

Technical Paper #1: How to transfer Schematics Properties from ORCAD to Allegro Layout Software
Emre Uludemir, Founder © 2006 Baykal Technology, Inc.

Introduction:

This technical paper explains how to transfer some useful schematics properties to Allegro Layout tool from ORCAD Schematics Capture tool version 9.2 or newer. In this example we use ORCAD 10.5 and Allegro SPB 15.5. Schematics properties are certain fields associated to nets, which are used for distinguishing these nets from the rest of the schematics nets on the design. Typically, these assigned properties would have some features different than rest of the nets.

Which ORCAD properties to pass into Allegro:

Through out the years I have been designing many boards, I found that many boards have the following types of nets. We can call these nets as net classes. In general, we can simplify that most of the designs have the following class of nets:

POWER
CLOCK
DIFFERENTIAL

These nets most of the time should have different design rules for layout. ORCAD Schematics Net properties tell these differences to layout tool. In Allegro there are three types of design rules for layout:

- 1-) Spacing Rule Set
- 2-) Physical Rule Set
- 3-) Default: Nets that do not have any property assignments in schematics will have the default constraints assignment in Allegro.

Most common ORCAD net properties to pass to Allegro:

NET_PHYSICAL_TYPE
NET_SPACING_TYPE
PROPOGATION_DELAY
RELATIVE_PROPOGATION_DELAY

In ORCAD, the text name assignment inside the NET_PHYSICAL_TYPE property will be passed into Allegro. In Allegro, those nets will have the same Layout Physical Rule Set. Similarly, the text property name inside NET_SPACING_TYPE will pass to Allegro through the Allegro Netlister, and nets which have this property will have the same Layout Spacing Rule Set.

PROPOGATION_DELAY and RELATIVE_PROPOGATION_DELAY values can be entered as distance or actual time delay. These properties are very useful for PLL feedbacks or to pass certain critical net routings.

The purpose of classifying nets in different classes in design through ORCAD Net Properties is to control the layout process. When these properties entered and passed into Allegro, any violations in Layout Database will show as DRC(Design Rule Check) errors.

How to pass ORCAD Net Properties to Allegro:

After deciding which nets to separate from default design rules and how many different net classes you would have, now you can enter them. Let's say that you have decided to have the following net classes, CLOCK, POWER and Default. In some applications, you might have two or more different layout rules for CLOCKS, where you might classify these clocks as CLOCK, and CLOCK_MORE_SENSETIVE classes to distinguish them. In this example, each type of net has the same design rule set for layout.

Each net class will have one assigned NET_SPACING_TYPE or NET_PHYSICAL_TYPE type in our example. This means that, in our net classes, all nets in that class will have the same NET_SPACING_TYPE and NET_PHYSICAL_TYPE properties.

Here is how we tell ORCAD to pass distinctive properties to Allegro:

(1) We have classified our schematics design nets into three distinctive classes:

- CLOCK
- POWER
- DIFFERENTIAL

(2) We have picked two ORCAD net properties to identify our classes:

- NET_PHYSICAL_TYPE
- NET_SPACING_TYPE

(3) Now in ORCAD go to flat nets and enter properties for each net :

- CLOCK → NET_PHYSICAL_TYPE
- POWER → NET_PHYSICAL_TYPE

- CLOCK → NET_SPACING_TYPE
- POWER → NET_SPACING_TYPE

(4) When you pass netlist to Allegro, above properties will be assigned as Allegro Extended Design Rules: Although, these properties are passed to Allegro, PCB designer still needs to manually enter following rule sets into extended design rules:

CLOCK
POWER

