

Converting from Cadence Allegro to Agilent's ADS

Abstract

This paper describes how to transfer the layout of a multi-layer test board from Cadence Allegro into Agilent's ADS for the purpose of simulating critical nets and deriving S-parameters.

The Allegro layout is exported as Gerber and drill data; a program called NETEX-G reads the Gerber/Drill, retraces the connectivity of each net and outputs the data in EGS archive format - a database native to Agilent's Advanced Design System.

NETEX-G also allows the user to specify a window or a selected set of nets for extraction. This ability is quite important as it is not possible to simulate the entire board in a reasonable amount of time.

ARTWORK CONVERSION SOFTWARE, INC.

Introduction

Many PCB layouts are done using Cadence's *Allegro* program. At the same time Agilent's *Advanced Design System* (ADS) is used extensively for RF and microwave design, circuit layout and simulation. ADS includes a field simulator program known as *Momentum*. In order to use *Momentum* one first must create a circuit geometry in ADS's layout module.

Currently there is no direct path between Allegro's database and the ADS database so the conversion must pass through some intermediate file format. Attempts to use a format that can be both exported by Allegro and imported by ADS (such as DXF) have not worked – primarily because there is a mismatch between the way Allegro describes certain geometries and the way ADS supports the same geometries.

Artwork offers a program called NETEX-G which was designed to read Gerber/Drill data and export a new geometry file that is organized by net. Further, NETEX-G can manipulate geometries - for example it can take a polygon with cutouts and convert it to a group of polygons with no cutouts that cover the same region. Such boolean operations are essential when converting from one database to a different one in cases where both data formats do not represent polygon cutouts in the same fashion.

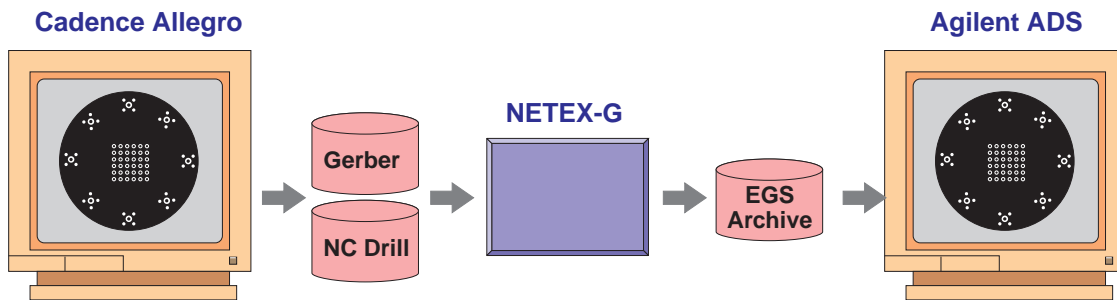


Figure 1: A layout in Allegro can be moved to ADS by exporting Gerber/NC Drill from Allegro. The Gerber/Drill data is brought into NETEX-G which processes the geometry and outputs an EGS Archive file. ADS imports the EGS archive file.

In order to support ADS Artwork has recently added the EGS Archive format to the existing DXF, GDSII and Ansoft formats already supported by NETEX-G. The rest of this paper describes in detail both the steps required for a successful transfer and points out special issues that arise in doing the conversion.

Why Gerber?

Why use Gerber/NC Drill instead of writing a direct interface from Allegro to ADS?

NETEX-G was written as a general purpose tool; since all PCB/package layout tools write Gerber/NC Drill NETEX-G can be used with any PCB tool for any layout without restriction. Writing a direct converter requires very detailed understanding of both databases and regular updates to the converter when either database is modified. The developer also must then write many direct converters. By using a standard format such as Gerber the number of formats that need to be supported is greatly reduced.

Example - High Frequency IC Test Board

In order to illustrate the process, we'll use a high speed test board - a round board on which an IC is mounted. The PCB has multiple coaxial connectors which feed test signals to the IC. In order to characterize the IC's frequency response one must "subtract" the effects of the test line. To do this one creates a S-parameter model of the test board. The board shown below was designed using Cadence Allegro PCB Expert.

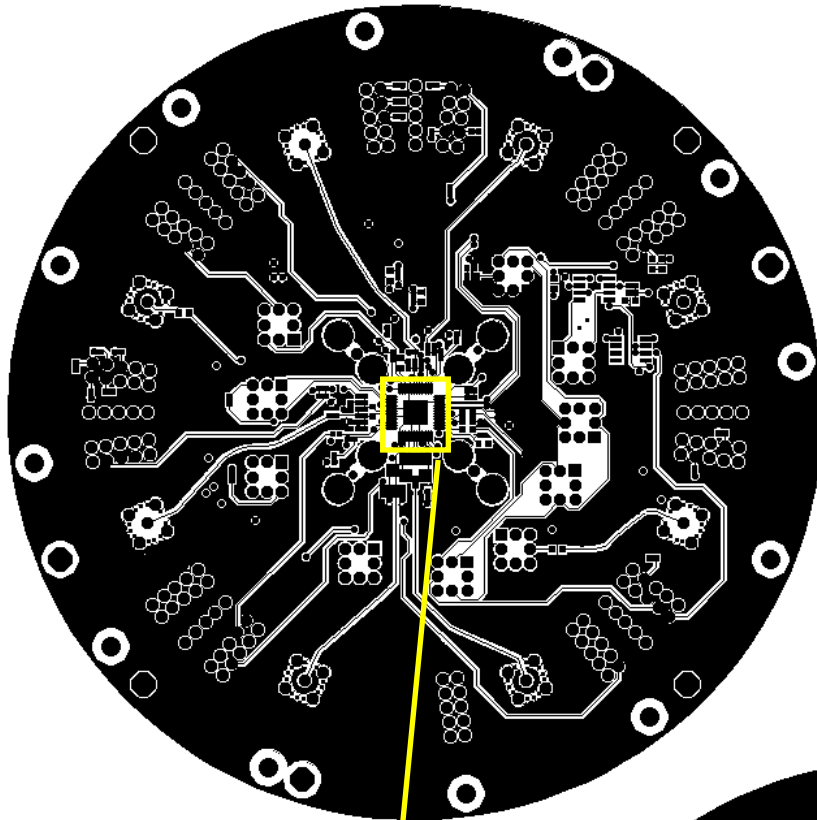


Figure 3: bottom view of test board. The DUT is mounted in the center of this side of the board.

