

NETEX-G USER INTERFACE

Main Dialog

All functions are launched from this main dialog. The stackup layers are selected and configured and the various other extraction parameters are defined.

View - Pressing on the View button opens that file in GBRVU for display.

Layer Name - Defines the name for this layer. Not always useful (i.e. GDSII)

Layer Type - Conductor or Dielectric are most common. Wirebond layers are treated in a special fashion.

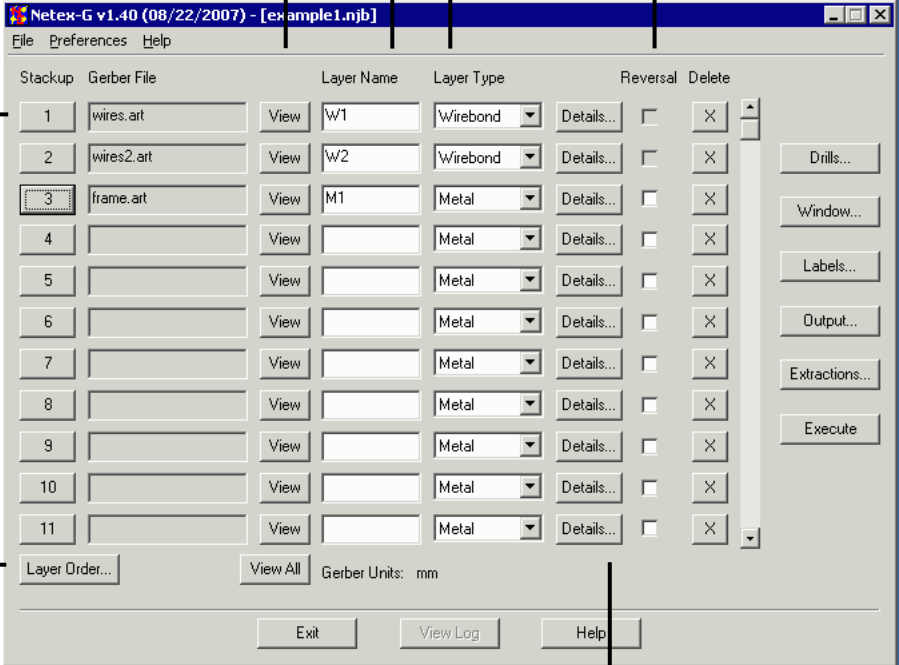
Reversal - Some Gerber files are drawn with reversed polarity such as ground planes - check if that is the case.

Stackup - pressing on one of the Stackup buttons opens a File Browser so that you can select the Gerber file for that layer.

The user normally must insert a dielectric layer between each conductor layer. You cannot import a Gerber file for dielectric layers.

Layer Order - opens a dialog allowing you to move layers around in their stack-up position or to delete layers.

Details - Some output formats (ASCII, ANF) support info about the layer's thickness and electrical properties.



Drills - defines which layers that the drills pass through.

Window - use to define a window for analysis.

Labels - define net & node labels (and corresponding coordinate) either manually or by importing an AIF file.

Output - select type of output, filename and location.

Extractions - user can select certain nets for extraction.

Execute - starts the NETEX-G process.

Layer Details Dialog

This dialog defines the thickness, material and electrical characteristics of each layer. The basic net extraction process does not require such information. However some output formats make use of this information: ANF, XFL, 3Di and ACIS.

Layer Details

Name: W1
File: wires.art
Thickness: 0
Material: gold
RGBA: E9BF41F5 hex
Conductivity: 0.999
Permittivity: 0
Permeability: 0

Wire Model
Name: long
Model Type:
☐ JEDEC3 H1: 0 L1: 0
☒ JEDEC4 H1: .1 L1: .2
H2: .05 L2: .4

Wires
Connect to Next Metal: ☒
Connect to Layer: 1
Wire Diameter: 0.025

Die Information
Name: nxp2003 Connect to Layer: 3
Thickness: 0.12 ☒ Corners (x1 y1 x2 y2)
-1.4,-1.4,1.4,1.4

OK Help Cancel

Metal and Dielectric Layers

Thickness - enter a thickness for the layer in the same units as the Gerber input files (inch or mm)

Material - assign a name for the material of this layer. The color and electrical properties are “tied” to the material so you can reuse it for similar layers.

RGBA Color - assign a RGB value (hex triplet 0-255) defining the color for this layer. The last value represents transparency for 3D use. (0 transparency = opaque)

Conductivity - enter the conductivity.

Permittivity - enter the permittivity.

Permeability - enter the permittivity.

Wire Layers

Connect Next Metal - if checked, NETEX-G will automatically generate a via from the outer point of the wire to the next metal layer below the wire.

Connect to Layer - If not checked, the user is expected to enter a stackup level (that represents a metal layer) and NETEX-G will generate a via down to the user specified layer.

Wire Diameter - the diameter of the bond wire. (Note that the Layer Thickness parameter is ignored)

Wire Model

For 3D outputs (3Di and ACIS) you need to define a wire model and its parameters.

Jedec 3 - a simple wire model suitable for “short” wires.

Jedec 4 - a wire model that can be “arched” for longer wires.

Die Information

For 3D outputs (3Di and ACIS) you should define a die thickness and what layer it sits on. You should also define the lower left and upper right corners.

