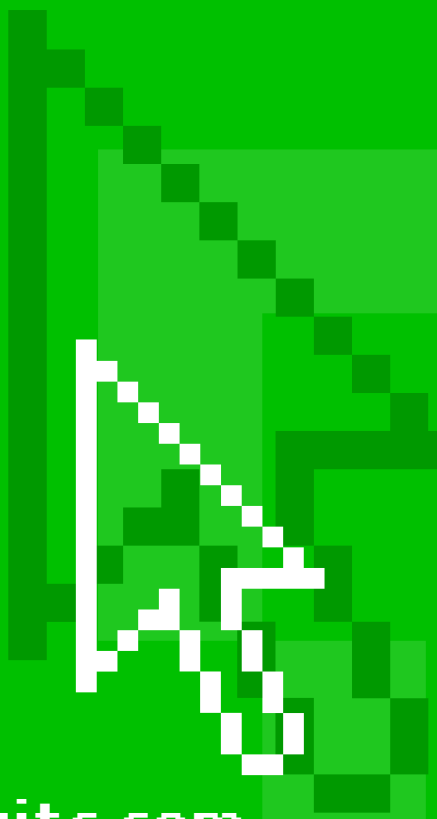
EURO CIRCUITS

Front-end

data preparation



www.eurocircuits.com

To preserve the environment and to make use of the many embedded links to articles on our web site, we recommended that you read this white paper in its digital form.

Front-end data preparation

What do PCB fabricators do with your data before they make your PCBs?

This paper explains the front-end engineering process that we use at Eurocircuits. All PCB fabricators carry out broadly similar operations even if they give them different names. We have included our own names for the various stages, so that if you are a Eurocircuits user you can follow the progress of your job through our front-end engineering process.

Why can't we use your Gerber and drill files as you uploaded them?

1. We need to analyse the data to confirm that it is manufacturable and to establish the production parameters of the job (we need to work out whether we can make it and how we will make it). This is the Data Analysis step.

► The Eurocircuits stages are "Analysis" (the automated input and data analysis process) and "Analysis CC" when our engineers carry out the Cross Check on the automated analysis.

2. We need to make changes to fit the data to the manufacturing processes (for example we need to increase the diameters of the holes to accommodate the plating on the hole walls). This is the Production Tool Generation step.

► The Eurocircuits stages are "Single image" and "Single Image CC" (Cross-Check).
Note: "Single Image" refers to what we will deliver to the customer. These may be single circuits or customer-specified panels.

3. We need to provide data in standard formats to drive our production and test equipment (photoplotters, drilling machines, plating lines, profiling machines, automatic optical inspection equipment and electrical testers).

What's in it for you?

If you can design your board to be manufactured more easily, the finished product will be more robust and less expensive. So we have included design tips and pointers throughout the process description. More information can be found on our home page **www.eurocircuits.com** under "Technology Guidelines".

Notes:

1. Most fabricators including Eurocircuits automate many or most of the steps described below. This makes the process faster and cheaper, and reduces the risk of error. To make the presentation clearer we have mostly ignored automation.

2. Design Tips are marked by ▷
   ► indicates comments which relate Eurocircuits' practice to the wider fabrication scene.

Part 1 – Data analysis

Note: we run the same data analysis before we make a quotation as well as when we receive an order.

Automatic data input and analysis

We use an automated software program to input your Gerber layout files and Excellon drill files as well as EAGLE BRD files. The program converts these into our internal front-end engineering format, checks that all layers are present and performs a preliminary analysis to establish the production parameters of the job.

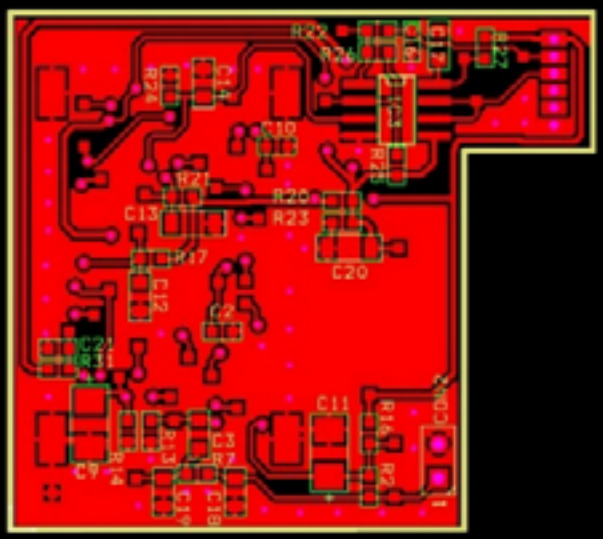
At this stage, our new PCB Visualizer software will allow you to see all layers of your board on screen.

► There are other data transfer formats in use (for example, Mentor Graphics/Valor's ODB++). We prefer Gerber as it is simple, unambiguous, and man-readable.