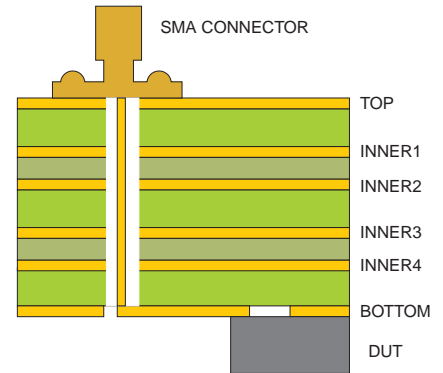


Figure 2 - the test board is a 6 layer design with 4 inner layers to handle power and ground. The DUT is mounted on the bottom. The illustration is not to scale: the thickness of the total board is typically a few mm.

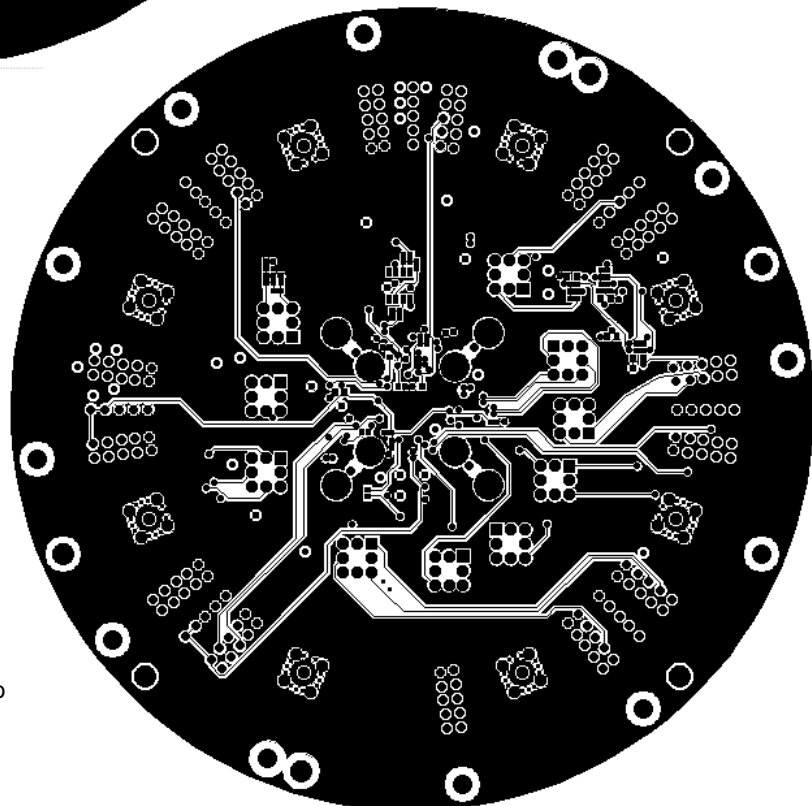


Figure 4: top view of board. The SMA coaxial connectors mount to this side of the board.

Exporting Gerber/NC Drill from Allegro

Exporting Gerber and NC Drill from Allegro is straightforward; after all the PCB cannot be built unless both are correctly produced and sent to the fabrication shop. To get Gerber files out, you must use the **Manufacturing | Artwork** dialogs. There are two tabbed dialogs. One is used to set up general parameters and the other defines the photoplot layers and how to build them.

