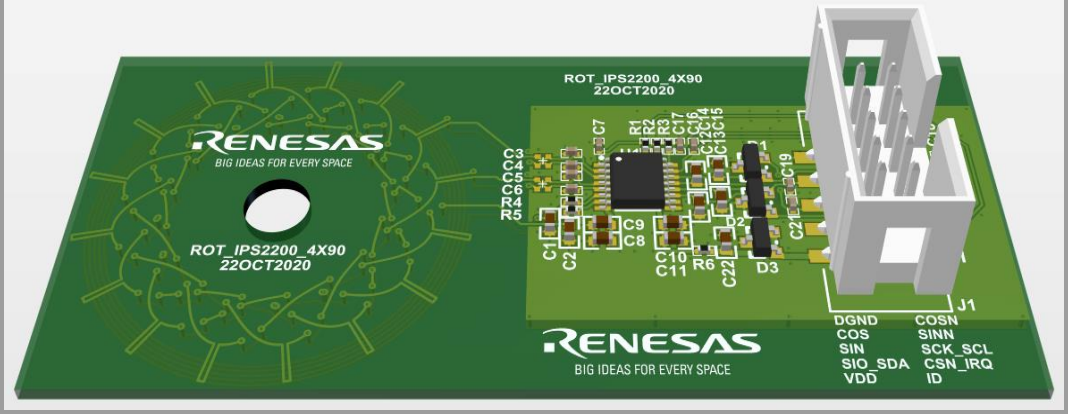
INDUCTIVE POSITION SENSOR (IPS) DESIGN GUIDELINES

2021-03-08

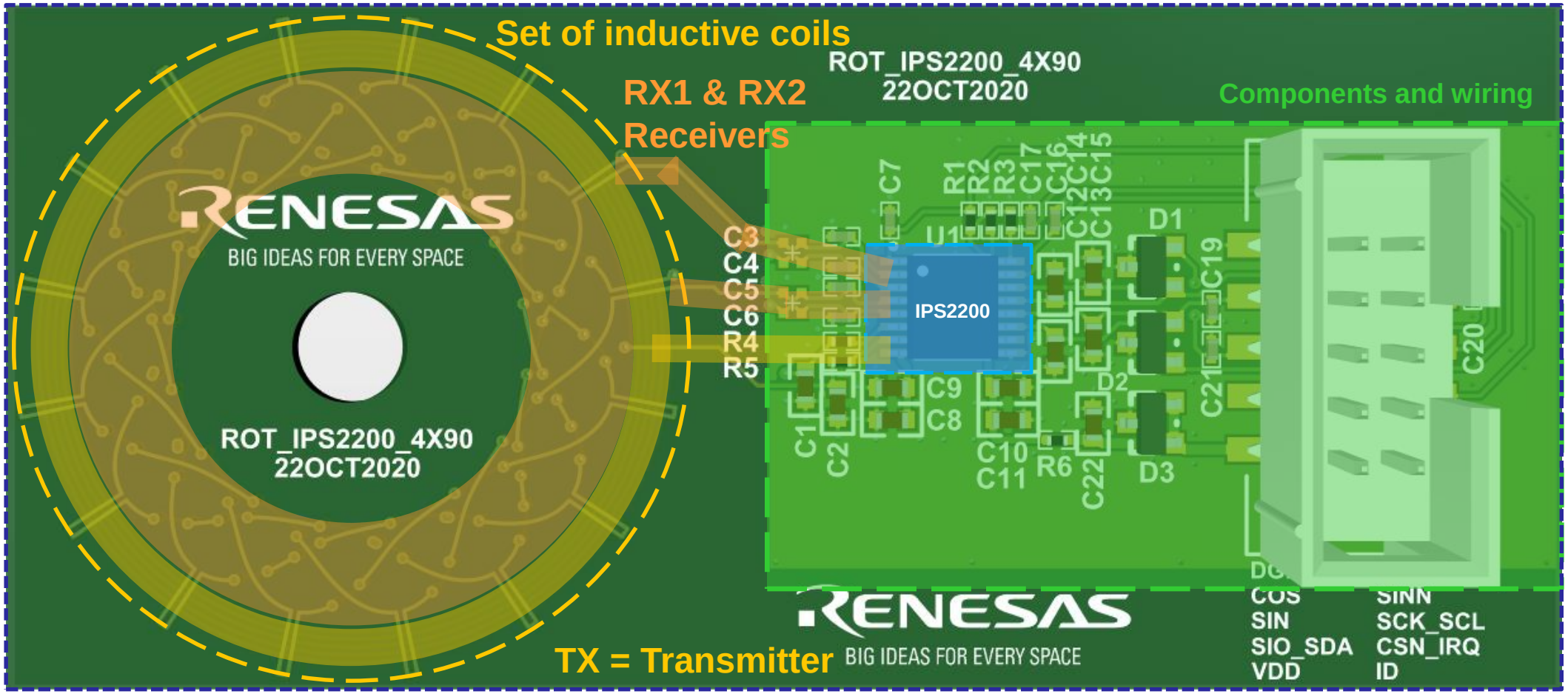
RENESAS ELECTRONICS CORPORATION



CONTENTS

WHAT IS AN IPS DESIGN?

PCB = Printed Circuit Board



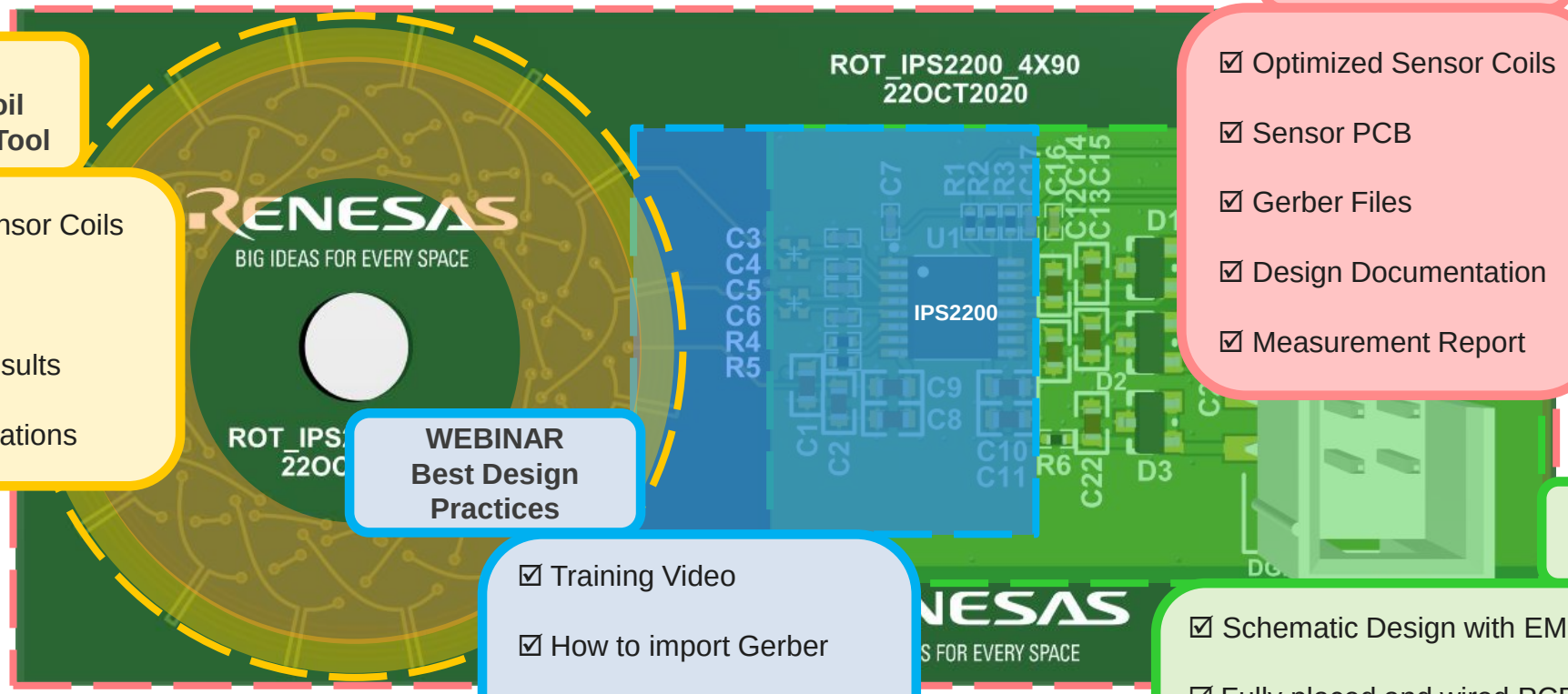
TOOLS FOR DESIGN GENERATION

ICOT
Inductive Coil Optimization Tool

- ☑ Optimized Sensor Coils
- ☑ Gerber Files
- ☑ Simulation Results
- ☑ 3 Air Gap Variations

CRB
Catalog Design

- ☑ Optimized Sensor Coils
- ☑ Sensor PCB
- ☑ Gerber Files
- ☑ Design Documentation
- ☑ Measurement Report



WEBINAR
Best Design Practices

- ☑ Training Video
- ☑ How to import Gerber
- ☑ How to connect Coils
- ☑ Design Hints
- ☑ Good and Bad Examples

PCB Design
TEMPLATES

- ☑ Schematic Design with EMC Components
- ☑ Fully placed and wired PCB Design
- ☑ Gerber Files
- ☑ Full Altium Project

https://www.renesas.com/buy-sample/locations

https://www.renesas.com/products/sensor-products/position-sensors

HOW TO IMPORT GERBER?

