# Using NETEX-G to Convert Gerber Data into Ansoft SI Tools

*ARTWORK CONVERSION SOFTWARE, INC.*

## Introduction

Engineers that need to simulate the high frequency behavior of they PCB or package layouts use Ansoft's 2D or 3D field analysis tools to create an equivalent circuit of each net.

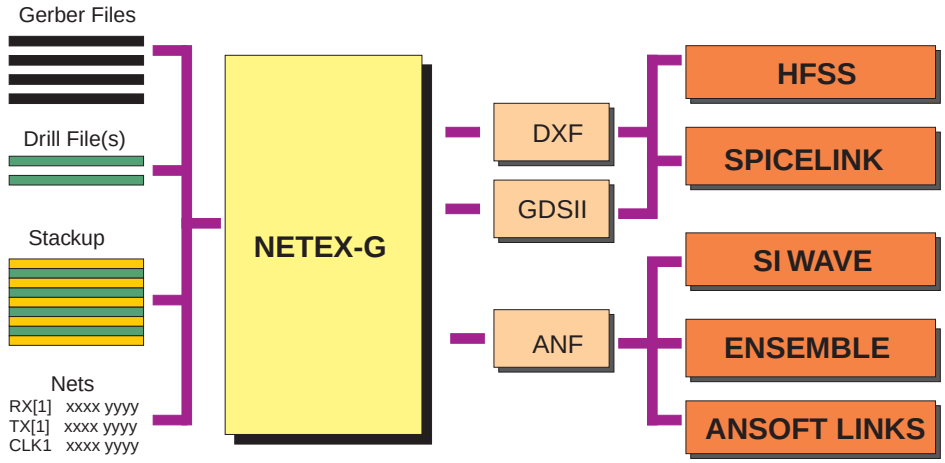
For many PCB and package layout programs there does not exist a "direct" interface to Ansoft that can extract both the needed geometries and the net information directly into the signal integrity tools.

Every PCB or IC package layout tool can export Gerber since without Gerber there would be no board.

Artwork Conversion has developed a program called NETEX-G (net extraction for Gerber) that uses the Gerber data (plus drill data) to rebuild the nets using boolean operators. Further, NETEX-G can attach probe points and net names to the data based on coordinate information. Finally, NETEX-G can collect adjacent conductors (the proximity net) needed to do capacitance calculations.

Unlike other programs that attempted this in the past, NETEX-G is extremely fast – results are obtained in a few minutes rather than a few hours. This is due to Artwork's advanced algorithms that were initially developed for the IC industry where the file sizes are two or three orders of magnitude greater.

### NETEX-G Flow Chart




| | |
|---|---|
| Gerber Data | provides the conductor geometry |
| Drill Data | provides the inter-layer connections |
| Stackup | used to determine connections |
| Nets | user selects which nets to extract |

| | |
|---|---|
| DXF | SpiceLink and HFSS |
| GDSII | SpiceLink and HFSS |
| ANF | SI-Wave, Ensemble and Ansoft Links |

Any PCB or IC package design tool can export Gerber and Drill data since this is required to fabricate the board. Netex builds connectivity and extracts user specified geometry by net. It can also extract any nearby geometry. The results can then be pulled into many of Ansoft's Signal Integrity tools.

## Inputs to NETEX-G

NETEX G uses the following input data:

#### 1. Gerber Files

These must be in RS274X format and must share common units, format and mode. The files for the various layers should be aligned. This is generally the same information used for producing the artwork.

#### 2. A Drill File

Drill data is used to build the vias - vertical interconnects between layers. Through holes, buried and blind vias are all supported by defining the start and end layer for the drill file. In actual operation the drill file is first converted to Gerber so that it can be merged withthe Gerber data defining the conductors. Of course, if you are analyzing a surface only no drill file is needed.

#### 3. Stackup Definition

The user must enter a "stackup" telling NETEX-G the physical order of the conductor layers. In addition to knowing the order, the user should also know the "thickness" of each layer -- both conductor and the dielectric between conductors.

#### 4. Net Coordinates

While not absolutely essential, in most practical cases the user wishes to analyze a particular net or group of nets. In order to select and isolate a net from the board it is required to specify the net name and a coordinate point that "sits" somewhere on the net. This information can be obtained in several ways. Sometimes an ASCII file is available (IC packages typically have bond finger coordinates) or the GBRVU program can be used to view and measure coordinates of the desired nets.

