CONTENTS

The Drill2GBR User Interface

Drill2GBR Reference

COPYRIGHT

DRILL2GBR Main Dialog Window

The drill files you wish to convert are loaded here using the ... button. Order of loading is not important as each drill file will create a single Gerber file.

This section is used to define the characteristics of the drill file. Some characteristics can be automatically deduced by the program -- others require user input.

This section is used to define the characteristics of Gerber files associated with the drill files. You need load only one of a group of Gerber files. All required information on the Gerber files can be obtained by the program when it scans the header of the RS274X file.

Drill2Gbr

Drill Files

... Delete
... Delete
... Delete
... Delete
... Delete
... Delete
... Delete
... Delete

Delete all

Starting D-Code: 200

Offset: 0 x 0

Apertures
- ○ Round
- ○ Polygonal

No. of Sides: 8

Map Drill Tools to Apertures

File Extensions

Drill Format

Format

Units
- ○ IN
- ○ MM

Zero Inclusion
- ○ Leading
- ○ Trailing
- ○ Decimal

Coordinates
- ○ Absolute
- ○ Incremental

Gerber Format

Format

Units
- ○ IN
- ○ MM

Zero Suppression
- ○ Leading
- ○ Trailing
- ○ Decimal

Coordinates
- ○ Absolute
- ○ Incremental

Load Reference File

Help   Execute   Exit

File Extensions: This button allowsthe user to set the default file extensions for drill and Gerber files making it easier to select the files you need.

Apertures - drills are mapped either to round apertures or a polygon with the users choice of sides. Polygon is often chosen when the target is going to be an electrical simulation program.

Offset - generally the drill hole coordinates and Gerber cooredinates line up -- but occasionally they do not. In this case you can apply the correct offset between the drill and Gerber data.

Starting D-Code. In order not to redefine D-codes that might be part of the Gerber files, the starting D-code is shifted to begin a D200. You can select a different value if you need to.

Map Drill Tools to Apertures: If the drill file lacks information on the diameter of the drill tool you can open a dialog here allowing the user to enter such information manually.

Loading the Drill Files

To select one or more drill files click on the small browse button (...) and a file selection dialog will open. Then select one or more drill files. There is no standard extension for drill files – some might end in .drl, others (from Cadence) in .tap but the actual suffix varies from each source.
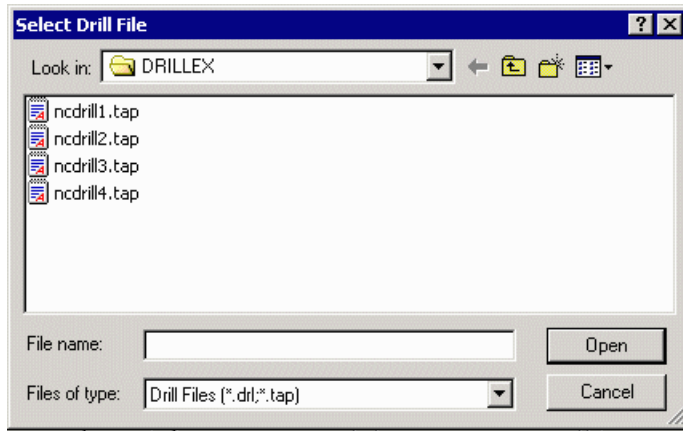
One can select multiple drill files by holding the Ctrl or Shift key down while selecting.

Immediately upon loading, drill2gbr scans the files and attempts to determine which tools are used, the tool size, the units and the coordinate mode. If this information is readable it will propogate the correct values in the dialog box. If the program cannot successfully scan the drill files an error message will pop up.

Loading a Reference Gerber File
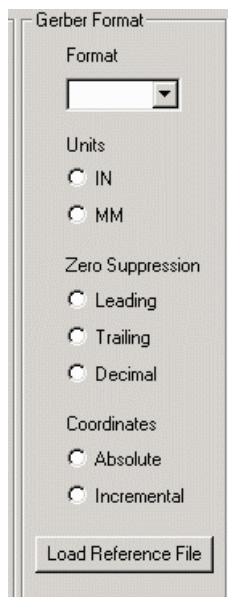
The usual purpose of converting drill to Gerber is to inspect or overlay the drill data with a set of Gerber files that have been generated for a PCB or IC package. It is important then, that the conversion use the same units, format etc.. as the Gerber files. By loading a "reference" Gerber RS274X file you can capture the Gerber settings automatically. In this example, the file called Layer1.art represents the top layer of the IC package. By selecting it as the reference file, the units, coordinate mode and zero suppression are automatically determined.
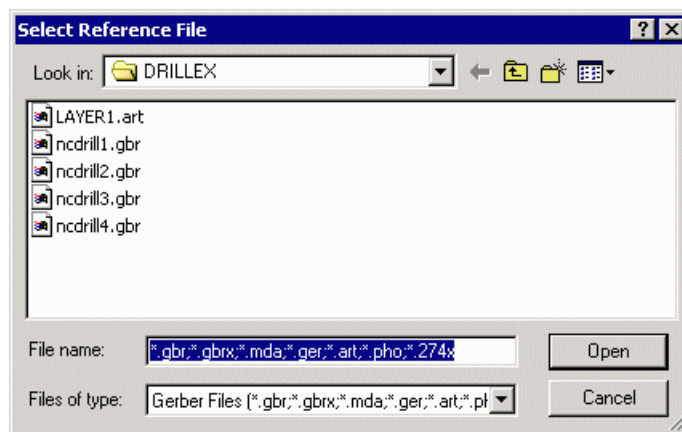
Mapping Drill Tools to Apertures

