AC – DC echipamente electronice srl
Cluj-Napoca, str. Muncitorilor, nr. 18, ap 8
tel: 0745-659909 www.acdcelectronics.ro

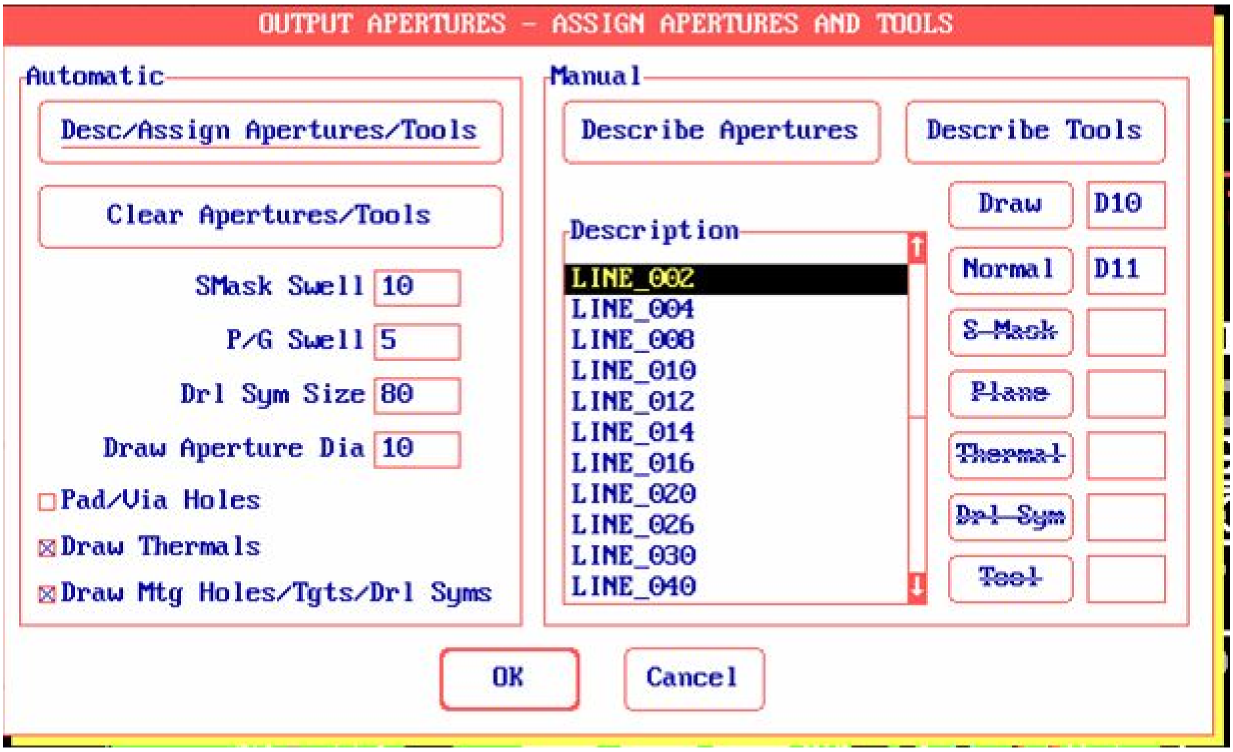
Gerber files achievement in

TANGO pcb plus v2.22

The achievement of Gerber files in TANGO pcb is laborious. There are a lot of buttons and options but help informations are few.

This material completes our tutorial concerning the implementation of TANGO on current Windows systems.