☑ Gerber Files of Optimized Sensor Coils

pcb_layout_advanced.zip
227 KB

Save and unpack ZIP container **1**

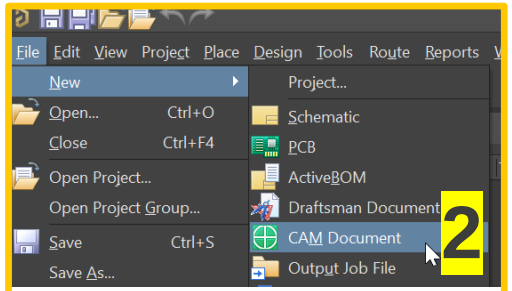
Dear User,
Your coil design optimization results are attached in the email.
Please send an email to „Teamsupport@coildesign“ if any questions.

Send the legal disclaimer as a PDF to the customer upfront.
The Legal Disclaimer can be found on Seismic as “Renesas Inductive Position Sen
Do not deliver the results and the Gerber files without the customer acknowledgir
Attach the disclaimer again when the results and the Gerber files are sent.

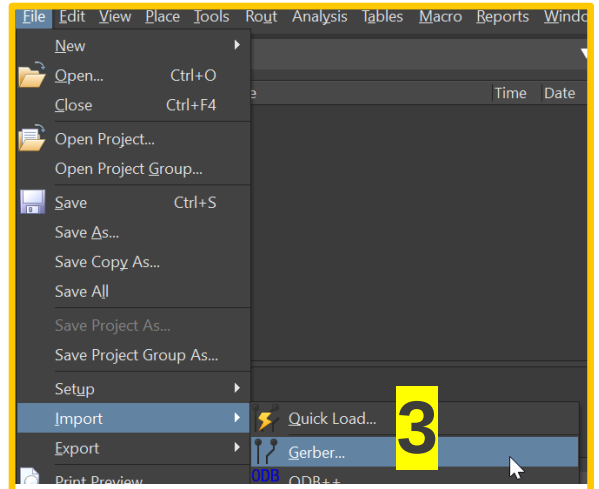
Note
The attached ZIP file is encrypted with you user password.

Regards,
AASBD Apps support team

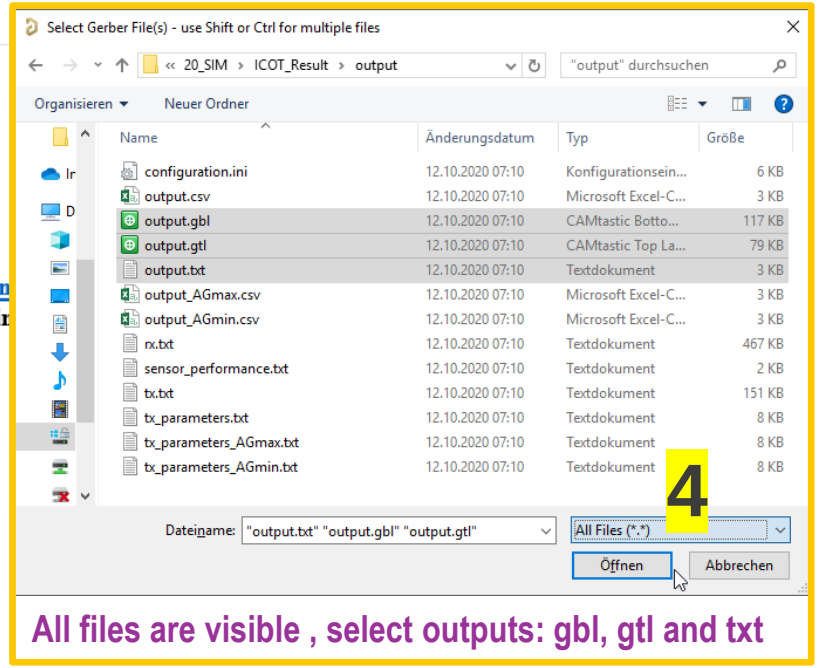
NB: This is a generated email please do not reply.



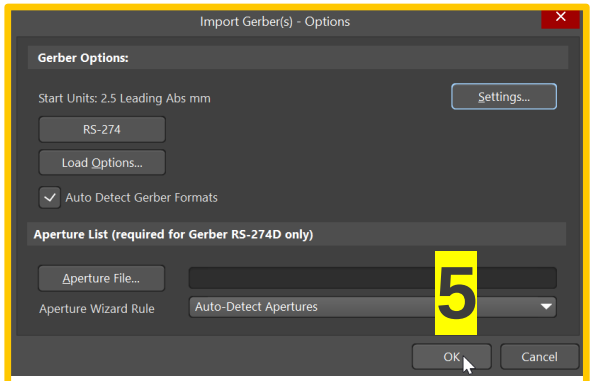
Create new CAM Document



Import Gerber…

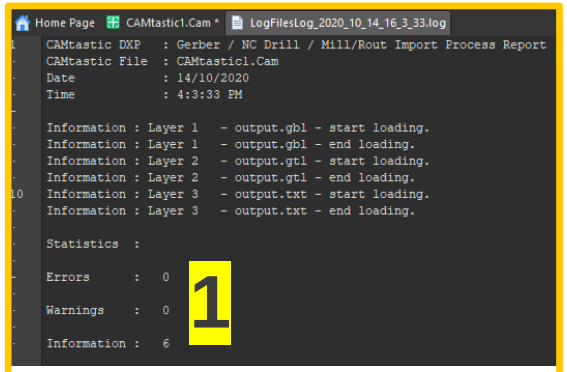


All files are visible , select outputs: gbl, gtl and txt

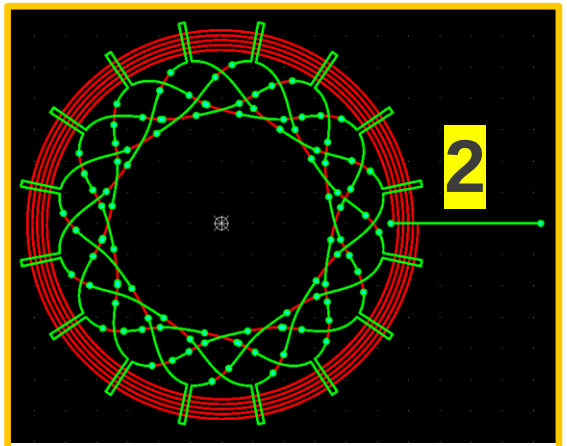


Just hit OK, Gerbers will be imported

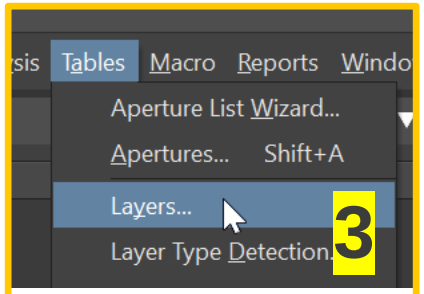
HOW TO SET THE LAYERS?



