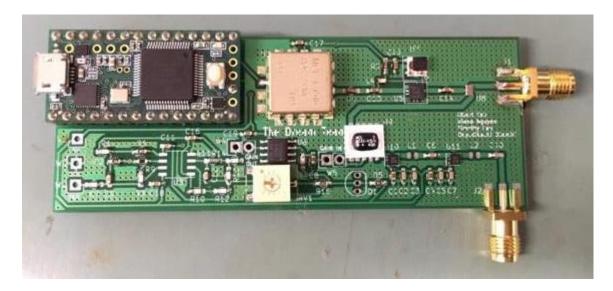
EEC 134AB

Application Note

Printed Circuit Board Design for High Frequency Systems

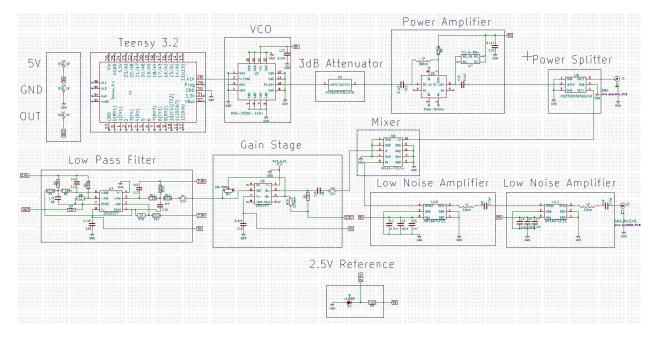
By: Timothy Lau



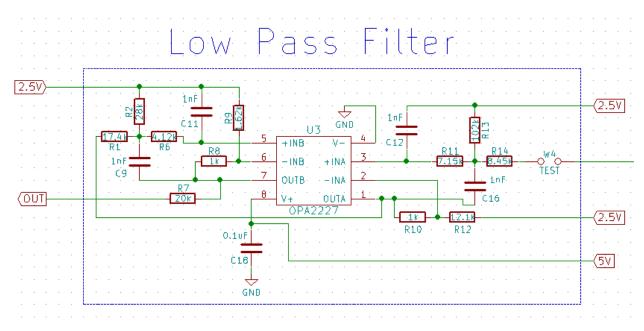
Introduction:

Printed circuit boards (PCB) are typically used in almost all modern electronic products and devices. They allow for circuits to be easily manufactured and assembled using automation. Furthermore, developing circuits with PCBs are cheaper and faster to manufacture than any current methods of circuit design. PCBs can be singled sided, double sided, or even multilayered to efficiently use space to save on costs, lower power consumption, and provide better performance.

Schematic Design:

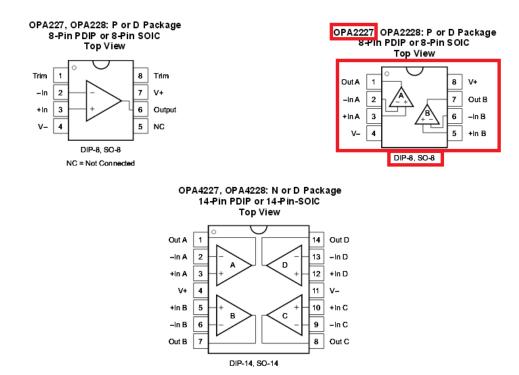


The first step to designing PCBs is to develop a circuit schematic. One of the most important aspects of schematic design is keeping the circuit clean and easy to read as this will help a great deal when it comes to PCB layout and debugging. Also, sectioning off circuit designs into modules will allow for transparency and even instantiation of modules as seen with the Low Noise Amplifiers in the image above. The schematic above was designed in KiCad 4.0.4.

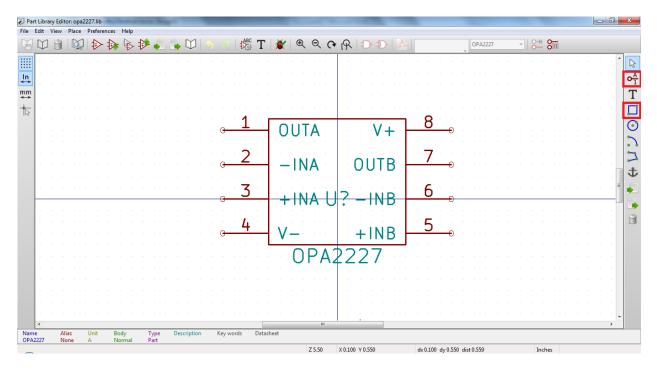


To start schematic designing, lets take a closer look at the Low Pass Filter module. Here, the main IC in this module is the OPA2227 High Precision, Low Noise Operational Amplifier.