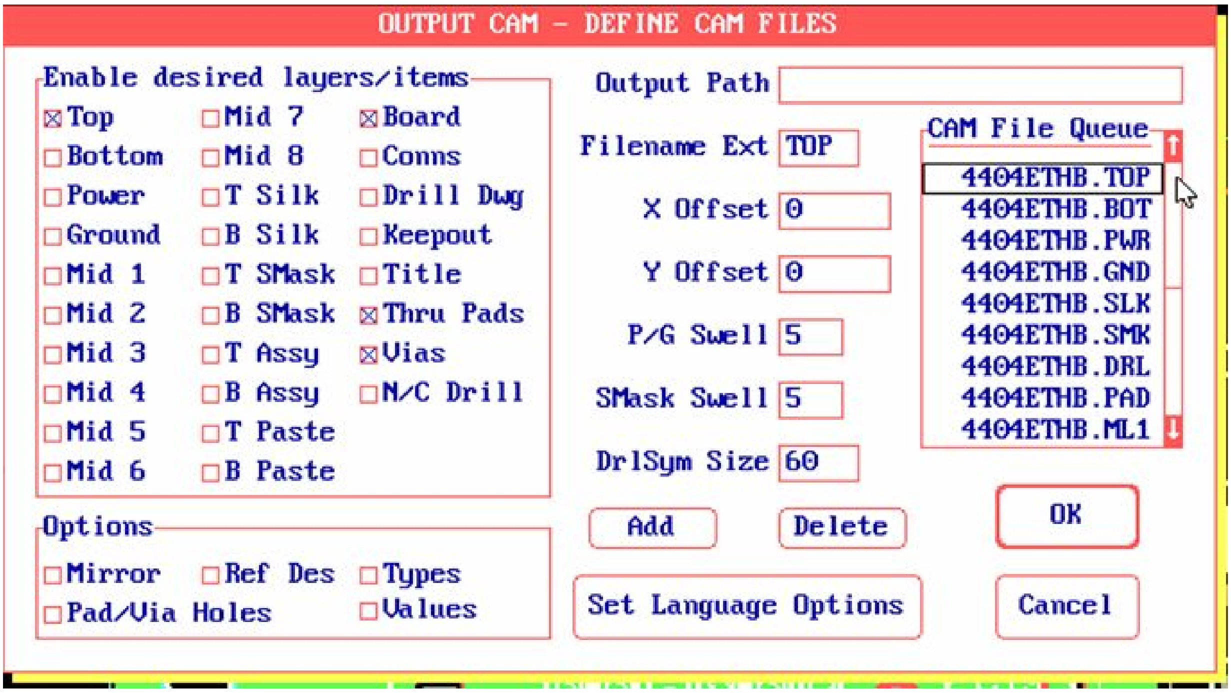
Open the Output/Apertures window.



Press Clear/Apertures/Tools button and confirm the action. Select Draw Thermals and Draw Mtg Holes. Press Desc/Assign Aperturtes/Tools button.

When Draw Aperture Dia is 0 change the value in 2…10 range.
Press button Desc/Assign Apertures/Tools and OK.

Open Output/CAM window.



In CAM File Queue there are 12 Gerber files about which there are few help informations. The first four are intuitive. TOP layer including trails, Board geometric outline, Thru Pads and Vias. As the bottom layer BOT. PWR and GND are the intermediate power supply layers.

The Gerber 374D files' aim is the achievement of PCB boards with metallized holes. Top and bottom copper layers are ussualy used.

- The top copper layer TOP including trails, pads and vias.
- The bottom copper layer BOT including trails, pads and vias.
- The solder mask SMT of TOP layer, including pads' surfaces.
- The solder mask SMB of BOT layer, including pads' surfaces.
- The board's geometric outline BRD.
- The NCD file defines the coordinates and diameters of the holes (drills).

