

TESTABILITY ANALYSIS SERVICE

GENERAL INFORMATION

UNI-FIX Test Fixtures offers testability analysis of your PCBA as a means to prevent wasted dollars and time in developing test fixtures and programs for a product that has limited access. In many cases, this limited access may be to crucial nets, which could severely limit the test.

With this service, UNI-FIX will analyze your PCBA and return to you a series of files, which will tell you if your assembly can be tested with a single or dual-side access fixture, and which nets cannot be accessed. If desired, we can also tell you how many nets will require probing with .100", .075" or .050" probes. We can supply you with information on net access, which is currently blocked by soldermask or components, and even recommend where net access may be obtained by removing soldermask. All this is done in a "Test Fixture" environment to insure a fixture that will be both repeatable and reliable.

If you are developing programs on a Agilent 3070, and do not have CAD data available, UNI-FIX can supply you with the information you need to input to the HP3070 for program development.

To perform this service, UNI-FIX requires the following information in ASCII text format:

Gerber Files *

Current Revision

ALL Layers including Mask, Paste and Silk-screen

Gerber Aperture List (or Gerber in RS274X Format) *

Current Revision

Must Include D-Code Definitions

CAD ASCII Database File *

IPC-D-356-A preferred

Assembly Drawing *

Current Revision

Hard Copy (legible), PDF or DXF (preferred) format

UNI-FIX TESTABILITY ANALYSIS SERVICE

Page 2 of 4

To reduce cost and lead-time of this service, as well as increase accuracy, the following information is requested but is not mandatory:

Net List **

Current Revision

Schematics **

Current Revision

Hard Copy (legible), PDF (preferred) or DXF format

Drill File

Current Revision

Fabrication Drawing

Current Revision

Hard Copy (legible), PDF or DXF (preferred) format

Bare PCB

Loaded PCBA **

The files indicated with an asterisk (*) are required files for Testability Analysis. Some files listed with a double asterisk (**) may be interchanged, but cost and lead-time may be affected. The Gerber files and aperture list are always required, however the CAD ASCII database file may be substituted with a schematic and assembly drawing or with a netlist and assembly drawing. Doing testability without the CAD ASCII database file will increase cost and lead-time of the study.

In some cases the testability study may be used to output a net list for the customer if one is not available, or to output the required files for HP3070 program development when CAD is not available. In performing this task without the CAD ASCII database file, UNI-FIX will arbitrarily assign pin numbers for various devices that may not match the original database file (such as pin numbers of capacitors and resistors). In this case, the pin-outs will match the net lists generated by UNI-FIX and therefore they should be used for test program development.

To ease the translation of the UNI-FIX testability reports, the Gerber files supplied to UNI-FIX will be edited to use the same datum and orientation as the original CAD database or PCB fabrication drawing.

The testability study generates arbitrary net numbers for all single and multiple-pin nets on the board. By overlaying the IPC-D-356-A database file, UNI-FIX can create a cross-reference report in Excel format that will list all information by the original net names. The reference is then output in four (4) formats, standard net list (xxxx.NET), HP3070 net list (xxxx.HP), GenRad net list (xxxx.CKT) and Mentor net list (xxxx.MNT). The study also generates a listing of all accessible holes on the board (xxxx.ALL) and the net it is related to (with reference to the UNI-FIX generated net numbers). In addition, the study generates the optimum probe position based on the desired outcome (xxxx.PTS).

UNI-FIX TESTABILITY ANALYSIS SERVICE

Page 3 of 4

The testability also generates a component/pin file (xxxx.CPT) which lists the coordinates of each pad/pin of each device, the type of device (through hole or surface mount), the side of the board that the device resides on and the arbitrary net number of the device.

Finally, a report file (README.xxx) is generated which will give the testability information including the percentage of accessibility in the current revision, requirements for single or dual-side access fixtures, the nets which are not accessible, the nets which could be accessible with modification of soldermask layers and the coordinates at which mask should be removed.

TOLERANCE ISSUES

In order to obtain a repeatable, reliable test, tolerance "stack-up" in both the board and the test fixture should be considered in the board design. The *average* test fixture manufacturer has the following manufacturing tolerances that should be considered:

Drill Tolerance	+/- .001"
Tooling Pin Hole Tolerance	+.000"/-.015"
Tooling Pin Installation Tolerance	+/- .001"
Receptacle Installation Tolerance	+/- .001"
Probe Radial Play	+/- .003"

.0135" Total Fixture Tolerance

*(Please note that the total manufacturing tolerance at UNI-FIX Test Fixtures is .0105")

When the fixture tolerances are added to the board manufacturing tolerances, you can determine the minimum target size for test as well as the proximity to devices. As the test fixture manufacturer has .0105" to .0135" tolerance in manufacturing, the tolerances in board fabrication need to be fairly tight to ensure repeatable test. The items to be considered in fabrication of the UUT include the following:

Cross-Diagonal Tooling Hole Tolerance	+/- .003"
Tooling Hole Size	+.003"/-.000"
Front/Back Registration Tolerance	+/- .005"
Pad Size Tolerance	+/- .001"
Pad Location Tolerance	+/- .002"

.025" Total Board Tolerance

With the board and fixture tolerances listed above, the tolerance stack-up is +.020"/-.0185" between fixture and board. In addition, it is recommended that the target size should be at least .010" larger than the total fixture and board manufacturing tolerances to allow for any marginal fabrication tolerances that have been allowed to pass the PCB inspection. In the case shown here, the minimum recommended target size should be .030" (.010" larger than the maximum UUT/fixture tolerance). Additional board manufacturing tolerances would require an increase in target size to insure a repeatable, reliable test.

UNI-FIX TESTABILITY ANALYSIS SERVICE
Page 4 of 4

UNI-FIX has built repeatable, reliable fixturing for .010" diameter targets. This requires a number of factors to be considered in both the UUT and fixture design/fabrication. The board tolerance "stack-up" needs to be no more than .006" and the test fixture tolerance "stack-up" needs to be no more than .004". Engineering and fabrication of special probe guide plates is required to reduce the "radial play" of the probes, and special "diamond-cut" tooling pins are required to allow a closer fit of the tooling holes to the fixture without binding. Although it is possible to build a test fixture for these requirements, the cost and maintenance of the test fixture increase with reduction of target size, and the tolerances allowed in the fabrication of the board must be very tight to accomplish this.

DESIGN TIPS

The following basic design tips should be considered in the "mechanical" design for testability:

- 1) Place test points on one (1) side of the board whenever possible.
- 2) Test access side should be the side of the board with the lowest component profile.
- 3) Insure test target size is at least .010" larger than board and fixture builder tolerance stack-up.
- 4) Include a minimum of two (2) tooling holes in opposite corners (diagonally).
- 5) If board is larger than 100 square inches, add a third tooling hole.
- 6) Make sure tooling holes are unplated.
- 7) Make tooling holes .125" diameter (or greater) if possible.
- 8) Allow a maximum tooling hole tolerance of $+.003"/-.000"$.
- 9) Do not place test pads closer than .025" to the body of any component up to .375" tall.
- 10) Do not place test pads closer than .075" to the body of any component over .375" tall.
- 11) Do not allow solder mask to fill the opposite end of a via to be used for test.

If you have any questions, please contact us at (602) 843-8565.