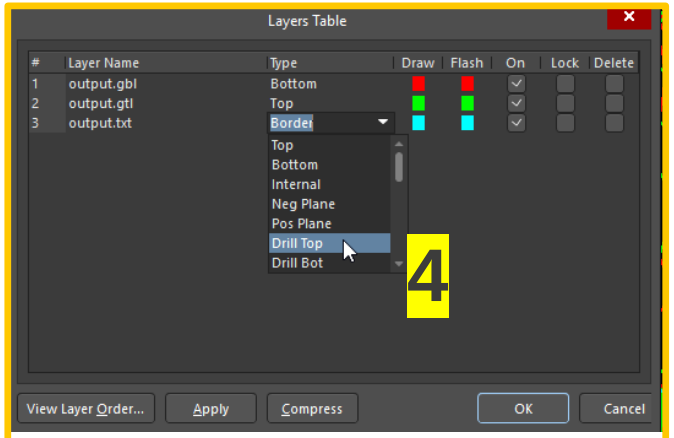
0 Errors and 0 Warnings → good



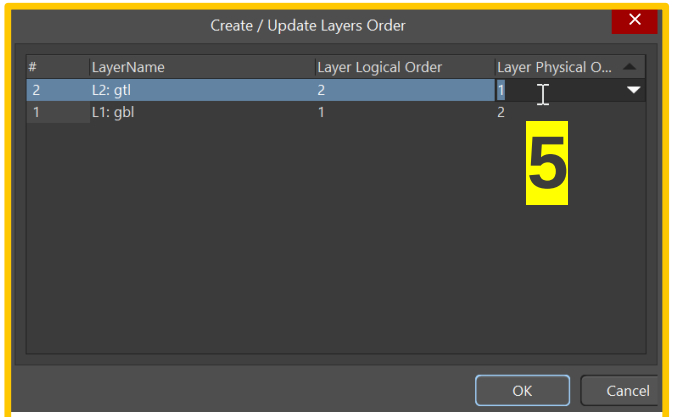
Back to CAMtastic, coils imported



Click Tables, Layers…

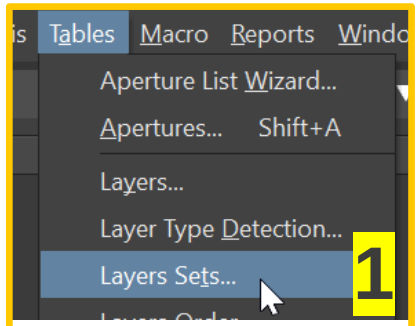


Set the TXT file for drilling layer

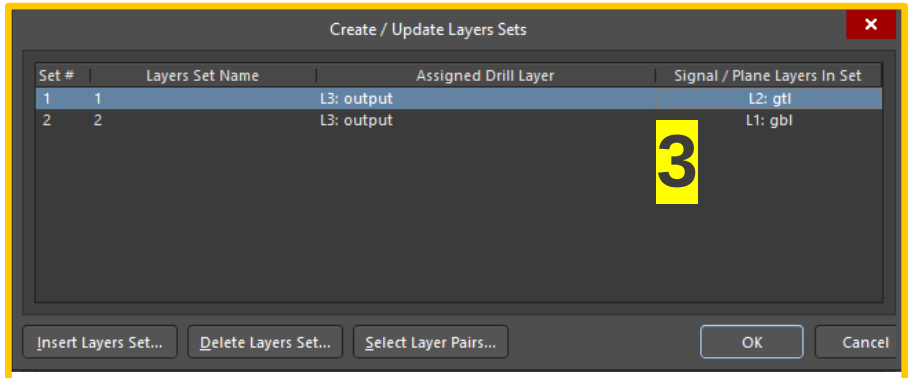


Set physical Layers: GTL→1, GBL →2

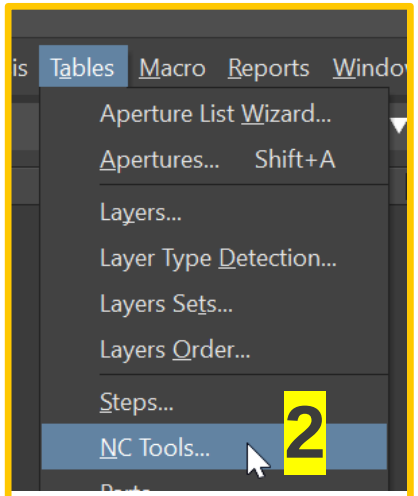
HOW TO SET THE VIAS?