General Parameters

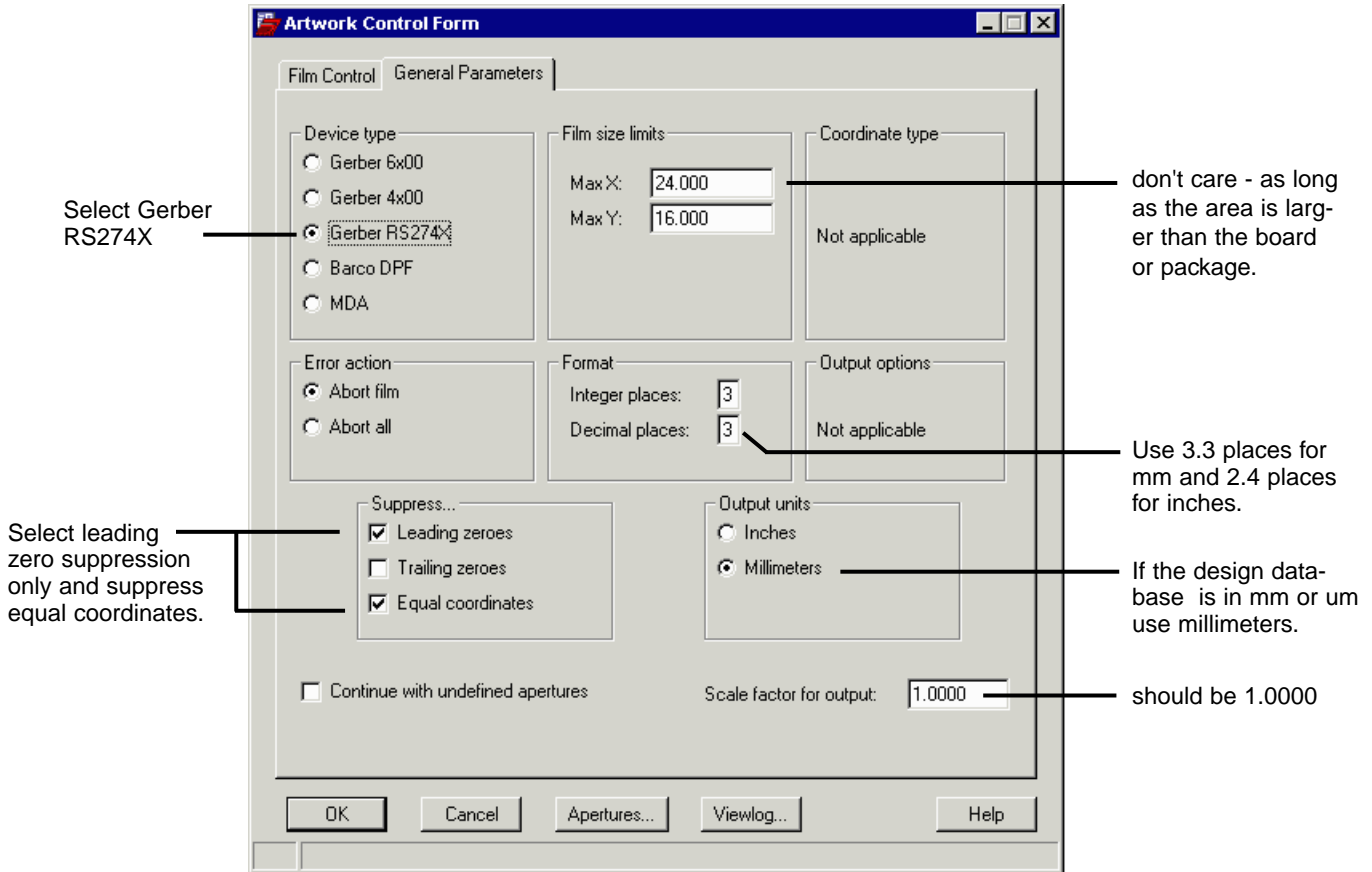


Figure 5: Allegro's Artwork Control Form Dialog

Film Control

The film control dialog allows the user to define a list of “films” and to define which classes/subclasses will be used for each film. While at first this may seem very complicated, there are only a few things to remember when creating Gerber files for NETEX-G.

1. NETEX-G only uses Gerber files for the conductor layers - all other films (solder mask, solder paste etc... are not needed)
2. For each conductor film one needs only three types of classes: pins, vias and conductor.
3. Do not offset, mirror or rotate any of the films.
4. Film polarity should be positive, even for power planes. Although negative polarity will work with NETEX-G one must take care to indicate this when running NETEX-G. Since we are using RS274X output, there is no advantage to selecting negative polarity.

