inductor and one transformer. Each design will be represented by a box with all the options available for it, and marked with the reference name given to it by the user and creation date.

A processed design will contain a miniature view of the top layer, and two groups of buttons, one placed to the right of the image and one below (Fig. 27).

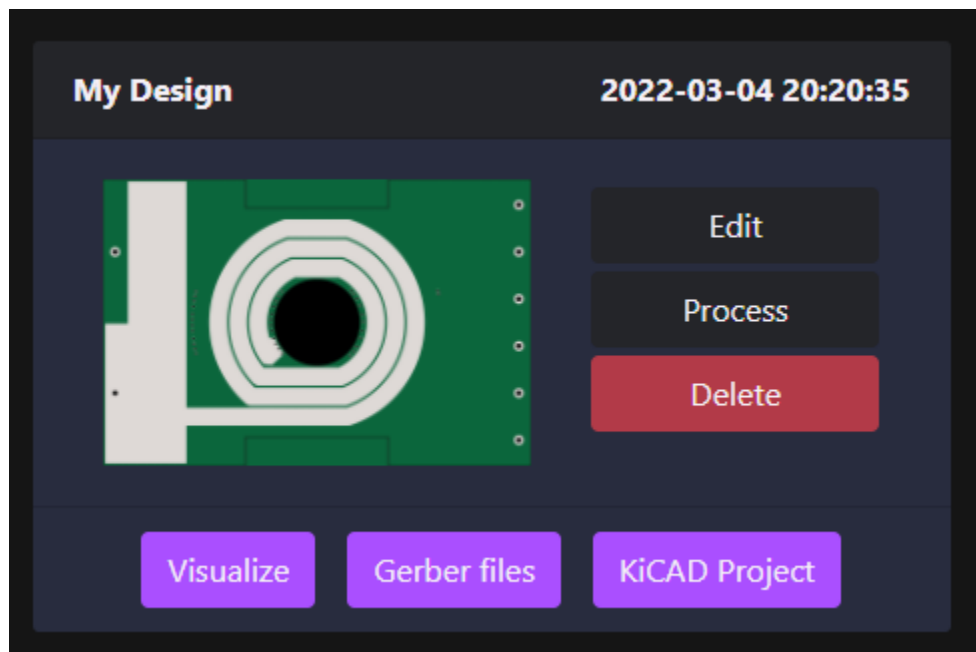


Fig. 27. Processed design in the dashboard.

The buttons in the group to the right of the miniature represent the different actions available to alter that design: edit it, process it and delete it. The buttons in the group below the miniature represent the different actions available for the outputs of the processed design: visualize the layers and download the Gerber files or the full project.

In the case of designs that have not been processed yet, the miniature picture and the buttons associated with the outputs will appear hidden until that design is processed (Fig. 28).

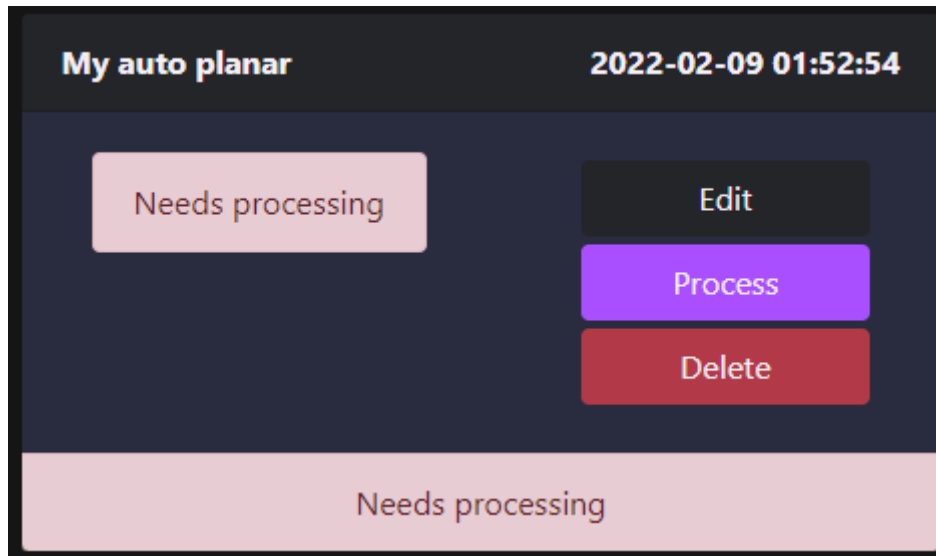


Fig. 28. Unprocessed design in the dashboard.

