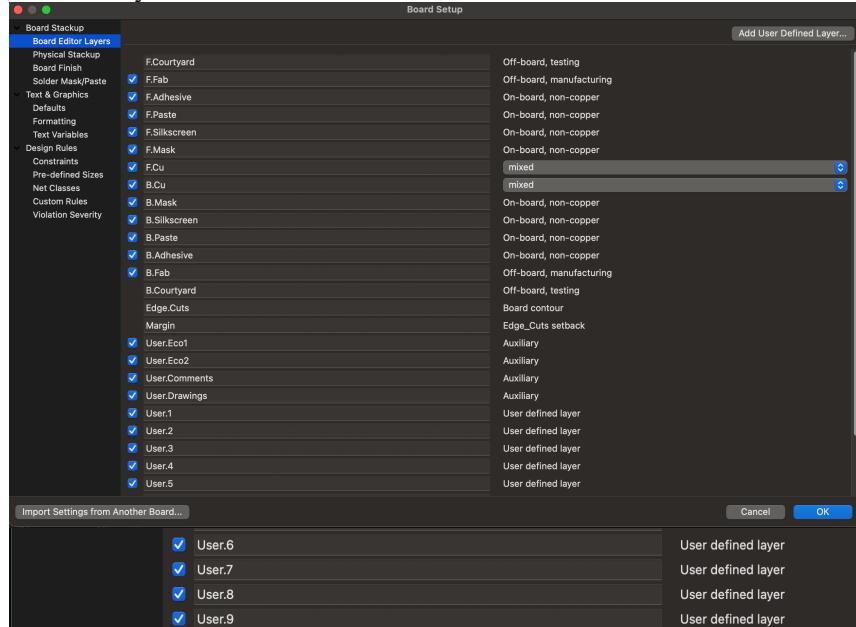


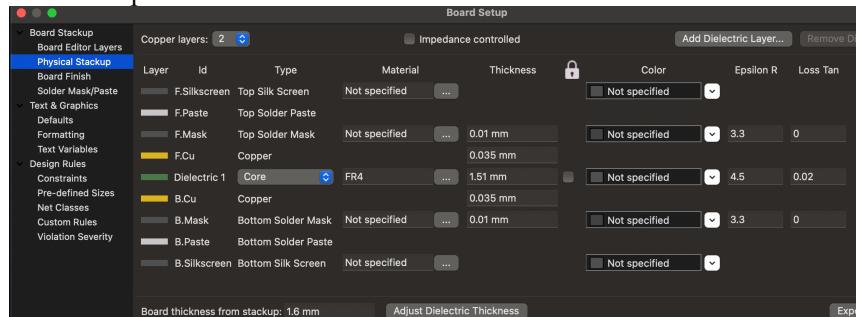
KiCad Settings Configuration

Board Stackup:

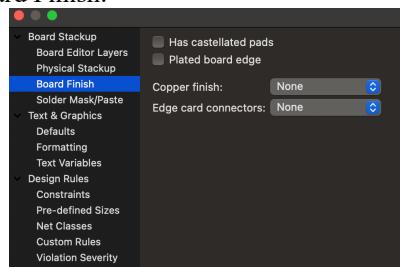
- Board Editor Layers:



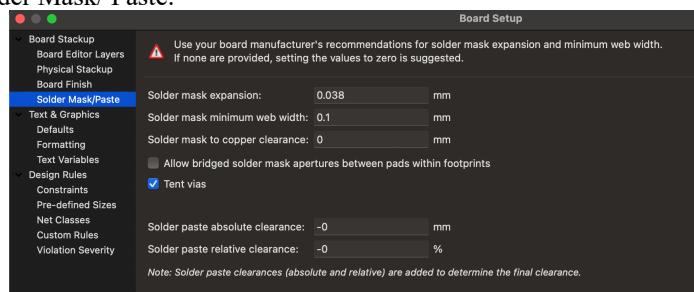
- Physical Stackup:



- Board Finish:

