CopperCAM

CopperCAM is a program for Microsoft Windows environment. It allows owners of a CNC machine to transform **Gerber** or **Excellon** files, produced by electronic CAD software, to drilling and engraving paths that isolate copper tracks. It can produce a CNC file and directly call up the milling module without requiring manipulations of intermediate CAM files.

CopperCAM is not an electronic router, not even a printed-circuit CAD system. Its capabilities are few compared to Galaad; it provides no design function, and consequently focuses on the simple preparatory milling works on isolation engraving and the drilling of a circuit that has already been designed.

◘ INSTALLATION

CopperCAM is available only as download from www.galaad.net website and is not provided on a physical support. The downloaded module is an auto-extractible installation program which allows you to choose the disk directory where the software will be installed. Installing makes **no changes in the registry**, except for creating shortcuts on the desktop and Windows "Start" menu.

◘ LICENCE

The licence is defined by a 20 digit code that corresponds to the user's personal data (name and address). This licence is given by a file COPPERCAM.LIC that is to be copied directly into the installation directory, default being "C:\COPPERCAM". A start-up, the program detects the licence file and directly transfers the information to the licence manager. Calling "Help / Licence' allows you to check the validity of the licence.

◘ CUTTER TOOLS

The very first thing to think about once CopperCAM is installed is **define the characteristics of the tools** that will be used for drilling and milling printed-circuit boards. Function "Parameters / Tool library" allows you to indicate each tool data. If you have a tool rack, number them and use the same numbers in the parameters, though each tool can be assigned a name.

Once tools are defined, it is necessary to **indicate which ones will be assigned different tasks** the program can offer: isolation engraving, drilling, hatching zones, card contouring, and centring on plots. The function "Parameters / Selected tools" gives access to a setup windows where, for each task, you may indicate which tool must be called, and its motion depth and speed. For isolation engravings, it is possible to add a small distance margin. For hatchings, the margin gives the mesh density, *i.e.* the recovery between two hatch lines.

It is possible to **limit the number of drilling tools** by driving boring cycles when the hole diameter is greater than the tool. However, this function must be used carefully: most drilling tools for epoxy are indeed not qualified for horizontal milling operations and therefore there is an important risk of break.

Three strategies can be followed for drilling: either use **one single tool** for all drills, with boring cycles if the hole is greater than the tool; or use **a limited number of selected tools** with, for each hole, a call to the tool which has the closest diameter, smaller (with circular boring to reach the diameter) or greater (without boring).

❑ OUTPUT

Function "Parameters / Output data format" allows you to choose the type of the data which will be produced at last for chaining to the milling driver. Different usual formats are offered, including a **fully definable post-processor** format.

CopperCAM can **create the output file and then immediately call the driver software**. This file is not of great interest since it is only an intermediate between CopperCAM and the machine driver. It will be overwritten at every new output, except if no name has been indicated. In such case, you will be prompted to give a name every time. Note that the file name is added automatically as argument to the program called. So it is useless to add it in the command line.

Some engraving machines are provided with a driver which displays them as virtual printers. In such cases, CopperCAM can call this printer driver and send produced data. It is also possible to output these data on a COM or LPT port for a direct link to the machine.

❑ GENERAL WORK SEQUENCE

The process consists in **opening a Gerber file** that has been produced by an electronic CAD software, then eventually one or more additional layers and an Excellon file that defines the drills. CopperCAM can automatically detect a track net that corresponds to the card final cut, and neutralise tracks that are not connected to any pads (texts, origins, *etc.*). These neutral tracks can also be engraved at the centre using a simple line. The program will then **calculate the isolation contours** around pads and tracks. The operator may rectify the result whenever necessary, by deleting or inserting isolation segments and by adding hatched zones where the whole copper must be removed. Once the calculation and rectification jobs are achieved, the software produces an **output file** under the predefined format and calls the module that dialogs with the machine.

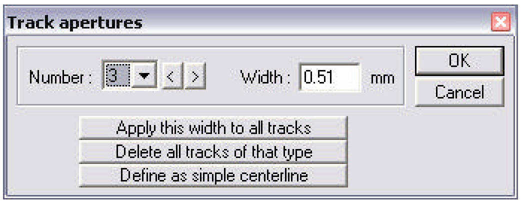
❑ GERBER FILES

The most widely known file format concerning electronic CAD is undoubtedly the Gerber format. This format is destined for Gerber Scientific Instruments photo-plotters and has become a real standard in that domain. The photoengraving of circuits using light-printing techniques induces specific considerations that appear in the format. Light-printing is performed by an optical head that focuses the light beam on the circuit after travelling through a diaphragm, at the locations where the copper should be preserved. The diaphragms generally have a predefined size and shape, the simplest being a basic circular disc. The pads may have more exotic shapes, but the tracks are printed using simple circular diaphragms of given diameters along the connecting path.

A Gerber RS274-D file (classical format) therefore contains **diaphragm numbers** that are either predefined in the optical library of the photoplotter, or referenced in the file itself or in an attached library file. Light-printing instructions are very simple: the flash head can be **moved, with the diaph-ragm shut**, to a given XY position (movement without light-printing), or **with the diaphragm open** (light-printed track), or even be sent to a position and the **diaphragm being opened then shut** to light-print a fixed point (pad). This makes a total of three positioning instructions, plus the number of the diaphragm that is currently used.