#### 5. Node Coordinates

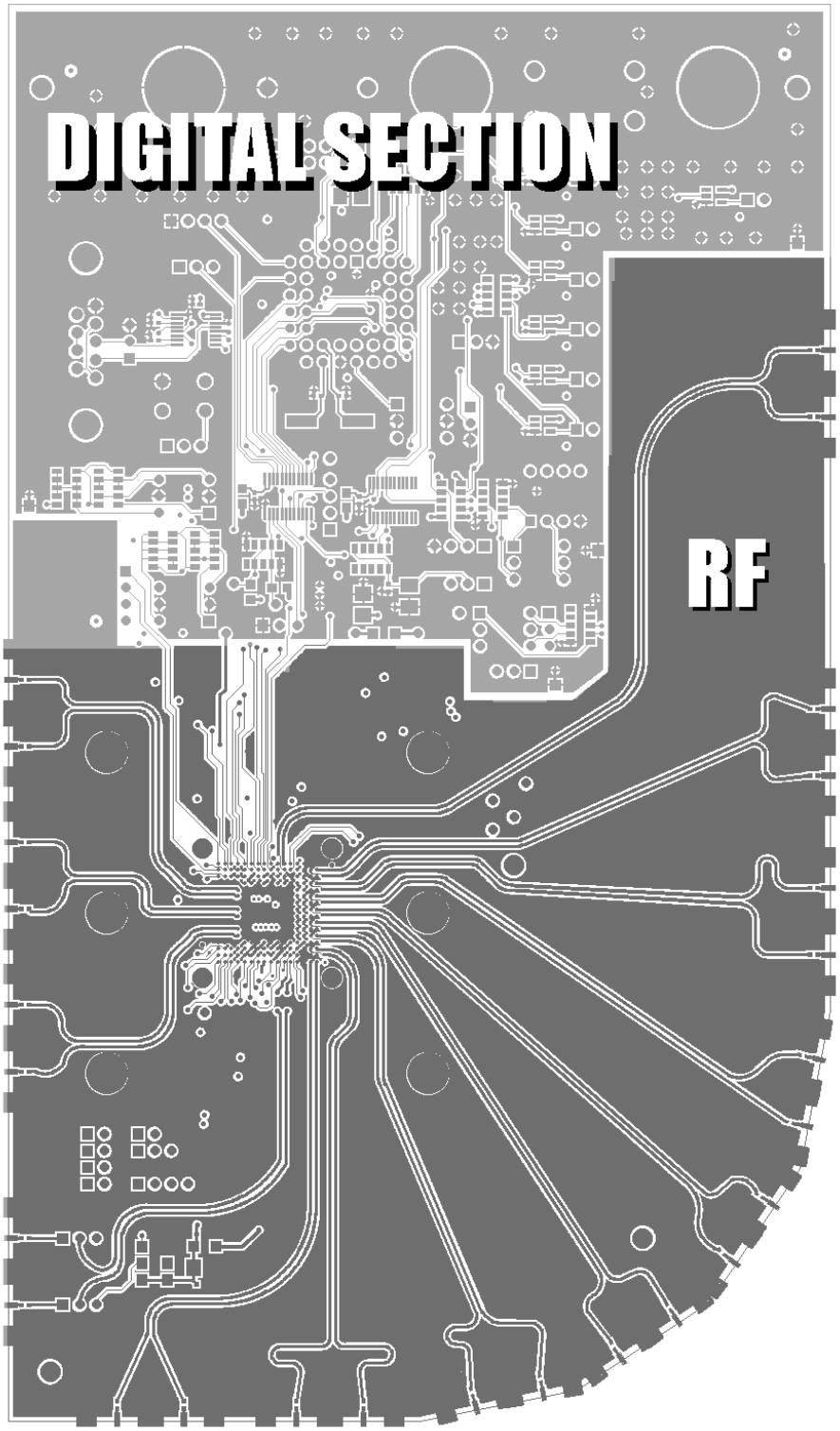
A Node is a coordinate point on a net. In most cases you will specify two nodes per net – an output and an input.

#### 6. Extraction Parameters

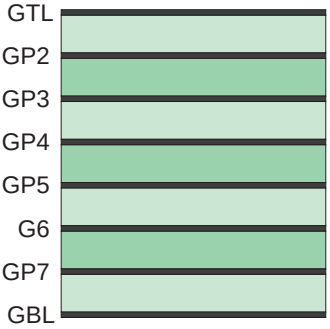
These are user defined parameters controlling the extraction. A data window can be specified, the range of the proximity net, smoothing, sliver removal and polygon types. The user can also specify an ASCII output, DXF (AutoCAD) output or a GDSII stream output.

## Example - High Speed IC Test Board

Below is shown a multi-layer board used for testing a high speed integrated circuit. The board has both a digital section and a RF section. Our interest is in examining the differential pairs driving the RF inputs and outputs for impedance, delay and possible coupling to other pairs. We don't care about the digital control circuitry.




Our goal is to analyze all of the controlled impedance differential pairs that connect the outside world to the IC.