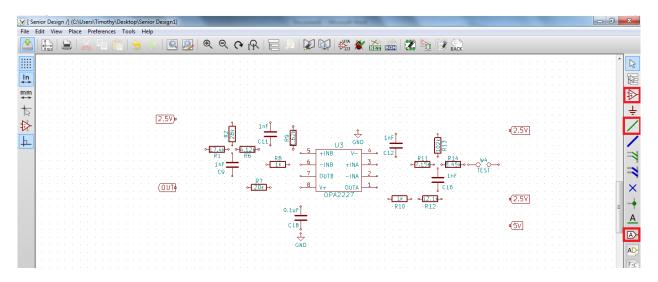
5 Pin Configuration and Functions



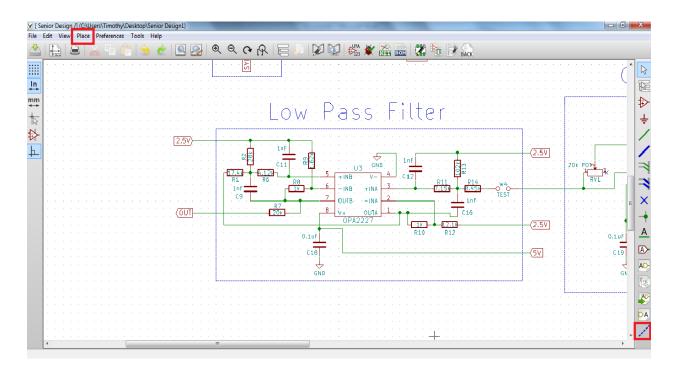
To get the pin configurations, we first consult the OPA2227 online datasheet. As seen above, we find the number of pins, pin configuration, and PCB footprint package. This allows us to create an OPA2227 block on our schematic.



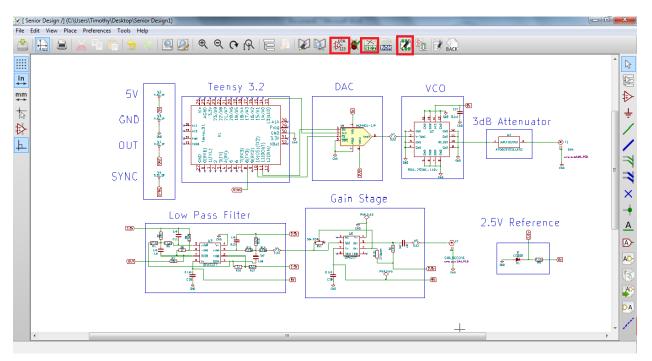
From KiCad, we enter the library editor and begin drawing our IC. From the datasheet, we follow the general pinout diagram and recreate it in our block. In the top right square is the 'Add pins to the component' option which allows for the user to enter input or output pins to their block diagram. After the correct number of pins is entered, they should be labeled accordingly. Once the pins have been placed, another option is the 'Add graphic rectangle to component body'. With this, the block is done and ready to be included in the library for use in our schematic.



Going back to our schematic diagram, other circuit components of the Low Pass Filter module can be placed using the option in the right bar called 'Place Component'. Additionally at the bottom of the right bar is an option for 'Place global label' which allows for wireless connections. For my schematic, I used these wireless connections to connect a global 5V power rail and 2.5V reference voltage. Once the component placing is complete, we can begin to connect them all using the 'Place wire' option located also in the right bar.