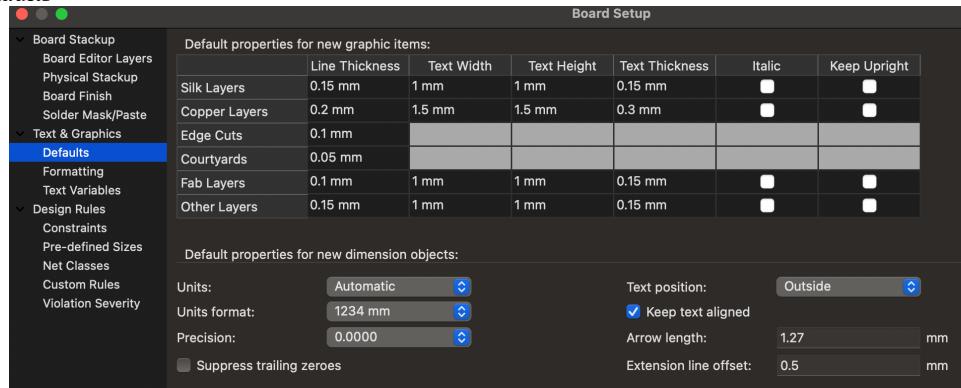


- Solder Mask/ Paste:

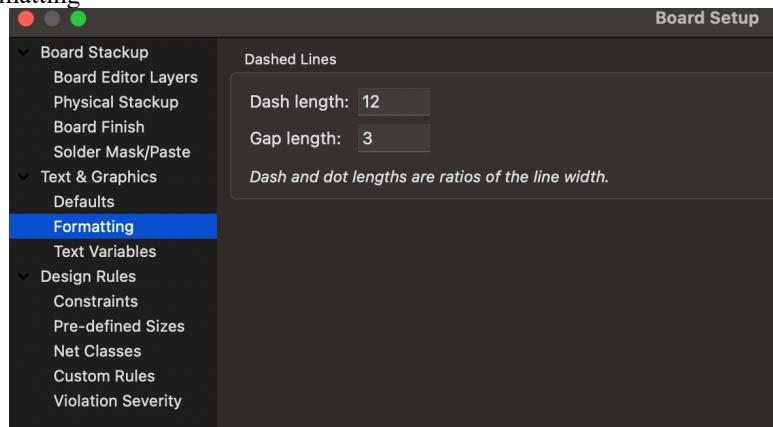


Text & Graphics

- Defaults

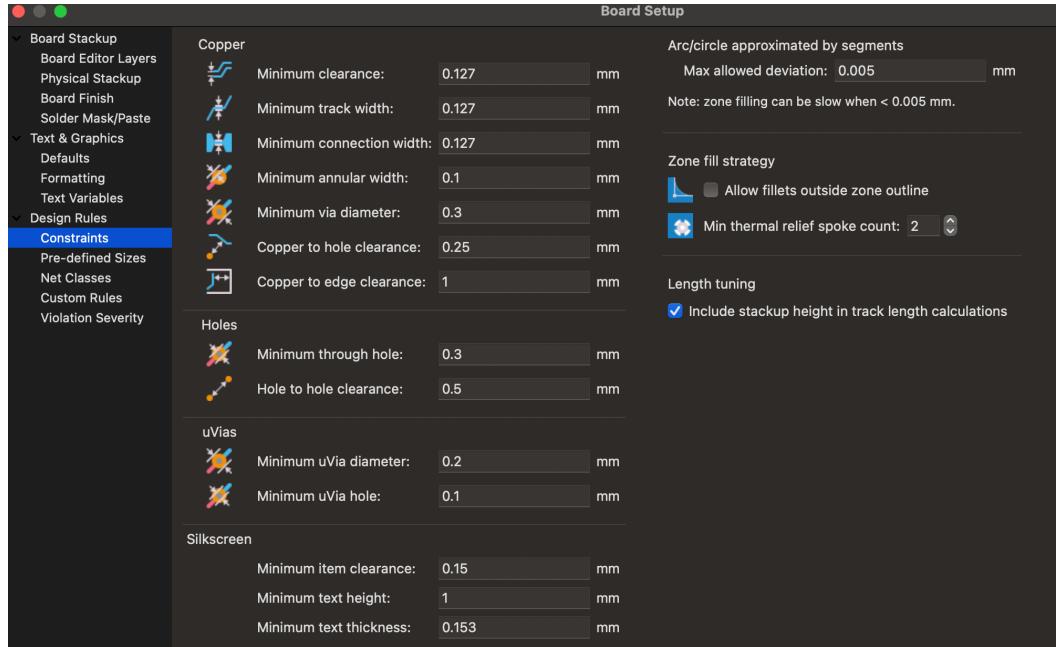


- Formatting



Design Rules

- Constraints:



- Pre-Defined Sizes:

The screenshot shows the 'Board Setup' dialog with the 'Pre-defined Sizes' tab selected. The left sidebar lists various design rules categories. The main area is divided into three tables: Tracks, Vias, and Differential Pairs. The 'Tracks' table has one row with 'Width' set to '0.127 mm'. The 'Vias' table has two rows with 'Diameter' set to '0.6 mm' and 'Hole' set to '0.3 mm'. The 'Differential Pairs' table is currently empty.

Tracks				Vias		Differential Pairs		
	Width			Diameter	Hole	Width	Gap	Via Gap
	0.127 mm			0.6 mm	0.3 mm			
	0.178 mm							
	0.254 mm							
	1 mm							
	2.032 mm							

- Net Classes:

Board Stackup

Name	Clearance	Track Width	Via Size	Via Hole	uVia Size	uVia Hole	DP Width	DP Gap
Default	0.2 mm	0.25 mm	0.6 mm	0.3 mm	0.3 mm	0.1 mm	0.2 mm	0.25 mm