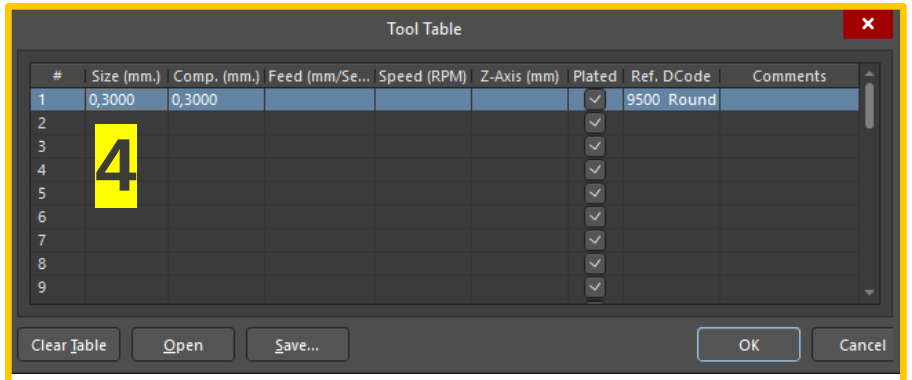
Click Tables, Layers Sets



Insert Layer Set, name them, assign Drill and Signal Layers

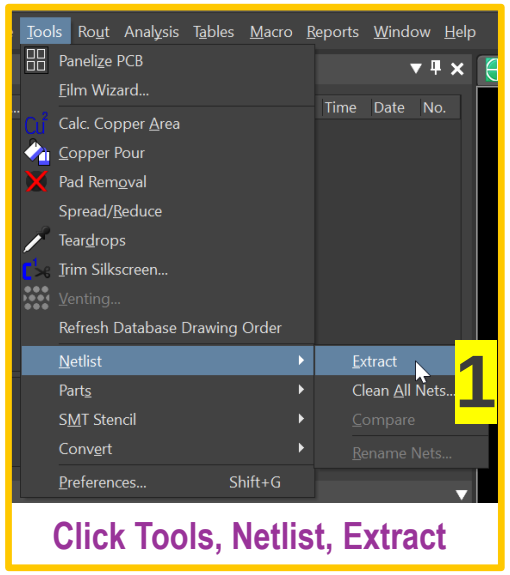


Click Tables, NC Tools

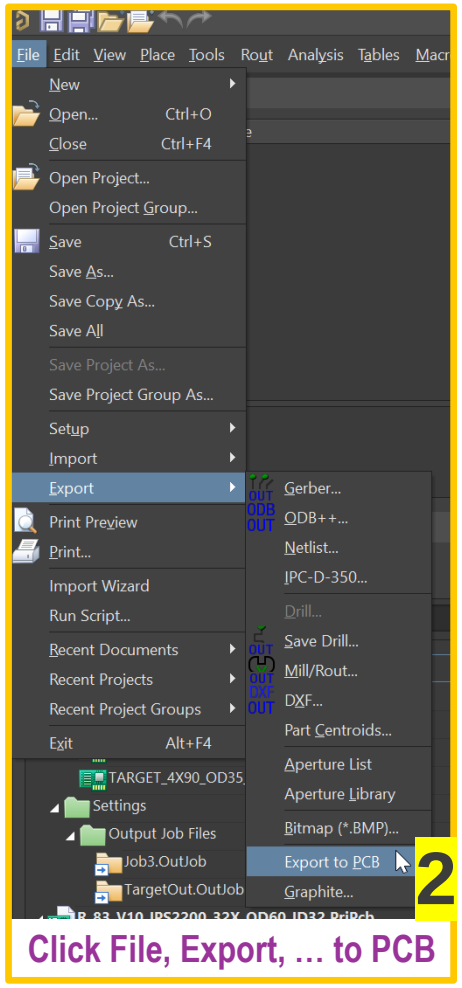


Set Via Drill diameter as simulated with ICOT

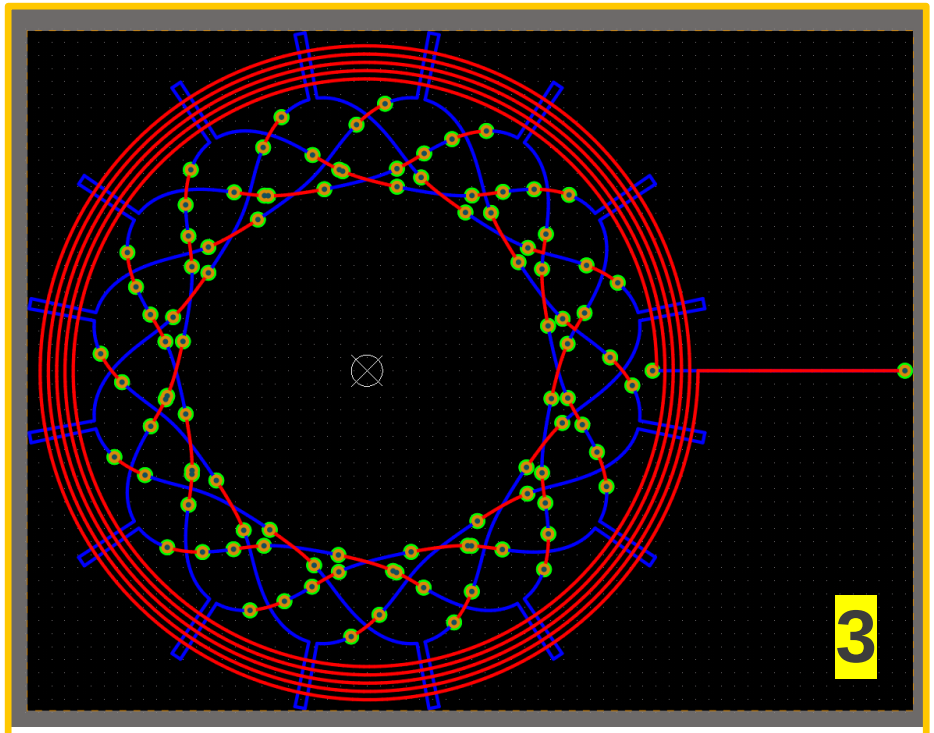
HOW TO EXPORT TO PCB?



Click Tools, Netlist, Extract

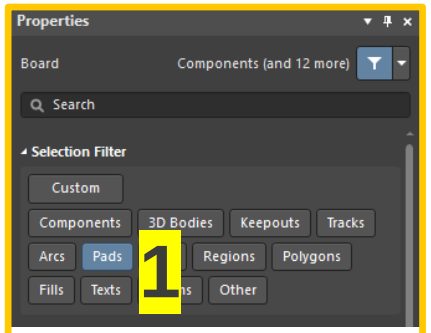


Click File, Export, … to PCB

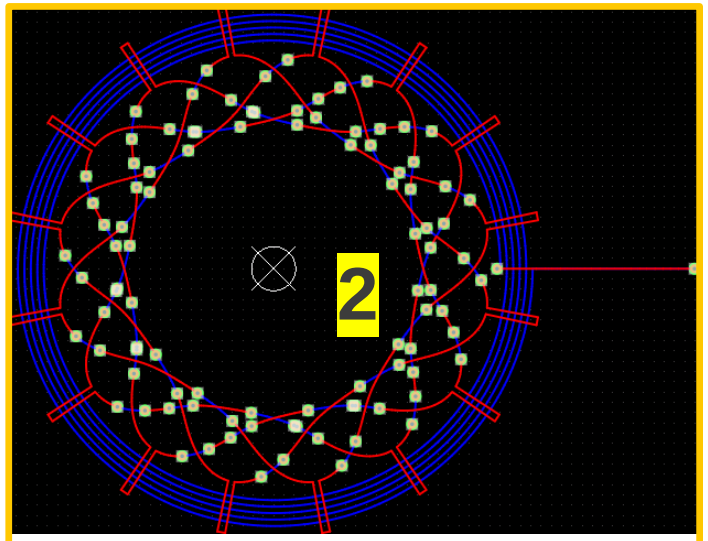


Sensor coils are now ready to use in Altium Designer as PCB

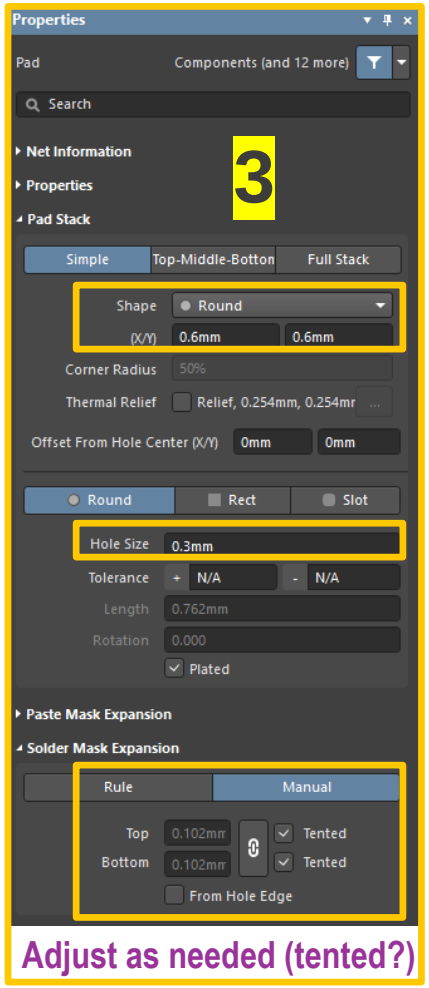
DESIGN HINT FOR ADJUSTING VIAS



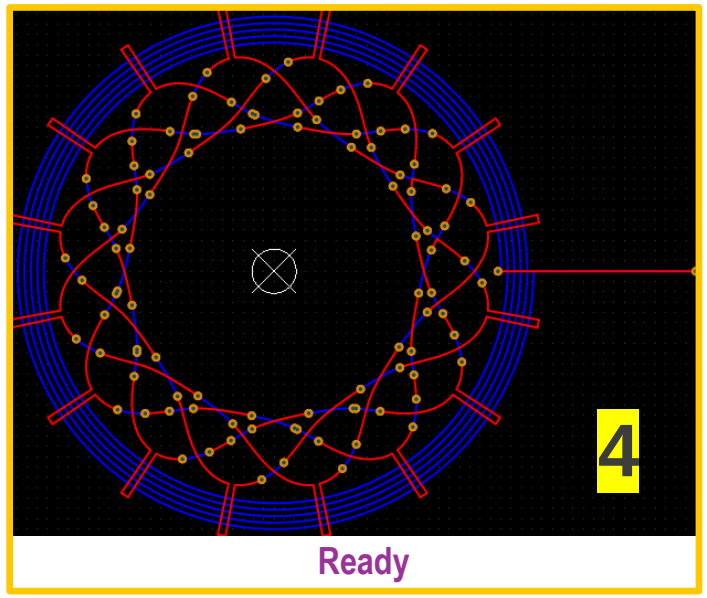
Select only Pads



Select all the Pads/Padstacks

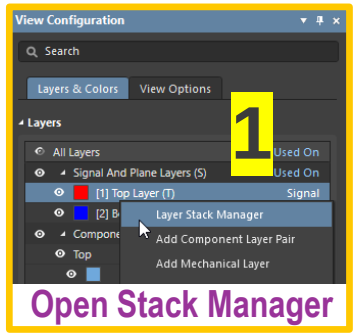


Adjust as needed (tented?)