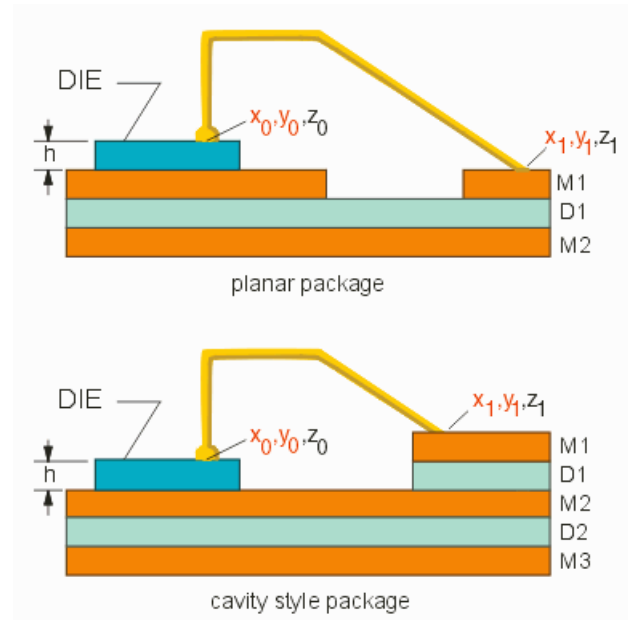
Wire Models

Wire models are used to take 2D wire data (start and end point) and create a 3D wire from it. You need only define wire models when creating 3D output formats such as 3Di and ACIS.

Z Heights

First, we assume that all wires originate on a die surface with a common Z-value. We will know the Z value by defining the height of the die body along with the conductor or dielectric layer that the die body sits on.

We will know the height of the other end of the wire by defining on which metal layer it falls on. NETEX-G looks "down" through the stackup from the wire end and figures out on which conductor layer it will land. Therefore it can support both planar packages and cavity or stepped packages.

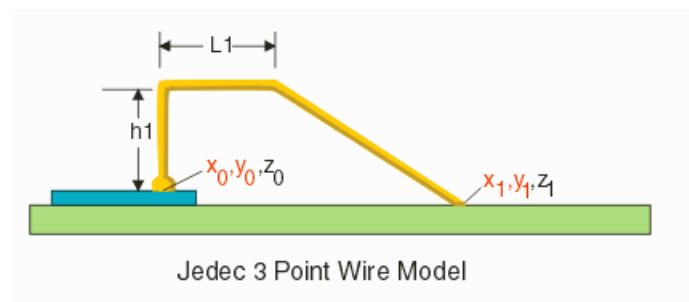


Z-Profile

The wire's z-profile is defined using a model with parameters. While Artwork has developed some very sophisticated user-defined models, we are not going to use those at present - instead we are going to use two very old models developed by JEDEC - the JEDEC 3 point wire model and the JEDEC 4 point wire model.

Jedec 3 Point Wire Model

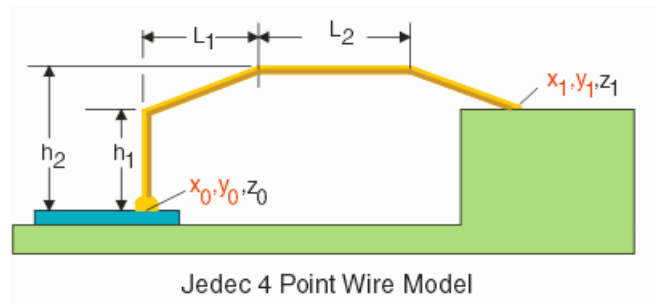
The JEDEC 3 point wire model requires just two parameters - $h1$, the height that the wire rises from the die pad and $L1$ the distance it travels horizontally before beginning a straight line descent to the package finger pad.



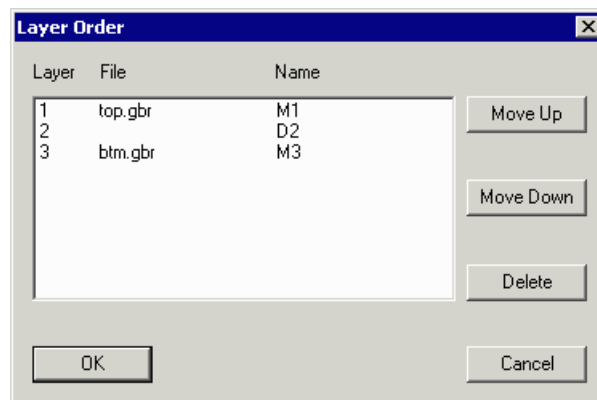
The three point model works well for relatively short wires. The illustration is not to scale as $h1$ and $L1$ are exaggerated for clarity.

Jedec 4 Point Wire Model

Longer wires generally have a bit of arc in them much like the arch of a bridge -- this helps to reduce the wire sag that might result in a short circuit. The JEDEC 4 point model has additional parameters that enable the user to approximate an arc as shown in the illustration below:



Layer Order Dialog



Use this dialog to move a layer up or down the stackup or to delete a layer (all other layers are then shifted as needed.)

First highlight the layer you wish to move or delete. Then press the appropriate button.

Hint: to insert a layer into the middle of the stackup first add the layer at the bottom of the stackup (from the main dialog) then use this dialog to push it up to where you want it.

Preferences

It is very important to check the preference settings to insure that your net extraction parameters are correct for the particular job at hand. Failure to do so, when the settings are inappropriate will result in bad results.

Max Points - specify the maximum number of points per polygon that should appear in the output data. If polygons with more vertices than this number are created they will be broken into two or more smaller ones.

Arc Resolution - defines how arcs in Gerber (e.g. a round pad) is broken into segments. Too fine a value and your output file will be large. Too gross a value and the output will be ragged.

Chord Error - is used to specify the allowed error between the ideal arc and the chord that spans it.

Smoothing - smoothing reduces the number of vertices per polygon by looking for closely spaced vertices and removing them.

Sliver - removes tiny slivers generated by numerical noise in the boolean operation. Pick a small value that won't remove data you wish to keep!

Max Via Size - NETEX-G attempts to recognize vias from round pads. However some large round pads are not vias. Any round pad larger than the specified size will not be treated as a potential via.

Scale Factor - default=1. Other values scale up or down the output. You do not need to apply a scale factor in order to change output units.

Working Directory - the directory where the program writes its temporary files.

Output Directory - the directory where the program writes the output files. We recommend that this be different from the working directory.

Output Units - output units default to the input Gerber units. However if other units are needed (such as in GDSII microns are preferred) they can be specified here.

