

Pulsonix RF Design Technology

RF Circuit Support in Pulsonix

The Pulsonix design tool supports RF (radio frequency) design through flexible design features integrated within the Schematic and PCB applications.

With the Pulsonix RF option, you can create RF specific features such as chamfered corners (where the inner corner is 90-degrees and the outer corner is 45-degrees), curved corners, flat ended tracks, and track back-off to avoid over-shoot.

Additional features include Gerber, DXF, IDF and STEP Import to generate complex shapes that have been calculated in external applications, such as Ansys, CST Microwave Studio, Simberian and RFSim99 from AD5GG. Many other products are also supported.

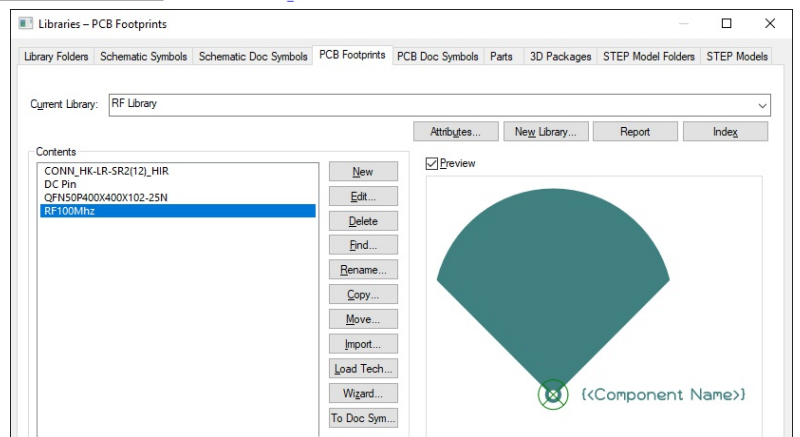
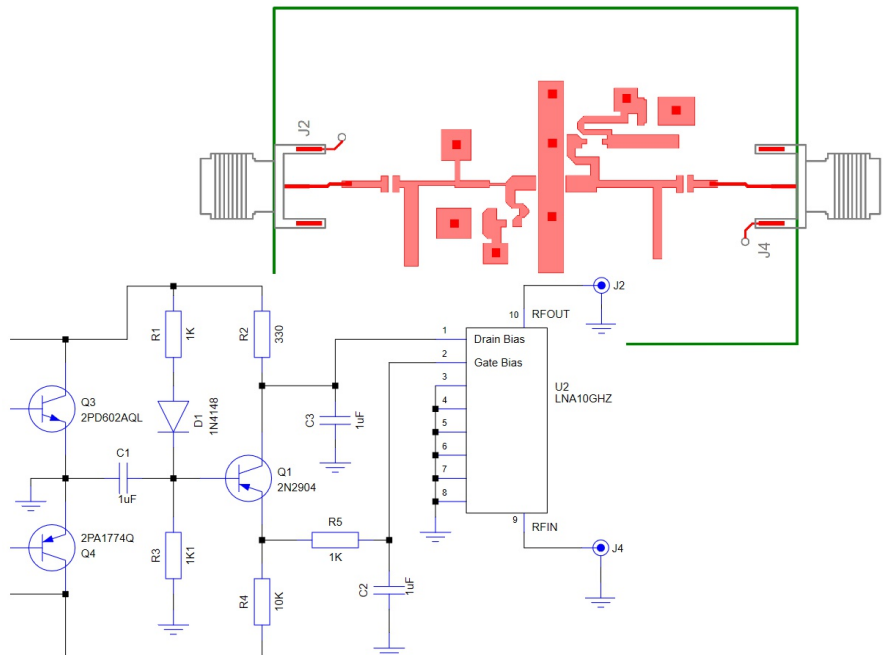
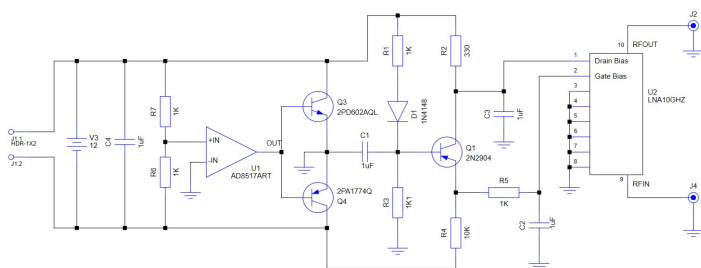
Powerful shape facilities within Pulsonix means complex shapes can be generated with ease and accuracy, these can be added to designs and libraries for completeness.