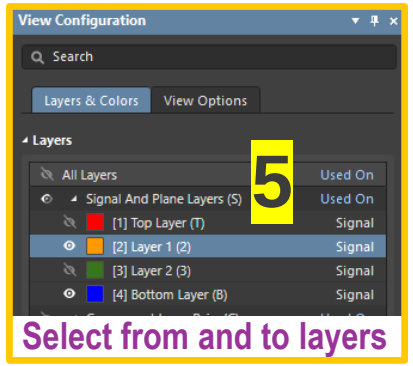
Ready

DESIGN HINT FOR SHIFTING LAYERS

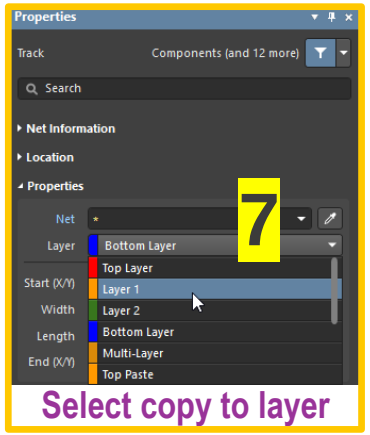


Open Stack Manager

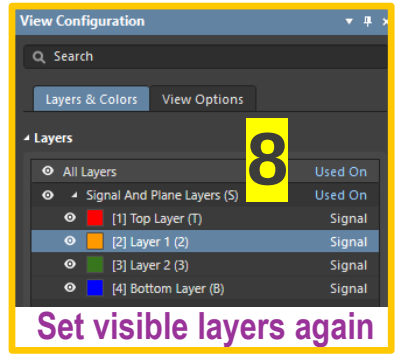
Select Tracks and Arcs



Select from and to layers

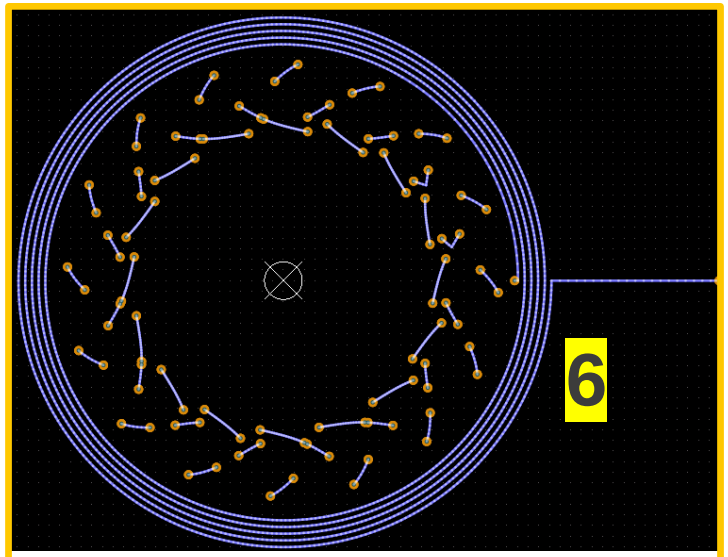


Select copy to layer

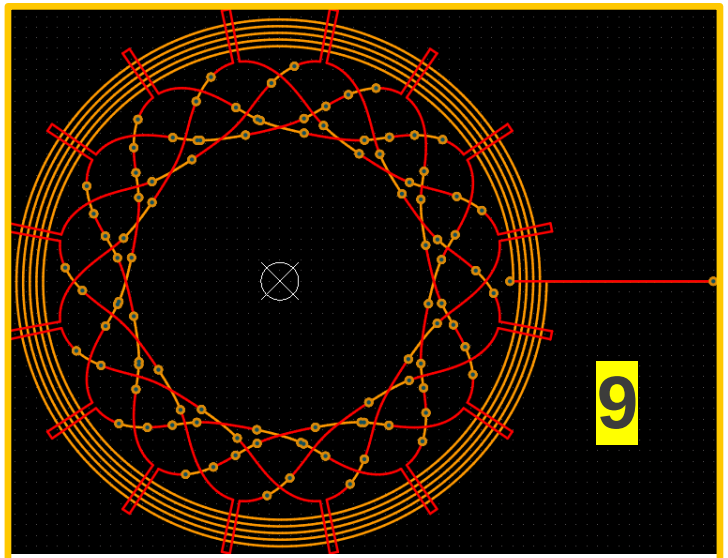


Set visible layers again

Insert Internal Layers



From layer is visible,select everything to copy



Coils have been copied from BOT to Layer2

#	Name	Material	Type	Weight	Thickness
	Top Overlay		Overlay		
	Top Solder	Solder Resist	Solder Mask		0.01016mm
1	Top Layer		Signal	1oz	0.03556mm
	Dielectric 2	PP-006	Prepreg		0.11mm
2	Layer 1	CF-004	Signal	1oz	0.035mm
	Dielectric 1	FR-4	Dielectric		1.11mm
3	Layer 2	CF-004	Signal	1oz	0.035mm
	Dielectric 3	PP-006	Prepreg		0.11mm
4	Bottom Layer		Signal	1oz	0.03556mm
	Bottom Solder	Solder Resist	Solder Mask		0.01016mm
	Bottom Overlay		Overlay		

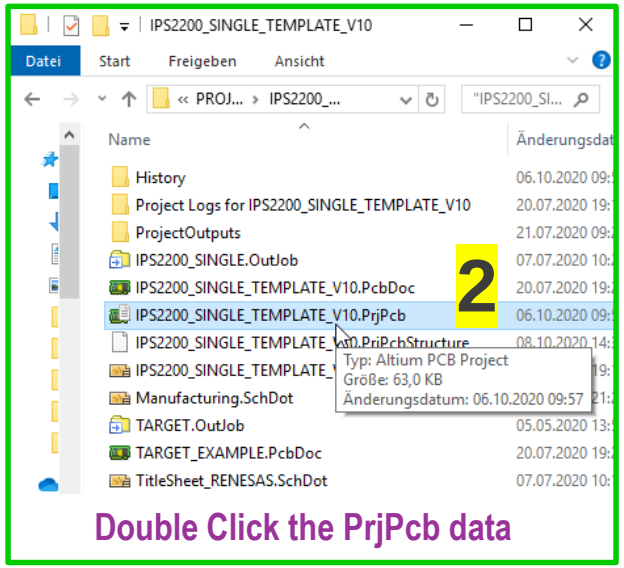
3

4 Layers are ready, Save and Exit Stack

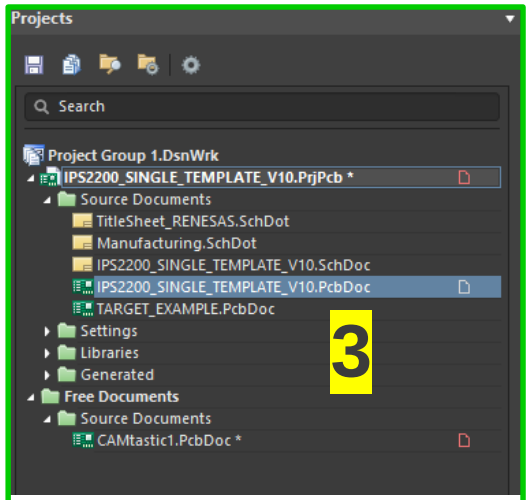
HOW TO OPEN THE TEMPLATE?

☑ Download Templates – Altium Project Files

RENESAS
BIG IDEAS FOR EVERY SPACE
WEBPAGE
Download
1



Double Click the PrjPcb data



Double Click the PcbDoc data

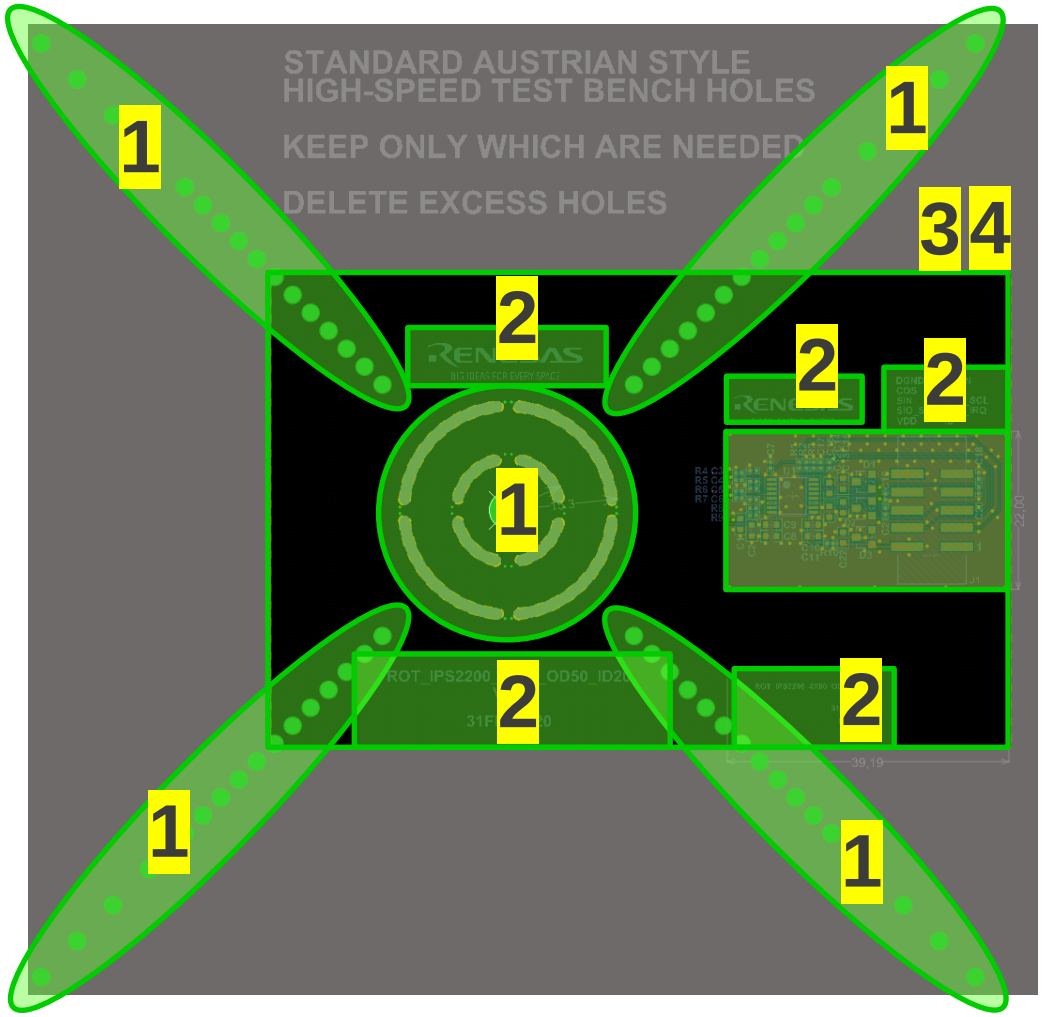
PCB Layout is ready for use

HOW TO ADJUST THE TEMPLATE?