▷ *Read our "Ten rules to provide perfect data for PCB production". This is a brief overview of how to provide good and clear data. Combine this with logically chosen file names for your Gerber data and you will prevent unnecessary misunderstandings.*

● All layers have a different offset

● All layers have the same offset



▷ *Use Extended Gerber (RS274X) with embedded aperture sizes and Excellon drill files with embedded drill diameters wherever possible. They are faster to process and there is less risk of error. A second useful format is EAGLE 6.x native format. This is an XML based format that we can transform to our internal CAM-format without having to use the intermediate format Gerber and hence this limits the risk of mistakes.*

Why do we change format?

1. There is a lot of production information needed for data analysis, tool generation and manufacture which is not included in Gerber or Excellon (for example layer types for automated processing or net-lists for data verification).

2. Gerber and Excellon data is supplied in many different versions. Data processing needs a standardised format.
So the more powerful data preparation packages ("CAM systems") use their own internal data formats. We use Ucamco's UCAM software with internal format DPF (Dynamic Process Format).

Analysis cross-check

Our engineers complete the analysis, confirm the build for a multilayer, verify the data and raise any questions with the user.

Include clear information in your data package to avoid losing time while we contact you for clarification. See our White Paper "PCB design guidelines" for more information on how to present data most clearly. All information concerning your design including the build-up should be clear from your digital data to allow automatic data processing. In the event that this is not enough to clarify your design, include a README file in the data package. It is a clear and simple way to provide information in the order, especially for the board lay-up if the file-naming is ambiguous.

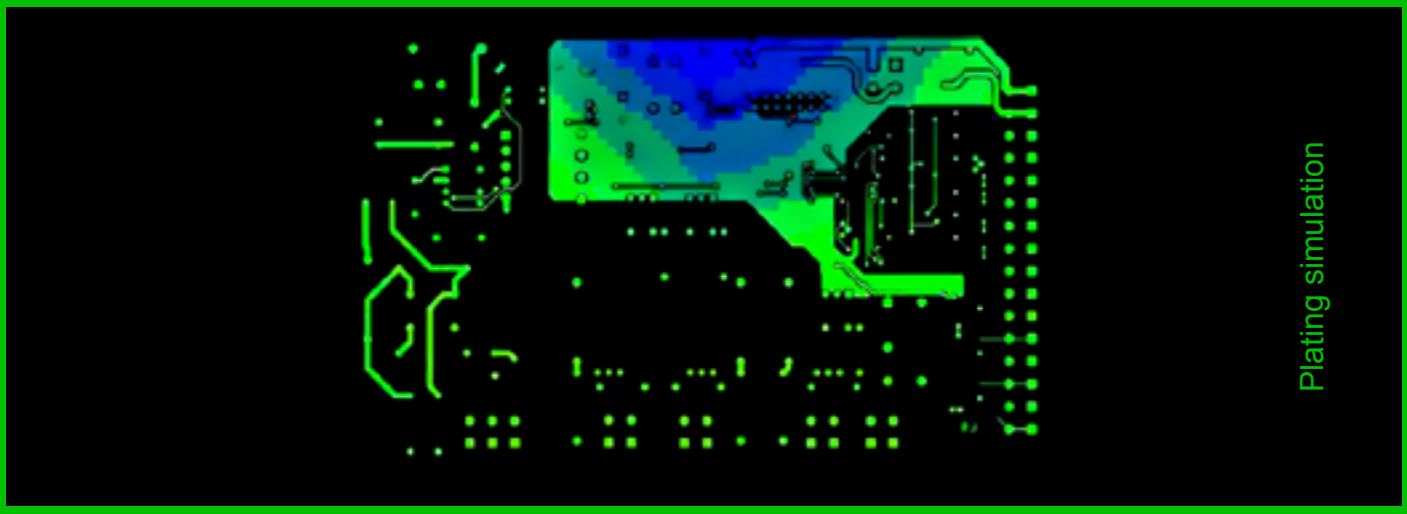
Our engineers check:

1. Does the job data match the order details (e.g. has the customer ordered a single circuit but supplied several circuits as a panel, or ordered one top silkscreen but supplied data for top and bottom?).