Library Components

Components in Pulsonix can be supported for RF design using standard tools within the product. Where standard RF shapes are required in the PCB footprint, they can be created by hand using facilities such as powerful Construction Lines and shape addition. Where shapes or components have been created externally, these can be imported using one of the Pulsonix Gerber import filters and native .plx, and the footprints saved for use.

Schematic Capture

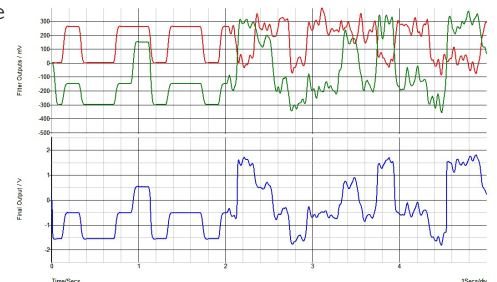
Create clear and concise schematic drawings in this easy-to-use design environment. A fully coherent schematic will mean front to back design integrity at all stages of the design. The Pulsonix schematic will generate the PCB data, it can also be used to create the Spice simulation netlist using the same component libraries, all within the one integrated application.



Spice Simulation

Generate Spice netlists from the Pulsonix schematic editor to your Spice simulator of choice. Pulsonix supports industry Spice application such as LTSpice, PSpice, SiMetrix, Micro-Cap etc. An integrated interface allows you to design the Spice type and also run the simulator engine from within Pulsonix.

From the Schematic, simulate the design using the integrated Spice interface



Layers and Layer Stacks

Layers with RF specific materials can easily be added to the Pulsonix PCB design. If layer stacks are created, calculated and verified externally to Pulsonix, these can then be imported into your PCB design using the IPC-2581 import option.

Name	Associated Layer	Class	Side	Bias	Net	Material	Thickness
Y Silkscreen Top	Silkscreen Top	None	None				0.00000
Y Top	Electrical	Top	X		Copper 1oz		0.035000
	Solder Mask Top	Solder Mask	Top	None		Mask (0.025)	0.025000
	Paste Mask Top	Paste Mask	Top	None			0.000000
Y Pin Names	Non-Electrical	Top	None				0.000000
Prepreg A	Construction	None	None		Prepreg (0.11)		0.110000
Prepreg B	Construction	None	None		Prepreg (0.11)		0.110000
Ground	Power Plane	Gnd	Inner		Copper 1oz		0.035000
Substrate	Construction	None	None		FR4 (1.0)		1.000000
Y Power	Power Plane	+5V	Inner		Copper 1oz		0.035000
Prepreg D	Construction	None	None		Prepreg (0.11)		0.110000
Prepreg C	Construction	None	None		Prepreg (0.11)		0.110000
Y Bottom	Electrical	Bottom	X				
	Silkscreen Bottom	Silkscreen	Bottom				
	Solder Mask Bottom	Solder Mask	Bottom				
	Paste Mask Bottom	Paste Mask	Bottom				
Y Pin Names (Bottom)	Non-Electrical	Bottom	None				
Y Documentation	Documentation						
Y Construction Lines	Documentation						
Drill Drawing	Drill Drawing	<Thru>					

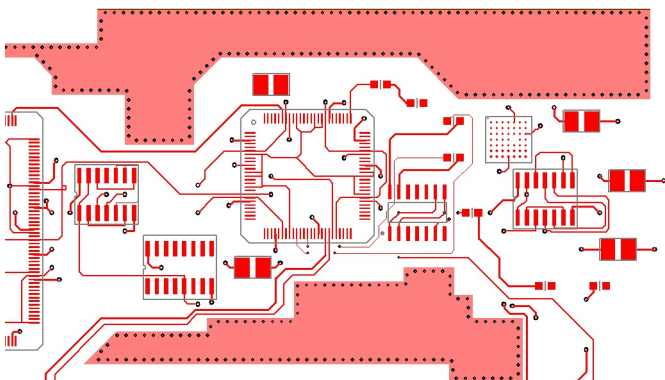
Board Materials

Pulsonix supports PCB and RF design materials along with their thickness, Electrical Conductivity and Dielectric Constant. When used on layers, these values are exported to the required formats for other systems, such as in the IPC-2581. Construction materials, foils and structures can be easily created and used on your layers.