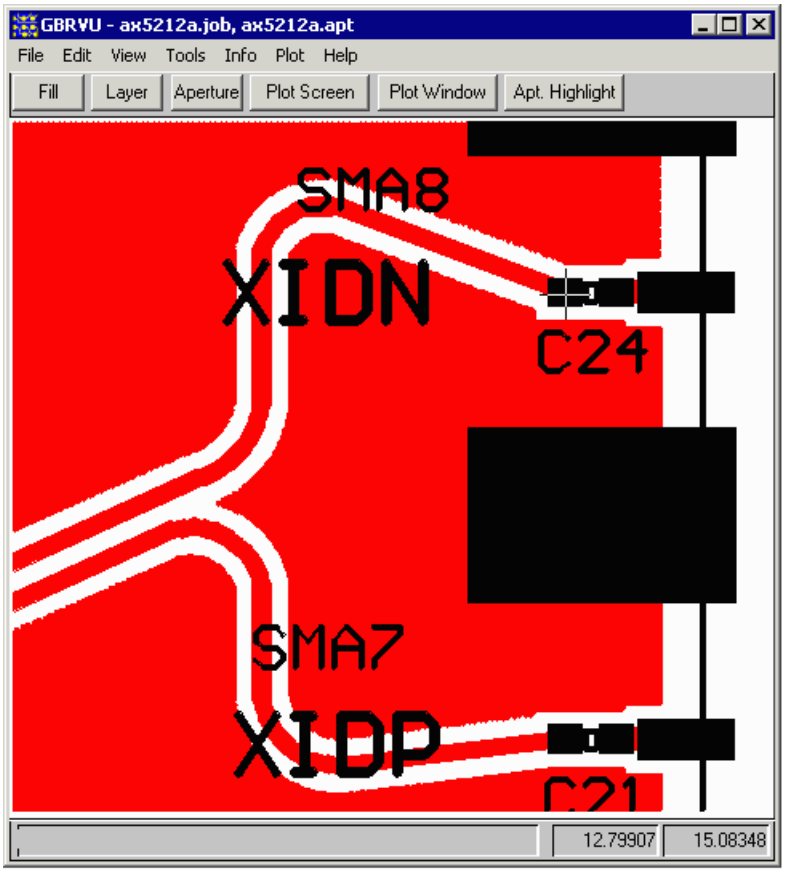

**Stackup**
Although this is a multi-layer board, inspection shows that the ground plane layer GP2 under the RF portion is solid copper – so there is no need to analyze the rest of the layers.

## Getting the Node/Net Coordinates

The board designer did not include a list of net names and a coordinate point. Using the top layer of the Gerber file (.GTL) and a Gerber file showing the top side assembly data (AST) we were able to use GBRVU to identify the desired net names and coordinates.

Carefully position the GBRVU crosshair over your desired net (and node) point and record the coordiantes from the status bar at the lower right against the net name.

The resulting table is shown below:

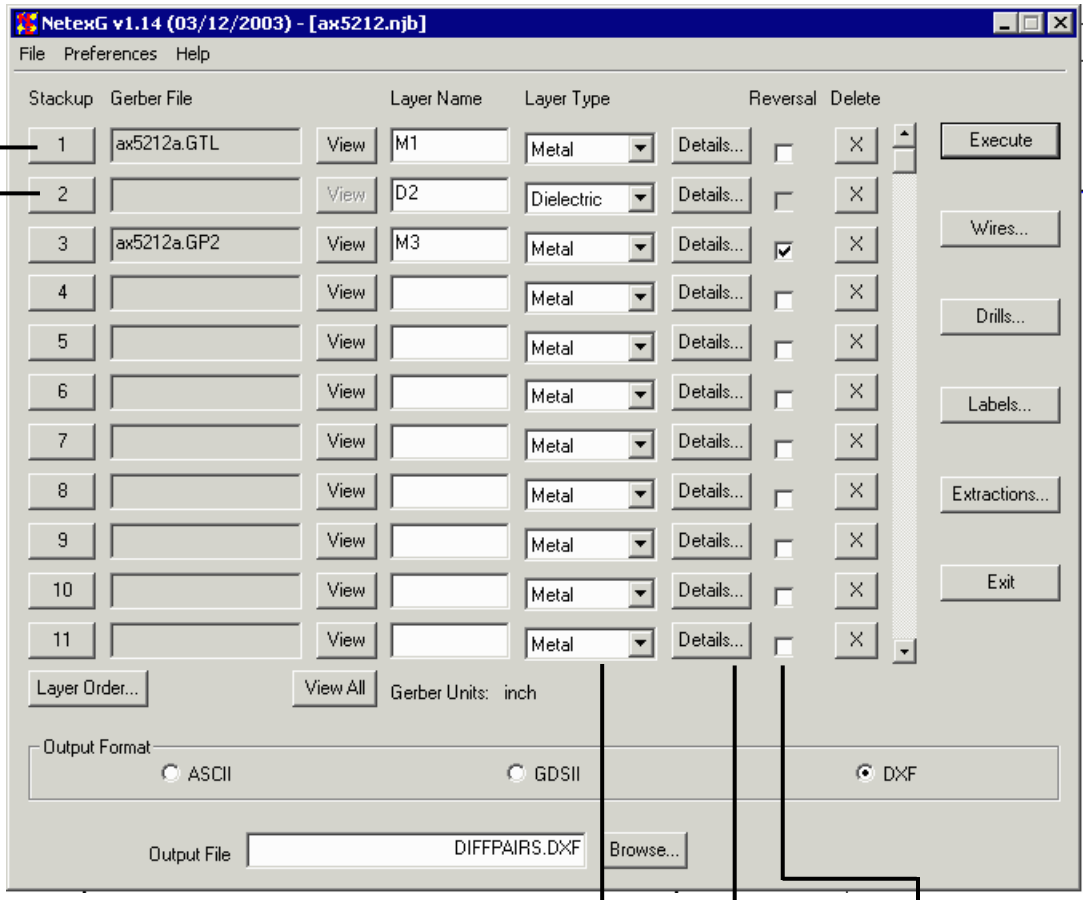



| | | |
|---|---|---|
| XIDN | 12.800000 | 15.080000 |
| XIDP | 12.800000 | 14.680000 |
| RXCN | 12.800000 | 14.010000 |
| RXCP | 12.800000 | 13.610000 |
| RXBN | 12.730000 | 13.000000 |
| RXBP | 12.650000 | 12.610000 |
| RXAN | 12.470000 | 12.170000 |
| RXAP | 12.250000 | 11.840000 |
| TXDN | 11.940000 | 11.510000 |
| TXDP | 11.600000 | 11.280000 |
| TXCN | 11.178000 | 11.080000 |
| TXCP | 10.760000 | 11.000000 |
| TXBN | 10.120000 | 10.930000 |
| TXBP | 9.720000 | 10.930000 |
| TXAN | 9.040000 | 10.930000 |
| TXAP | 8.640000 | 10.930000 |
| PREFCLKN | 8.120000 | 11.510000 |
| PREFCLKP | 8.120000 | 11.910000 |
| PTCP | 8.120000 | 12.910000 |
| PTCN | 8.120000 | 13.310000 |
| PTDP | 8.120000 | 13.885000 |
| PTDN | 8.120000 | 14.285000 |
| PRXP | 8.120000 | 14.860000 |
| PRXN | 8.120000 | 15.260000 |
| PVCXDN | 12.800000 | 17.235000 |
| PVCXDP | 12.800000 | 16.835000 |

Net Name vs. Coordinates - here are the net names and associated coordinates. I could have picked any coordinate that "sits" on the net but picked the end near the connector (actually on the inside of the capacitor) so that the same point can be used as a node since these points represent one end of the net.

## Running the NETEX-G Program

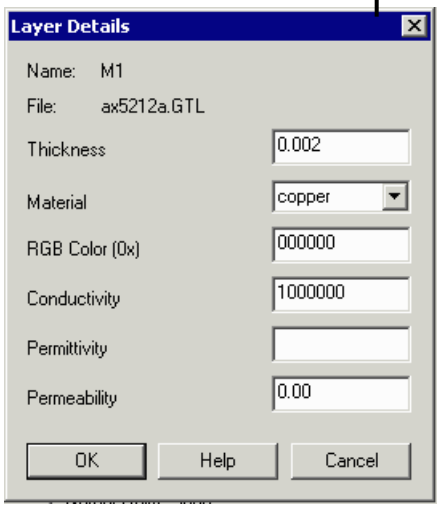
While this is not intended to be a manual on running NETEX-G, the basic settings and procedure are shown below. First define the Gerbers for the board stackup. Only after that go to the other dialog boxes.




Every layer must be assigned a type.


Ground planes are often output as "reversals." You need to inform NETEX of such layers so it can put them back to normal polarity.

2. Between each Gerber layer is a dielectric layer. No Gerber file is associated with a dielectric layer unless it is a solder mask.




Layer Details: every layer can be assigned a thickness, material and electrical properties. These are not mandatory for NETEX-G but other programs may want to read the values for electrical analysis.