Polygon Output - controls how polygons are written out:

Leonov: polygons are grouped - the outer polygon drawn CCW comes first followed by its child or children which are to be subtracted from it; they are drawn CW.

Cut Lines: If islands are needed a cut line is drawn entering and exiting the polygon along the same path. This mode can generate polygons with a very high vertex count. Use this mode for GDSII output.

No Cut Lines - some output formats do not support self touching polygons so this option cuts such polygons into two or more as needed.

Window - opens a dialog that allows the user to specify the data window either by entering coordinates or by launching GBRVU and enabling the user to select a window. There is also a "polygon" window that can be interactively defined from the GBRVU display.

Node Text - For certain output formats (DXF, GDSII, EGS) NETEX-G places text labels on nodes. The height parameter controls the text height and the layer offset creates the text on a layer number offset from the labeled layer by the given amount.

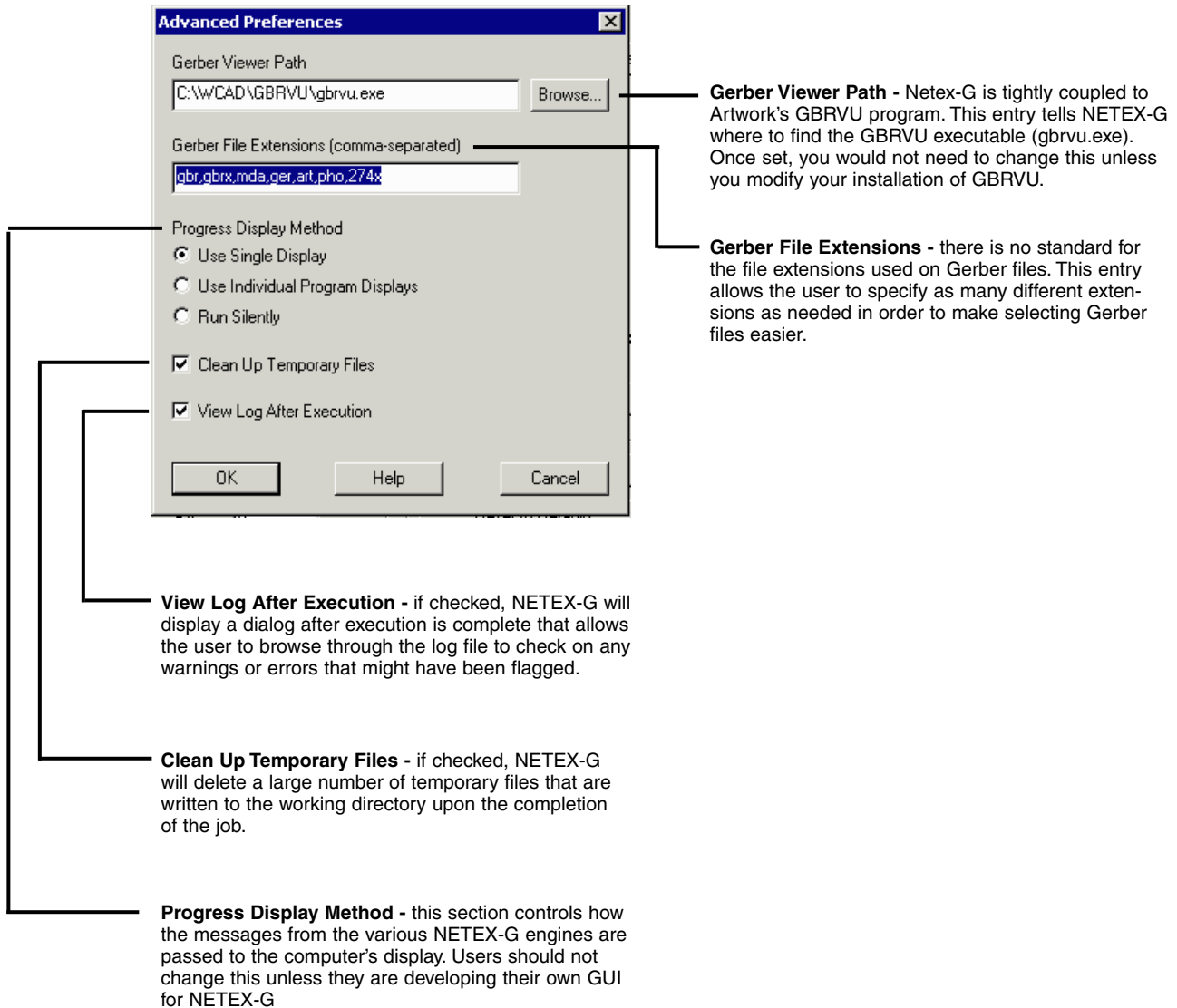
Batch Mode - The GUI will produce the command line and resource files for the NETEX-G engine but will not execute it.

String Matching - in the extraction module one can specify nets to extract using DOS style wildcards or UNIX regular expressions depending on this setting.

Current Preferences - clicking on Save as Default writes the current preferences to the Windows registry. Clicking on Reset to Defaults reads from the registry and writes the settings to the dialog box.

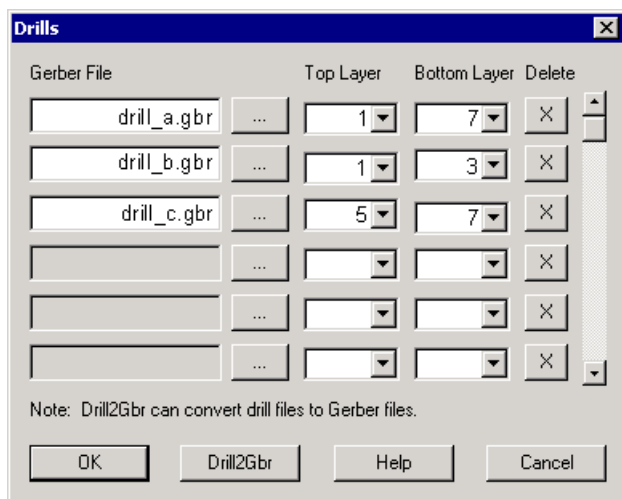
Advanced Preferences Dialog

These settings are grouped separately as they are rarely changed from job to job.