A more elaborate format, which keeps the ascending compatibility with that mentioned above, has been defined under the name of Gerber RS274-X or Extended-Gerber. This new format uses the same light-printing instruc-tions, but its major advantage is that it integrates in the file header all **geometrical indications about used diaphragms and even drills**. In fact, a Gerber RS274-X file requires neither an attached library file nor a table of predefined diaphragms. Any useful data that is required to create the printed circuit is contained in the file. Naturally, CopperCAM can read this heading information whenever available. If your own electronic CAD application offers an export function under Gerber RS274-X format, it is the one you should use.

Should you open a classical Gerber RS274-D file, no information related to diaphragms is available, and consequently you must redefine the geo-metrical properties of these diaphragms once the file is loaded, *i.e.* for each set of pads indicate the **shape** and the **size** of the diaphragm, and even the **drill** diameter if necessary, and indicate for each set of tracks, the **size** of the diaphragm, which corresponds to the track **width**. This must be done for each diaphragm referenced in the open file.



CopperCAM memorises your indications for the next file, so it is not necessary to redefine pad and track diaphragms at anytime if your electronic CAD application always reuses the same references. Obviously, this task has no object if you are using the Gerber RS274-X format, whose heading indications are picked up and immediately applied. However, as soon as the file is loaded, if a diaphragm that is used in the file is not referenced, then CopperCAM displays a dialogue box which allows you to complete the missing references.

So, in an extended Gerber file, it is possible to get not only the geometrical design information about pads and tracks, but also the drilling diameters of pads for soldering traditional components (component side and colder side). The only information that is still missing – and this is bad news for owners of a CNC milling machine – is the contour of the whole printed circuit board, with its origin points.

◻ EXCELLON FILES

Another file format concerns the **drilling process of pads** in a printed circuit, this format has been elaborated for Excellon Automation multi-drills. It is used less today with the surface-mounted components, but nevertheless it concerns the creation of electronic circuitry, so CopperCAM cannot ignore it.

Like Gerber files, Excellon files contain XY drilling co-ordinates and tool numbers whose diameters correspond to the holes. And like classical GRB files, EXL (or sometimes DRL or TXT) files unfortunately do not contain this drilling diameter information, so it is necessary to fetch it in attached files that are not standard, or indicate it manually once the file as been loaded, which is the basic solution that CopperCAM offers.

Furthermore, there are actually two different Excellon formats, one using real co-ordinates (XY numerical values are formatted and indicated in immediately usable units), the other being a bit older in co-ordinates with no trailing zeros on the right-hand side, which may induce position errors if the file does not contain header information about the numeric framing format. To avoid this problem, CopperCAM lets you select both formats under two distinct entries, which allows you to directly choose the correct numerical model. If your file looks wrong, load it again using the other available Excellon format, and all being well things should look better.

An Excellon file cannot represent a printed circuit, not even its pads, but only the drills. If your electronic CAD application provides an export function under Gerber or Excellon, please select Gerber which contains much more geometrical information.

Note that the Gerber RS274-X format may encode drilling diameters for pads, but this is optional and unfortunately not often used by CAD software. It is better to include drilling data to the Gerber file if your software can. This avoids handling a supplementary Excellon file and eventually realign drills with pads.

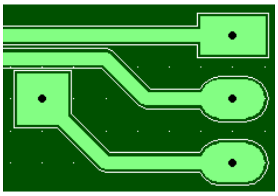
◻ ALIGNING LAYERS

CopperCAM is able to manage **four copper layers** simultaneously (pads and tracks), plus a **drilling plane** (layer #5) and a **card contour plane** (layer #6). Gerber or Excellon, open files do not always use the same origin point. Consequently, it may become necessary to **realign items** so they match one another. The first thing to do is rotate the current layer so it has the same orientation as the layer #1, by using the inverting and rotating commands.

On layer #1, select a **reference pad**. Then skip to the other layers and select a pad or a drill to be aligned to the reference. That can be done directly with the **right mouse button** on a pad.

Do not confuse **origin** and **reference pad**. The origin is the point of co-ordinates (0,0) of the circuit and therefore of the output milling file. The reference pad is used only for realigning different layers.

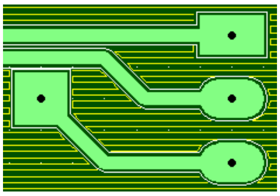
◻ CALCULATION OF ISOLATION CONTOURS



Isolating tracks and pads requires a preliminary calculation that defines **path contours** according to the end diameter of the cutter tool that will be used for the engraving task. This calculation manages collisions should trajectories overlap. This is the main purpose of CopperCAM which, from the geometrical information the Gerber file contains, will be able to create isolation paths and even allow you to remove the copper from all empty zones using hatchwork. These functions can be applied from "Machine / Calculate contours (or hatches)".

The distance between the tool axis path and the track or pad border is generally the engraving **tool end radius**. CopperCAM then calculates all isolation contours for engraving the circuit. This calculation is performed in several phases that may sometimes take some time depending on the complexity of the circuit and the computer speed.

◻ HATCHING EMPTY ZONES



Engraving contours is sufficient to isolate tracks and pads, but you may want to **remove the remaining copper** from the card surface or part of it, to ease soldering components, for example on very neighbour pads.

The hatching function can be called up only if the track and pad contours have already been calculated. The hatch margin is defined in selected tool parameters. It corresponds to the distance between two consecutive lines. The default value is the tool radius at the engraving depth (for a conical tool), which gives a recovery ratio of 50%. But you may of course choose a different margin. CopperCAM can also link hatch lines to build a zig-zag path whenever possible without milling a shortcut through a contour. Hatch ends approach the contours at about 10% of the hatching distance.

◻ MILLING

Calling function "Machine / Mill" produces the output file and automatically chains to the machining driver. The job sequence is definable through different sections, each of them corresponding to a task (drilling, engraving different layers, drilling centring plots, final cut).