

1.	Proc	ess of ir	nporting Gerbers to Altium Designer	2				
	1.1	Importi	ng Gerbers	2				
		1.1.1.	Create new CAM Document in Altium Designer	2				
		1.1.2.	Import Gerber Files	2				
		1.1.3.	Select all the Gerber data in the right location, then click open	3				
		1.1.4.	Change Gerber Import settings, and/or press OK	3				
		1.1.5.	Warnings and Errors should be 0, then close this log file	4				
		1.1.6.	Gerber Files are successfully imported	4				
	1.2	Import	Drill File	5				
		1.2.1.	Select the Drill importer	5				
		1.2.2.	Select the right folder with the Drill that belongs to the Gerbers	5				
		1.2.3.	Select the Drill File, then press OK	6				
		1.2.4.	Change Units if necessary, otherwise press OK	7				
		1.2.5.	Errors and Warnings should be 0, then close the log file	7				
2.	Adju	sting the	e CAM File	8				
	2.1	Setting	the Layers	8				
		2.1.1.	Set the correct layers then press OK	8				
		2.1.2.	Set the correct layers order, then press OK	9				
	2.2	Setting	the layer sets	9				
		2.2.1.	Add Drill and Plane Layers in set, then press OK	9				
	2.3	Extract	the Netlist	10				
	2.4	Export	to PCB	11				
3.	Adjusting the PCB							
	3.1	Deletin	g the excess pads that were imported by default	12				
		3.1.1.	Keep only one layer on visible (Top Layer is turned off in the example)	12				
		3.1.2.	In Properties menu only select the Pad as selection filter	12				
		3.1.3.	Select the entire imported sensor	13				
		3.1.4.	Delete the selected pads	14				
		3.1.5.	Turn the TOP layer back on	14				
		3.1.6.	For some strange reason there are always 2 pads	14				
		3.1.7.	Select one smaller pad, then right click and select Find Similar Objects	15				
		3.1.8.	Set the X Size and Y Size	15				
		3.1.9.	Now all 0.2mm pads are selected, press delete and eliminate them	15				
	3.2	Creatin	g vias and adjusting them	16				
		3.2.1.	Select all the pads with the same size	16				
		3.2.2.	Set Multi-Layer instead of TOP Layer	16				
		3.2.3.	Convert Pads to Vias	17				
		3.2.4.	Select Vias in the Selection Filter under Properties	17				
		3.2.5.	Select Vias	18				
		3.2.6.	Adjust Via parameters under Properties:	18				
	3.3	Adjust	the rest:	19				
	3.4	Don't fo	orget to Save the PCB!	19				
4.	Revi	sion His	tory	20				



1. Process of importing Gerbers to Altium Designer

1.1 Importing Gerbers

1.1.1. Create new CAM Document in Altium Designer



1.1.2. Import Gerber Files





1.1.3.	Select all the Gerber data in the right location, then click open

→ 🔹 ↑ 📜 « (C:) Windows > Simulations > _ICOT_support	Simulation_Gerbers	✓ U Search Simula	ation_Gerbers
rganize 🔹 New folde	er			©== ▼
Ouick access	Name	Date modified	Туре	Size
	🕀 output30a_outline.gbr	11.06.2024 16:24	CAMtastic Layer Gerb	1 KB
Desktop	🕃 output30a1.gbr	11.06.2024 16:24	CAMtastic Layer Gerb	40 KB
Downloads * Documents *	output30a2.gbr	11.06.2024 16:24	CAMtastic Layer Gerb	69 KB
File pa	me: "output30a2.gbr" "output30a.outline.gb	pr" "output30a1.gbr"	 Gerber Files (*.A*:*.G*:*.PHO)

1.1.4. Change Gerber Import settings, and/or press OK

	Gerber Import S	Settings ×
	Digits	Units
	Integer 4 🔻	<u>E</u> nglish
	Decimal 4 🔻	<u>M</u> ethe
	Туре	Zero Suppression
	<u>A</u> bsolute	<u>L</u> eading
	Incremental	<u>I</u> railing <u>N</u> one
Import Gerber(s) - Opt		0-
Gerber Options:		OK Cancel
Start Units: 4.4 Leading Abs mm RS-274 Load <u>O</u> ptions	<u>s</u>	ettings
✓ Auto Detect Gerber Formats		
Aperture List (required for Gerber RS-274D only)		
<u>A</u> perture File		
Aperture Wizard Rule Auto-Detect Apertures		
	ОК	Cancel