☑ Remove unused Mounting Holes

☑ Or add/adjust as needed **1**

☑ Remove/adjust Text, Logo, etc. **2**

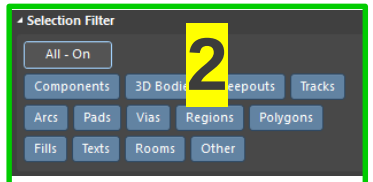


☑ Adjust Board Shape freely **3**

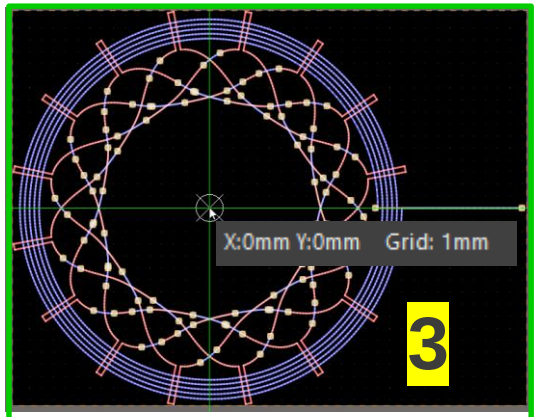
☑ Adjust Layer Stack **4**

HOW TO COPY/PASTE THE COILS?

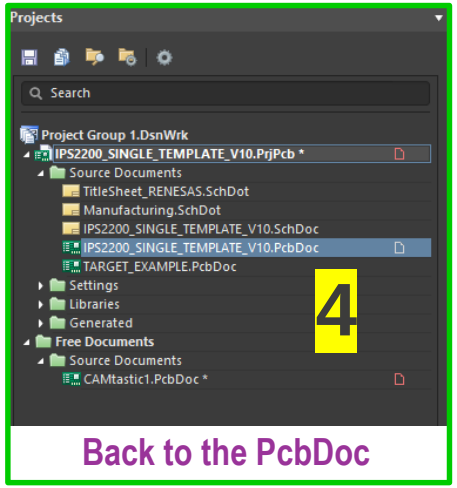
Back to the imported coils



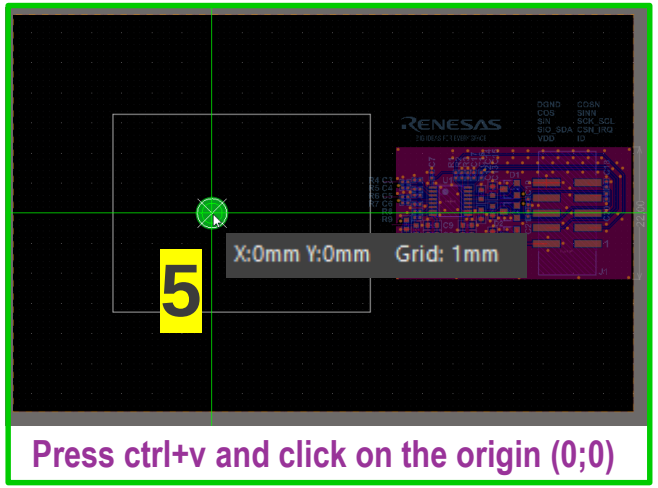
Filters all ON



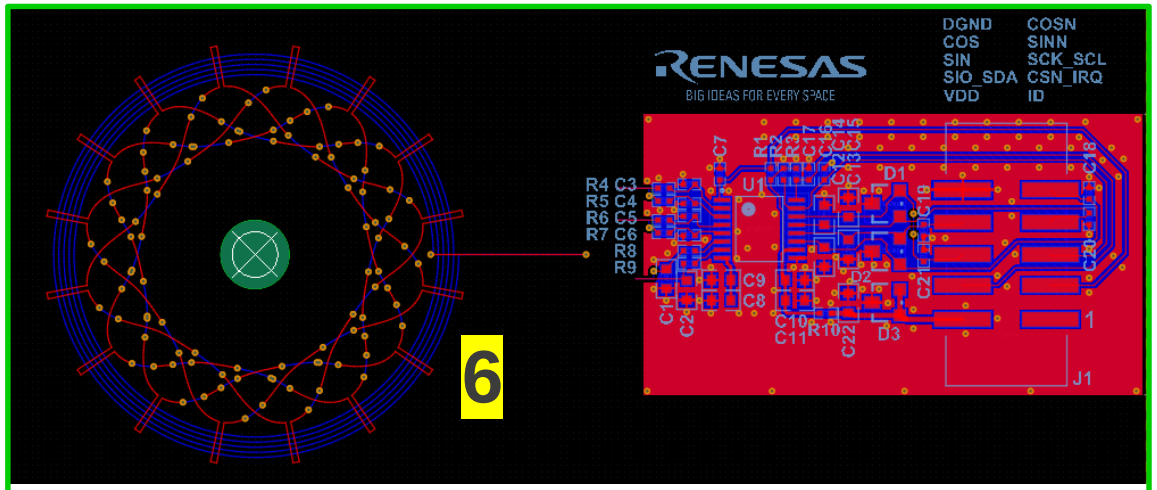
Select everything, press ctrl+c and click on the origin (0;0)



Back to the PcbDoc

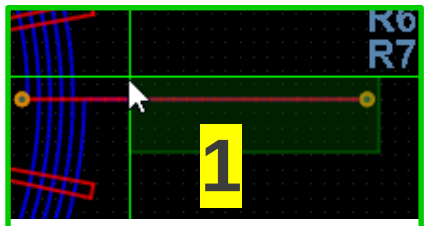


Press ctrl+v and click on the origin (0;0)

