



# How to Define a net specific area in PCB Editor

## 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.

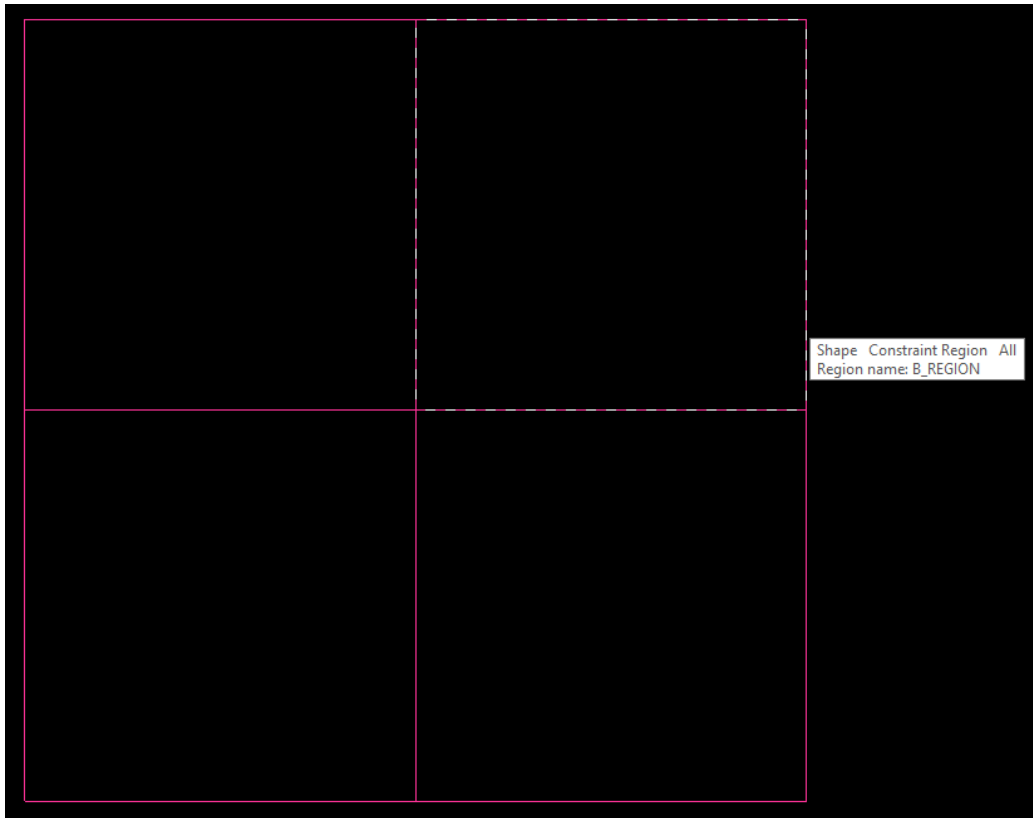
## How to define a net specific area in PCB Editor.

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 A\_CLASS, B\_CLASS, C\_CLASS and D\_CLASS. To do this LEFT CLICK the first net, then Shift + Left click the last net in Constraint Manager then right click > 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.

Dsn		net_areas	DEFAULT	0.2000	0.0000	0.1270	0.0000
NCIs		A_CLASS(20)	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		AGND	POWER	0.4000	0.0000	0.2000	0.0000
Net		AVCC	POWER	0.4000	0.0000	0.2000	0.0000
Net		A_AREF	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A_CS+	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A_CS-	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A_CT	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A_DB	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A_INV	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A_PVE	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A_PVSET	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A_REF	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A_REFIN	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A_RT	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A_SD	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A0	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A1	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A2	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A3	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A4	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		A5	DEFAULT	0.2000	0.0000	0.1270	0.0000
NCIs		B_CLASS(20)	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		BGND	POWER	0.4000	0.0000	0.2000	0.0000
Net		BVCC	POWER	0.4000	0.0000	0.2000	0.0000
Net		B_AREF	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B_CS+	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B_CS-	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B_CT	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B_DB	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B_INV	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B_PVE	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B_PVSET	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B_REF	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B_REFIN	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B_RT	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B_SD	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B0	DEFAULT	0.2000	0.0000	0.1270	0.0000
Net		B4	DEFAULT	0.2000	0.0000	0.1270	0.0000

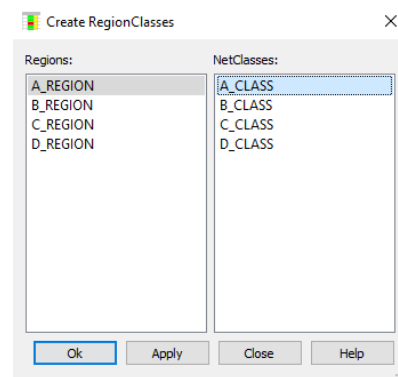
Next we want to define the Constraint Areas. Add the four required areas using Shape > Rectangle (Polygon or Circular), then define the class / subclass as Constraint Region / All (or layer specific) and then specify a Region name in the Assign to Region box. This example the regions are called A\_REGION, B\_REGION, C\_REGION and D\_REGION.

