

Hughes Circuits Design Rules – PADS Layout

Hughes Circuits, Inc. has provided these recommended design rules and via sizes for use in your PCB layouts. These design rules are a great starting point for your layout because they will help you to design a PCB that is optimized for manufacturability. These design rules correspond to Hughes Circuits’ “Standard” manufacturing level. The PADS “macro” file should be loaded into to your PADS Layout database before beginning placement of components, and before entering any other design rules. Depending on the requirements of your specific design, it may be necessary to modify these design rules and/or create additional design rules. Below is a description of each design setting, and the corresponding PADS Layout Default setting.

Hughes Circuits, Inc. Clearance rules

Clearance Rules: Default rules

Same net

All	Corner	Via
Via		5
SMD	5	5
Trace	0	
Pad	5	

Trace width

Minimum	Recommended	Maximum
5	8	250

Clearance

All	Trace	Via	Pad	SMD	Copper
Trace	5				
Via	5	5			
Pad	5	5	5		
SMD	5	5	5	5	
Text	5	5	5	5	
Copper	10	10	10	10	10
Board	50	50	50	50	50
Drill	10	10	10	10	10

Other

Drill to drill: 10 Body to body: 10

Buttons: OK, Cancel, Delete, Help

<i>Rule Type and Description</i>	<i>Hughes Circuits Preferred</i>	<i>PADS Layout Default</i>
---	---	-----------------------------------

Same net - Defines minimum spacing between objects of the same net

Via to Via	5mil	6mil
SMD to Corner	5mil	6mil
SMD to Via	5mil	6mil
Trace to Corner	0mil	0mil
Pad to Corner	5mil	6mil

Other - Miscellaneous spacing requirements

Drill to drill	10mil	6mil
Recommended	8mil	6mil

Trace width - Defines trace widths used to route nets

Minimum	5mil	12mil
Recommended	8mil	12mil
Maximum	250mil	12mil

Clearance - Defines spacing between all objects of different nets

Trace to Trace	Trace = all traces	5mil	6mil
Via to Trace	Via = all via pads	5mil	6mil
Via to Via		5mil	6mil
Pad to Trace	Pad = through hole pads	5mil	6mil
Pad to Via		5mil	6mil
Pad to Pad		5mil	6mil
SMD to Trace	SMD = surface mount pads	5mil	6mil
SMD to Via		5mil	6mil
SMD to Pad		5mil	6mil
SMD to SMD		5mil	6mil
Text to Trace		5mil	6mil
Text to Via		5mil	6mil
Text to Pad		5mil	6mil
Text to SMD		5mil	6mil
Copper to Trace	Copper = pours and planes	10mil	6mil
Copper to Via		10mil	6mil
Copper to Pad		10mil	6mil
Copper to SMD		10mil	6mil
Copper to Copper		10mil	6mil
Board to Trace	Board = board outline	50mil	6mil
Board to Via		50mil	6mil
Board to Pad		50mil	6mil
Board to SMD		50mil	6mil
Board to Copper		50mil	6mil
Drill to Trace	Drill = all drilled holes	10mil	6mil
Drill to Via		10mil	6mil
Drill to Pad		10mil	6mil
Drill to SMD		10mil	6mil
Drill to Copper		10mil	6mil

Hughes Circuits, Inc. Via Pad Stack settings

Pad Stacks Properties - STANDARDVIA

Pad Stack Type
☐ Decal ☒ Via

Decal name:
 LARGERVIA
 SMALLERVIA
 STANDARDVIA

Pin: Plated: Sh.: Sz.: Layer:
 CNN 24 <Start>
 CNN 24 <Inner Layers>
 CNN 24 <End>

Add Delete Add Delete

Parameters
 Use Global Defaults

Pad style: Pad

☐ Pad size relative to drill size

Diameter: 24

Preview:

Vias
 Name: STANDARDVIA
☒ Through ☐ Partial

Start layer:
 End layer:

Drill size: 12 ☒ Plated

Slot Parameters
☐ Slotted
 Length:
 Orientation:
 Offset:

Decal Units
☒ Mils
☐ Metric

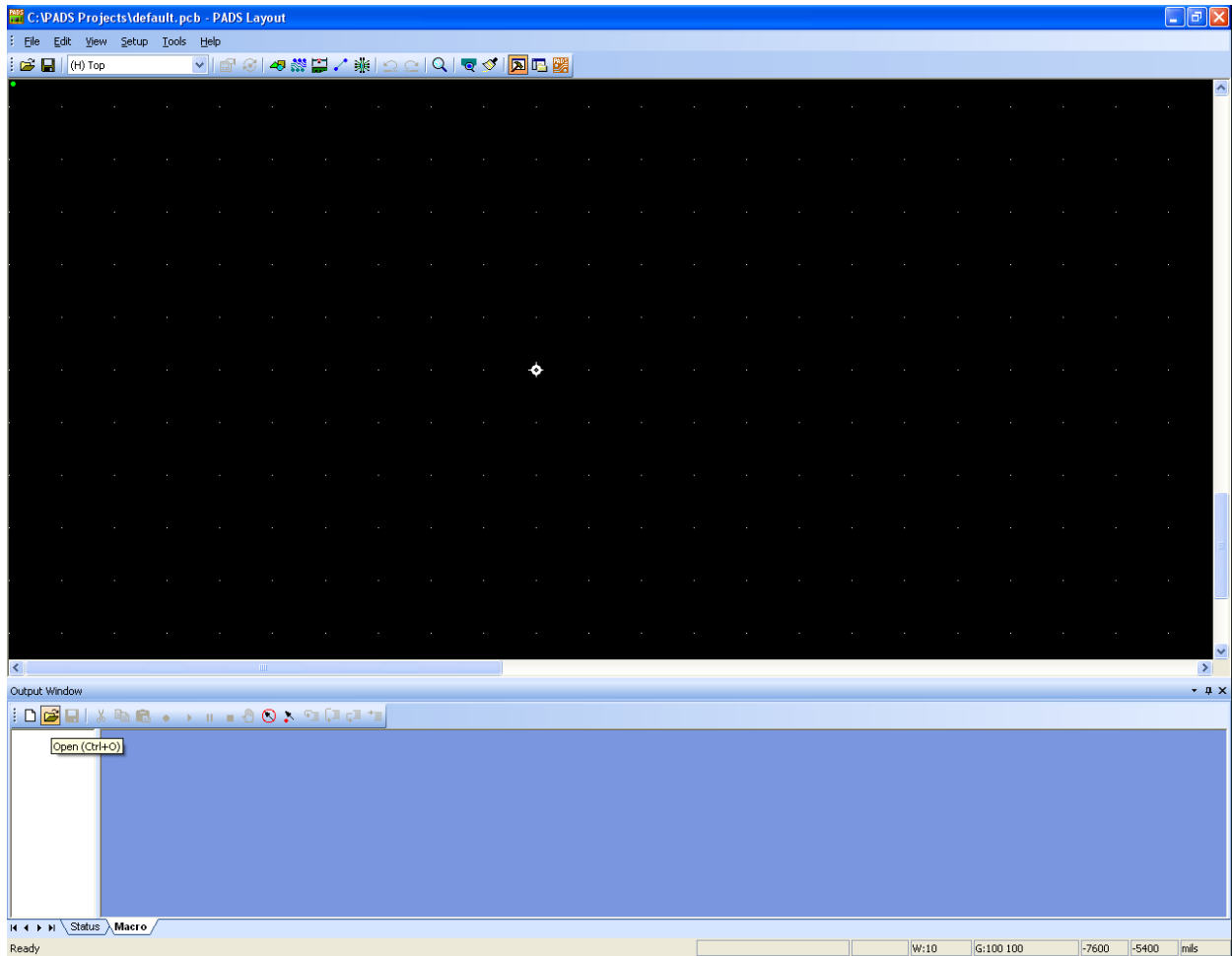
OK Cancel Help List List All

Pad Stacks Properties>Via - Defines multiple via sizes for use in routing nets

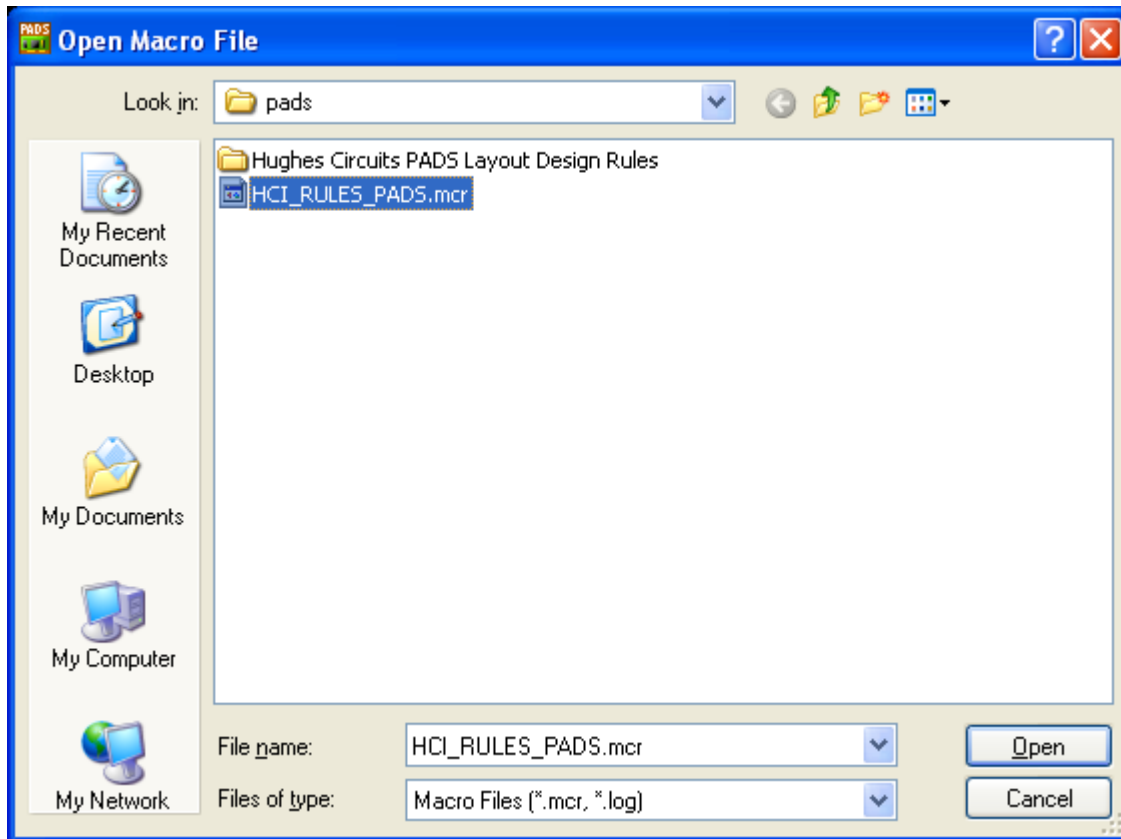
STANDARDVIA	24mil Pad / 12mil Hole	50mil Pad / 37mil Hole
LARGERVIA	32mil Pad / 20mil Hole	NONE
SMALLERVIA	22mil Pad / 10mil Hole	NONE

Instructions for Loading Hughes Circuits, Inc. PADS Macro File

Open the “Output Window” and click the “Macro” Tab.



Click *Open*, and select the HCI_RULES_PADS.mcr file.



Press *Run* to replay the macro, and set your design rules and via sizes .