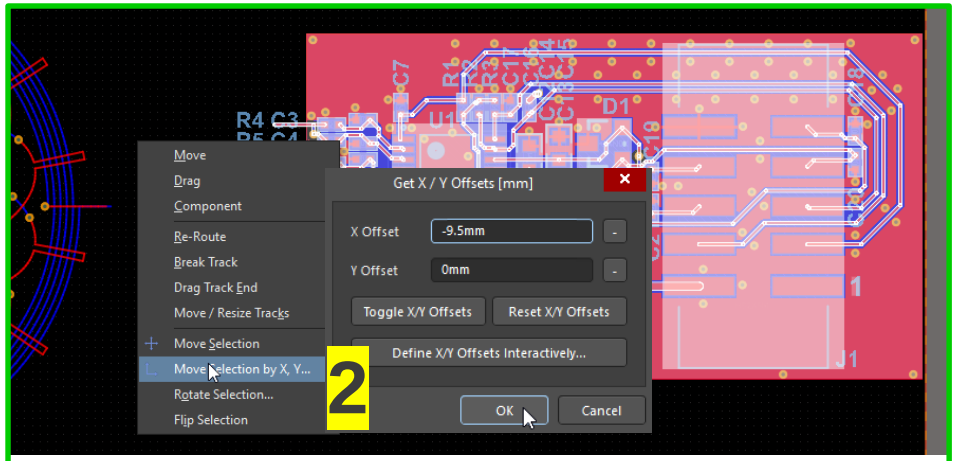


Sensor Coils are now copied to the PCB Layout Template

DESIGN HINT FOR COMPACTING THE BOARD



Delete long unused Traces



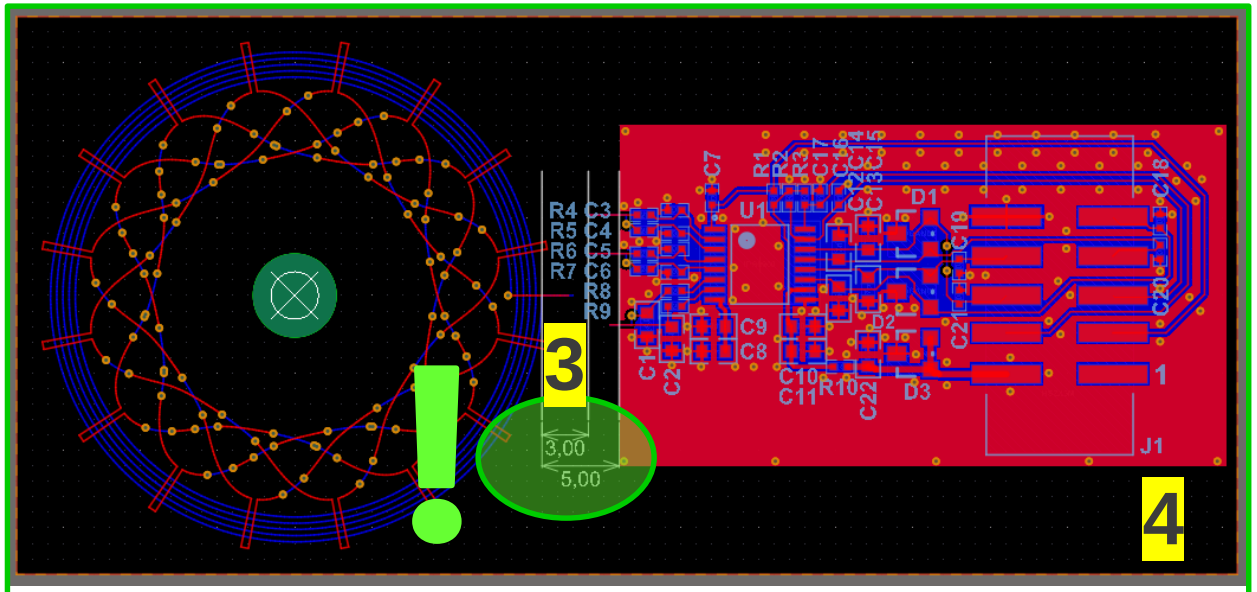
Select everything and move them closer to the Coils

To eliminate the unwanted influence of large metallic objects:

☑ Keep **3mm** minimum distance from copper planes to the coils!

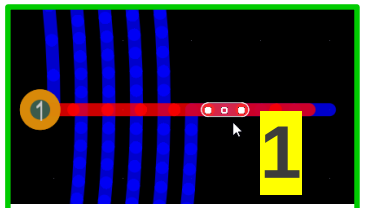
☑ Keep **5mm** or larger safety distance if your design lets you!

3

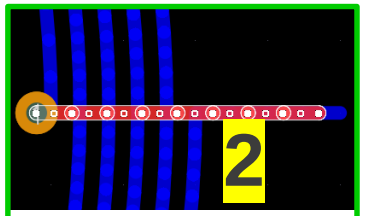


Coils and the Template Layout are now close to each other

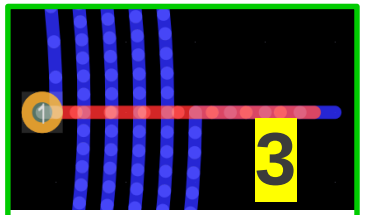
HOW TO RENAME THE COILS?



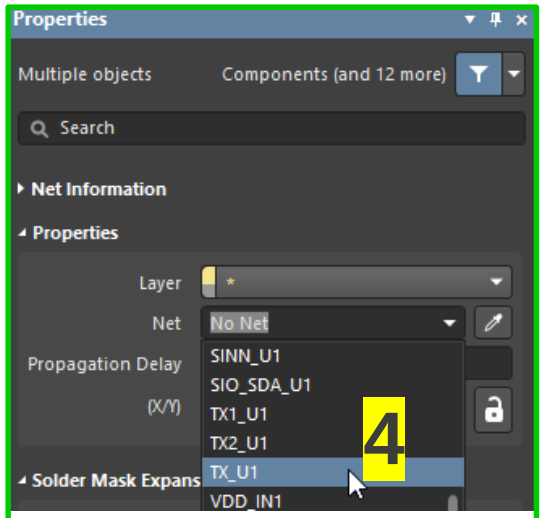
Select a via/segment



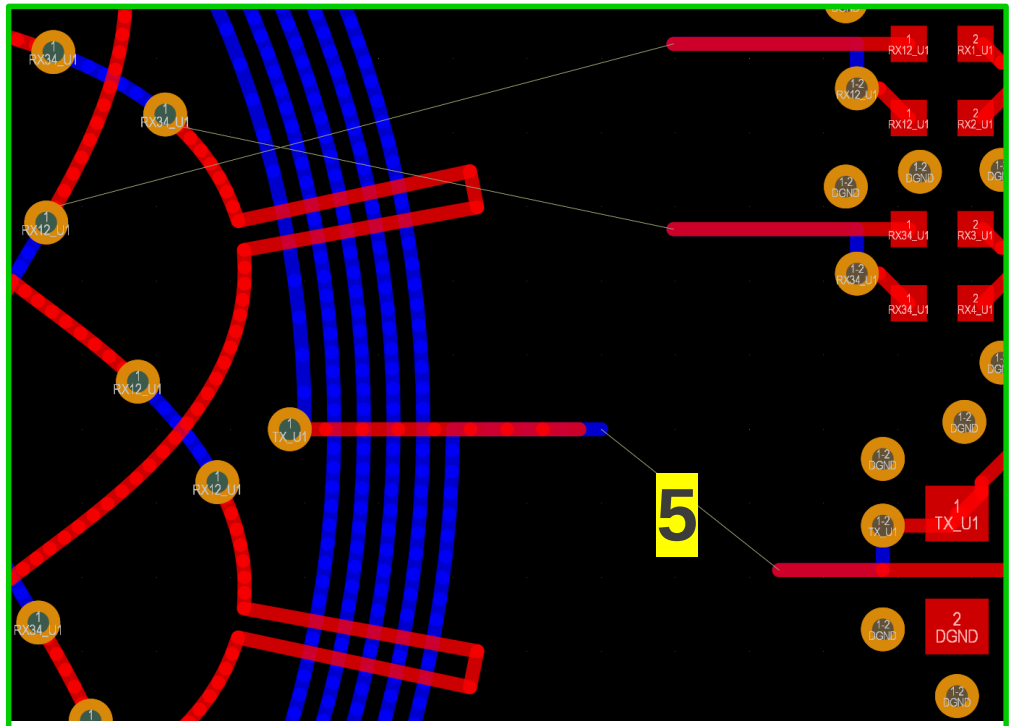
Press Tabulator once



Press Tabulator twice

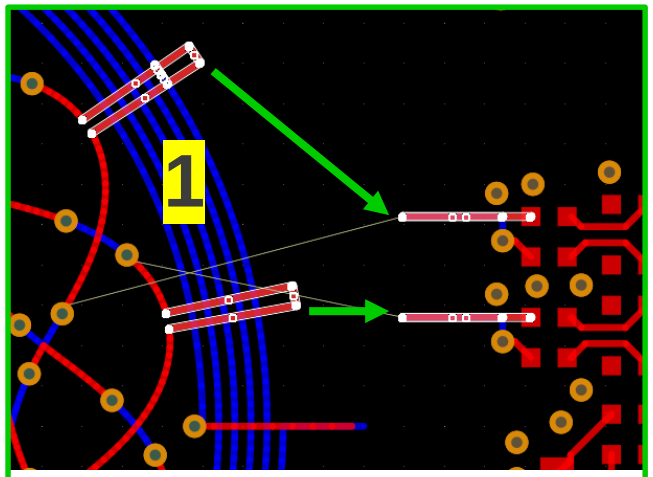


Select Net Name for each Coil

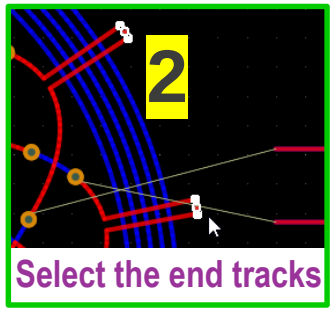


TX_U1, RX12_U1 and RX34_U1 are ready for connection

HOW TO CONNECT THE COILS?