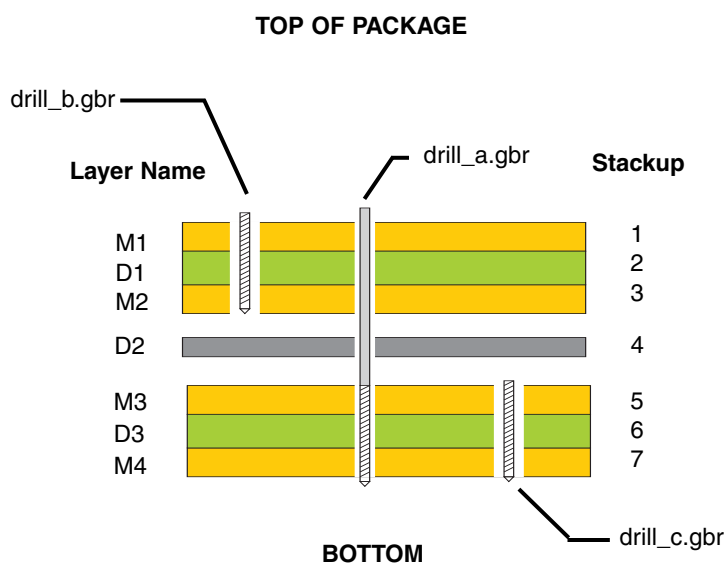


Drills Dialog

Gerber files only provide "horizontal" information about the layout. Drill files provide the vertical interconnect information. Each drill file must be first converted into Gerber using the DRILL2GBR utility. Then it must be identified in this dialog and the user must specify top layer and bottom layer that the drill passes through. It is assumed that the drill passes through all intermediate layers. For designs with buried or blind vias you might have two or more drill files.



Note that the Top Layer and Bottom Layer refer to stackup position; not to conductor position. Since there is typically a dielectric between each layer a 4 layer board will have 7 stackup positions and M4 will be on stackup position 7.



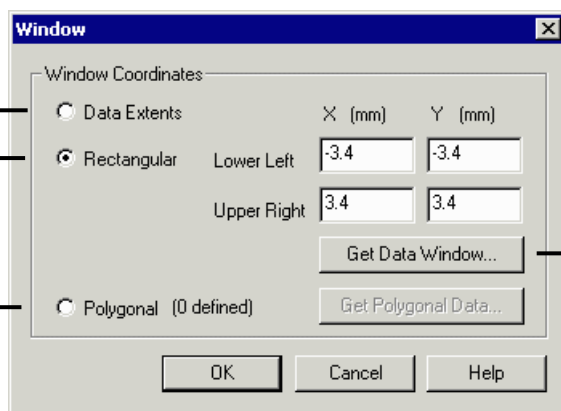
Window Dialog

Sometimes the NETEX-G user will not want to analyze the entire board or package and wants to window out a region to examine. In other situations a design will include a plating bar that shorts all nets into one large net and in order to get results one must cut away inside the plating bar. The Window dialog is used for this purpose.

Data Extents - use the extents of the input data files to define the analysis window. (Default)

Rectangular - user enters the lower left and upper right coordinates of a rectangular window.

Polygonal - if this button is selected the user then clicks on Get Polygonal Data ...

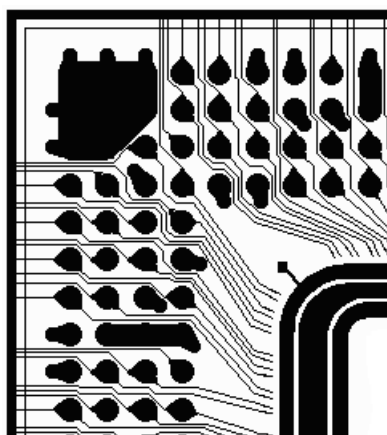


Get Data Window ...

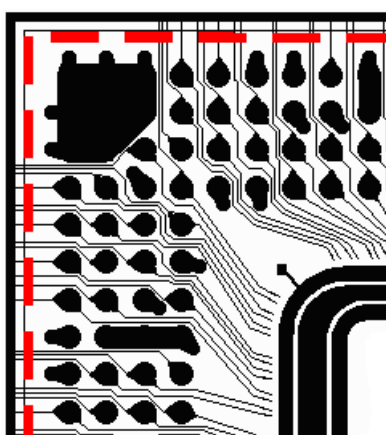
This button opens GBRVU and displays all Gerber layers. The user can select the rectangular data window using the mouse.

Get Polygonal Data - clicking on this button opens GBRVU and displays all of the Gerber files in the stackup. The user defines a polygon with a series of right clicks. Only data falling inside of the polygon will be processed. This option is useful for designs where a rectangular window would capture too much circuitry.

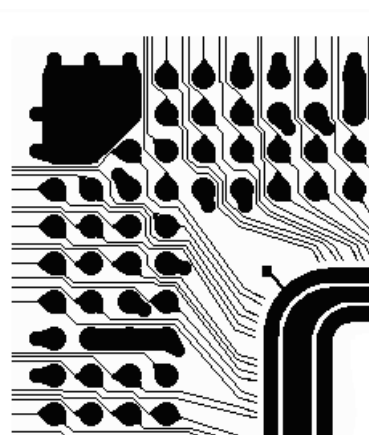
Using a Window to Remove Short (Plating Bar)



the gerber files for this BGA included the plating bar -- effectively shorting all nets together.



The user defined a rectangular window just inside of the plating bar.



NETEX-G removes any elements outside of the window essentially freeing up the individual nets.