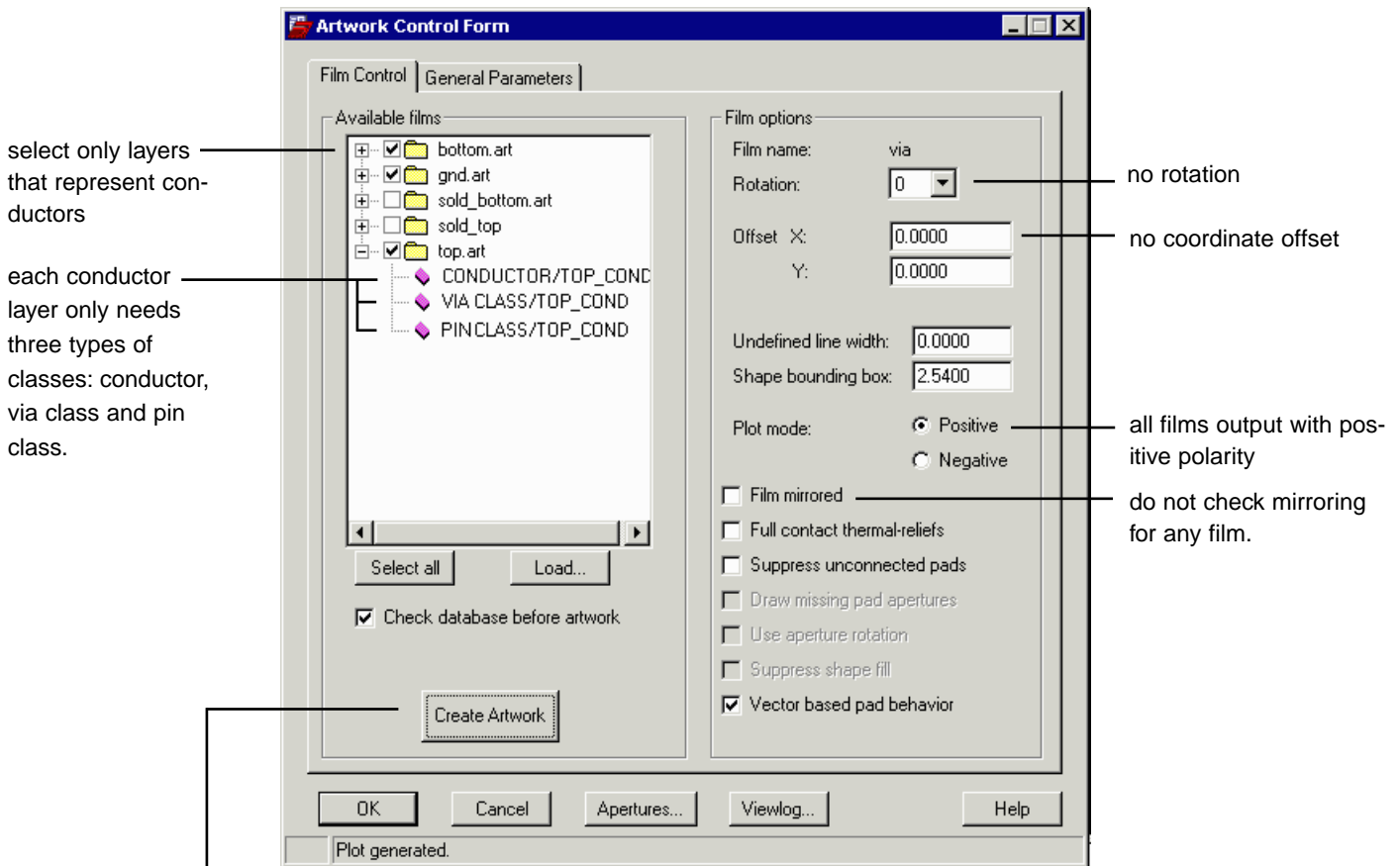


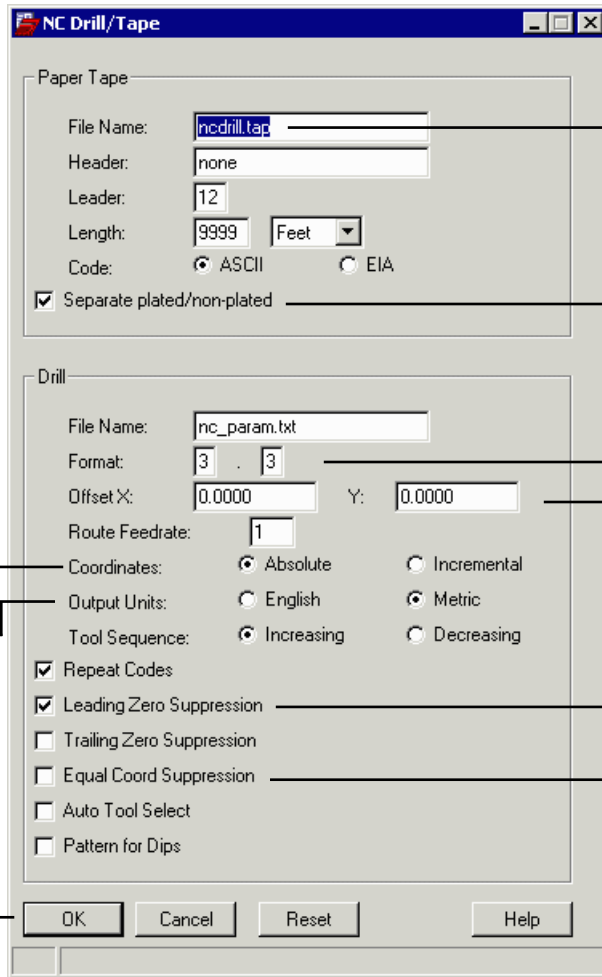
Figure 6: Allegro's Film Control Dialog

When all parameters are correctly set and the films you need checked, then press Create Artwork. Allegro will write a series of .art files to the current working directory.

Exporting NC Drill from Allegro

The Gerber files defines the horizontal geometry -- NC Drill files define how the individual layers are connected vertically. Each drill hole results in a plated through hole that connects two (or more) layers together. To output NC Drill data one must first set up the drill parameters and then create a drill "tape." To set up the drill parameters use the **Manufacturing | NC | Drill Parameters** selection to open the Drill Parameters dialog box.

Drill Parameters Dialog



if multiple drill files are needed (for blind or buried vias) then *Allegro* will name them ncdrill1.tap, ncdrill2.tap and so on.

always check this box - otherwise if there are some non-plated through holes they will appear in the same drill file as the plated through holes.

for this example (where the design is in mm) I set the drill format to 3.3 and the drill units to metric.

do not offset the drill data from the design database.

use leading zero suppression.

it is OK to use Equal coordinate suppression if you like but not required.

use absolute coordinates.

match output units to the design database - in this case the design is in mm so select metric.

Once all settings are correct, click on OK. The settings are written to the file ncpam.txt Now you are ready to create the drill tape(s).

Figure 7: Allegro's Drill Parameter setup dialog

To write the file to disk use **Manufacturing | NC | Drill Tape** to open the NC Tape dialog. Make sure the scale factor = 1 and press **Run**.

