

Tutorial 1

Netlist Analyzer: Terms and Concepts

Introduction

The Netlist Analyzer is a graphical package for the generation and comparison of netlists throughout the life cycle of a Genesis job. The Netlist Analyzer can be used for the following tasks:

- To verify that the netlist generated after data input is identical with the netlist provided by the designer
- To compare netlists in the same job/step or between different jobs
- To locate shorted or broken nets
- To check netlist violations on-line while editing a job in the Graphic Editor, ensuring netlist integrity throughout the process.

This tutorial is the first in a series of tutorials which describe the Genesis Netlist Analyzer. This tutorial describes concepts and terms and does not require you to be in front of your computer. The other tutorials will describe how to use the Netlist Analyzer, providing many practical examples.



NOTE:

Information not contained here can be found in the on-line documentation: see Doc. 0506 The Netlist Analyzer and Release Notes.

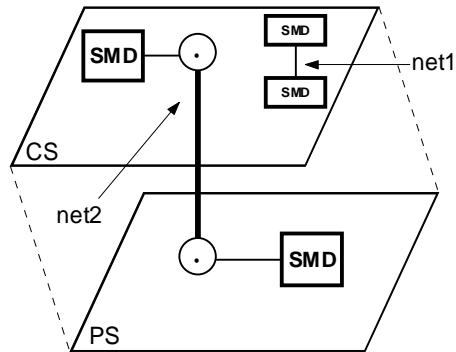
Objectives

This tutorial introduces you to general terms and concepts to help you understand the Netlist analyzer. In this tutorial you will learn about:

- Nets and netlists
- Why you need to check the board's netlist?

Netlist

A **Netlist** is a list of nets which define the conductivity interconnection scheme of a bare circuit board.



A doubled sided board with 2 nets: the netlist is net1, net2

Why you Need to Check the Board's Netlist?

It is important to emphasize here that netlist integrity is a critical factor in PCB production. Unlike other errors which can result in lower yields or less reliability, netlist errors will almost always result in non-functional boards. Therefore it is essential to catch netlist errors as early as possible in the design-manufacturing cycle and correct them.

When do you Need to Check the Netlist?

There are two cases when the netlist should be checked.

- 1** When getting a job from a CAD designer, we want to confirm that the netlist given by the designer is identical to the netlist according to the graphical data.
- 2** While editing the board and also when we finish editing, we want to make sure that our editing operations have not caused any netlist violations. This includes manual editing as well as DFM functions.

What is a CAD Netlist and CAD Net Points?

The **CAD netlist** is the original netlist supplied by the designer from the CAD software database. The CAD Netlist can be received in various formats, for example:

- IPC-D-356
- ODB++ from Enterprise 3000
- Mentor Graphics neutral file.

The original net names used in the CAD system are unchanged in the Genesis 2000 database, facilitating communication exchanges with the designer. Most designers expect the CAD Netlist to be the final authority on how the manufactured board should function.

A CAD Netlist contains a list of nets. Each net has a unique name. Sometimes a net's name is related to its functionality. Here are some examples: **+3.3V** can be a name for a net of 3.3 volts, **CLK_62.5_MHZ** can be a name for a net of a digital clock pulse and **GND** can indicate a net used as an electrical ground.

Here are some examples of net names found in a CAD netlist.

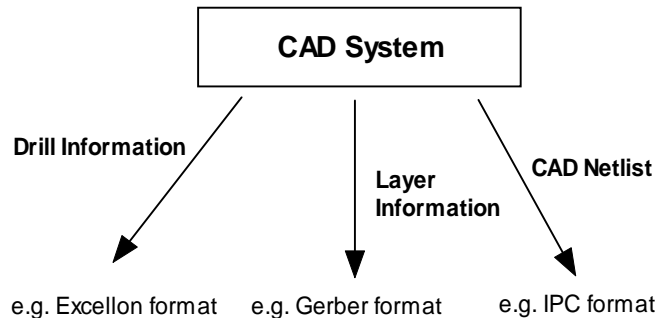
+1.8V_DSP
+1.8V_FPGA
+3.3V
CLK_62.5_MHZ
SSLO_RESET
VCC
GND

CAD Net Points

The CAD netlist supplied by the designer includes also the points of each net, which are called net points. Net points are coordinates along the net, mostly coordinates on pads in outer layers, or coordinates on plated drills which were set by the designer as points which define the connectivity of

Possible Netlist Problems with Data from Designers

In addition to the CAD netlist, the CAD system also gives information about the board layers and the drills. This information is graphic information and it is given in various formats (e.g., ODB++ or Gerber for layers and Excellon for drill).



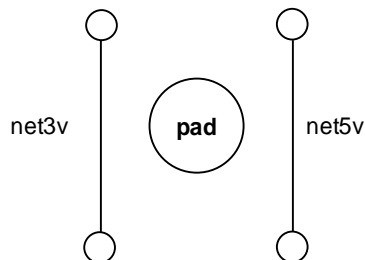
Sometimes there are differences between the CAD Netlist and the graphics (i.e., layers and drill information). These differences may be caused by format transformation errors, or because of an error in the CAD Netlist. We want to detect such errors to avoid fabrication damage.