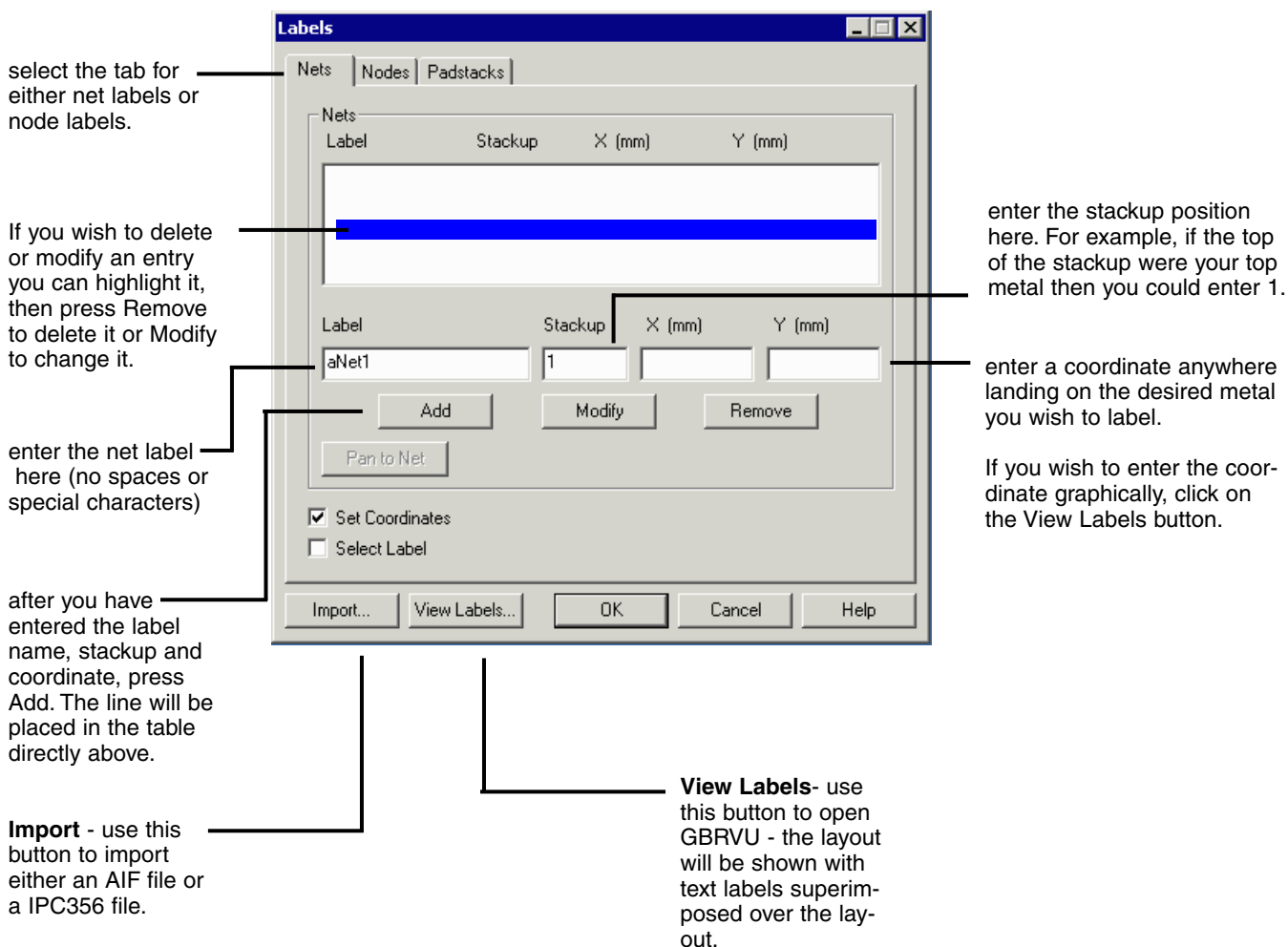
Net and Node Labels

Gerber data carries no net or node information with it. So without some external input NETEX-G can separate out the various nets by connectivity but cannot name them in a way that might correspond to a schematic or netlist.

Netex-G allows you to manually enter net and node names and an associated coordinate. It will then figure out where the label falls and associate the label with the net.

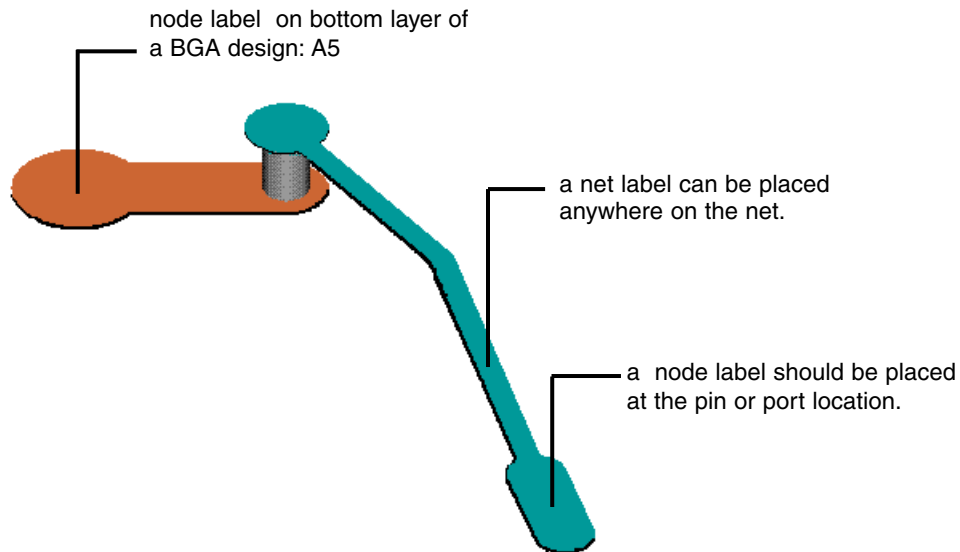
In many cases one has a table of coordinates and associated net names and coordinates and associated node points: for IC packages this might be an AIF file and for board layouts this could be an IPC356 test point file.