1.1.5. Warnings and Errors should be 0, then close this log file

\bigcirc	CAMtastic1.Cam ³	*	Log	_2024	_7_9	9_14_46_52.log
1 • •	CAMtastic DX CAMtastic Fi Date Time	KP Lle	: Ge : CA : 9/ : 2:	27/20 46:5	: / sti)24 52	NC Drill / Mill/Rout Import Process Report c1.Cam PM
- •	Information		Layer	1		output30a_outline.gbr - start loading.
	Information Information Information		Layer Layer Layer	1 2 2	-	output30a_outfine.gbr - end foading. output30al.gbr - start loading. output30al.gbr - end loading.
10 • •	Information Information		Layer Layer	3 3		output30a2.gbr - start loading. output30a2.gbr - end loading.
	Statistics					
- •	Errors		0			
•	Information		6			

1.1.6. Gerber Files are successfully imported



1.2 Import Drill File

1.2.1. Select the Drill importer



1.2.2. Select the right folder with the Drill that belongs to the Gerbers

· · · · · · · · AM	File Import - NC Drill	×		
Battery	Select Files to Load		_	
BJT NPN 4 MGI	Select Import Directory PPORT\R_341_IPS2550_40x9_0D229_ID209\20_SIM\03_4L_fullop File(a) in Directory	ot\ 🖻	Ju	ul.9.24
Select Directory	File(s) in Directory			×
\leftarrow \rightarrow \checkmark \uparrow \blacksquare « (C:) Windows \Rightarrow S	imulations > _ICOT_support > Simulation_Gerbers v	Search Simul	ation_Gerbers	Q
Organize 🔹 New folder				?
Temp 🖈 ^ Name	^ Date modified	Туре		Size
90_INPUT	No items match your search.			
240528_GKNPM_				
websim 🗸 <				>
Folder: Simulation_G	erbers			
		Select Folder	Cano	el



File Import - NC Drill	×
Select Files to Load	
Select Import Directory	
C:\Simulations_ICOT_support\Simulation_Gerbers\	e),
File(s) in Directory	
output30a.drl	
Drill Files (*.DR*;*.ROU;*.RTE;*.NC*;*.TX*)	
Loading Options	
Detect Aperture Formats (RS-274-D)	
Start Units: 4.4 Leading Abs mm	
Gerber Options Default Units	

1.2.3. Select the Drill File, then press OK



IP 4 Pins T	NC Drill Import	Settings ×
Import Drill Data 🛛 🗙	Digits	Units
Settings Start Units: 4.4 Leading Abs mm	Integer 4 💌 Decimal 4 💌	<u>E</u> nglish <u>M</u> etric
Shape/Default Hole Size	Туре	Zero Suppression
0,0320:0,0320 <u>T</u> ool Table	 <u>A</u>bsolute <u>I</u>ncremental 	 Leading Irailing None
OK Cancel		OK Cancel

1.2.4. Change Units if necessary, otherwise press OK

1.2.5. Errors and Warnings should be 0, then close the log file

\oplus	CAMtastic1.Cam	*	Log_2024_7_9_14_53_0.log
	CAMtastic DX CAMtastic Fi Date Time	(P ile	: NC Drill Import Process Report : CAMtastic1.Cam : 9/7/2024 : 2:53:0 PM
	Information Error: Line Information	: La no. : La	ayer 4 – output30a.drl – start loading. 64 – – Invalid tool number definition: TOC ayer 4 – output30a.drl – end loading.
0	Statistics		
	Errors		1
	Warnings		0
	Information		2

2. Adjusting the CAM File

2.1 Setting the Layers



2.1.1. Set the correct layers then press OK

		Layers Table		×
# 1 2 3 4	Layer Name output30a_outline.gbr output30a1.gbr output30a2.gbr output30a.drl	Layers TableTypeBorderTopbottomDrill TopTopBottomInternalNeg PlanePos PlaneDrill TopDrill TopDrill Bot	Draw Flash On Lock I	Delete
View I	Layer <u>O</u> rder <u>A</u> pply	<u>C</u> ompress	ОК Са	ancel