A drill file should reference one or more drill tools. The drill tool select command can also include the diameter of the tool – some applications do not bother to include the diameter information. Below you see two different select tool commands. On the left Tool 01 has been selected. On the right Tool 01 has been selected but the additional info, C0.100, indicates that this tool is circular and 0.1 in diameter.

```
Select Tool 1          Select Tool 1 of Diameter 0.100

T01                    T01C0.100
```
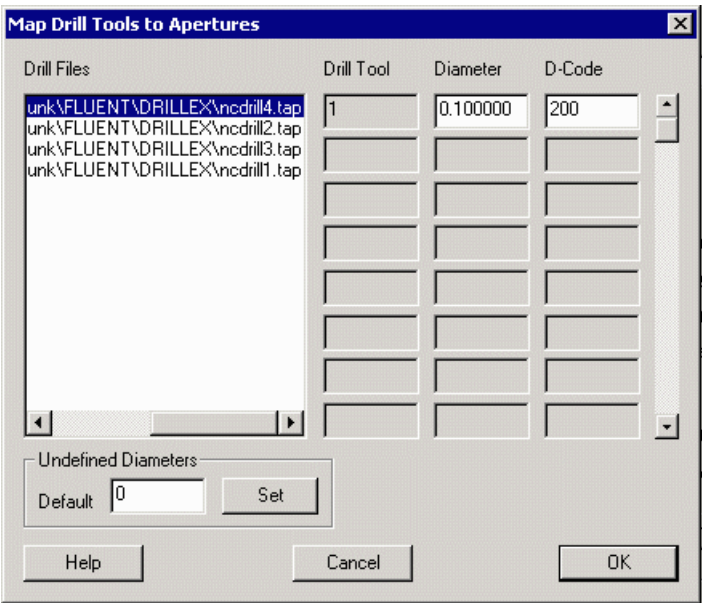
If the diameter is not specified then the user should enter this from the drill2gbr dialog.

For each drill file, the dialog will list the tools that were found. If a diameter was specified it will appear in that row. If not, the user should enter a diameter.
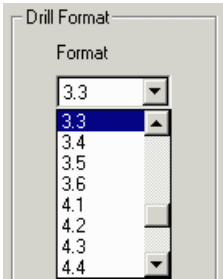
Setting the Drill Data Format

Back when drill data was stored on paper tape it was important to minimize the amount of data on the tape. One way this was done was by eliminating the decimal point in the coordinate data. In order to re-insert the decimal point it was neccessary for the drill file reader to be told how many digits were on the right and how many on the left of the decimal point. This information is known as the data format.

Drill Coordinate on Tape = 55567

```
Format          Coordinate Value
                after decimal re-insertion


2.3             55.567
3.2             555.67
1.4             5.5567
```


So when running drill2gbr it is necessary to tell the program the data format. If the wrong format is entered the drill coordinates will be off by factors of 10X - too big or too small. The format used for the drill data does not always match the format used for the Gerber data.

Drill Units

Drill units are either in mm or inches. The units of a drill file can often be determined by examining the header of the file. There are different ways of indicating the units:

M70 or M71 - if you see either of these (never both) in the beginning of the file you can determine the units. M70=inches and M71 = millimeters

INCH, METRIC - if you see either INCH or METRIC in the header of the file you can tell the units from this command. Typically INCH or METRIC is followed by a command and a second command such as TZ i.e.