The Label Dialog Box



What is the difference between a Net and a Node?

A net is a label that identifies all of the metal pieces that are electrically connected. A node is a particular point on a net. For example, on a BGA layout, the bond finger, trace, via and ball pad all share the same net. But for creating a SPICE model or lumped element equivalent you would still have to identify at least two nodes -- typically a point on the bond finger and another point on the ball pad.



Editing the Job File to add Net/Node Labels

Sometimes a user may have a list of node or net labels and their coordinates available as an ascii table or file. It is possible to manually edit the NETEX-G job file and add these. The correct syntax is shown below:

```
B_NETS
CLOCK5  56.0  67.9  2
VSS      30.4  55.8  2
VCC      20.5  60.5  2
E_NETS
```

Column 1 - Net name. Limit it to 32 characters and do not use spaces or special characters other than: () - _ []

Column 2,3 - Coordinates. Enter an X,Y point that falls anywhere on the net you wish to label. This must be in the units that match your input Gerber files (i.e. inches or mm)

Column 3 - Stackup layer. Enter the stackup position where the program should look for the net. It doesn't make sense to enter a stackup position that represents a dielectric.

The net section must begin with b_nets and end with e_nets.

Importing an AIF File

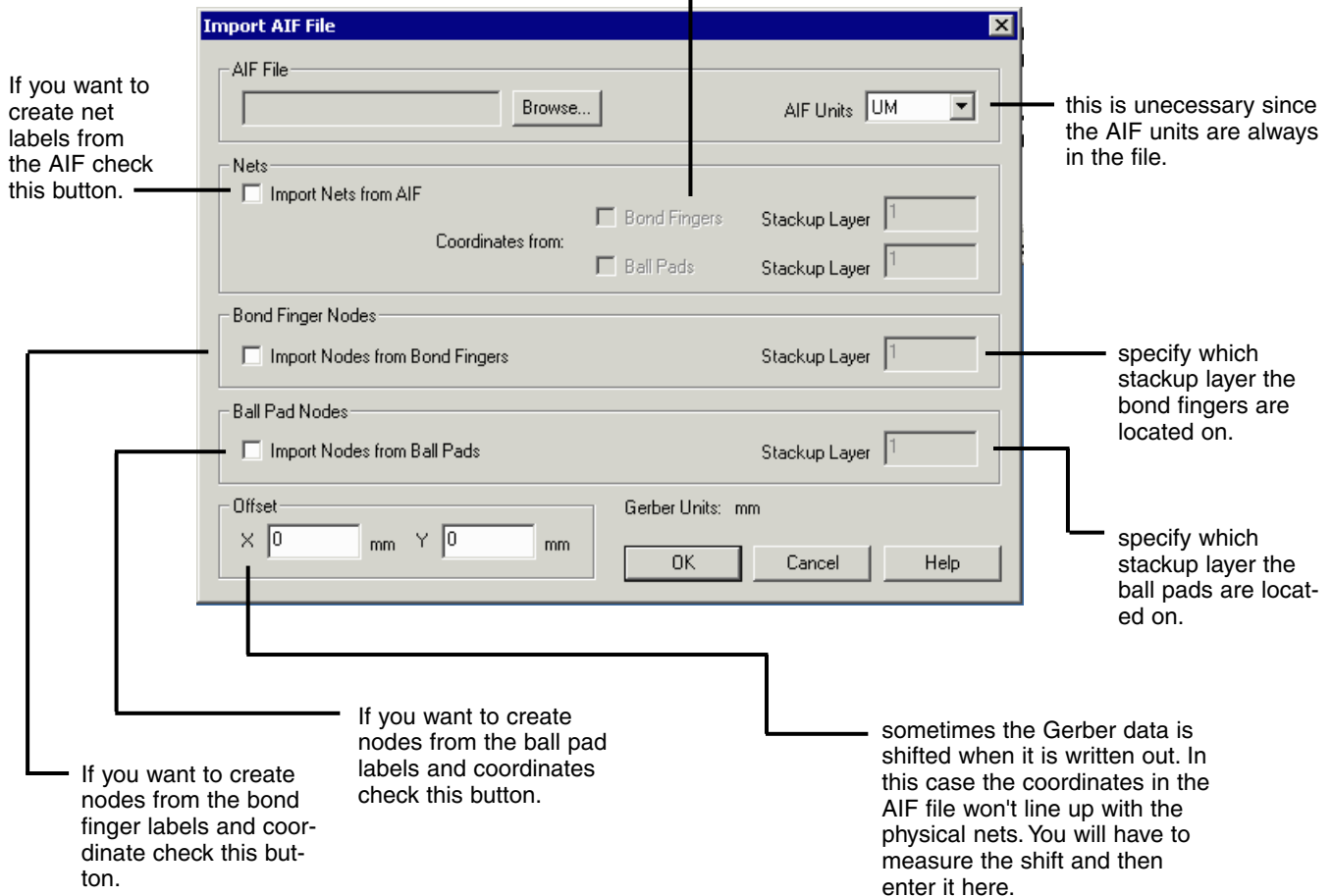
An AIF file turns out to be very useful for labeling a package design in Gerber. The AIF includes:

- netnames
- name and coordinates of each bond finger
- name and coordinates of each ball pad

This means that you can:

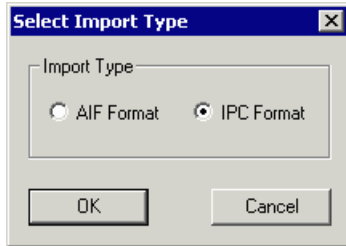
- use the bonfinger number and coordinate as a node label
- use the ball pad label and coordinate as a node label
- use the netname and either of the above coordinates for net label.

you can get a coordinate for the net either from the bond finger or the ball pads (or both!)
Since not every net is connected to the bond fingers (power and ground) the ball pads are more useful if you want all nets.