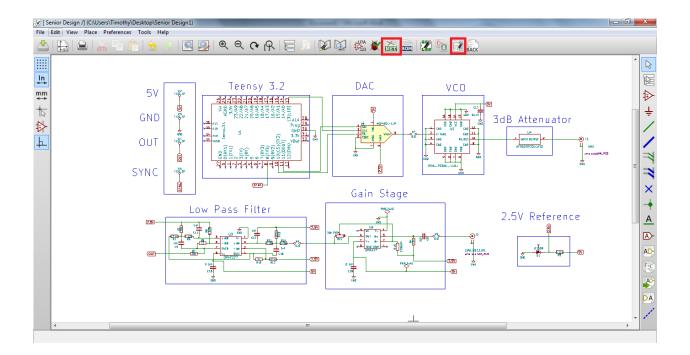
With the module wired up, we can then place a graphic enclosure using the 'Place graphic lines or polygon' option on the bottom right. Once done, a graphic text can be placed in the top right under the 'Place' tab. With this the module is complete and the rest of the modules can be created to complete the system.



Once the entire circuit schematic is complete, we will press 'Annotate schematic components' in the top to label our components. Next, we will 'Run CvPCB' to associate each component in our netlist with a footprint for PCB layout.

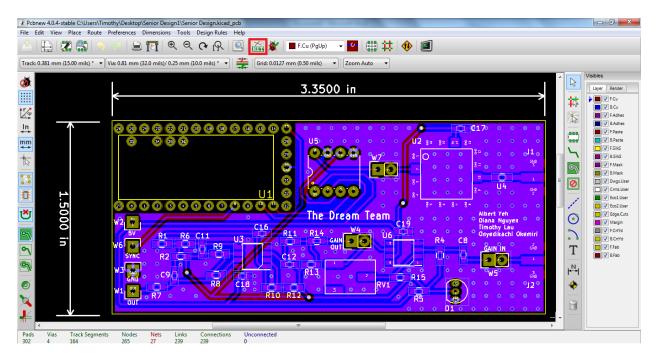
Cvpcb 4.0.4-stable Pro	ject: 'C:\Us	ers\Timothy\D	esktop\Senior Design1	1\Senior Design.pro'	X	()	Pn Footpri	nt: Housings_	SOIC:SOIC	8_3.9x4.9mm_Pi	tch1.27mm				- 0	23
File Preferences Help								Q Q (2.0							
📥 🕸 🔯	• •	∣ ≱⇒ <mark>≬</mark> ≱×≬						~ ~ (600						
10pinIC ^ 23BR10KLFTR	9 10	D1 - J1 -		t1009new:Lt1009new	-	^										
3361P-1-203GLF	10	J2 -	SMA RECEIVE : s				1 ^r									
Air Coils SML NE	12	R1 -	-	Resistors SMD:R 0603 HandSolderi			In									
Buttons Switches	13	R2 -		Resistors SMD:R 0603 HandSolderi			\leftrightarrow									
Buttons Switches =	14	R4 -		Resistors SMD:R 0603 HandSolderi			mm									
Buzzers Beepers	15	R5 -		Resistors SMD:R 0603 HandSolderi												
Capacitors SMD	16	R6 -		Resistors SMD:R 0603 HandSolderi			· 12									
Capacitors THT	17	R7 -		Resistors SMD:R 0603 HandSolderi	Ш.											
Capacitors Tanta	18	R8 -		Resistors SMD:R 0603 HandSolderi			Ø				OCC*	+				
Choke Axial Thro	19	R9 -		Resistors SMD:R 0603 HandSolderi	Ш.						- KEF"					
Choke Common-Moc	20	R10 -		Resistors SMD:R 0603 HandSolderin			$ \mathcal{X} $									
Choke Radial Thr	21	R11 -		Resistors SMD:R 0603 HandSolderi			<u></u>									
Choke SMD	22	R12 -		Resistors SMD:R 0603 HandSolderi	Ш.		ø									
Choke Toroid Thr	23	R13 -	102k : B	Resistors SMD:R 0603 HandSolderi	Ш.	m.					ÈRE-F*	<u>ن م</u>				
Connectors	24	R14 -	8.45k : F	Resistors SMD:R 0603 HandSolderin	Ш.											
Connectors_Harwi	25	R15 -	220ohm : R	Resistors_SMD:R_0603_HandSolderin	Ш.	Ε										
Connectors Hiros	26	RV1 -	20k POT : 3	3361P-1-203GLF:3361P-1-203GLF	Ш.											
Connectors_JAE	27	U1 -	Teensy_3.1 : m	nodules:Teensy-3.1	=			010	-8	3 92	4.9mr	n P	itch	1 2	7 m n	n
Connectors_JST	28	U2 - F	ROS-2536C-119+ : R	ROS-2536C-119+:ROS-2536C-119+					<u> </u>	<u></u>				-	/ 1110	
Connectors_Mini-	29	U3 -	OFA2227 : H	iousings_SOIC:SOIC-8_3.9x4.9mm_P												
Connectors_Mole>	30	U4 - 2	AT0603T03ECATD : P	Resistors_SMD:R_0603_HandSolderin	Ш.											
Connectors_Multi	31	U5 -	MCP4921-E/P : P	Power_Integrations:PDIP-8	Ш.											
Connectors_Phoer	32	U6 -	OPA227 : H	lousings_SOIC:SOIC-8_3.9x4.9mm_P	Ш.											
Connectors_Samte	33	W1 -	TEST_1P : P	Pin_Header1x1:Pin_Header_Straigh	Ш.											
Connectors_TE-Cc	34	W2 -	TEST_1P : P	Pin_Header1x1:Pin_Header_Straigh												
Connectors_Termi	35	W3 -	TEST_1P : F	Pin_Header1x1:Pin_Header_Straigh												
Connectors_WAGO	36	W4 -	TEST : F	Pin_Header_Straight_1x02:Pin_Head												
Converters_DCDC_	37	W5 -	TEST : F	Pin_Header_Straight_1x02:Pin_Head												
Crystals	38	W6 -		Pin_Header1x1:Pin_Header_Straigh												
Diodes_SMD _	39	W7 -	TEST : F	Pin_Header_Straight_1x02:Pin_Head	-	-	REE**			1.1.0			0.1	C 1.1		
< <u> </u>	•		III	•				9x4.9mm_Pit	ch1.27mm	Last Chang Mar 18, 201		Layer F.Cu	Pads 8	Status 	Angle 0.0	Attr Inse
omponents: 39, unassign	ed:0	Filter list:		Filtered by key words+pin count: 211			L Z 6.63	X 0.00000	0 Y 0.00196	9 dx 0	000000 dy 0.00196	9 dist 0.002)	Inches		