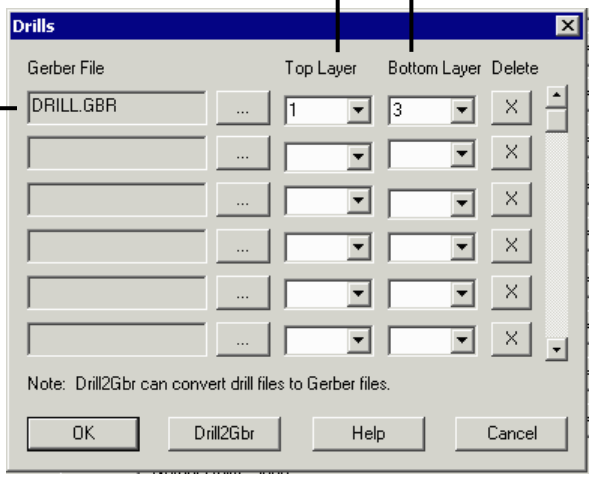
1. select the Gerber files using this button to pop open a dialog box. Enter the files from physical top to bottom.

# Running NETEX-G cont ...

## Vias from Drill

Once the stackup is defined the user needs to use the drill dialog to create vias. Drill files must first be converted into a Gerber file whose units, data mode and zero suppression match that of the conductor layers. This can be done with the drill2gbr utility shipped with NETEX-G..

Enter the name of the drill file (in Gerber) here.



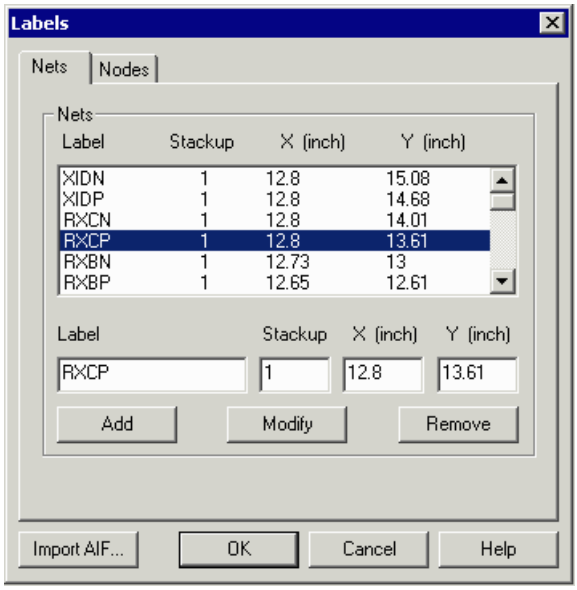

specify the top and bottom layer that the drill passes through. Remeber to use the stackup position.

In this example we are dealing with only the first two conductor layers:

| | |
|---|---|
| GTL | 1 |
| DIELECTRIC | 2 |
| GP2 | 3 |

## Labeling the Nets and Nodes

In order to be able to extract just the net pairs we want to analyze it is necessary to specify a point on the net and associate it with a name. That is why we went to the trouble of building such a table using GBRVU. There are two ways to enter the data -- from the dialog box or by editing the job file. We'll show the dialog box approach here:

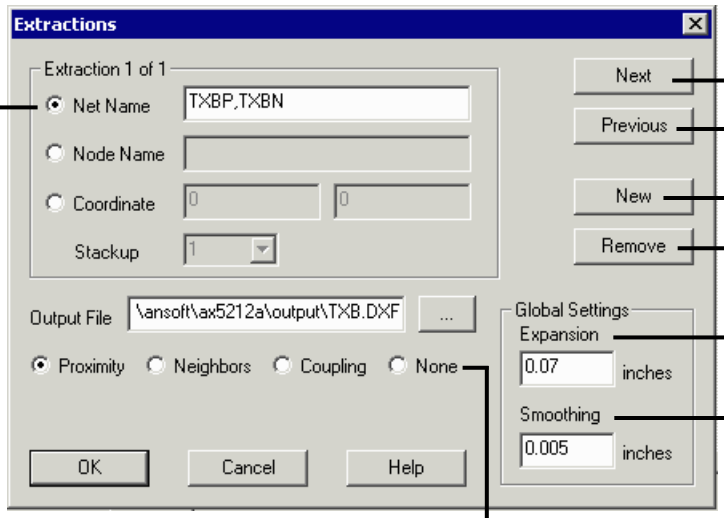



enter the net label, stackup layer and an X,Y coordinate that falls on the net (on that layer).

**Hint**: If there are a large number of net or nodes to define you can edit the job file directly!

## Specifying the Extractions

This dialog is used to tell NETEX-G which nets to extract. We don't want one giant geometry with all of the nets in it – this would be too large to analyze in a reasonable time. Instead we are going to request a dozen pairs of nets – one extraction for each differential pair.




Use Next and Previous to move back and forth from extractions already defined.

Use the New button to define a new extraction.

Use the Remove button to delete an existing extraction.