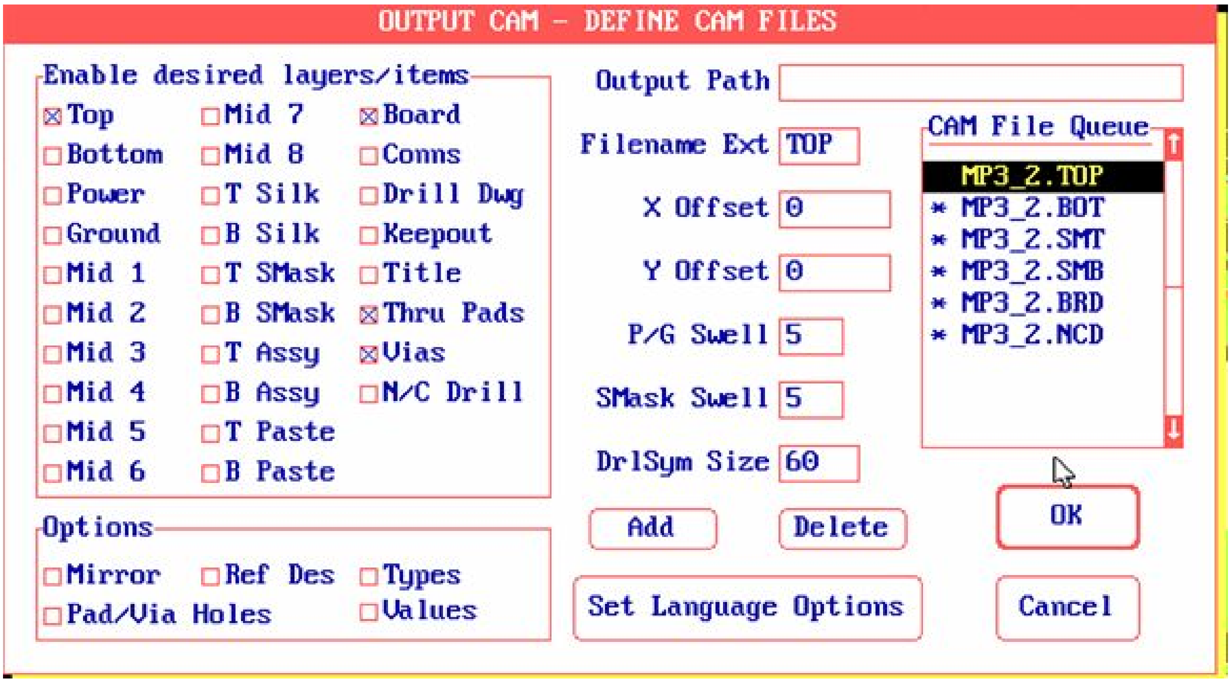
Observation: In the case of small and medium series it is useful the components' mask including the components' geometric outlines and names.

Our solution is to **Delete** all existing files and to redefine only the files we are interested in:

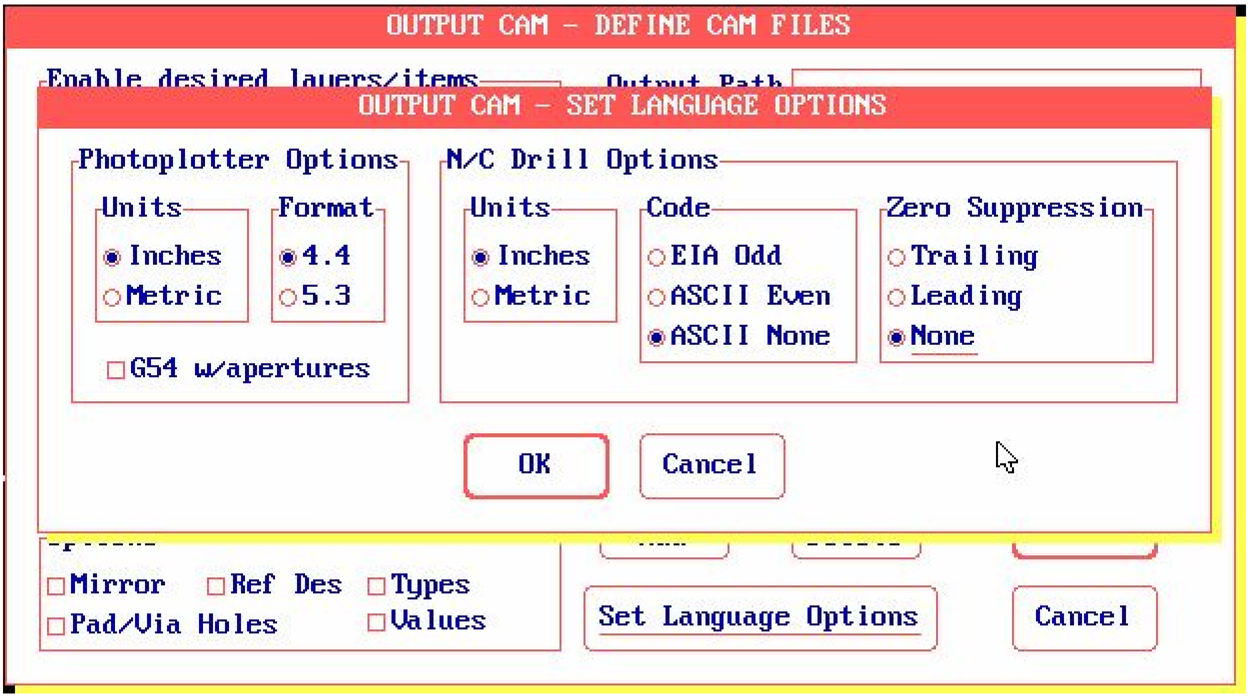
- In Filename Ext write TOP. Select Top, ThruPads, Vias. Press Add.
- In Filename Ext write BOT. Select Bot, ThruPads, Vias. Press Add.
- In Filename Ext write SMT. Select TSMask, ThruPads. Press Add.
- In Filename Ext write SMB. Select BSMask, ThruPads. Press Add.
- In Filename Ext write BRD. Select Board. Press Add.
- In Filename Ext write NCD. Select N/C Drill. Press Add.