In CvPCB, we simply find the footprint in the footprint library and associate them with our circuit components. Referring back to our OPA2227 datasheet in the Low Noise Amplifier module, we find that the PCB footprint can either be DIP-8 or SOIC-8. In this instance, I have chosen the SOIC-8 package to associate my component with.

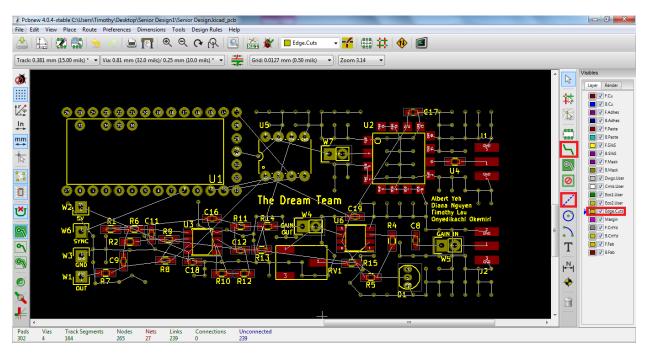


Once done with CvPCB, we return to our circuit schematic and choose 'Generate netlist' at the top and we can begin the PCB layout by clicking the 'Run PCBnew' option.

PCB Layout:



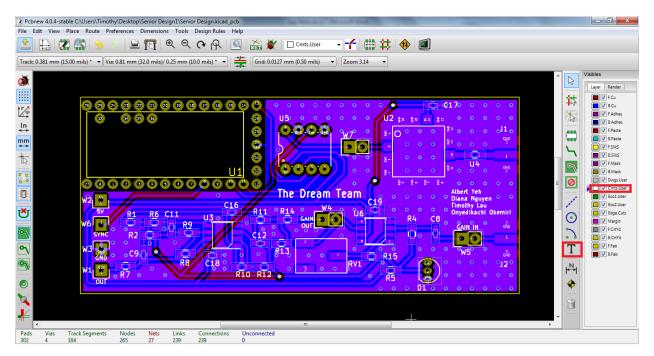
In PCBnew, we first begin by choosing the option 'Read netlist' at the top. This will drop all of the circuit components into one spot. After the circuit components are in one spot, they can be repositioned and rotated if necessary. For the PCB layout, it is best to reposition all the components into the general area of where they are in the circuit schematic. This will ensure that components go where they are supposed to and allow for easier debugging in the future.