INCH, TZ

If drill2gbr finds either of these commands in the drill file it will fill in the appropriate radio button in the main menu. If the radio button is blank then drill2gbr did not find the units info and the user must select one or the other.
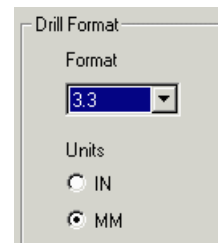
Drill Zero Inclusion

Another method of reducing the drill file size is to remove either the extra leading zeros or trailing zeros. However in order to re-insert the decimal point (which is also missing) one must know whether the leading zeros or trailing zeros have been removed. Sometimes this information is present in the header of the file:

INCH,TZ - indicates trailing zeros are included
INCH, LZ - indicates leading zeros are included

One major difference between Drill and Gerber is that for drill the header tells which zeros are included and for Gerber the header specifies which zeros are excluded.

It is also possible that both leading and trailing zeros are included. There are even some rare cases when both leading and trailing zeros are missing but the decimal point is then included.
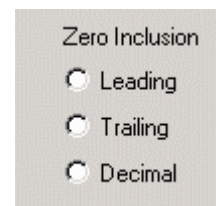
If after loading the drill files, one of the radio buttons is checked, the drill2gbr was able to determine which zeros are included. If none of the radio buttons are checked the user will need to determine this and check the correct button.
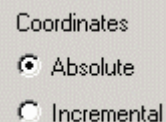
Drill Coordinate Mode

The x,y coordinates found in the drill file can be either absolute or relative.

>**Absolute** - each coordinate is expressed in the absolute x,y coordinates where the drill hole should be located.

>**Relative** - the first coordinate pair is absolute. Each successive data line represents the difference between the previous location and the current location.

Sometimes the header of the drill file will have a G90 or G91 where G90 indicates absolute coordinates and G91 indicates relative coordinates. In this case, drill2gbr will know how to make the conversion. However if the G90/G91 is missing from the drill file the user will have to choose absolute or relative.

Offset Drill Coordinates

The purpose behind converting drill data to Gerber data is so that one can overlay the drill holes on the Gerber layout. From time to time one will run across drill data that has been offset some unknown amount from the coordinates of the Gerber data. Drill2gbr allows the user to compensate for this by entering an offset value for the conversion.

Procedure

Run the drill2gbr conversion with no offset. Then view the resulting drill file together with one of the reference Gerber files using GBRVU.
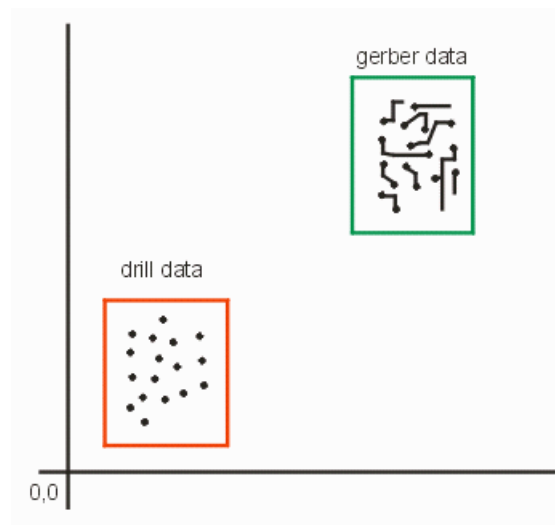
If possible, identify a drill hole that corresponds to a flashed pad in the Gerber data and carefully measure the difference. Use the value of the difference as the offset and run drill2gbr again. Now overlay the drill file and Gerber file in GBRVU again. Since the two files are much closer you should be able to zoom in very tightly and veri-fy that the offset value has aligned the drill and the Gerber. If there is a small difference, measure it and then use this difference to mod-ify your original offset values.