2. Are the layers consistent with each other (e.g. are there drill holes missing from through-hole component pads or vias)

3. Is the copper distribution within our manufacturing parameters, especially if the data is a customer-designed panel

4. Does the design data meet the specifications of the chosen service? A full list of checks is set out in our blog.

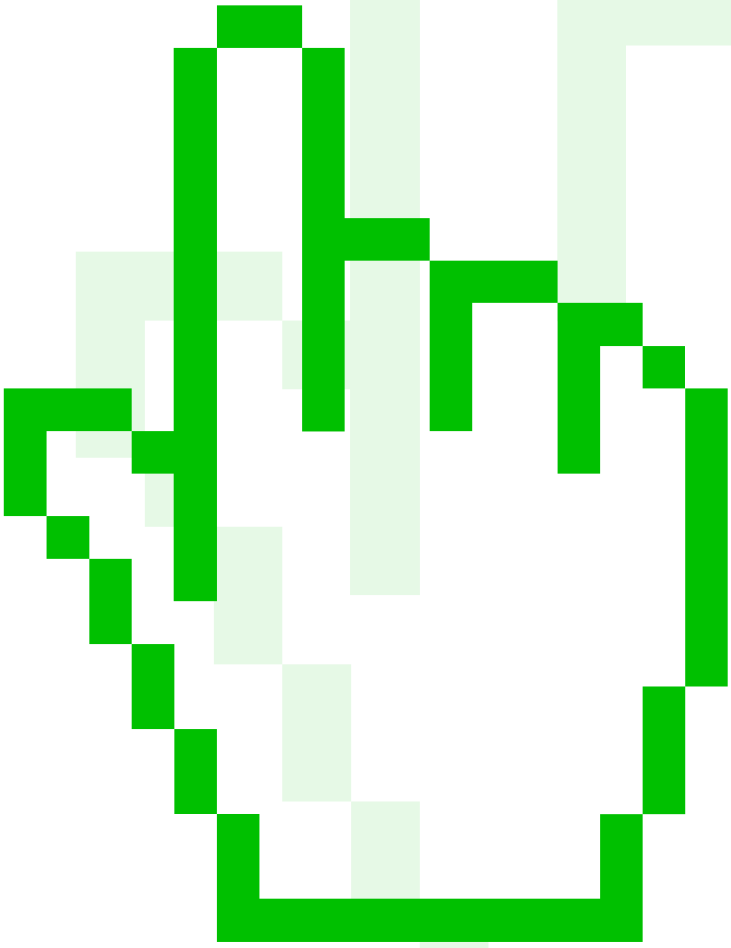


► Eurocircuits have different specifications for the different pooling and non-pooling services. Other fabricators have similar specifications for different costing bands. Our STANDARD pool and TECH pool specifications correspond pretty well with most fabricators' standard and tight or high-density capability/costing bands.

▷ *Build our DRC minimum values into your CAD system to ensure the most cost-efficient and best-priced design. For EAGLE users we provide sets of DRU files including these values. For Altium users also.*

▷ *Different minimum track and especially isolation values are required for different copper thicknesses (see PCB Design Guidelines p. 7). Ensuring a clean etch and good insulation between tracks and pads requires a minimum isolation distance. This distance depends on the copper thickness. Etching down, even with modern etching systems, also produces some sideways etching into the wall of the track or copper feature. We compensate for the loss of width by modifying the production films, but there remains an effective minimum gap dependent on the foil thickness. On outer layers when we plate through the holes we also plate 25 – 30 microns of copper onto the tracks and pads, so 18 micron start copper foil will end up +/- 45 microns and 35 micron start copper foil will end up 60 – 65 microns. This means that in many cases you can use a thinner foil and still achieve the thickness of copper you need, e.g. for current-carrying capacity.*

▷ *To ensure a robust end product with optimum plating, no drill breakout and, where relevant, good solderability, board manufacturers look for a minimum annular ring of copper around the hole. For optimum quality we measure this ring from the production hole (the TOOLSIZE) which is oversized from the finished size (the ENDSIZE) to allow for the plating in the holes. For inner layers the annular ring required is larger than for outer layers to compensate for any movement in the material during bonding. For the values required see PCB Design Guidelines p. 7.*



What happens when we find an error in the data (points 1 – 3 above)?

1. If the data error means that we cannot complete the analysis, we stop the process immediately and report the data issue to the customer (raise an exception). Typical examples are: no drill data, no board outline, panel instead of single circuit.

2. Otherwise we complete the analysis and raise an exception at the end (or for an inquiry, put it in the manufacturability report).

What happens if we find a violation of the service parameters (point 4 above)?

We evaluate the violation to see:

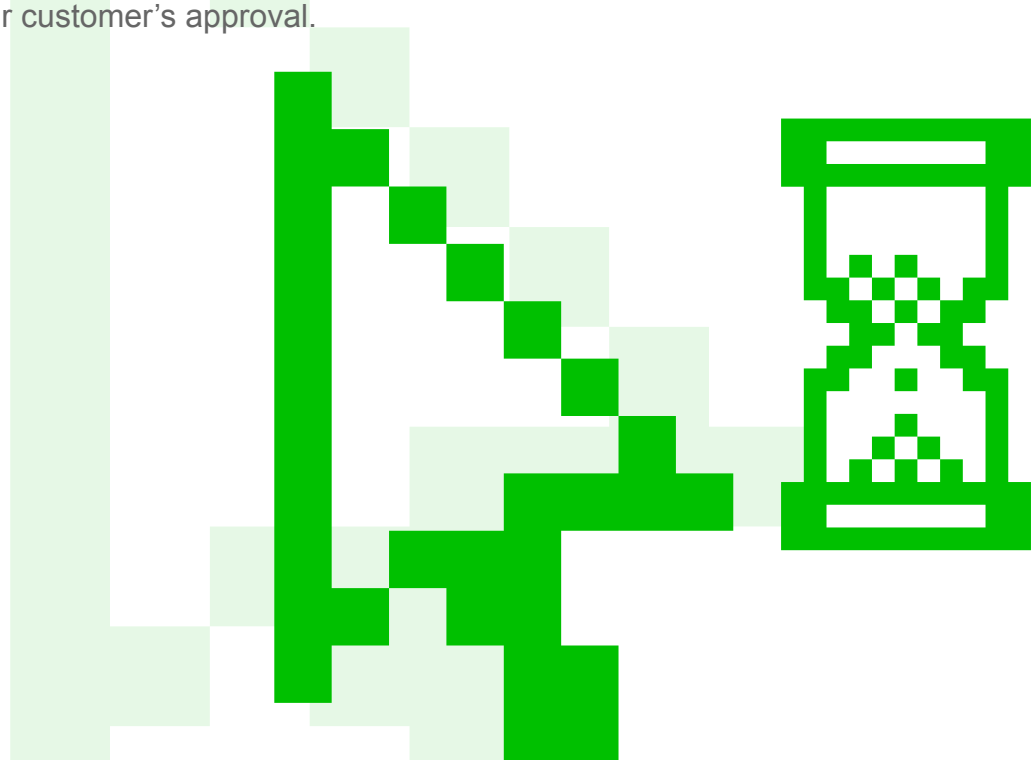
1. Can we repair it without affecting the board functionality?
2. Would the repairs be better done by the customer on his CAD system? I.e. are there too many repairs needed or are they too complex to carry out as part of normal data preparation.

There is a full list of the checks we run and what we regard as repairable on our blog.

► These are Eurocircuits' repair procedures which we like to make clear to our users. Other fabricators may use different procedures which they may or may not make explicit. If you are concerned that your design may be changed in ways you do not want you should ask what their policies are.

For pooling service orders we make the permitted repairs without further comments. For the rest we halt the job and provide the customer with a detailed report of the issues with screen dumps and advice on possible repairs (raise an exception).

For inquiries and On demand orders we produce a detailed manufacturability report describing the issues and proposing solutions. If the only violations are repairable under our procedures we list what we have done for our customer's approval.



Part 2 – Production tool generation: Single Image preparation

We now take the clean data and modify it to build the production tooling.

Prepare the job data

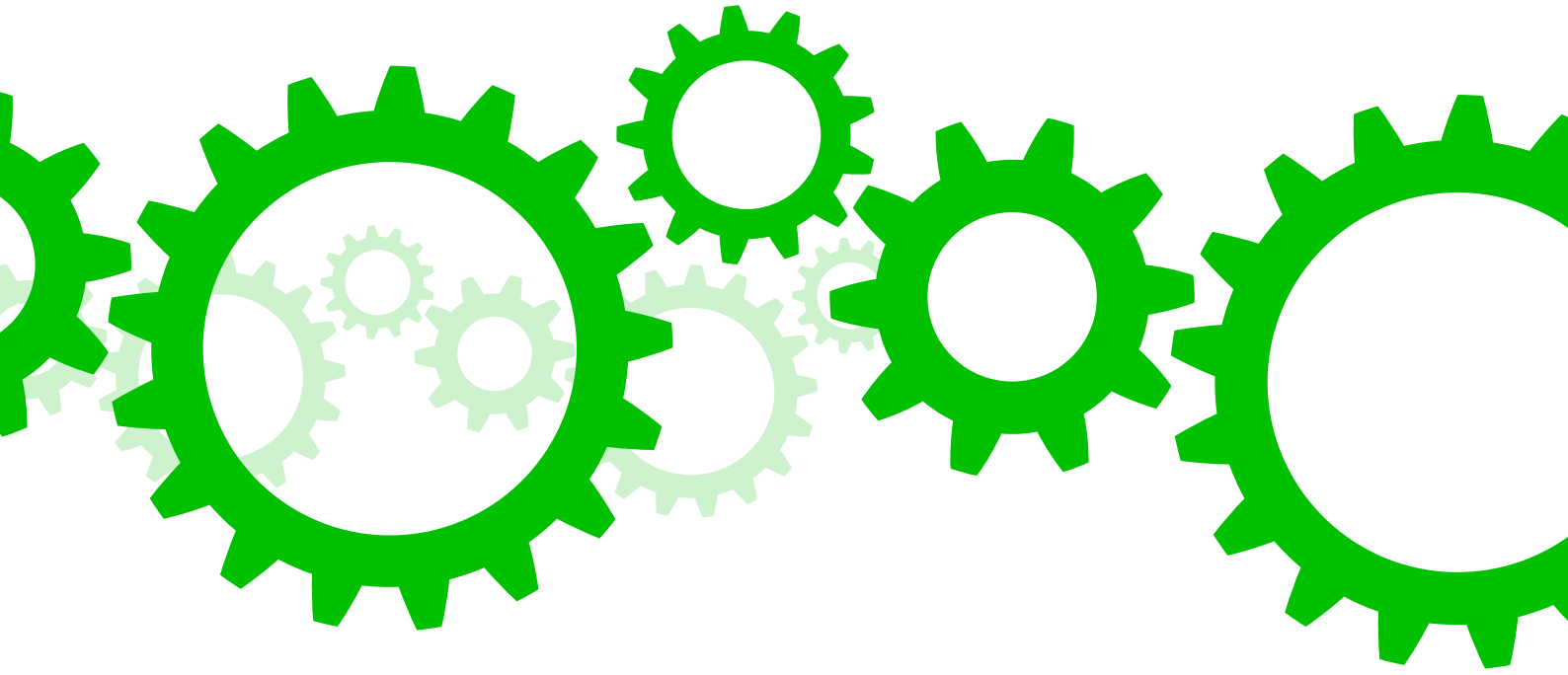
1. Make security copies.