Processing A Design

Processing a design is quite trivial, we just need to press the process button and the system will read all the design specifications introduced in the previous sections, providing an estimation of the time it will take and launching the process (Fig. 29).

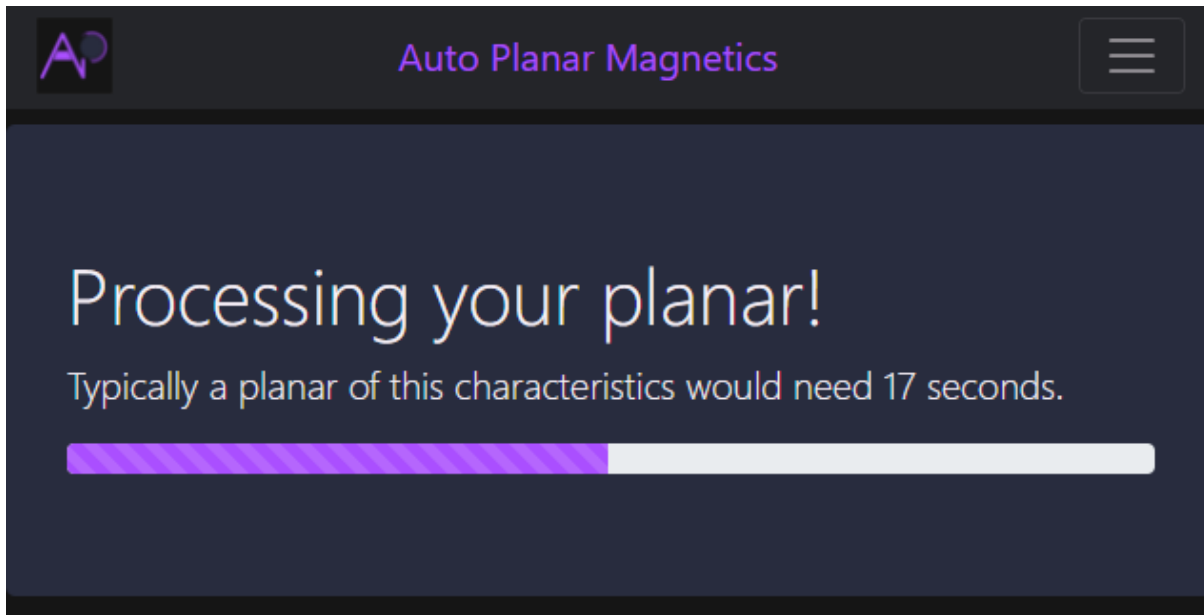


Fig. 29. Progress bar for our magnetic.

As this time estimation is based collectively on the design runtimes recorded for all users' designs, as time goes on and more designs are run, this estimate will become more accurate. Once the design is finished we will be notified and we can choose what to do with our generated design (Fig. 30).

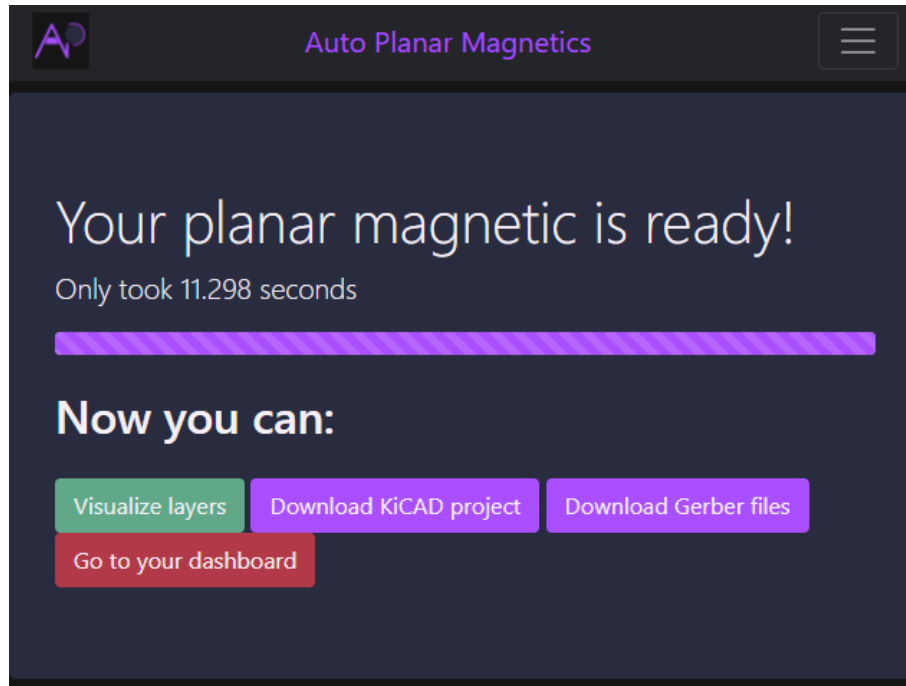


Fig. 30. Our planar magnetic is ready!