The Expansion distance determines how far to grap capacitively coupled metal for the "proximity" net.

A separate smoothing parameter applies to the proximity net. Typically this value can be larger than the smoothing for the extracted nets to reduce vertex count.

### Defining the Extractions

There are several ways to do this -- using Net names (that have been assigned in the label section) using Node names (again, that have been assigned in the label section) and finally entering the coordinates directly.

In this case we are using the net names and have specified both pairs of a differtial pair. We could also have used a wildcard expression such as XID* or XID?.

### Output Options

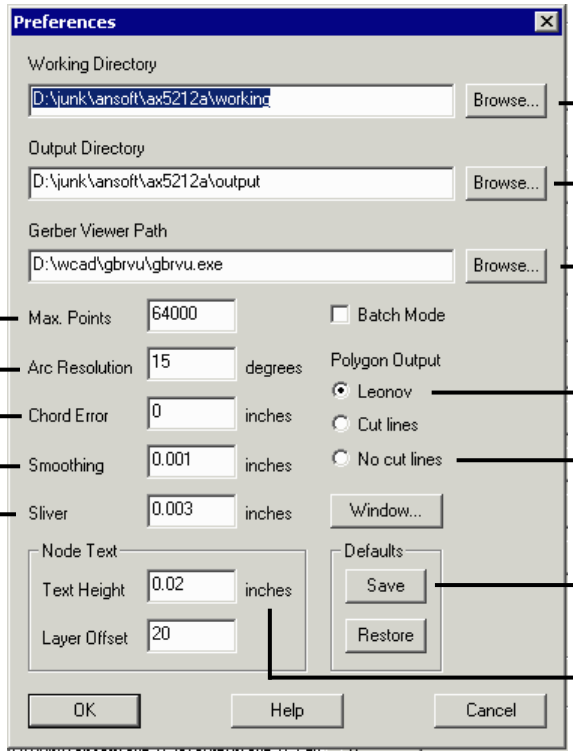
**None** -- only extract the specified nets.

**Proximity** -- calculate an additional net called the proximity net, which consists of all conductors that fall within the expansion range of the nets to be extracted.

**Neighbor** -- extract any "neighbor" nets that fall within the expansion region. The entire net is extracted.

**Coupling** -- build a coupling table for the entire design showing which nets couple to which nets (based on the expansion distance ...)

# Preference Settings




NETEX-G creates lots of intermediate and temporary files and they will be written into the working directory.

Specify the directory where you wish the output files to be written.

Specify the gbrvu.exe path so that NETEX-G can launch GBRVU when needed.

For output to Ansoft you can select either Leonov (you will have to do cutouts manually) or No Cut Lines.

Save - saves the window values to the registry for use later.

Control the height of text labels (i.e. the nodes) that appear in the DXF file from this settings.

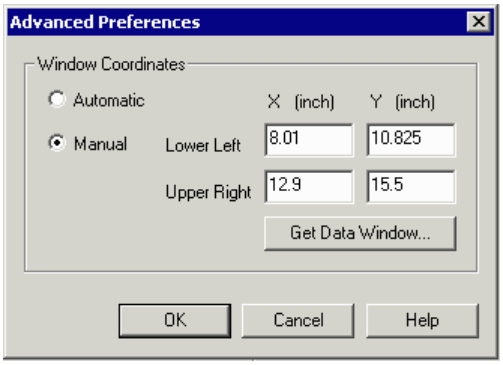
Sliver - removal of very small slivers that may appear due to boolean errors or noise.

Smoothing - controls the removal of "excess" vertices. A distance.

difference between chord and actual arc (another way of specifying the fracturing of arcs

resolution (in degrees) used when breaking up arcs.