The first operation is to make a security copy of the data. We will use this later to check that we haven't made any mistakes during data preparation. We build and save a netlist from the Gerber data as well as a copy of the layers.

▷ *If your CAD system allows you to output a netlist in IPC format include it in your data package. We can use it to check for any errors in the Gerber data supplied.*

2. Add the material information

We attach the files to the standard build for the order. This imports the correct material thickness, copper thickness, solder-mask and legend colour etc. If the job needs a special build the engineer will make the build now.



Prepare the mechanical data for board manufacture

1. Drill data.

We need to modify the hole size so that the hole after plating meets the customer's specified size and tolerances. See the blog article about drill data and copper image for more details

▷ *Always specify finished hole sizes. Different board manufacturers use different rules for calculating the oversize, so if you oversize at the design stage the figures you use may not suit your chosen supplier.*

▷ *Where possible specify standard drill tolerances +/- 0.1 mm for component holes and +0.1/0.3 mm for via holes. Tighter tolerances are possible but increase the cost of the finished circuit as they are not allowed in the pooling services.*

2. Profile and internal cut-out/slot data.

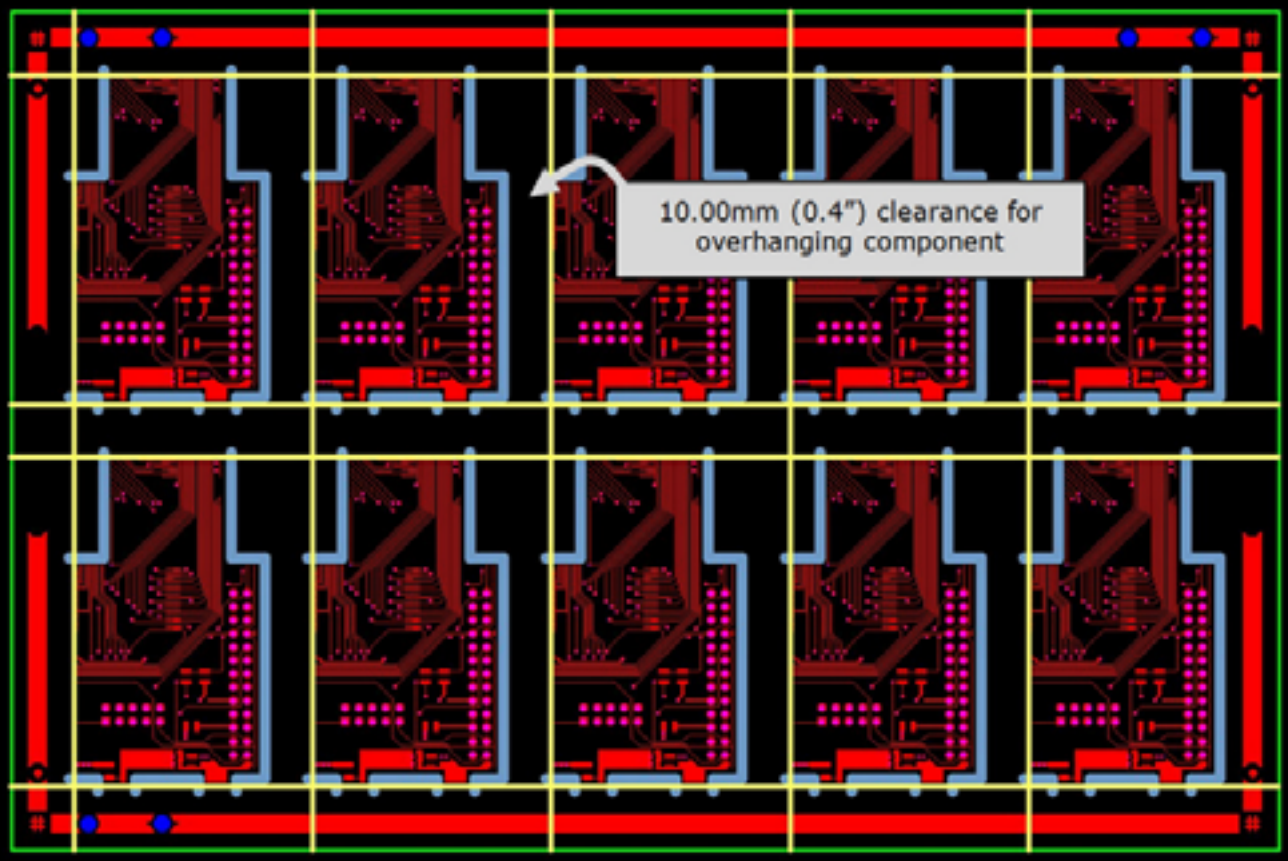
Our engineers convert your board's profile and cut-out information into a path for the router cutter. We use a standard 2 mm cutter for the profile and if required smaller cutters for internal slots and cut-outs (internal milling). If you have asked for the boards to be delivered in a panel the engineers add break-out tabs to hold the circuits in place during transport and assembly. The tabs are patterned to combine a secure hold with a clean snap-off.