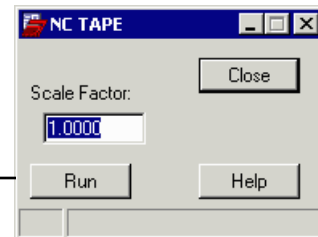


Figure 8: Allegro's Drill Tape generation dialog

Converting Drill to Gerber for NETEX-G

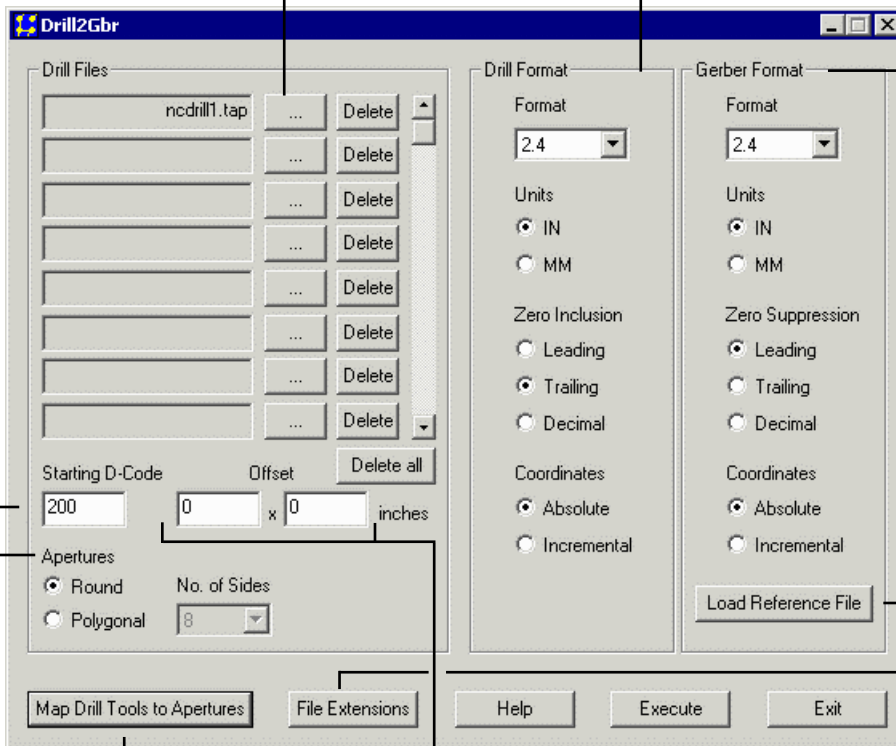
NETEX-G does not accept NC Drill files directly (this decision was made because NC Drill files vary greatly and it would be quite difficult to guarantee correct reading of these files.) The user must first convert the NC Drill files into an equivalent Gerber file in order to use it in NETEX-G.

This is done using the utility drill2gbr.

The drill files 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.

Figure 9: Drill2gbr's main dialog



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.

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.

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.

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

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.

Running NETEX-G

At this point we have exported Gerber and Drill from Allegro and run the drill data through the drill2gbr converter. All information needed to create the EGS Archive file is now prepared. NETEX-G is started and the data is loaded into NETEX-G's main dialog box.

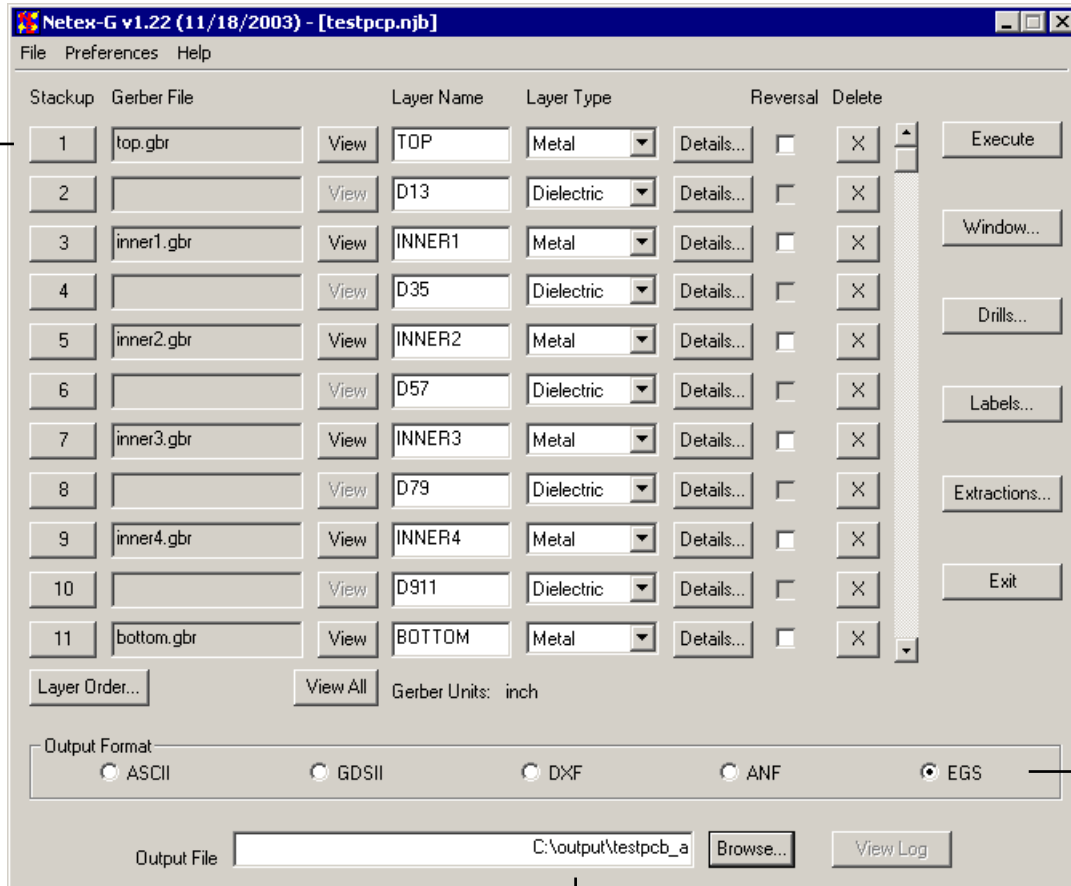


Figure 10: NETEX-G main dialog

Gerber Inputs - each conductor layer is entered into the "stackup" in its physical order starting at the top.