Name	Thickness	Electrical Conductivity	Dielectric Constant (Er)	Special Use
Copper 1/2oz (0.018)	0.018	595900		
Copper 1oz (0.035)	0.035	595900		
FR4 (0.2)	0.200		4.3	
FR4 (0.3)	0.300		4.3	
FR4 (0.4)	0.400		4.3	
FR4 (1.0)	1.000		4.3	
Mask (0.025)	0.025			
Plating	0.030+	595900		Hole Plating
Prepreg (0.11)	0.110		4.2	
Thin Plating	0.020+	495900		Micro-Via Hole Plating

Complex Shapes and Shielding

Create complex shapes with Pulsonix PCB for RF for amplifiers, antennas, attenuators, passive and active filters, striplines and waveguides using the powerful tools provided. Add vias to copper shapes for stapling and shielding. Vias can also be added around tracks for shielding.



Track Impedance

Built-in design calculators are available to help calculate the track impedance during design based on the given track width and copper thickness. This calculator can also be used to calculate track thickness based on a required track impedance.

Scientific Track Width and Resistance Track Impedance Via Resistance

PCB Track Geometry: Microstrip Differential Track Pair Hide

Calculate: Impedance

Inputs:

Dielectric Thickness (H): 30 thou

Track Thickness (T): 1.01597 oz (per sq. f)

Track Width (W): 10 thou

Relative Permittivity (Er): 4

Results:

Characteristic Impedance: 110.302 Ohms

Capacitance: 1.22917 pF per: inch

Inductance: 14.9546 nH per: inch

Propagation Delay Time: 135.579 psec per: inch

Integration With External Verification Tools

Pulsonix supports interfaces to many external tools for PCB simulation, including impedance matching, crosstalk analysis as well as Spice simulation. Pulsonix supports export formats, such as ODB++ , IPC-2581, STEP and Spice. Where your process requires specialist RF design tools, Pulsonix can import data created as Gerber or DXF to ensure accuracy is maintained.

Manufacturing Outputs

Export industry formats from Pulsonix to manufacture your RF designs using Gerber, ODB++ , IPC-2581, NC-Drill, BOM, pick & place and assembly outputs. All exports are fully user customisable and tailored to your requirements.

Generate Plot - CAM Plots

Name	Enabled	Device	Process	Scale	Rotate	Mirror	Position	Area
Top	<input type="checkbox"/>	Gerber	Layer Top	1.000	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>
Power	<input type="checkbox"/>	Gerber	Layer Die Core Bottom	1.000	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>
Bottom	<input type="checkbox"/>	Gerber	Layer Bottom	1.000	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>
Silkscreen Top	<input type="checkbox"/>	Gerber	Layer Silkscreen Top	1.000	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>
Solder Mask Top	<input type="checkbox"/>	Gerber	Layer Solder Mask Top	1.000	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>
Paste Mask Top	<input type="checkbox"/>	Gerber	Layer Paste Mask Top	1.000	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>
Silkscreen Bottom	<input type="checkbox"/>	Gerber	Layer Silkscreen Bottom	1.000	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>
Solder Mask Bottom	<input type="checkbox"/>	Gerber	Layer Solder Mask Bottom	1.000	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>
Paste Mask Bottom	<input type="checkbox"/>	Gerber	Layer Paste Mask Bottom	1.000	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>
Documentation	<input type="checkbox"/>	Windows	Layer Documentation	1.000	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>
<Through Hole>	<input type="checkbox"/>	Excellon	Layer Span <Through Hole>	1.000	No Rotation	<input type="checkbox"/>	Auto Shift	<Design Extents>
Drill Drawing	<input type="checkbox"/>	Gerber	Layer Span <Through Hole>	1.000	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>