▷ *There is no need to include a cutter path in your data. It is time-consuming for you to construct and in any case different manufacturers use different diameter cutters for profiling. And it will be incomplete as it doesn't include start and end points and for a panel the order in which each cut will be made. Each manufacturer will optimise the cutter path in his own way. More info in our the article on the webside under PCB layout data (indicating slots and contours)*

The alternative to profile routing for a customer's delivery panel is V-cut or scoring. We make a V-shaped cut in each side of the PCB leaving about 1/3 of the material to hold the panel together. Scoring can only be used for straight cuts across the panel, but it can be combined with routing for more complex profiles. As the circuits about each other it is more economical of space but

▷ *The V-shaped cut means that there can be no copper within 0.40 mm of the edge of the board.*

LIGHT BLUE: Milling (o break-routing) to obtain the required PCB shape
YELLOW: V-cut



3. Mechanical drawings

Although our drilling and profiling machines are driven by digital data we still need drawings for tool set-ups and for quality checks. These drawings showing a complete set of dimensions and hole and slot sizes are also available to our users as an online PDF, part of our visual feedback policy.

Prepare copper, solder-mask, silkscreen (legend) and paste layers.

1. Outer layers

We “clean” the outer layer data and carry out the repairs we identified earlier. Cleaning means converting any drawn/painted features into proper pads and polygons and removing minor copper defects (pinholes, slivers and peelables). These minor defects can cause shorts and opens in production if small pieces of etch/plating resist or copper break off the circuit and stick back in the wrong place.

▷ *Avoid drawn and painted features wherever possible. In RS274X you can define any pad or polygon you need. Slivers and pinholes are a common by-product of copper flood procedures but we can clean them automatically.*

2. Inner layers.

As well as cleaning and repairing the data as for outer layers we also remove all non-functional pads and make sure that all thermal pads are properly connected .We remove non-functional pads to reduce the risk of internal shorts and to increase the reliability of the plated hole.

3. Solder-mask

There is often a trade-off here between a good clearance between the solder-mask window and the pad and good coverage over adjacent copper. For the details see “PCB Design Guidelines p. 15 and our blog.