Here are examples of 4 possible problems (violations): shorts, breaks, missing and extra.

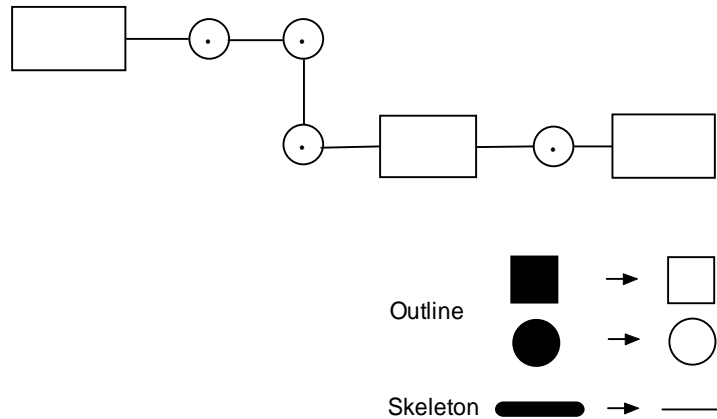
“Short” Problem

A **Short** occurs when points from two or more nets, in the CAD Netlist are connected to a single net according to the graphical data.

For example, suppose you had 2 nets in the original layer and a pad between them, like this:



The process that follows the net connectivity from point to point also generates a shape for each graphic feature in the net (outline for pads and surfaces, skeletons for lines and arcs).

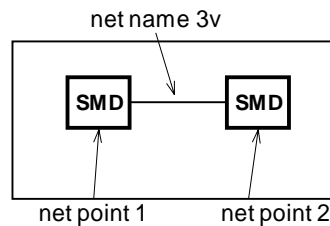


Why is there a Problem Comparing the CAD Netlist with the Current Netlist?

As explained previously, in order to detect netlist problems, comparison is made between the netlist created by Genesis and the CAD Netlist given by the designer.

Now we will discuss a problem which often happens when comparing a CAD Netlist and Current Netlist.

Let us first look at an example of a good case and then we will discuss the problem. Suppose we get this layer from the designer.



According to the CAD Netlist we have one net here called **3v** and two net points (netpoint 1 and 2).

What is the Current-Based CAD Netlist?

It order to overcome the problem of extra net points, we create a third type of netlist. This netlist uses only net points from the CAD netlist and finds their connectivity using the real features on the layer according to the graphical data. Therefore, it is called the **Current-Based-CAD Netlist** (i.e., it is calculated like the Current Netlist but it is based only on the CAD net points). When this netlist can be compared equally to the original **CAD Netlist**, we can be confident that the electrical connectivity of the original design has been preserved and one avoids the false alarms of “extra” violations.

What is a Reference Netlist?

This netlist serves as a basis from which to determine if netlist changes have resulted from editing operations. A **Reference Netlist** is created from any one of the above netlists. It can be set to be either the CAD Netlist, the Current Netlist, or Current-Based CAD Netlist. It is used as a reference during editing to ensure that changes made in the Graphic Editor do not violate netlist conductivity rules.

The recommended workflow is to first compare the CAD Netlist with the Current-Based CAD Netlist. Then, if the comparison passes without violations, it is recommended to set the Reference Netlist to be the Current Netlist.

This Reference Netlist will be as a reference to verify that our editing has not caused any netlist violations. From now on, every graphical change we make in the board will reflect the Current Netlist and by comparing the Current Netlist with the Reference Netlist, we can be sure that we have not caused any netlist violations.

What is On-line Netlist Check?

When editing a board, sometimes one wants to verify on-line that netlist violations have not occurred. This procedure is called **On-line Netlist Check**. On-line netlist check has 3 modes:

- 1 **None** - No on-line check is performed.

- 2 **Deferred mode** - The Current Netlist is compared to the Reference Netlist after some editing operations. The user decides when to carry out the comparison.
- 3 **Immediate mode** - In this mode, checking for netlist violations can be done before the editing operations. (i.e., if we are going to perform some editing actions which are going to cause a netlist violation, the Genesis system will not allow us to perform this action.

What is Registration of a CAD Netlist?

In many cases, a CAD netlist may not be aligned (registered) with the graphical data. If netlist points and features on the graphical layer do not overlap, then registration is needed.

In the next tutorial we will discuss how to register a CAD Netlist.

Summary



In this tutorial you were presented with general terms and concepts behind the Netlist Analyzer. You learned about:

- Nets and netlists
- Why you need to check the board's netlist
- When you need to check the netlist
- A CAD Netlist and CAD Netpoints
- Netlist problems with data from designers (short, break, missing and extra problems)
- How Genesis detects netlist problems
- A Current Netlist (what does it mean, how is it calculated and what are netlist shapes)
- The problem comparing the CAD Netlist and Current Netlist
- The Current-based CAD Netlist
- A Reference Netlist
- On-line netlist check
- Registration of a CAD netlist

This tutorial discussed general terms and concepts needed to understand the Netlist Analyzer. The next tutorial will show how to use the Netlist Analyzer.