In case the component's mask is needed, type IMK in In Filename Ext. Select T Silk, Ref Des. Press Add.

The Output/CAM window shows as follows. Only files marked with an asterix will be saved.



Press Set Language Options button.



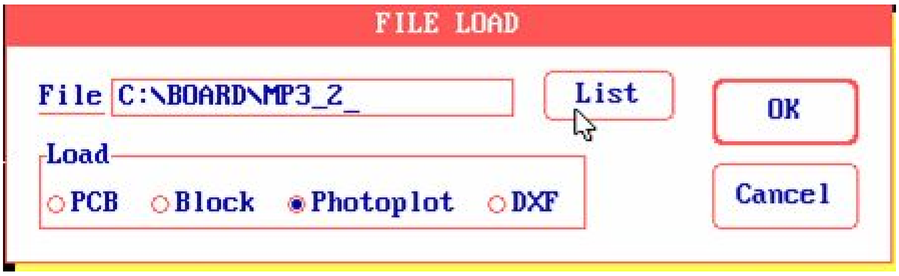
Select ASCII None and None. Press OK. Thus, the .NCD file becomes a text file.

Press the OK button of Output/CAM window. The files are saved in TANGO directory.

We need another text file: apertures containing the photoploter’s light spot dimensions. Select Output/Reports. Select only Aperture Informations and press OK. The file with .REP extension is saved in the same director as the main board .PCB file.

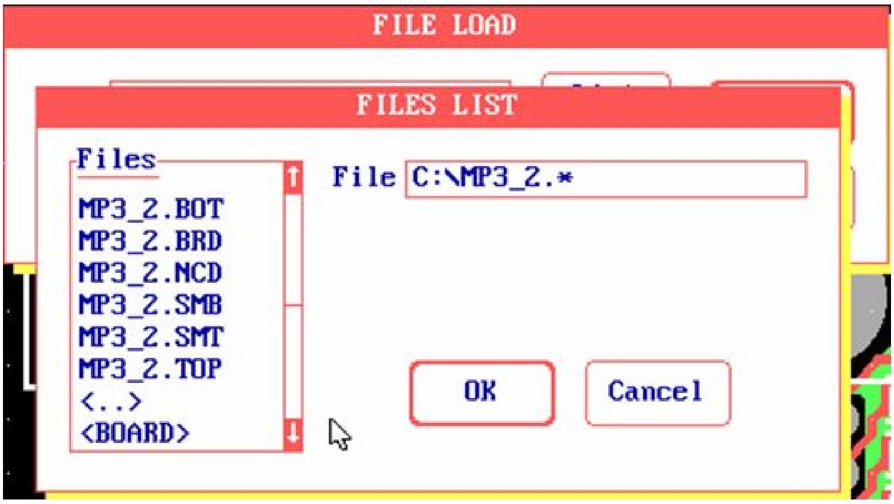
Viewing the Gerber files

In File Load select Photoplot.

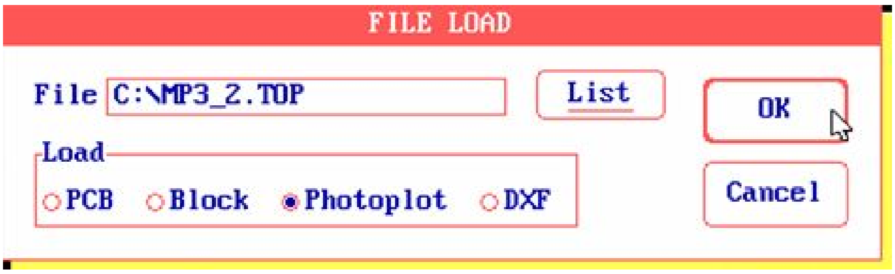


Observation: TANGO is implemented with DosBox 0.74, see our [tutorial](). The virtual disc C:\ is the real path D:\TANGO.