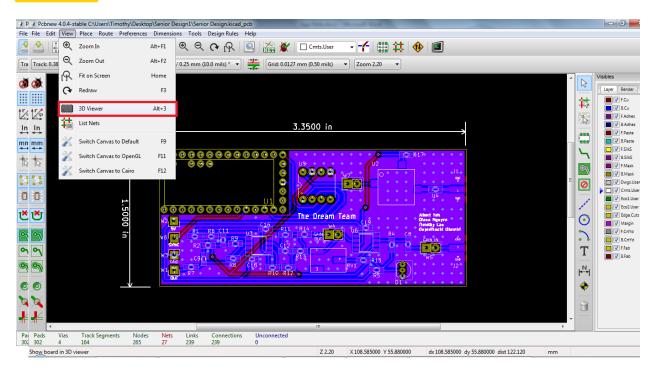
After the components are positioned, click on the Edge Cut layer and choose the 'Add graphic line or polygon' feature to add a physical edge to the PCB. With the physical edge of the board defined, we can then select 'Add tracks and vias' to connect our components. Be sure to have no 90 degree turns for traces but rather have then at 45 degrees instead to minimize loss.

Pcbnew 4.0.4-stable C/Users/Timothy/Desktop/Senior Design1/Senior Design.kicad_pcb	_ 0 <mark>_ x</mark> _
File Edit View Place Route Preferences Dimensions Tools Design Rules Help	
🖄 🔛 🗱 🥱 🔅 🖨 🛐 🍳 🔍 🍳 🥐 🧛 🔍 🔛 🎆 🕷 🗆 Conts.User 💿 🕇 🖽 🗱 🚸 🗐	
Track: 0.381 mm (15.00 mils) * 🔻 Via: 0.81 mm (32.0 mils)/ 0.25 mm (10.0 mils) * 🔹 🗱 Grid: 0.0127 mm (0.50 mils) 🔹 Zoom 3.14 🔹	
	Image: Second system Image: Second system
Pads Vias Track Segments Nodes Nets Links Connections Unconnected 302 4 164 265 27 239 56 183	

After the traces and vias have been placed, we can then begin to place our Via Holes. Click the 'Add Footprints' feature on the right and add drill holes connecting both the front and bottom of the copper ground planes. This is to connect the two ground planes and prevent any unwanted resonations in the circuit. Once the Via Holes are placed, the feature 'Add filled zones' can be used to flood the circuit and ground the areas between the circuit components.



With this, the overall layout of the PCB is mostly complete and graphic text can be added as necessary in the 'User Comments' layer on the right using the 'Add graphic text' feature. The PCB layout is complete and we can now begin our 3D model rendering.

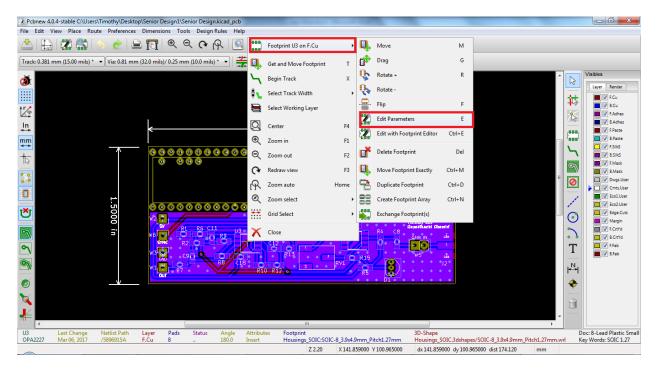


3D Model:

To do a 3D model rendering, simply go to the 'View' tab at the top and select the '3D Viewer' option.

ewer		
eferences		
📲 🎕 🔍 🔍 🗛 🕺 🖧 🕺 🖧 😤 🦑 ≉		
000000000000000000000000000000000000000		•••• ••••••
	O U5 · · · · · ·	
		J1.
U1	○ '○'○'○'○'	U4 .
	The Dreem Team	nuert Yeh
w2* 1	The Dream Team	* * Diana Nguyen * *
C <u>1</u> 6	W4	Timothy Lau Onyedikachi Okemiri
		4 C8
SYNC R2 📻 P		
W3		
GND C9 🚺	R13 - R15	W5' I
	RV1	J2*
	0 R12	
OUT	R5	D1

3D Rendering with no OPA2227



As in our example above, we will now add a 3D model to the OPA2227. To do so, we go back to our PCB layout and select the OPA2227 footprint and choose the 'Edit Parameters' tab.

