



DYCONEX

ADVANCED CIRCUIT TECHNOLOGY

TOOLING REQUIREMENTS



Table of Contents

2.	Introduction	3
3.	Transferring Data to DYCONEX	3
3.1	E-mail	3
3.2	FTP	3
4.	Data Requirements	4-7
4.1	Input Formats	4
4.2	File Naming	5
4.3	NC Drill Data	5
4.3.1	Basic requirements	5
4.3.2	Plated and none-plated holes	5
4.3.3	Blind Vias / Buried Vias	5
4.4	Apertures	5
4.5	Multiple Flashed Pads	6
4.6	Net List Data	6
4.7	Data Polarity in Barco UCAM	7
5.	Documentation Requirements	7-8
5.1	Header File	7
5.2	File Description	8
5.3	Aperture List	8
6.	Fabrication Drawings	8-9
6.1	Blank Drawing	8
6.2	The Drill Drawing	9
6.3	The Delivery panel Drawing	9
7.	Summary	9

2. Introduction

This Tooling Requirements Manual aims to provide a short overview of the PCB tooling requirements, processes and capabilities at DYCONEX in Bassersdorf.

We will not assume to cover every last issue with this manual, as we are aware that every project as well as it's designer are unique. Open and accurate communication, though, are very important keys for the success of any project.

Meeting the requirements and following the recommendations in this manual will lead to clarity, shorter tooling time, reduced costs and a significant reduction of potential misunderstandings, oversights and/or errors.

Aside from the information that we must have, there is also information that would be useful to have such as the field in which the board is used, and information about the actual application.

3. Transferring Data to DYCONEX

Data transfer is possible in several ways. If none of the following transfer methods is applicable, please contact us.

3.1 E-mail

The best way to transfer data to DYCONEX is via e-mail. Please send the data (max. 15MB) to the Key Account Manager in charge of your account. If you do not have that information then please use the following address: sales@dyconex.com

Considering the low data throughput during the transfer, we strongly recommend compressing the data.

(To download Ultimate Zip www.ultimatezip.com/download.htm)

3.2 FTP

Please send an e-mail to sales@dyconex.com to inform us of recent uploads (or for the first transfer to receive login user name and password). Note that for security reasons all the files (including your own) in the target directory are hidden.

Transfer with an FTP Client Program

<i>Host name:</i>	ftp.dyconex.com
<i>User name/Password:</i>	dycodata/dycodata
<i>Directory:</i>	...\download ...\upload
<i>Transfer type:</i>	binary

4. Data Requirements

The DYCONEX tooling centre uses the CAD/CAE program UCAM from Barco ETS.

4.1 Input Formats

The following file formats can be processed. The ranking refers to data import speed and data quality.

PCB formats (preferred):

- 4.1.1 Gerber (with separate aperture list)
- 4.1.2 Extended Gerber
- 4.1.3 Barco UCAM (.dpf) (Please include the *.job file)
- 4.1.4 ODB++

Standard CAD formats (only if no other formats are available):

- 4.1.5 .hpgl
- 4.1.6 .dxf
- 4.1.7 .dwg

Required data encoding: ASCII

To reduce pre-tooling and tooling time (and minimise the non-recurring costs), the following should be considered:

- The files should include no unexpected control characters or header entries
- The record length should fit in one line in order to edit the file with one of the usual editors
- A consistent end of block character is requested
- There is no restriction concerning the used units as long as the coordinate format has a suitable resolution
- To avoid a complicated alignment process, please adjust all files with the same CAD offset
- We recommend you do not mirror CAD data
- The layers should be viewed from the topside of the PCB

Please note that fast data import does not necessarily mean faster or easier editing of the data.

4.2 File Naming

File and layer names are as varied as projects and designers. None-the-less we do have customized names for Copper artwork, masks, drill and mill files at Dyconex. We recommend simple and logical naming of layers and NC data.

4.3 NC Drill Data

4.1.1 Basic Requirements

Make sure to include drill size information with tolerances in a separate document. Such a document should also have information about the file names, the number of holes, as well as the mention of «from» and «to» layers. Please specify if the given diameters are drilled hole size or finished hole size.

4.1.2 Plated and None-plated Holes

Try not to use the same tool numbers for plated and none-plated holes. If a separate file for none-plated holes is created (recommended), continue the tool number sequence from the plated holes.

4.1.3 Blind Vias / Buried Vias

Using this technology inevitably means more drill files.

Please make sure that the drill information for each of these files is separately documented.

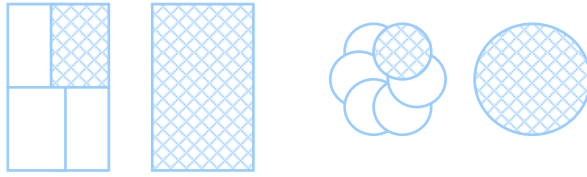
4.4 Apertures

Contours are extremely difficult to edit. DYCONEX tooling must edit the design data for manufacturing (e.g. if the finished hole size is given, engineering calculates the drill Ø necessary to achieve that finished size).

When possible, avoid using contours to define the artwork, even if they reduce the data size. Simple standard apertures simplify tooling work a great deal.

If special apertures are needed, please specify the shape (with dimensioning) as clearly as possible and include a mechanical drawing.