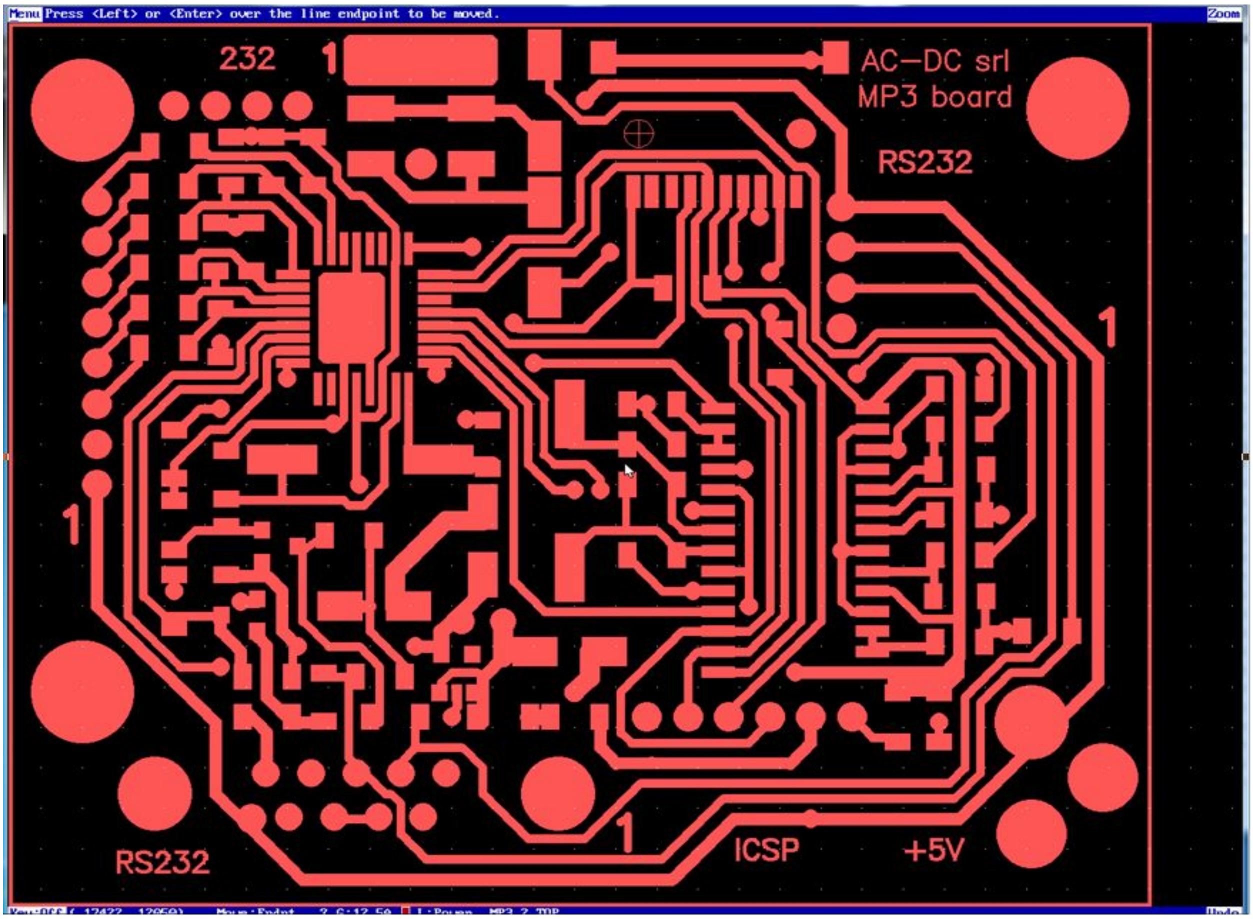
Press List button, select the virtual C:/ disk and get the list of the Gerber files that interest us.