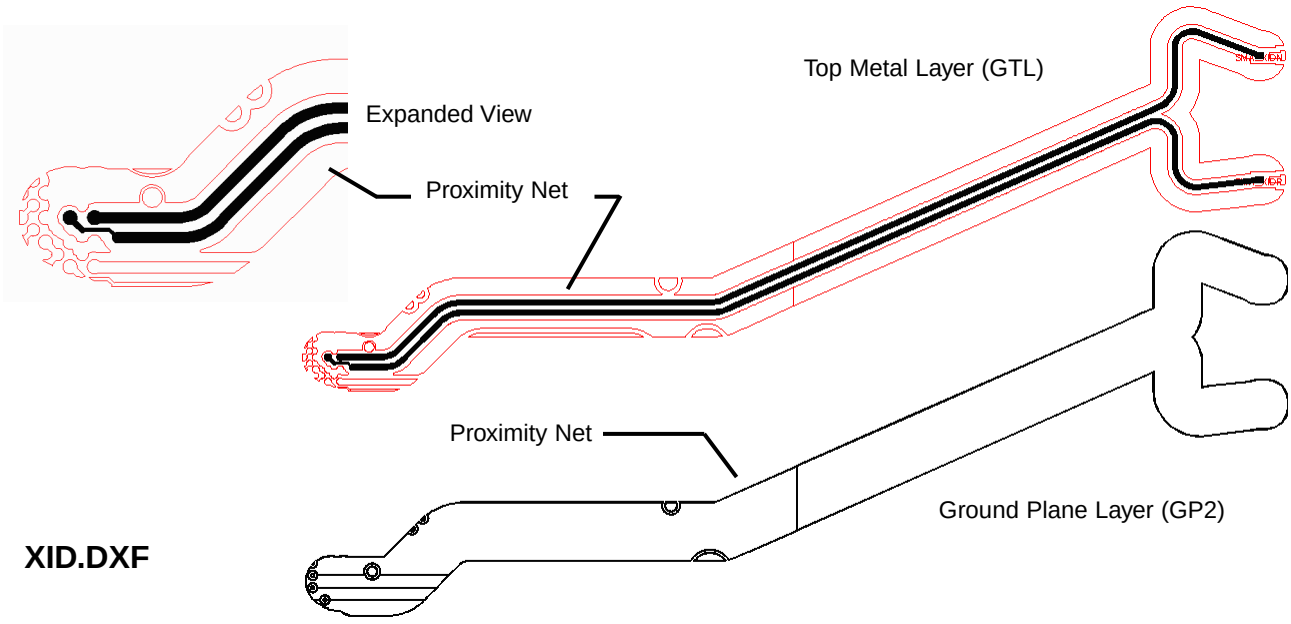
maximum number of points allowed in an output polygon.




The clipping window is defined from the Advanced Preferences dialog. One can either enter coordinates manually or use GBRVU to select them with the cursor.

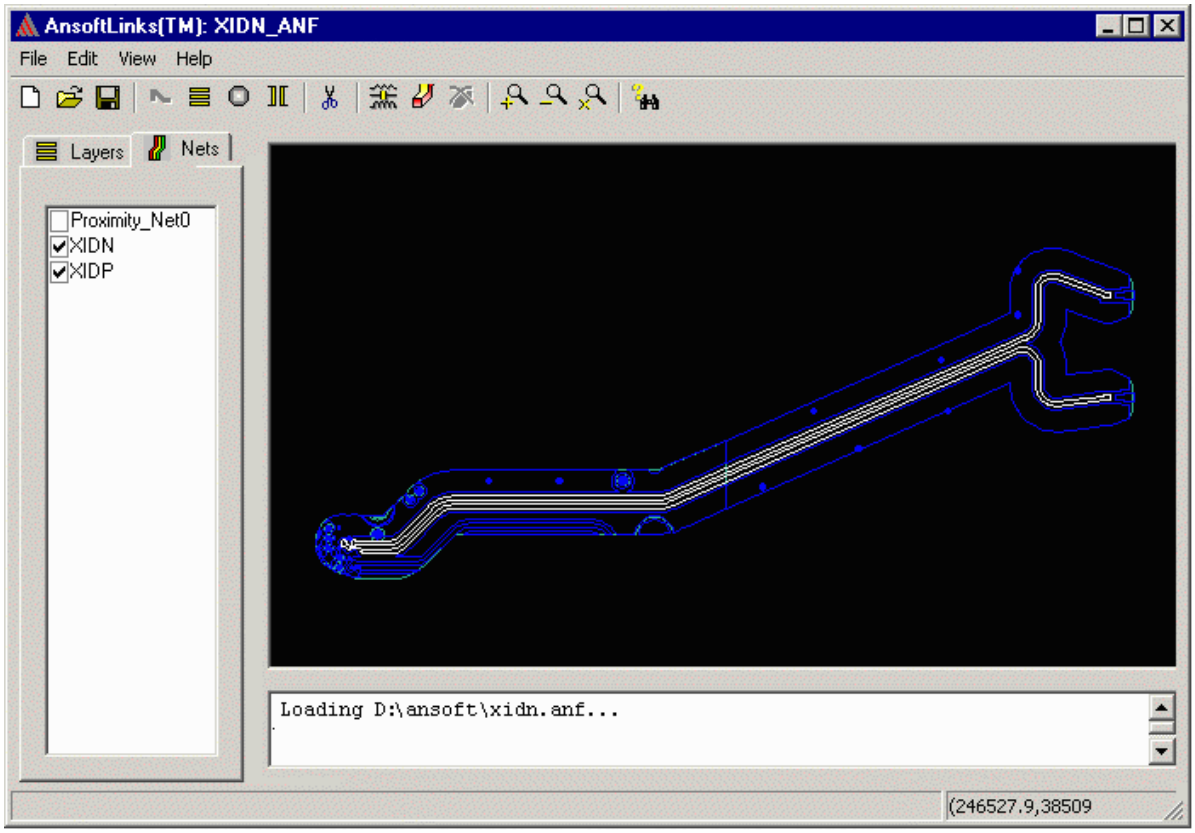
For large jobs setting a clipping window that just contains the nets to be extracted will speed up the extraction.

