

<https://techdocs.altium.com/display/ADOH/Importing%20and%20Exporting%20Rules>

Altium Designer (17.0) - G:\My Drive\regulador fuente paritad\esquema\PCB1.PcbDoc - esquemaco.PjPcb: Not signed in

DXP File Edit View Project Place Design Tools Route Reports Window Help G:\My Drive\regulador fuente paritad\esquema\PCB1.PcbDoc - esquemaco.PjPcb: Not signed in

Workspace1.DsnWk Workspace

esquemaco.PjPcb Project

Files Structure

esquemaco.PjPcb

Source Documents

PCB1.PcbDoc

Sheet1.SchDoc

Settings

Libraries

Generated

Free Documents

PCB Rules and Constraints Editor (mm)

Design Rules

- Electrical
 - Clearance
 - Clearance
 - Short-Circuit
 - Component Clearance
 - Un-Routed Net
 - Un-Connected Pin
 - Fabrication Testpoint
 - Modified Polygon
- Routing
 - Width
 - Width
 - Routing Topology
 - Routing Pairs
 - Routing Layers
 - Routing Corners
 - Routing via Style
 - Fanout Control
 - Differential Pairs Routing
 - Mask
 - Plane
 - Testpoint
 - Manufacturing
 - High Speed
 - Placement
 - Signal Integrity

Name	P.	Ena.	Type	Category	Scope	Attributes
Assembly Testpoint	1	✓	Assembly Testpoint	Testpoint	All	Under Comp - Allow
Assembly Testpoint	1	✓	Assembly Testpoint	Testpoint	All	Testpoint - One Requi
Clearance	1	✓	Clearance	Electrical	All	Clearance = 0.3mm
Component Clearance	1	✓	Component Clearance	Placement	All	Horizontal Clearance =
Differential Pairs	1	✓	Differential Pairs	Routing	All	Pref Gap = 0.254mm
Fabrication Testpoint	1	✓	Fabrication Testpoint	Testpoint	All	Under Comp - Allow
Fabrication Testpoint	1	✓	Fabrication Testpoint	Testpoint	All	Testpoint - One Requi
Fanout_BGA	1	✓	Fanout Control	Routing	IBGA	Style - Auto Directio
Height	1	✓	Placement	Placement	All	Pref Height = 12.7mm
HoleSize	1	✓	Hole Size	Manufacturin	All	Min = 0.025mm Ma
HoleToHoleClearanc	1	✓	Hole To Hole Clearanc	Manufacturin	All	Hole To Hole Clearanc
LayerPairs	1	✓	Layer Pairs	Manufacturin	All	Layer Pairs - Enforce
MinimumSolderMas	1	✓	Minimum Solder Mask	Manufacturin	All	Minimum Solder Mask
NetAntennae	1	✓	Net Antennae	Manufacturin	All	Net Antennae Toleran
PasteMaskExpansion	1	✓	Paste Mask Expansion	Mask	All	Expansion = 0mm
PlaneClearance	1	✓	Power Plane Clearance	Plane	All	Clearance = 0.508mm
PlaneConnect	1	✓	Power Plane Connect	Plane	All	Style - Relief Connect
PolygonConnect	1	✓	Polygon Connect	Style Plane	All	Style - Relief Connect
RoutingCorners	1	✓	Routing Corners	Routing	All	Style - 45 Degree M
RoutingLayers	1	✓	Routing Layers	Routing	All	TopLayer - Enabled B
RoutingPriority	1	✓	Routing Priority	Routing	All	Priority = 0
RoutingTopology	1	✓	Routing Topology	Routing	All	Topology - Shortest
RoutingVia	1	✓	Routing via Style	Routing	All	Pref Size = 1.27mm
ShortCircuit	1	✓	Short-Circuit	Electrical	All	Short Circuit - Net Ali
SilkToBoardRegionC	1	✓	Silk To BoardRegion	Electrical	All	Silk to Board Region C
SilkToSolderMask	1	✓	Silk To SolderMask	Manufacturin	All	Silk to Silk Clearance =
SilkToSolderMaskC1	1	✓	Silk To Solder Mask C1	Manufacturin	IsPad	Silk to Solder Mask C1
SolderMaskExpansion	1	✓	Solder Mask Expansion	Mask	All	Expansion = 0.127mm

New Rule Delete Rule(s) Duplicate Rule Report...

Rule Wizard... Priorities... Create Default Rules OK Cancel Apply

click derecho sobre Design Rules y ahi se puede importar y exportar reglas de pcb.