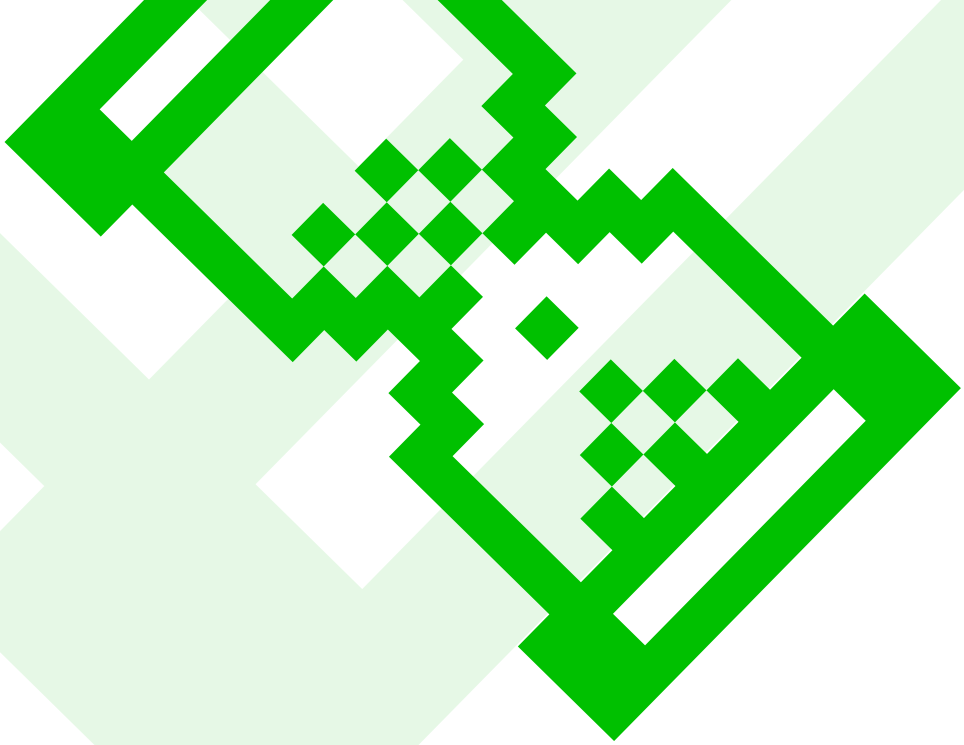
▷ *You can deliver the solder-mask windows at the same size as the pads and we will make the necessary adjustments.*

4. Silk-screen (legend)

CAD systems often place component labels over pads. To ensure good solderability we cut the legend back to 0.1 mm from the edge of the soldermask window. The software tidies the remaining text to give a clean image.

5. Paste layers.

If the customer has supplied a solder-paste layer we check it against the copper data. If there is no solder-paste layer supplied we build one by selecting all the surface-mount component pads. If the customer has asked for a delivery panel the paste layer is stepped and repeated at the same time as the other layers. It is then available for download to make third-party stencils.



Add identification codes

Users often need to know who made a PCB, when it was made, and whether it meets certain safety standards. We screen our internal order number unobtrusively onto the board. You can then look this up on our website to see when the board was made, what materials were used and what inspection stages it passed. This information may also be stored on the board as a barcode. Where a board has to meet UL flammability standards we add our UL marking as well.

Prepare special layers

These include data for hard-gold plated edge connector fingers, peelable solder-mask, via-fill and carbon pads. There is more information on these in our PCB Design Guidelines and in our blog.

Make the delivery panel

If the customer has asked us to put the boards into a delivery panel we do this when we have prepared all the data for the individual circuits. For the different options see our PCB panel guidelines.

▷ *You can also supply your own panelized data.*

Security check

1. The engineer finally verifies his completed production data against the reference layers and reference netlist that he saved at the beginning of the process.

2. A second engineer then cross-checks the data for a second time against the customer's order, any customer instructions, and the specifications of the chosen service.

Part 3. Build the production panel and output the tool data

Board fabricators don't make each circuit by itself; they combine them on large production panels. This, of course, makes handling much more efficient but also the panel borders contain features critical for manufacture, including:

- sets of tooling holes for imaging, drilling and profiling machines,
- fiducials for automatic optical inspection,
- test coupons for destructive and non-destructive quality testing,
- barcodes with data, for example for automatic calculation of plating currents.

The borders also allow the panels to be held securely in the electro-plating jigs and on production and test equipment.

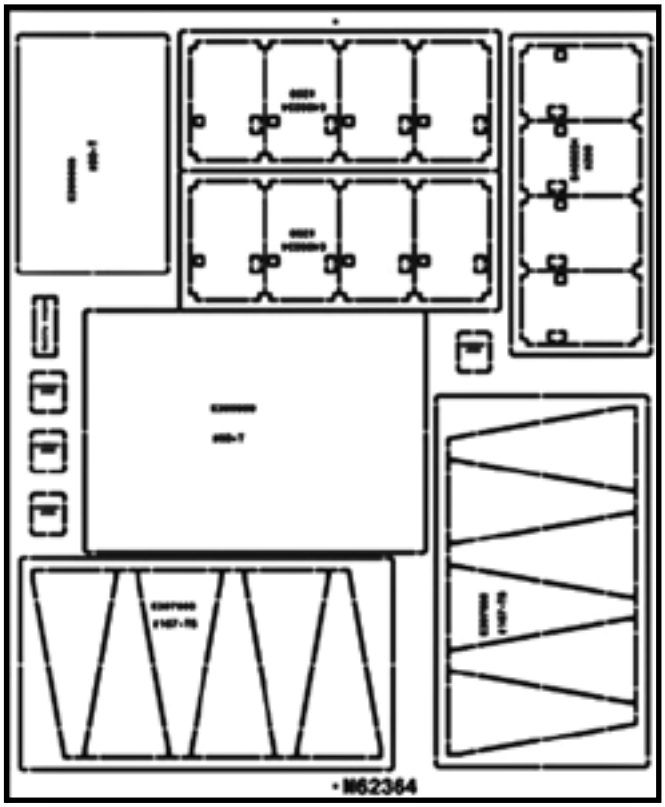
Most fabricators make their production panels by stepping and repeating the same circuit. Eurocircuits combine several different designs from different customers into one production panel – order pooling or panel sharing. There is more information on how we build the pooling panels in our blog.