Overlaying Drill and Gerber Data

To verify that the drill to Gerber conversion went correctly, load the drill files and a reference Gerber file together using GBRVU. **Hint:** In order to properly see the drills, select your drill files first (holding down the Ctrl Key) and then the Gerber file last. Since they are painted in reverse order of selection the Gerber file will be painted first and the drills will be painted last so they are visible.
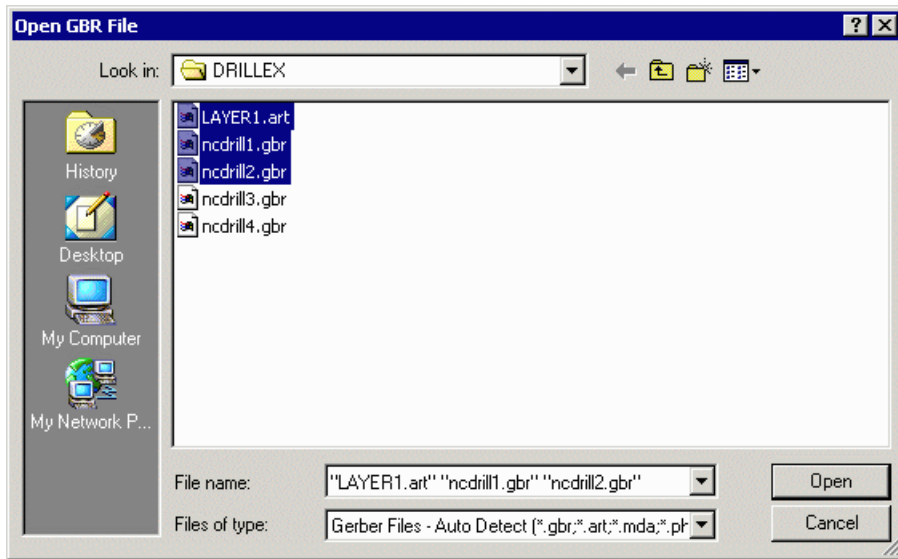
In this example we loaded ncdrill1.gbr and ncdrill2.gbr which were generated from Cadence .tap files together with layer1.art. (we didn't load ncdrill3 or ncdrill4 because they were inner layer vias.)

You can clearly see that the drills line up with the via pads on thetop metal layer of the IC package.
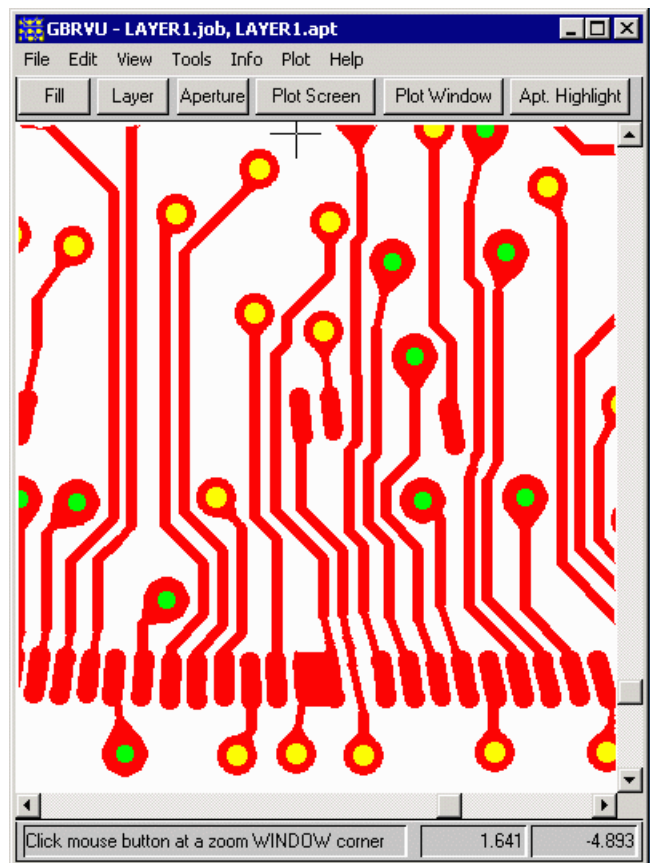
Recognizing a Drill File

It is useful to be able to open a file and determine first, if it actually is a drill file, and second what information it contains and what might be missing. Drill files must be ASCII files so they can be opened in Notepad, Write or Word. Be careful when saving them (if you make modifications) that the output format is text and not RTF or some other non-ascii format.

A typical drill file might look like the one below (comments in italics are not part of the file)

```
M71                              M71 = millimeters (M70 = inches)
%                                % = start of data
G05                              drill mode (as opposed to route mode)
G90                              data coordinates are absolute (G91=relative)
T001                             T = tool select followed by tool number
X018600Y016200                   X,Y = drill coordinates
X018600Y020200
X018600Y018200
X018600Y022200
X018600Y024200
X018600Y026200
X018600Y028200
X018600Y030200
X018600Y032200
X018600Y034200
X018600Y036200
X018600Y038200
X018600Y040200
X018600Y042200
X018600Y044200
X018600Y046200
.
.
.
M00                              M00 = halt program (or M30 stop & rewind)
```

In the above file it is clear that neither leading nor trailing zeros have been suppressed. One cannot tell the exact format of the data but you can tell that the sum must = 6. so the format could be 2.4 or 3.3 or 4.2. If you have some idea of the extents of the board you can eliminate one or more of the possibilities. In this particular example the correct format is 3.3.

The tool select command, T includes a tool number but does not include the diameter defintion. If you see a "C" in the tool command, the value that follows the C is the tool diameter.

```
T001C0.050
```

indicates that this is tool #1 and it has an 0.050 diameter tool.