Visualising The Design

If we want to take a look at the layers generated by the online tool, we can click on the "Visualize Layer" button at the end of the design or in the corresponding design box in the dashboard. The visualizer is quite simple, consisting of a little tool where the different layers are shown and can be changed for the next or previous layer (Fig. 31). It must be noticed that the last layer, the bottom one, will be shown mirrored.

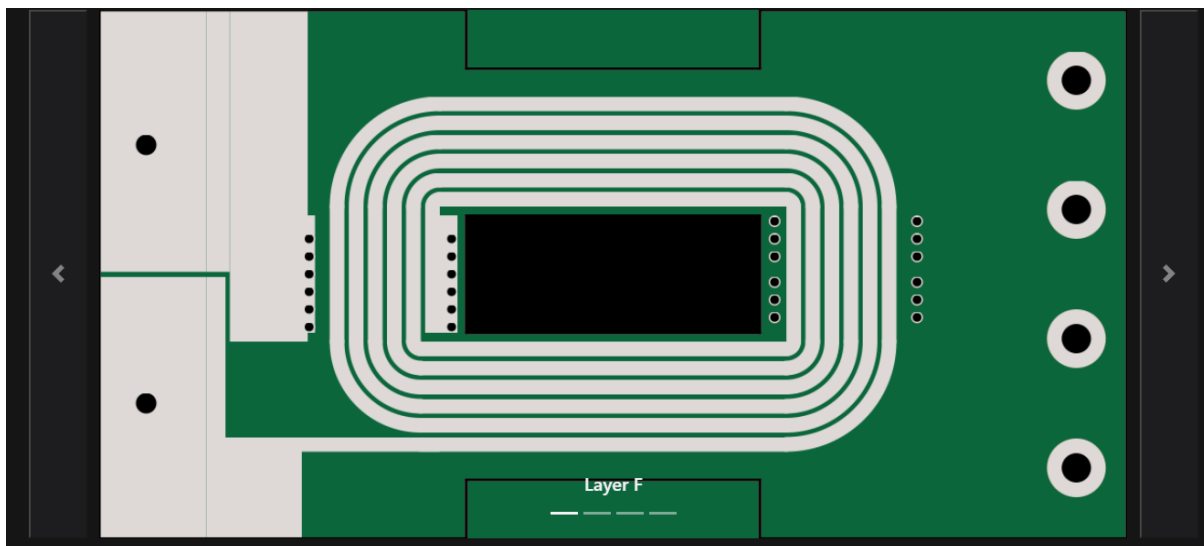


Fig. 31. Layer visualizer.

Downloading It

A processed design can be downloaded from the finalized design or from the corresponding design box in the dashboard. Two formats are supported right now:

- Gerber Files: The Gerber files extracted from the processed design and compressed in a zip, ready to be sent to a PCB manufacturer.

- KiCAD 6 project file: The project file for the latest version of the Open Source KiCAD. This project is totally editable and can be modified by the user as desired. This format is also supported by Altium, in case the user prefers a paid software.

Reference

AutoPlanar can be accessed online at <https://autoplanar.com/>.

About The Author



Alfonso Martínez currently serves as the CTO of Frenetic and is the creator of Auto Planar. He has more than 13 years of technical experience spread between the electronic engineering and computer science fields. In 2018 he co-founded Frenetic in Spain where he has been responsible for developing its AI-based tool for designing magnetic components. In this role he began learning about power electronics and detected an opportunity for automation of planar magnetics layout, which led to his creation of AutoPlanar.

Prior to this, Alfonso cofounded the start-up Swarm64 in Berlin, where he created an FPGA-based architecture to optimize the performance of databases. That company was acquired by ServiceNow. And previously he developed hardware and firmware in the field of the Internet of Things at a startup in Madrid. Alfonso holds an MSc in industrial engineering from the Polytechnic University of Madrid.

For more on magnetics design, see these How2Power Design Guide search [results](#).