🗶 Foot	print Properties	-	_		_	2	3		^ -	3						- 0 x
Prope	rties 3D settings						_ 1	田 井	ጭ∣[2						
	iape Name						3	D Shape:					X			
Hou	sings_SOIC.3dshapes/S	OIC-8_3.9x4.9m	m_Pitch1.27	mm.wrl				🔊 - 🗼 « kica	ad ≯ mo	odules 🕨 packages3d 🕨	- ↓ ₂	Search packages3d	R			
									v folder			· · · · ·		^	\mathbf{k}	Visibles
	ult Path (from KISYS3D) rogram Files\KiCad\sh			-3d				~							45	Layer Render
	-	are (kieda (inioae	nes (package	354				🔰 releases	^	Name		Date modified	Туре 🖍		韝	F.Cu
-3D S	Scale and Position									Divers.3dshapes		10/22/2016 11:04				B.Cu
Sha	pe Scale:						6	Libraries		Fuse_Holders_and_Fuses.		10/22/2016 11:04			1€	F.Adhes
X:	1.000000							🄹 Apps 📄 Documents		Hall-Effect_Transducers_I						E.Paste
Y:	1.000000							Documents Music		Housings_DFN_QFN.3dsF	napes	10/22/2016 11:04				B.Paste
	1.00000							Pictures		Housings_DIP.3dshapes		10/22/2016 11:04				Esilia
Z:	1.000000							Videos		Housings_QFP.3dshapes		10/22/2016 11:04				B.SIKS
Sha	pe Offset (inch):				_			In videos	=	Housings_ROHM.3dshap	es	10/22/2016 11:04			Cر ا	F.Mask
	0.000000				Add	3D Shape		Homegroup	11.0	Housings_SIP.3dshapes		10/22/2016 11:04		=		Dwgs.User
	0.00000				Remov	e 3D Shape		• Homegroup		Housings_SOIC.3dshapes		10/22/2016 11:04 10/22/2016 11:04			0	Cmts.User
Y:	0.000000						- III .	Computer		Housings_SSOP.3dshapes Inductors.3dshapes	5	10/22/2016 11:04			1	Ecol.User
Z:	0.000000				Edit	Filename		Local Disk (C:)		Inductors.3dshapes IR-DirectFETs.3dshapes		10/22/2016 11:04			1	Eco2.User
L.	0.000000											10/22/2016 11:04	File tol 🔻		\odot	Edge.Cuts
Sha	pe Rotation (degrees):						-			III			•		$\widetilde{}$	F.CrtYd
X:	0.000000								File nam	e:	-	Vrml and x3d files (*.wr	*.x3d) 🔻		• •	B.CrtYd
Y:	0.000000											Open 🔻	Cancel		Т	F.Fab
																Brao
Z:	0.000000							• • • b1 •	• • •			+			н	
															+	
				0	ĸ	Cancel						<u>Î</u>			V	
															Ĥ	
₩				6										-		
U3	Last Change	Netlist Path	Layer	Pads	Status	Angle	Attributes	Footprint			3D-Shape				Do	oc: 8-Lead Plastic Small
OPA222	27 Mar 06, 2017	/5896915A	F.Ću	8		180.0	Insert			3.9x4.9mm_Pitch1.27mm		OIC.3dshapes/SOIC-8_3.9		27mm.wrl	Ke	y Words: SOIC 1.27
								Z 2.2	20 >	(141.859000 Y 100.965000	dx 141.8590	000 dv 100.965000 dist 1	74.120 m	m		