## Results




**XID.DXF**

We requested two nets: XIDP and XIDN (the postive and negative halves of a differential pair ) and also requested a proximity net that extends 0.070 inch out past the nets.



### The NETEX-G Job File

The Job File is an ascii file containing all of the information needed to run NETEX-G. If you are careful not to disturb the syntax you can edit this file manually.

```
B_LAYERS
1   M1   METAL           0.002     copper   0x000000 10000.0   0.0     0.0
2   D1   DIELECTRIC      0.035     fr4      0x000000 0.0       4.0     0.0
3   M2   METAL           0.002     copper   0x000000 10000.0   0.0     0.0
E_LAYERS
```

The layer section defines the stackup. Only the first 4 columns contain information used by NETEX-G. The balance are used for Ansoft to describe the material.

```
B_LAYER_INPUT
1  "D:\ansoft\ax5212a.GTL"  normal
3  "D:\ansoft\ax5212a.GP2"  reverse
E_LAYER_INPUT
```

The layer input section contains the path/name of the input files. If a Gerber file needs to be reversed the flag "reverse" appears.

```
B_DRILL
"D:\ansoft\DRILL.GBR"  1  3
E_DRILL
```

The drill section defines the file or files derived from drill data. It also includes the start and end layer for each drill file.

```
B_NETS
XIDN          12.800000    15.080000  1
XIDP          12.800000    14.680000  1
RXCN          12.800000    14.010000  1
RXCP          12.800000    13.610000  1
RXBN          12.730000    13.010000  1
RXBP          12.650000    12.610000  1
.
.
.
PVCXDN        12.800000    17.235000  1
PVCXDP        12.800000    16.835000  1
E_NETS
```

The net section is used to lable nets. This includes the so called "name" , a coodinate point anywhere on the net and a stackup layer.

```
B_NODES
SMA_PVCXDN    12.800       17.235       1
SMA_PVCXDP    12.800       16.835       1
SMA_XIDN      12.800       15.080       1
SMA_XIDP      12.800       14.680       1
SMA_RXCN      12.800       14.010       1
SMA_RXCP      12.800       13.610       1
SMA_RXBN      12.730       13.000       1
SMA_RXBP      12.650       12.610       1
.
.
.
E_NODES
```

The node section is used to lable nodes. This includes the node "name" , a coodinate point anywhere on the net and a stackup layer. Nodes can be used to identify locations on the net that will be your reference points for S-parameter or lumped element models.

```
B_EXPANSION
0.070000
E_EXPANSION
```

The expansion distance is used to create a proximity net during net extraction. Any conductor that falls into the expanded region is grabbed for the proximity net.

## The NETEX-G Job File cont ...

```
B_EXTRACT BYNAME  "D:\ansoft\XIDN.DXF" PROXIMITY
XIDN
XIDP
E_EXTRACT

B_EXTRACT BYNAME  "D:\ansoft\RXC.DXF" PROXIMITY
RXCN
RXCP
E_EXTRACT

B_EXTRACT BYNAME  "D:\ansoft\RXB.DXF" PROXIMITY
RXBN
RXBP
E_EXTRACT
.
.
.
```

The extraction section defines particular nets to extract. You can specify a net either by coordinates or by using a net label or node label. You can also specify if you want a proximity net.

As many extractions as desired can be specified.

```
B_SETTINGS
WORKDIR           "D:\ansoft\working"
OUTPUTDIR         "D:\ansoft\output"
MAXPOINTS         64000
ARCRESOLUTION     15.000000
CHORDERROR        0.000000
SMOOTHING         0.001000
SLIVER            0.003000
POLYOUTPUT        LEONOV
WINDOW            8.01     10.825000     12.9     17.83
NODETEXTHEIGHT    0.020000
NODETEXTOFFSET    20
PROXSMOOTHING     0.015000
OUTPUTFORMAT      DXF
OUTPUTFILE        "D:\ansoft\xid_all.anf"
E_SETTINGS
```

The Settings section stores the remainder of the program settings. These values are saved in the job file and are generally also saved in the registry for use again and again.


**For more information contact:**

Steve DiBartolomeo
Applications Manager
Artwork Conversion Software, Inc.
831 426-6163
steve@artwork.com
www.artwork.com