		•	
	Creat	e / Update Layers Order	×
#	LayerName	Layer Logical Order	Layer Physical O 🔺
2	L2: output30a1	2	1
1	L3: output30a2	3	2 🔨
			1
			2
			OK Cancel

2.1.2. Set the correct layers order, then press OK

2.2 Setting the layer sets



2.2.1. Add Drill and Plane Layers in set, then press OK

			Create / Update L	ayers Sets		×					
Set #	Layers S	Set Name	Assigr	ned Drill Layer	Signal / Plane Laye	ers In Set					
1		l	L4: output30a		2: output30a1,L3: c	output30a2	S	elect Lave	r Pairs		×
							L2: output L3: output	30a1 30a2			
Insert L	ayers Set	<u>D</u> elete Layers Set	. <u>S</u> elect Layer Pa		ОК	Cancel	<u>C</u> lear All		ОК	Canc	el





2.3 Extract the Netlist



2.4 Export to PCB





3. Adjusting the PCB

- 3.1 Deleting the excess pads that were imported by default
- 3.1.1. Keep only one layer on visible (Top Layer is turned off in the example)



3.1.2. In Properties menu only select the Pad as selection filter

Properties			▼ ₽ ×
Board	Comp	oonents (and 12 mc	ore)
Q Search			
General	Parameters	Health Check 🗸	
▲ Selection	Filter		Î
Custom			
Compone	ents 3D Bo	odies Keepouts	
Tracks	Arcs Pade	s Vias Regio	ins
Polygons	Fills	exts Rooms	Other



3.1.3. Select the entire imported sensor elect Second Corner







3.1.4. Delete the selected pads





3.1.6. For some strange reason there are always 2 pads

imported on each layer. A correct sized, like in our case 0.6mm diameter and a smaller one with 0.2mm diameter. The latter needs to be deleted.





3.1.7. Select one smaller pad, then right click and select Find Similar Objects



3.1.8. Set the X Size and Y Size

(or just one of them) to Same. Usually the rest of the settings should be unchanged, then press OK

Shape (All Layers)	Round		Any	
X Size (All Layers)	0.2mm	N	Same	-
Y Size (All Layers)	0.2mm	•	Same	
Drill Type	Drilled		Anv	

3.1.9. Now all 0.2mm pads are selected, press delete and eliminate them





3.2 Creating vias and adjusting them

3.2.1. Select all the pads with the same size (exclude the header pads on the right)



3.2.2. Set Multi-Layer instead of TOP Layer

Properties				▼ ₽	×
Pad	Co	mp	ponents (and 12 more)	T	•
Q Search	า				
D	esignator	1			
	Layer		Top Layer	•	
	Net		Top Layer		
Flect	rical Type		Bottom Layer		
			Multi-Layer		
Propagatio	ion Delay		Top Paste		
Pin Packad	ge Length		Top Ovorlav		





3.2.3. Convert Pads to Vias

3.2.4. Select Vias in the Selection Filter under Properties

Properties			▼₽×
Board	Comp	onents (and 12 mc	ore) 🔽 🗸
Q Search			
General F	Parameters	Health Check \checkmark	
▲ Selection Fi	lter		
Custom			
Componen	ts 3D Bo	dies Keepouts	
Tracks	Arcs Pads	5 Vias Regio	ons
Polygons	Fills Te	exts Rooms	Other

3.2.5. Select Vias



3.2.6. Adjust Via parameters under Properties:

Diameter should be correct as simulated, Hole Size should be adjusted to match PCB manufacturing requirements, also you can have tented or capped vias, it doesn't matter in terms of inductive sensing performance



Gerber Import to Altium



3.3 Adjust the rest:

Net Names, adjust the output headers, connect the sensing element to the IPS chip, finalize the layout, etc.

3.4 Don't forget to Save the PCB! 🙂



4. Revision History

Revision	Date	Description
1.0	Jul. 22, 24	Initial release.