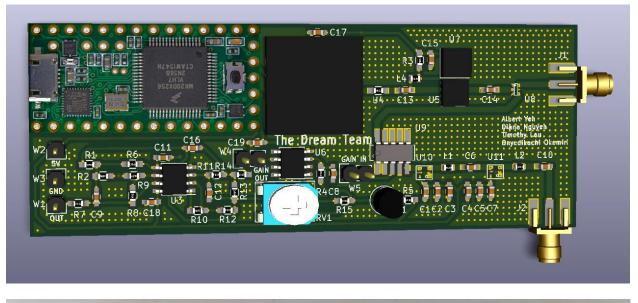
From this window, we select the '3D Settings' tab and choose 'Add 3D Shapes' below. This will open the KiCad 3D model library. If a model does not exist, they may be found online.

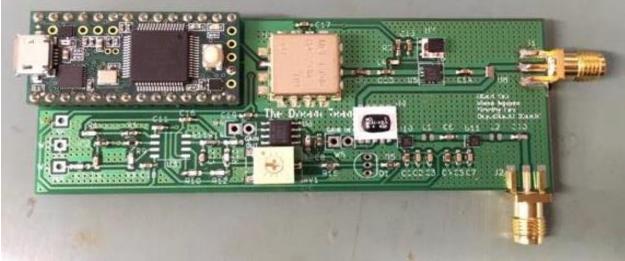
K Footprint Properties		
Properties 3D settings	1 🕂 🛗 🗱 (金) 🗐	
3D Shape Name	3D Shape:	×
Housings_SOIC.3dshapes/SOIC-8_3.9x4.9mm_Pitch1.27mm.wrl		
	Search	packages3d P
Default Path (from KISYS3DMOD environment variable)	Organize 🔻 New folder	BII - D Render
C:\Program Files\KiCad\share\kicad\modules\packages3d	🔋 releases 🔺 Name 🏠 Dat	
3D Scale and Position		22/2016 11:04 File fol
Shape Scale:		22/2016 11:04 File fol
X: 1.000000		22/2016 11:04 File fol
Y: 1.000000		22/2016 11:04 File fol
Y: 1.000000		22/2016 11:04 File fol = Silic
Z: 1.000000		22/2016 11:04 File fol
Shape Offset (inch):		22/2016 11:04 File fol
X: 0.000000		22/2010 11:04 File to
Remove 3D Shape	Todanigs_Sole.3danapes	22/2016 11:04 File fol
Y: 0.000000 Edit Filename		De contro da la contro de la co
Z: 0.000000	M 1 1011 (C)	22/2016 11-04 File fol
		Argin
Shape Rotation (degrees):		F.CrtYd
X: 0.000000	File name:	x3d files (*.wrl *.x3d) 🔻 🔹 🔪 🔲 🗸 B.CrtYd
Y: 0.000000	Open	Cancel T SFab
Z: 0.000000		
Z: 0.000000	<u> </u>	
OK Cancel		1 Îl
	III	
	tributes Footprint 3D-Shape	Doc: 8-Lead Plastic Small
OPA2227 Mar 06, 2017 /5896915A F.Cu 8 180.0 In		pes/SOIC-8_3.9x4.9mm_Pitch1.27mm.wrl Key Words: SOIC 1.27
	Z 2.20 X 141.859000 Y 100.965000 dx 141.859000 dy 100	.965000 dist 174.120 mm

After a model is imported, it can be positioned accordingly and scaled to fit the 3D rendering.

👔 3D Viewer	
File Preferences	
IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII	
W2 W2 W2 W2 W2 W2 W2 W2 W2 W2	
WZ C16 R11 R14 W4 U6 Timethy Line 5V R1 R6 C11 U3 R14 W4 U6 R4 CB W6 R1 R1 R14 W4 U6 R4 CB Charles Allow W6 R1 R1 R14 R4 CB Charles Allow Charles Allow W6 R2 R3 R3 R3 R3 R3 R3	
W6 TR1 R6 C11 U3 R1 R1 C C C C C C C C C C C C C C C C C	
SYNC R2 HE CL2	
GH0 R8 C18 R9 R1 RV1 R1 Y2 W1 R7 R10 R12 RV1 R5 R1 1	

OPA2227 with OPA2227





Afterwards, the rest of the 3D models can be placed onto the rendering to complete the design and be used as a guide for PCB assembly.