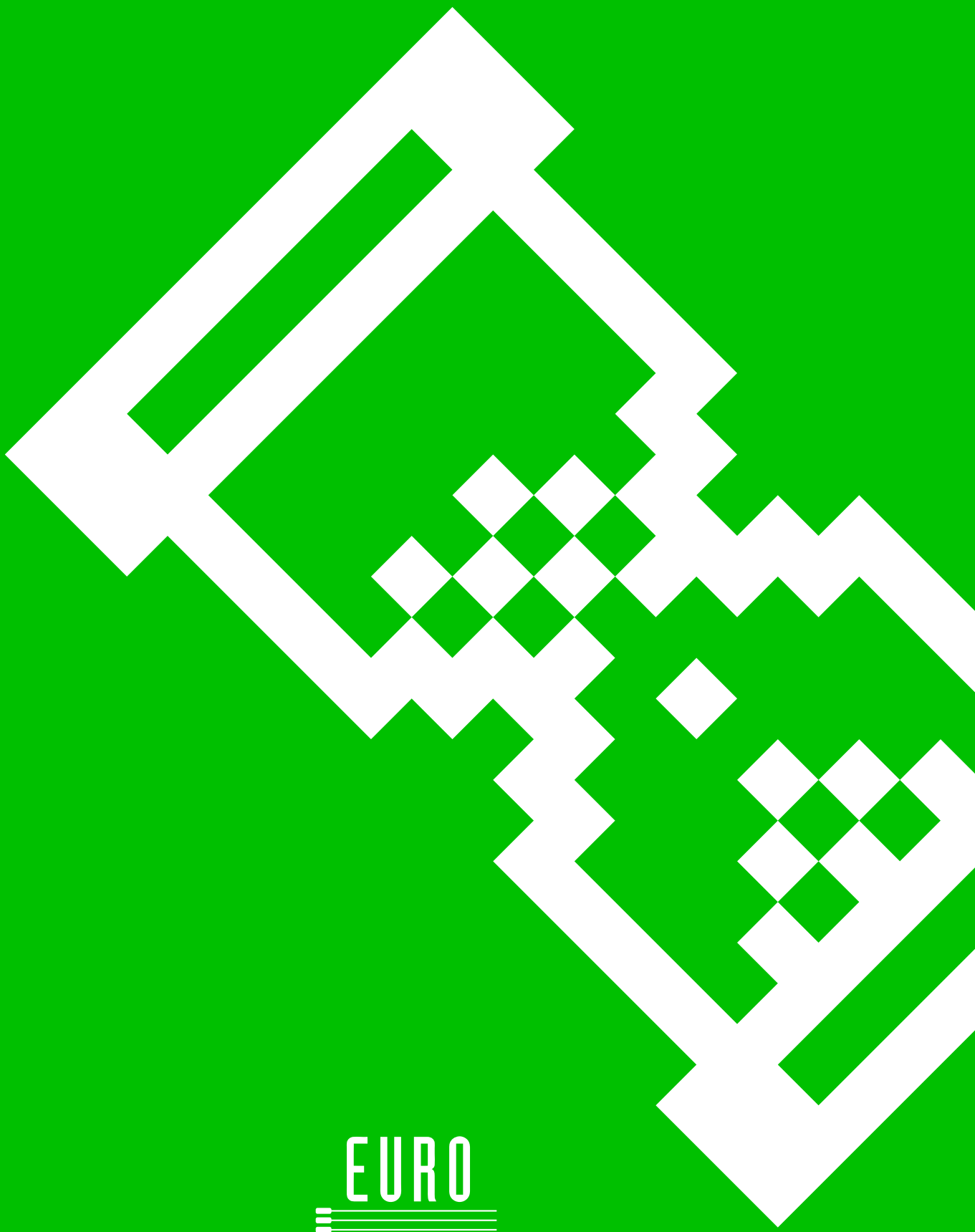
There are still more operations to be carried out on the data in the panel to cut production times and improve quality.

For example:

- Drill and rout path optimization algorithms find the shortest path to cut machining time.
- Plating simulation combined with the addition of extra copper patterns (robber-bars) helps to ensure optimum plating thickness across the panels.
- Dynamic etch compensation ensures accurate track widths. We increase the width of tracks and other features to compensate for the sideways component in the etching process.

Finally we output all the necessary files for the different production and test machines. There is a full list of these processes and outputs in our blog.





EUROCIRCUITS

www.eurocircuits.com