Importing an IPC356 File

The IPC 356 file can be produced by many PCB design tools. This ASCII file contains test point coordinates, net names and other information used to test both bare and populated PCBs. NETEX-G can read the IPC356 file and use it to label nets and nodes.



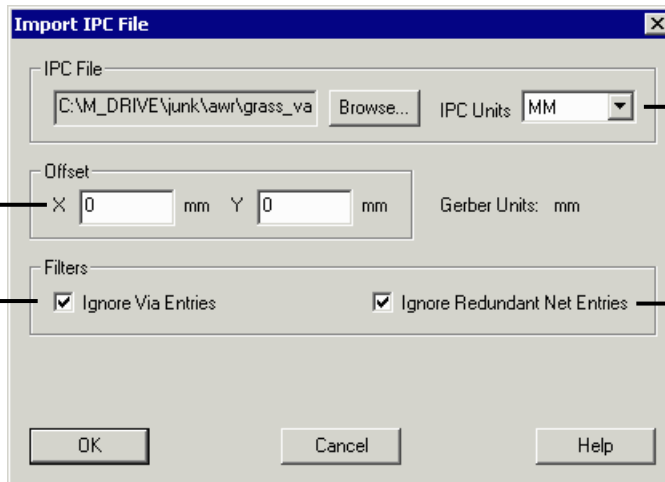
From the Labels dialog box select **Import**

Then select the option **IPC Format**.

Offset - in principle, there should be no need to set an offset. However we have come across IPC 356 files whose coordinate system is offset from the Gerber file coordinates.

Ignore Via Entries - IPC files often have hundreds or even thousands of via entries and these are generally not useful as node or net labels.

Use the **Browse** button to select the IPC file to load.



IPC Units - If the units appear automatically, then you do not need to specify them.

Ignore Redundant Net Entries - The same net may be labeled many times in an IPC file. By checking this option, only the first label for the net is used.

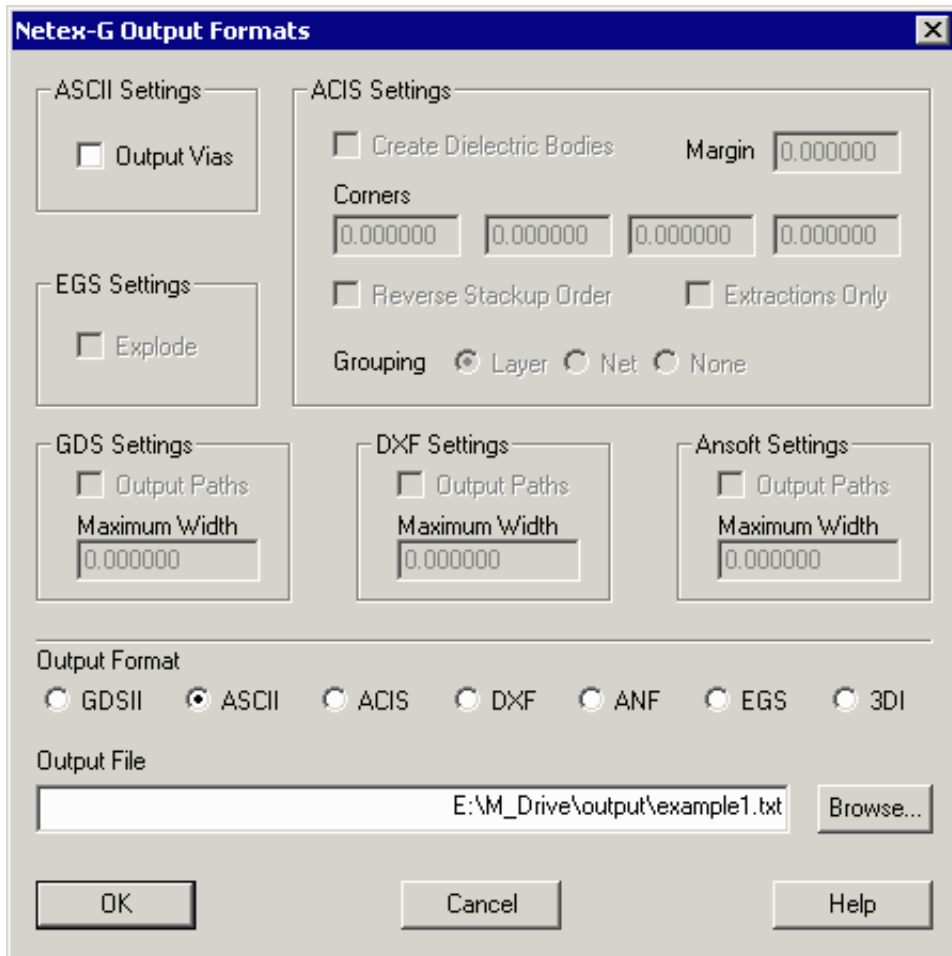
Sample IPC356 File

```
C  Date:10/18/2005 08:00:12 AM
C  Database: 4210.pcb
C
P  JOB    4210.pcb
P  UNITS  CUST 1 _____ units are read from this line
P  VER   IPC-D-356
P  IMAGE PRIMARY
P  layer coordinates
327AFC      U1      -27      A01X+020973Y+022999
327AFC      R1      -2      A06X+027623Y+024638
327AFC      C12     -2      A06X+027623Y+025654
317AFC      VIA          D0200PA00X+021615Y+023241
327AFC_CONN R1      -1      A06X+028766Y+024638
327AFC_CONN JP1     -51      A06X+033147Y+036576
327ANT      C5      -1      A01X+023622Y+007429
327ANT      U3      -8      A01X+022509Y+012608
net name    ref designator pin number
```

Sample IPC356 File - there are two versions of IPC356 - the original and a revision A. The syntax is slightly different but the IPC importer should support either.

Output Formats

Netex-G supports many different output formats. Each format requires a license; so if the option is grayed out in the dialog box, the license is not present.