## How to Define a net specific area in PCB Editor



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 right click on the A\_REGION > Create > Region Class, the left click the A\_REGION and the A\_CLASS then Apply, Then left click the A\_REGION and the B\_CLASS then Apply. Repeat for the C\_CLASS and the D\_CLASS. Once complete you will see the Region Classes below the Region names. Repeat for the other three Regions B, C and D.

Objects			Referenced Physical CSet	Line Width		
Type	S	Name		Min	Max	Min
*	*	*	*	*	*	*
			mm		mm	
Dsn		net_areas	DEFAULT	0.2000	0.0000	0.1270
Rgn		A_REGION				
Rgn		B_REGION				
Rgn		C_REGION				
Rgn		D_REGION				



Objects		
Type	S	Name
*	*	*
Dsn		net_areas
Rgn		A_REGION
RCIs		A_CLASS
RCIs		B_CLASS
RCIs		C_CLASS
RCIs		D_CLASS
Rgn		B_REGION
Rgn		C_REGION
Rgn		D_REGION

## How to Define a net specific area in PCB Editor

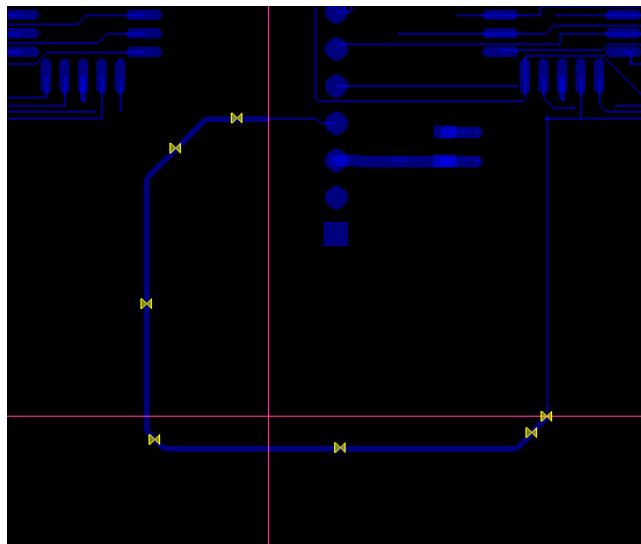
The next step is to create a Physical Cset that doesn't allow Etch. Copy the Default Physical Cset (Right click > Create > Physical Cset) to create a new rule called NO\_ETCH and change the Etch Allow to False.

Objects			Referenced Physical CSet	Line Width		Neck		Differential Pair				Vias	BB Via Stagger		Allow		
Type	S	Name		Min	Max	Min Width	Max Length	Min Line Spacing	Primary Gap	Neck Gap	(+)Tolerance		(-)Tolerance	Min	Max	Pad-Pad Connect	Etch
*	*	*	*	mm	mm	mm	mm	mm	mm	mm	mm	mm	mm	mm	*	*	
Dsn		net_areas	DEFAULT	0.2000	0.0000	0.1270	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	VIA-0_45-...	0.1270	0.0000	ALL_ALLOW...	TRUE
PCS		DEFAULT		0.2000	0.0000	0.1270	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	VIA-0_45-...	0.1270	0.0000	ALL_ALLOW...	TRUE
PCS		NO_ETCH		0.4000	0.0000	0.2000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	VIA-0_5-0_2	0.1270	0.0000	ALL_ALLOW...	FALSE
PCS		POWER		0.4000	0.0000	0.2000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	VIA-0_5-0_2	0.1270	0.0000	ALL_ALLOW...	TRUE

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.

Objects			Referenced Physical CSet	
Type	S	Name		
*	*	*	*	*
Dsn		net_areas	DEFAULT	0.200
Rgn		A_REGION		
RCls		A_CLASS		
RCls		B_CLASS	NO_ETCH	0.400
RCls		C_CLASS	NO_ETCH	0.400
RCls		D_CLASS	NO_ETCH	0.400
Rgn		B_REGION		
RCls		A_CLASS	NO_ETCH	0.400
RCls		B_CLASS		
RCls		C_CLASS	NO_ETCH	0.400
RCls		D_CLASS	NO_ETCH	0.400
Rgn		C_REGION		
RCls		A_CLASS	NO_ETCH	0.400
RCls		B_CLASS	NO_ETCH	0.400
RCls		C_CLASS		
RCls		D_CLASS	NO_ETCH	0.400
Rgn		D_REGION		
RCls		A_CLASS	NO_ETCH	0.400
RCls		B_CLASS	NO_ETCH	0.400
RCls		C_CLASS		
RCls		D_CLASS	NO_ETCH	0.400

The setup is complete. Test the design by routing a net from the B\_Class 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  
Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

#### Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

**NOTICE OF DISCLAIMER:** Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible 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 claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.