4.5 Multiple Flashed Pads



A design that has standard figures (circles, rectangles) created with multiple flashes, may encounter processing problems (e.g. for the netlist generation script, which tries to set a test-needle at the centre of every flashed pad). The tooling-operator has then to convert such multiple flashed pads manually to single flashed ones. This does increase the tooling cycle time, and Design Rule Check (DRC) time.

When possible, (single-) flash as many standard figures as possible

For special apertures, multiple standard figure flashes are preferred over draws and/or contours.

4.6 Net List Data

The question often arises if it is possible to use a net list output from the design side to develop the electrical test program or to compare with the net list generated at DYCONEX. Net lists generated by design systems often have the following difficulties:

- The net list contains all net points and not only end points
- The coordinates have an offset to our CAM data
- The net list data do not include solder mask information
- The net list data do not include routing contour information
- There are multiple test points on one pad
- The design contains test pads (in-circuit tests) which are not in the net list as net point
- The net list version does not always equal the design version (very difficult to verify)

For these reasons, a net list will be generated by the tooling station, so as to meet the requirements specified by the responsible contractor for electrical testing.

Depending on the complexity of the design, a net list comparison can be very complicated but not impossible if required. Furthermore, comparing net lists does not reduce the necessity of the visual comparison.

4.7 Data Polarity in Barco UCAM

Barco UCAM uses positive data only. Therefore negative or mixed data have to be converted into positive data with the reverse tool, creating several problems:

- The positive data created by UCAM (out of negative or mixed data) consists primarily of contours. As stated above, these «apertures» are extremely difficult and time consuming to modify.
- The resolution of the transferred positive data is quite often not adequate, since Barco UCAM converts arcs into segments. Fine structures will not be translated properly

To avoid these problems:

- Avoid negative or mixed data
- Refrain from designing with two positive layers that have to be merged negatively.

5. Documentation Requirements

The Layout data is essential to the tooling of a PCB. There is, however, a list of additional information, which we require in order to properly analyze the data and plan the manufacturing of your board. Failing to deliver this information in whole or in part will cause delays in the process.

5.1 Header File

The header file should contain the following information:

<i>Part number</i>	A valid Part number or a customer's board ID
<i>Blank board revision</i>	A valid board revision marker or a customer's version label
<i>Artwork date</i>	The latest artwork modification date
<i>Contact</i>	Name, phone number and e-mail address
<i>Design class</i>	Specify the IPC design class and the smallest design structure
<i>Plating type</i>	Defines the required plating process
<i>Layer construction</i>	Defines the required board thickness, number of layers, base material and build-up
<i>Electrical test</i>	Is an electrical test required?
<i>Solder mask type</i>	Please specify the solder mask type required
<i>Format information</i>	Coordinates, units, leading or trailing zeros
<i>Board size</i>	Please specify board and delivery panel sizes (x and y, including units and tolerances)

5.2 File Description

The file description should contain the following information:

- A list associating every file name to the corresponding layer name.
- Specific instructions regarding merge partner and merge sequence.
- Any other relevant information.

This information is important as there are often no layer identification labels on the CAD data, resulting in difficulties to assign layers correctly in the build-up.

Such a list will also enable us to verify that all of the files are in the data package.

5.3 Aperture List

When regular Gerber files are used as format, please specify for each aperture:

- Shape and dimensions (with units) for rectangular and circular apertures
- A mechanical drawing with all the relevant dimensions for special apertures
- All d-code positions with their associated aperture

6. Fabrication Drawings

Please be sure to include the following fabrication drawings:

- Blank drawing
- Drill drawing
- Delivery panel drawing

6.1 Blank Drawing

The blank drawing is needed to extract board outline and dimensions for internal route features, fiducials and tooling holes.

The following points should be regarded:

- All mechanical components must be so described and dimensioned so as to be able to reconstruct them during programming.
- Make sure to add tolerances to the dimensions.
- Mark the drawing revision in the information block.
- Include any fiducials in the drawing with dimensions as a control reference for copper artwork and mechanical parts.

6.2 The Drill Drawing

The drill drawing should show the board outline with all drilled holes.

The following points should be considered:

- Choose clear symbols for each drill Ø.
- Mark the drawing revision in the information block.

It is of course possible to combine the blank and the drill drawing, providing that the details are clear.

6.3 The Delivery panel Drawing

If a special panel layout is requested, a delivery panel drawing must be added.

In such a case the drawing should contain:

- Stepping information (X and Y)
- Position of single board in the delivery panel (offset to reference point)

7. Summary

Clarity and simplicity are the keys to speed (cost) and integrity of data transfer and preparation whether it be for quotation or for manufacturing.

The more information we have at the offset of a project, the more accurate the quotation will be and the easier and more efficient the tooling and planning will be.

Someone once said: «The only stupid question is the one you don't ask.»

If you have any questions at all or if something in this manual is not clear, please do not hesitate to contact us.

For questions please contact your Key Account Manager:

sales@dyconex.com

DYCONEX AG

Grindelstrasse 40

8303 Bassersdorf

Switzerland

Phone +41 43 266 1100

Fax +41 43 266 1101

mail@dyconex.com

www.dyconex.com

DYCONEX Inc.

1839 S. Alma School Rd., Suite 205

Mesa, AZ 85210-3024

USA

Phone +1 480 775 6878

Fax +1 480 775 6880