GDSII - outputs a GDSII stream file. Elements of each net are stored in their own structure - the net name is the structure name. Vias are represented as geometries on the dielectric layers.

ASCII - NETEX-G's native ASCII format. If the Output Vias option is checked, vias are output as entities instead of geometry on each layer.

EGS - Agilent EEsof's native layout format. Used to import data into ADS/Momentum.

DXF - AutoCAD's DXF file format. Elements of each net are stored in their own block with the block name equal to the net name.

ANF - Ansoft's Neutral File format. (In the Preferences, geometry should be set to Leonov polygons) For users of SI-Wave select the option: **Output Paths**.

Output Formats cont ...

3Di - 3Di is Artwork's 3D format for package and IC structures. It requires that the user specify layer thicknesses and supports 3D wire bond models (Polygons should be set to Leonov in the Preferences Section.)

Artwork supplies a 3Di Viewer which can be used to view and plot 3D models and extractions.

ACIS - ACIS is Spatial Technology's 3D format and is widely read by 3D mechanical and analysis tools. Options include:

Create Dielectric Bodies - normally the dielectric body between conductor layers is not rendered as a 3D object. Checking this option forces rendering of the dielectric.

Margin - If dielectric bodies are created, a margin must be specified (related to the data extents) as there is no CAD data defining the dielectric region between conductors.

Corners - Instead of specifying a margin, it is also possible to define the Lower Left and Upper Right coordinates for the dielectric bodies.

Reverse Stackup Order - when checked the order of the stackup (top to bottom) is reversed.

Extractions Only - when checked, no ACIS will be generated for the complete output - only for net extractions. This is because ACIS file is very large and many users do not need it for the entire board.

Grouping - 3D bodies in ACIS can be ungrouped (**None**), grouped by layer (**Layer**) or grouped by net (**Net**)

Output Paths Option

NETEX-G decomposes all input data (flashes, draws, areas) into boundary data. Therefore the output of NETEX-G consists solely of boundaries and vias which are derived from drill data. Some output formats, in particular ANF, support traces. In order to support the end users who need traces for their analysis, the Output Path option can be checked.

This function attempts to determine where traces should be used instead of boundaries and inserts traces in those locations. However it may not be able to exactly determine the original starting or ending point of the trace because the boundary data includes both the trace end point and surrounding metal generated by the flash.

Extractions Dialog

The user of simulation software may not want or need the entire layout passed to his software. Instead, he may desire to focus on a few critical nets. Netex-G allows the user to specify one (or a few) nets to extract and output separately. To do so, click on the **Extractions ...** button from the main menu to open the Extractions Dialog.

Nets to Extract can be specified by coordinate, net label or node label. A wildcard such as v* can also be used.

If a coordinate is used, then the stackup position must also be specified.

A unique output file is specified for each extraction.

It is possible to define more than one extraction per run. Use the **New** button to create a new extraction.

The Expansion distance determines how much adjacent conductor is "grabbed" for the proximity net.

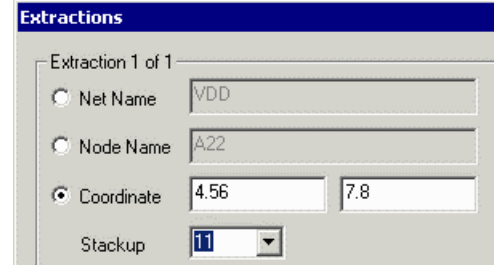
Smoothing parameter applies to the Proximity net and can be set larger than smoothing in the Preferences section.

In addition to the nets that are extracted the user can specify a proximity net that consists of all conductor (on all layers) that lies within the specified "expansion" distance.

Methods of Specifying the Nets to Extract

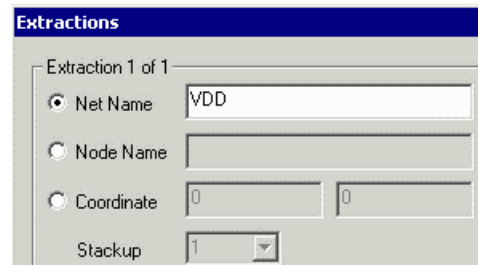
There are several ways to specify the nets one wishes to extract.

By Coordinate - specify a coordinate anywhere on the geometry. Of course, since the geometry for that net may be spread across many different layers it is necessary to specify the stackup layer to look on.



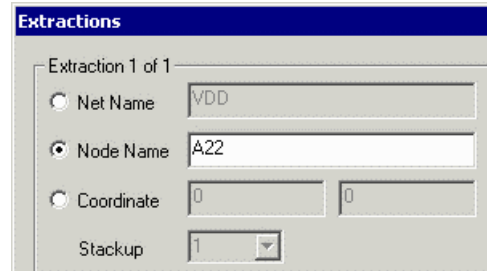
The 'Extractions' dialog box, titled 'Extraction 1 of 1', shows the 'By Coordinate' method selected. The 'Net Name' field is empty, 'Node Name' is 'A22', and 'Coordinate' is set to '4.56' and '7.8'. The 'Stackup' dropdown is set to '11'.

By Net Name - If you have taken the trouble to assign net labels to geometries (again using a coordinate on the geometry) then one can specify the net name. This is really an indirect way of specifying a coordinate.



The 'Extractions' dialog box, titled 'Extraction 1 of 1', shows the 'By Net Name' method selected. The 'Net Name' field is 'VDD', 'Node Name' is empty, and 'Coordinate' is set to '0' and '0'. The 'Stackup' dropdown is set to '1'.

By Node Name - If you have taken the trouble to assign node labels to geometries (again using a coordinate on the geometry) then one can specify the node name. This is really an indirect way of specifying a coordinate.



The 'Extractions' dialog box, titled 'Extraction 1 of 1', shows the 'By Node Name' method selected. The 'Net Name' field is 'VDD', 'Node Name' is 'A22', and 'Coordinate' is set to '0' and '0'. The 'Stackup' dropdown is set to '1'.