Introduction

Many PCB’s today use split Power and Ground planes. Users may want the Ground planes to only cover nets that are associated with that Ground net. A good example is an Analog and Digital ground. You can use the Constraint Region function inside PCB Editor to allow nets of a specific NET CLASS only in this area. To use this command you must be running either an Allegro PCB Designer License or and OrCAD PCB Designer Professional license. This feature is not supported with an OrCAD PCB Designer Standard License.

How to define a net specific area in PCB Editor.

To start you need to group your nets into netclasses in PCB Editor. In the example below we have four regions A, B, C and D so we create four netclasses ACLASS, BCLASS, CCLASS and DCLASS. To do this LMB (left mouse button) the first net, then Shift+LMB the last net in Constraint Manager then RMB (right mouse button) Create – Class, define the name, the nets are then listed under the Class name. Ensure when you create the class that it is created for both physical and spacing rules.

Next we want to define the Constraint Areas. Add the four required areas using Shape – Rectangle (Polygon or Circular), then define the class / subclass as Constraint Region / All (or layer specific) and then specify a Region name in the Assign to Region box. This example the regions are called A_REGION, B_REGION, C_REGION and D_REGION.
Now we have the regions and netclasses we need to create region classes for all the regions to the netclasses so for Region A we create a region class to netclass A, B, C and D. To do this go to Constraint Manager – Physical – Region All Layers and RMB on the A_REGION – Create – Region Class, the LMB the A_REGION and the ACLASS then Apply, Then LMB the A_REGION and the BCLASS then Apply. Repeat for the CCLASS and the DCLASS. Once complete you will see the Region Classes below the Region names. Repeat for the other three Regions B, C and D.

The next step is to create a Physical Cset that doesn’t allow Etch. Copy the Default Physical Cset (RMB – Create - Physical Cset) to create a new rule called NO_ETCH and change the Etch Allow to False.

Apply the new Physical Rule (NO_ETCH) to the Region classes that are not allowed in that specific area. So for example for Region A you would apply the rule to B, C and D.
How to Define a net specific area in PCB Editor

The setup is complete. Test the design by routing a net from the BClass into any of the other regions. You will see DRC’s when the net is not in the region allowed. The DRC Error produced is S-N meaning Etch Not Allowed.

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134

Notice of Disclaimer: Parallel Systems is providing this design, code, or information “as is.” By providing the design, code, or information as one implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from any claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.