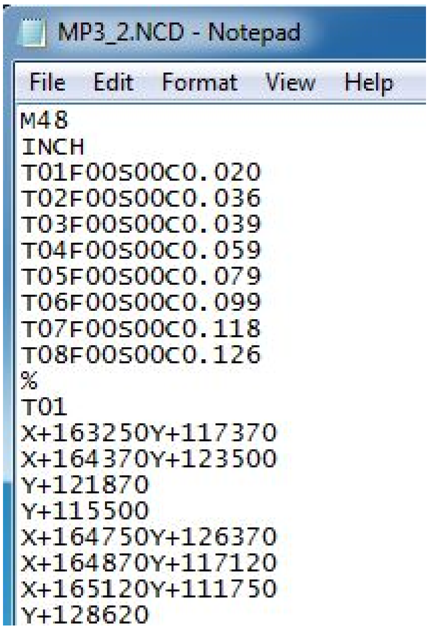
Select top layer MP3_2.TOP, press OK and return to File Load window.



Finally press OK. The MP3_2.TOP file is as follows.



There are two files that can be viewed with Notepad. The drilling .NCD file is as follows:



MP3_2.NCD - Notepad

File Edit Format View Help

```
M48
INCH
T01F00S00C0.020
T02F00S00C0.036
T03F00S00C0.039
T04F00S00C0.059
T05F00S00C0.079
T06F00S00C0.099
T07F00S00C0.118
T08F00S00C0.126
%
T01
X+163250Y+117370
X+164370Y+123500
Y+121870
Y+115500
X+164750Y+126370
X+164870Y+117120
X+165120Y+111750
Y+128620
```

The Aperture Information .REP looks like this:

```
MP3_2.REP - Notepad
File  Edit  Format  View  Help

   Aperture Information                                                   MP3_2.PCB


                          Aperture and Tool Descriptions
   ==========================================================================

   Code    Shape    X     Y      Hole   Type    Comment
   --------------------------------------------------------------------------
   D10     Ellipse  002   002    000    Both    Ellipse  X_002 Y_002 H_000 FD
   D11     Ellipse  004   004    000    Both    Ellipse  X_004 Y_004 H_000 FD
   D12     Ellipse  008   008    000    Both    Ellipse  X_008 Y_008 H_000 FD
   D13     Ellipse  010   010    000    Both    Ellipse  X_010 Y_010 H_000 FD
   D14     Ellipse  018   018    000    Both    Ellipse  X_018 Y_018 H_000 FD
   D15     Ellipse  020   020    000    Both    Ellipse  X_020 Y_020 H_000 FD
   D16     Ellipse  024   024    000    Both    Ellipse  X_024 Y_024 H_000 FD
   D17     Ellipse  030   030    000    Both    Ellipse  X_030 Y_030 H_000 FD
   D18     Ellipse  040   040    000    Both    Ellipse  X_040 Y_040 H_000 FD
   D19     Ellipse  050   050    000    Both    Ellipse  X_050 Y_050 H_000 FD
   D20     Ellipse  080   080    000    Both    Ellipse  X_080 Y_080 H_000 FD
   D21     Ellipse  062   062    000    Flash   Ellipse  X_062 Y_062 H_000 FL
   D22     Ellipse  046   046    000    Flash   Ellipse  X_046 Y_046 H_000 FL
   D23     Ellipse  082   082    000    Flash   Ellipse  X_082 Y_082 H_000 FL
   D24     Ellipse  066   066    000    Flash   Ellipse  X_066 Y_066 H_000 FL
   D25     Ellipse  086   086    000    Flash   Ellipse  X_086 Y_086 H_000 FL
   D26     Ellipse  070   070    000    Flash   Ellipse  X_070 Y_070 H_000 FL
```

The Gerber informations are saved in PCB.DFN file.

Gerber files preparation for boards achievement

The achievement of double layers boards with metallized holes requires at least 7 Gerber files. The components' mask is sometime useful and increases this number to 8.

TANGO pcb ensures four layers including the intermediate copper power supply layers PWR and GND.

The Gerber files with TOP and BOT extension do not explicitly declare that realy are the top or bottom layers, the extensions being subjectiv defined by the designer. To clarify this it is useful to write some texts on these layers for ex. the product name. On the bottom layer the text is reversed written.

Archive the Gerber files. Send the archive to specialized companies in manufacturing electronic circuits boards. It is very important to communicate. The put in practice observations of these specialist will certainly improve the quality of the electronic circuit boards.