A dielectric must be placed between each conductor layer.

The Output File - the output file name/location is specified here.

Output Format - for ADS use the EGS output format.

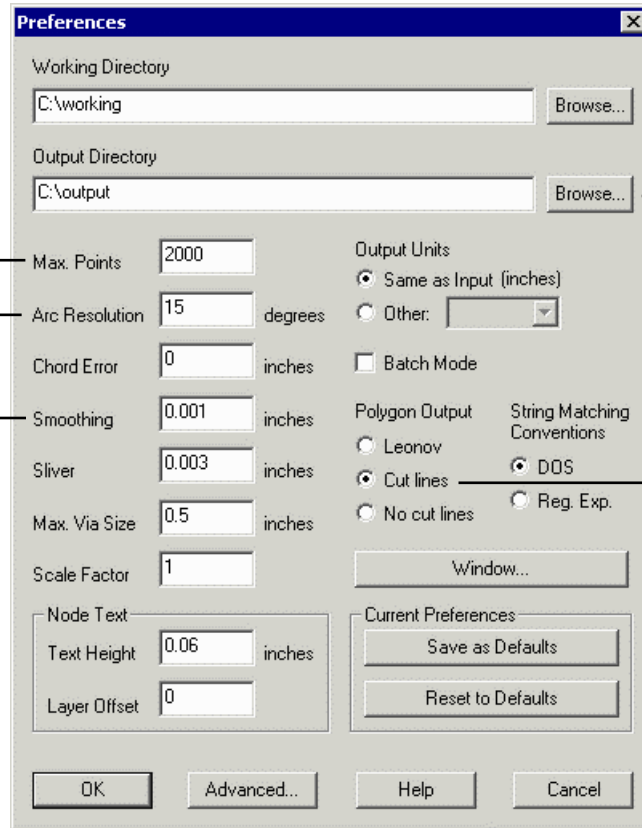
Drill - use the Drill dialog to enter a drill file and specify which conductor layers it passes through.

NETEX-G Preferences

Max Points - max number of points per polygon. For EGS output a value of 2000 is a reasonable setting.

Arc Res - controls smoothness of arcs. If set too fine, output will take too long to simulate in Momentum. If set too coarse the output may not resemble the true layout.

Smoothing - removes closely spaced vertices in the output polygons. Again, this is used to reduce the number of vertices that the Momentum field simulation must process.



Working Directory - where temporary and intermediate files are written to.

Output Directory - where the output file(s) are written.

Polygon Output - if EGS is the target format you should select Cut Lines.

Figure 11: Netex-G Preferences Dialog

NETEX-G Drill

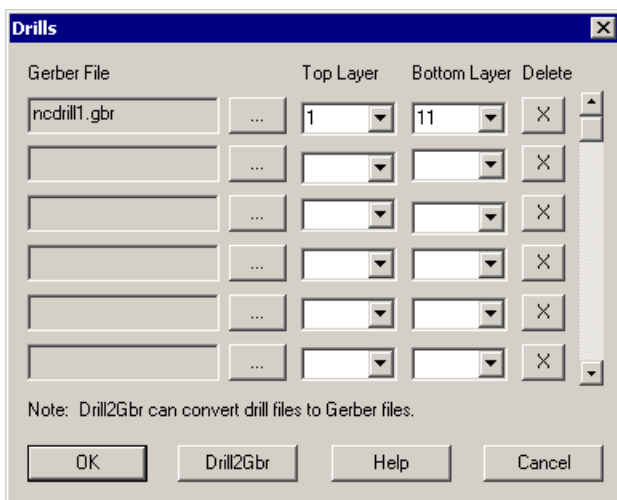


Figure 12: NETEX-G's drill dialog.

The drill dialog is used to define the vertical interconnections between layers. First, select the drill file (it must have already been converted to Gerber format). Then specify which layers it passes through.

In this example the drill is a through hole - it passes from the very top (stackup position 1) to the very bottom (stackup position 11).

NETEX-G will use this to create the required vias between layers. If you have blind or buried vias you can enter these files also.

Loading the EGS Archive into ADS

From Agilent's ADS, use the **File | Import | EGS Archive** dialog and select the file test_pcb_a. After importing (and turning off all layers except for the bottom one you will see something like the screen shot shown below.