POWER	0.2 mm	2.032 mm	0.6 mm	0.3 mm	0.3 mm	0.1 mm	0.2 mm	0.25 mm
SIGNAL	0.2 mm	0.254 mm	0.6 mm	0.3 mm	0.3 mm	0.1 mm	0.2 mm	0.25 mm

Netclasses:

Pattern	Net Class
GND	POWER
SIGNAL	SIGNAL
POWER	POWER
/R1_LED1	SIGNAL
/ESP_LED1	SIGNAL
/R2_LED2	SIGNAL
/ESP_LED2	SIGNAL
/R3_LED3	SIGNAL
/ESP_LED3	SIGNAL
/R4_LED4	SIGNAL
/ESP_LED4	SIGNAL
/R5_LED5	SIGNAL
/ESP_LED5	SIGNAL
/R6_LED6	SIGNAL
/ESP_LED6	SIGNAL
/R7_LED7	SIGNAL
/ESP_LED7	SIGNAL

Netclass assignments:

Pattern	Net Class
/R8_LED8	SIGNAL
/ESP_LUNA_LED8_TXD	SIGNAL
/LUNA_ESP_RXD	SIGNAL
/ESP_MD_B_AIA	SIGNAL
/ESP_MD_B_AIB	SIGNAL
/ESP_MD_B_BIA	SIGNAL
/ESP_MD_B_BIB	SIGNAL
/ESP_MD_A_BIB	SIGNAL
/ESP_MD_A_BIA	SIGNAL
/ESP_MD_A_AIB	SIGNAL
/ESP_MD_A_AIA	SIGNAL
/ESP_VL_A_XSHUT	SIGNAL
/ESP_VL_B_XSHUT	SIGNAL
/ESP_VL_C_XSHUT	SIGNAL
/MPU_ESP_INT	SIGNAL
/ESP_MPU_VL_SCL	SIGNAL
/ESP_MPU_VL_SDA	SIGNAL
+3.3V	POWER
+5V	POWER
/CHRG_BB_IN_P	POWER
/CHRG_BAT_POS	POWER
/BB_EN	SIGNAL
+9V	POWER

Nets matching 'GND':
GND

- Violation Severity:

Board Setup			
	Electrical		
Board Stackup	Items shorting two nets:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Board Editor Layers	Tracks crossing:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Physical Stackup	Clearance violation:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Board Finish	Via is not connected or connected on only one layer:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Solder Mask/Paste	Track has unconnected end:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Text & Graphics	Thermal relief connection to zone incomplete:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Defaults			<input type="radio"/> Ignore
Formatting			<input type="radio"/> Ignore
Text Variables			<input type="radio"/> Ignore
Design Rules	Design for Manufacturing		
Constraints	Board edge clearance violation:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Pre-defined Sizes	Hole clearance violation:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Net Classes	Drilled holes too close together:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Custom Rules	Track width:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Violation Severity	Annular width:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
	Drill out of range:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
	Micro via drill out of range:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
	Courtyards overlap:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
	Footprint has no courtyard defined:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
	Footprint has malformed courtyard:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
	Board has malformed outline:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
	Copper sliver:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
	Solder mask aperture bridges items with different nets:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
	Copper connection too narrow:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Solder Mask/Paste	Schematic Parity		
Text & Graphics	Duplicate footprints:	<input type="radio"/> Error	<input checked="" type="radio"/> Warning
Defaults	Missing footprint:	<input type="radio"/> Error	<input checked="" type="radio"/> Warning
Formatting	Extra footprint:	<input type="radio"/> Error	<input checked="" type="radio"/> Warning
Text Variables	Pad net doesn't match schematic:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Design Rules	Missing connection between items:	<input checked="" type="radio"/> Error	<input type="radio"/> Warning
Constraints			<input type="radio"/> Ignore
Pre-defined Sizes			<input type="radio"/> Ignore
Net Classes			<input type="radio"/> Ignore
Custom Rules			<input type="radio"/> Ignore
Violation Severity			<input type="radio"/> Ignore
	Signal Integrity		
	Trace length out of range:	<input type="radio"/> Error	<input type="radio"/> Warning
	Skew between traces out of range:	<input type="radio"/> Error	<input type="radio"/> Warning
	Too many vias on a connection:	<input type="radio"/> Error	<input type="radio"/> Warning
	Differential pair gap out of range:	<input type="radio"/> Error	<input type="radio"/> Warning
	Differential uncoupled length too long:	<input type="radio"/> Error	<input type="radio"/> Warning
			<input type="radio"/> Ignore
	Readability		
	Silkscreen overlap:	<input type="radio"/> Error	<input checked="" type="radio"/> Warning
	Silkscreen clipped by solder mask:	<input type="radio"/> Error	<input checked="" type="radio"/> Warning
	Silkscreen clipped by board edge:	<input type="radio"/> Error	<input checked="" type="radio"/> Warning
	Text height out of range:	<input type="radio"/> Error	<input type="radio"/> Warning
	Text thickness out of range:	<input type="radio"/> Error	<input type="radio"/> Warning
			<input type="radio"/> Ignore
	Miscellaneous		
	Items not allowed:	<input type="radio"/> Error	<input type="radio"/> Warning
	Copper zones intersect:	<input type="radio"/> Error	<input type="radio"/> Warning
	Isolated copper fill:	<input type="radio"/> Error	<input type="radio"/> Warning
	Footprint is not valid:	<input type="radio"/> Error	<input type="radio"/> Warning
	Padstack is not valid:	<input type="radio"/> Error	<input checked="" type="radio"/> Warning
	PTH inside courtyard:	<input type="radio"/> Error	<input type="radio"/> Warning
	NPTH inside courtyard:	<input type="radio"/> Error	<input type="radio"/> Warning
	Item on a disabled copper layer:	<input type="radio"/> Error	<input type="radio"/> Warning
	Unresolved text variable:	<input type="radio"/> Error	<input type="radio"/> Warning
	Footprint component type doesn't match footprint pads:	<input type="radio"/> Error	<input type="radio"/> Warning
	Footprint not found in libraries:	<input type="radio"/> Error	<input checked="" type="radio"/> Warning
	Footprint doesn't match copy in library:	<input type="radio"/> Error	<input checked="" type="radio"/> Warning
	Through hole pad has no hole:	<input type="radio"/> Error	<input type="radio"/> Warning
			<input type="radio"/> Ignore