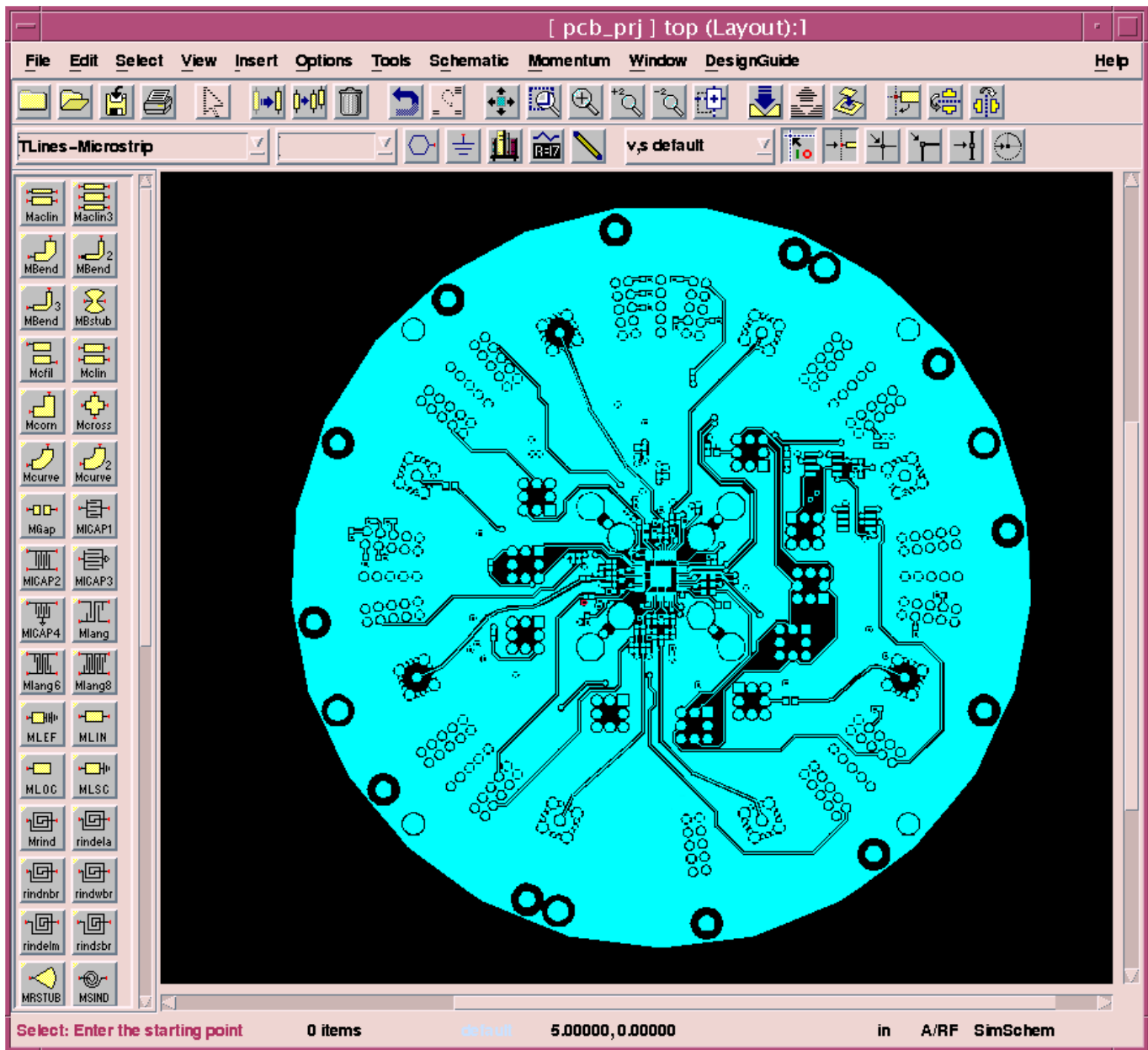


Figure 13: Screen shot of the test board translated from Allegro by NETEX-G and loaded into ADS.

Selecting Only a Small Window for Analysis

It is not useful to move the entire layout to ADS for analysis in Momentum for two reasons: only a limited number of circuit tracks are of interest; attempting to run a 3D field simulation on the entire board would take too much time.

There are only 5 or 6 critical high speed circuit tracks on this board that need to be simulated and there is no need to simultaneously simulate these nets because there is no interaction between them.

The illustration at right shows the bottom layer of the test board and highlights one of the high speed nets (in red). An outline (yellow) shows the area we would like to “clip” out in order to focus our analysis solely on this net.

How to Clip Out an Irregular Area

By default, NETEX-G converts the entire database. However the user has the option to select either a rectangular or polygonal window that defines the extents of the data. Anything that falls outside of the window is clipped and discarded.

Because of the radial geometry a rectangular window will select too much irrelevant data - a polygonal selection can select the desired net and enough surrounding groundplane to do an accurate simulation.

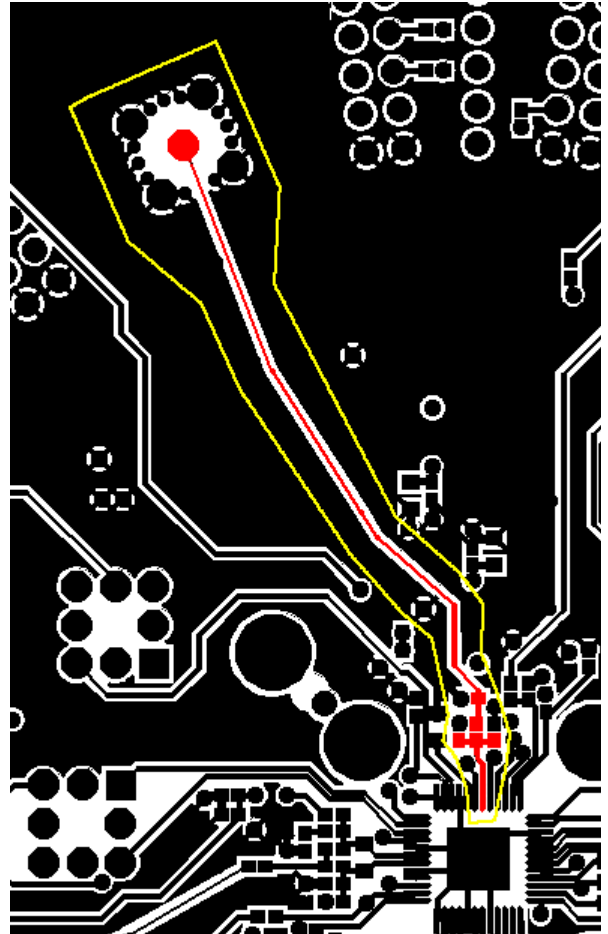


Figure 14: the circuit trace highlighted in red is one we wish to simulate in Momentum. The yellow boundary indicates the area we wish to clip out.

From the main NETEX-G dialog, click on the **Window** button to open the Window dialog. Select the radio button option labeled **Polygonal**. The click on the **Get Polygonal Data** button. This will open your Gerber files in GBRVU and make it possible to select a polygon shaped window.

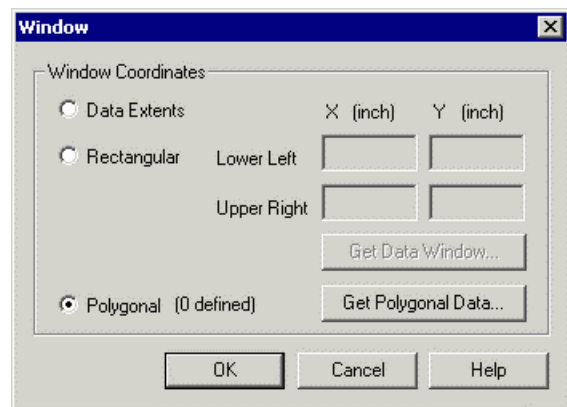
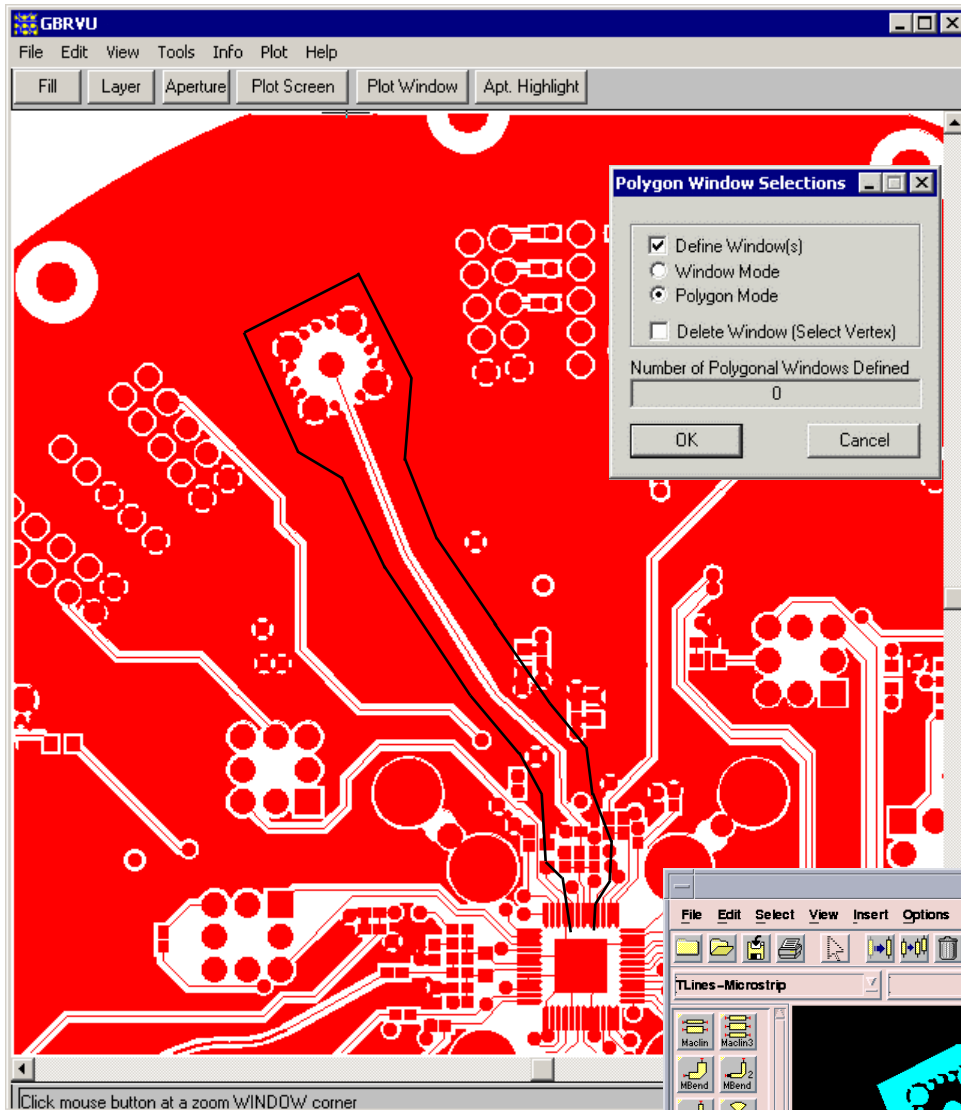


Figure 15 - NETEX-G’s window dialog.

How to Clip Out an Irregular Area cont ...

GBRVU will open and automatically load the Gerber files that are present in the NETEX-G stackup. You can turn off any layers that are not needed to visualize the area you wish to select. Zoom in to the area of interest. Now click on **Define Windows(s)** and select the **Polygonal Mode** option.



Using the right mouse button begin building your polygon. The left mouse button can still be used for panning as needed between vertex picks. To close your polygon return to the first point and click on it (again, with the right mouse button.) You will see the dialog pop back up. Click on **OK**.

Figure 16: GBRVU used to define polygonal window.

Figure 17 - Running NETEX-G with this new polygonal window results in a much smaller file as one can see in the ADS layout screen shot at right.

This file will be much easier to prepare and run through Momentum.