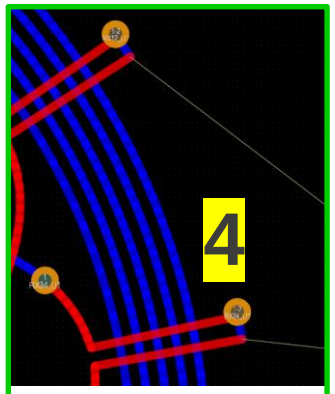


Always take the 2 closest Exits

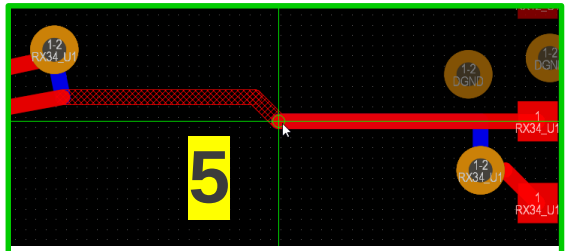


Select the end tracks

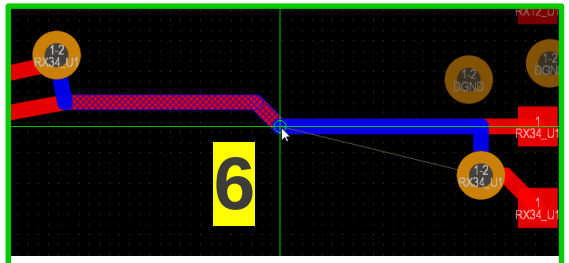
Move to 2nd Layer



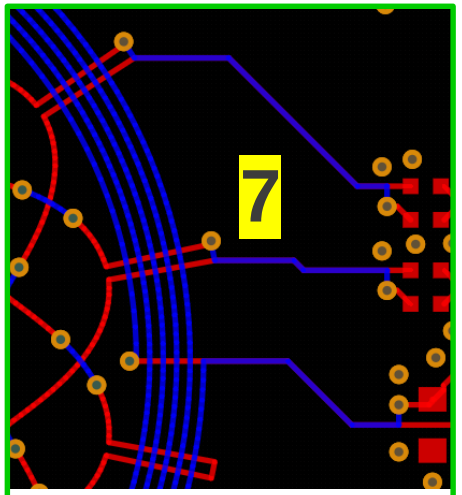
Place Vias



Finish routing on TOP…



…and on the 2nd Layer



Connect all TX & RX coils

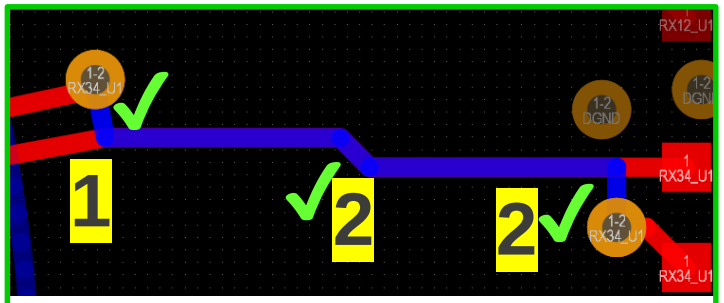
DESIGN HINT FOR COIL ROUTING

To eliminate the unwanted influence of bad wiring:

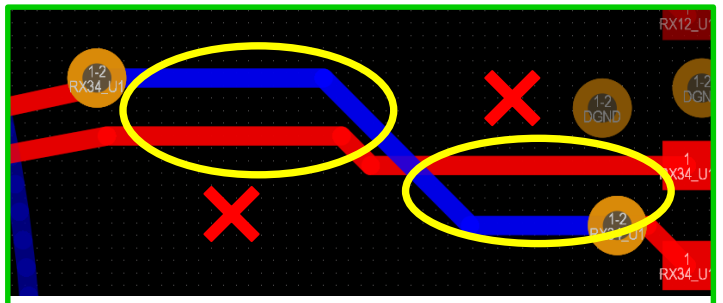
1 ☑ Always use the Exits as simulated

2 ☑ Always route the Coils overlapped on neighboring Layers!

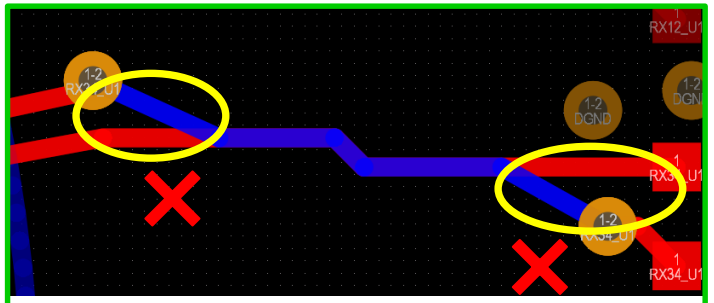
☑ Don´t change the Renesas Layout Template



Good: overlapped traces, Exit as generated



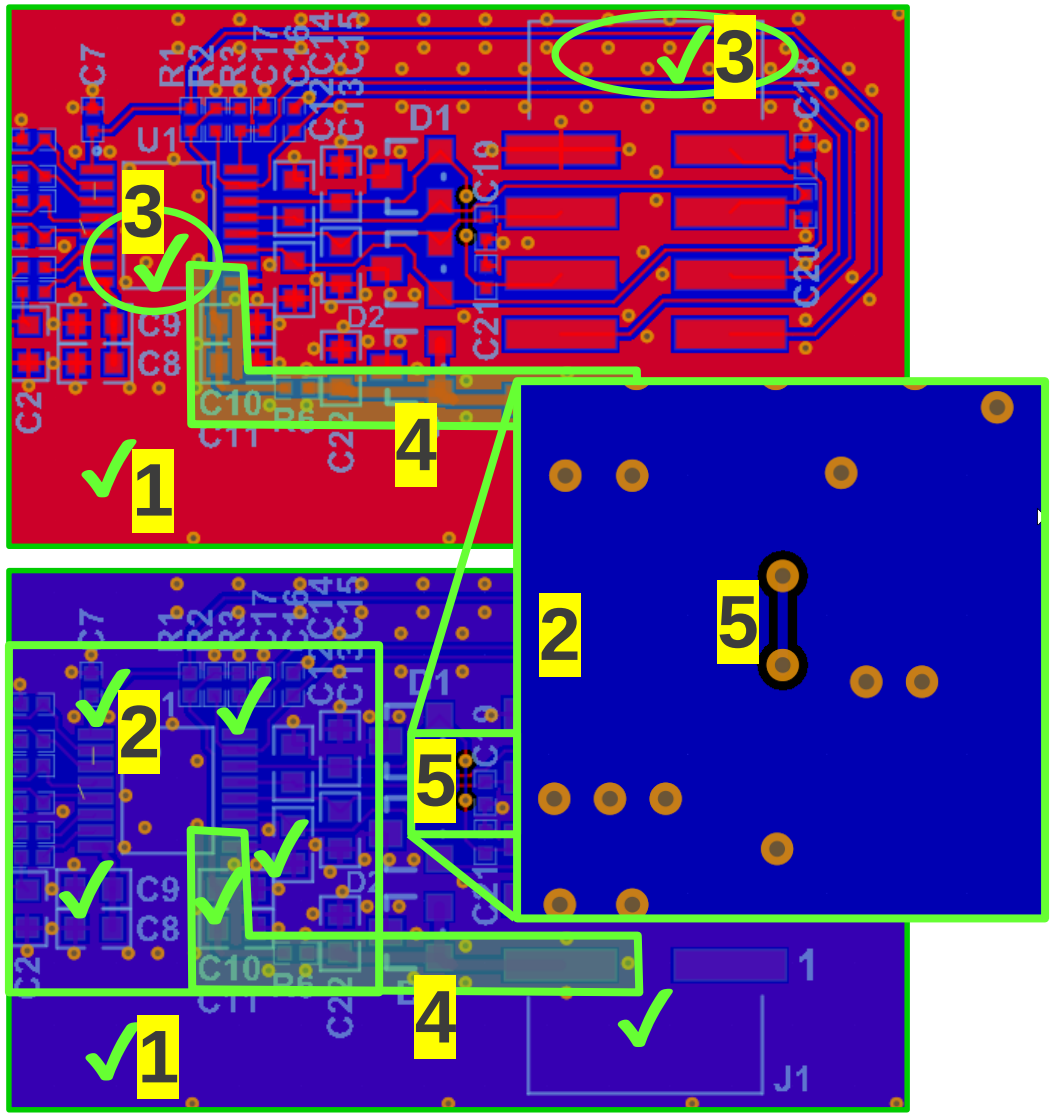
Bad: Traces are NOT overlapped



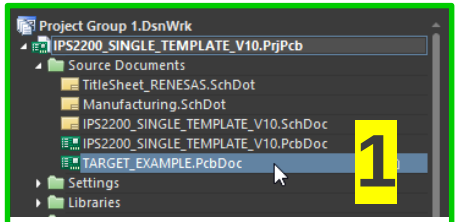
Bad: Traces are not all the way overlapped

DESIGN HINT FOR GND PLANES

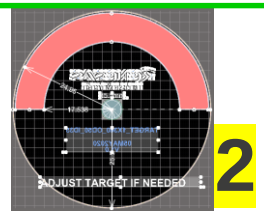
1. ☑ Always fill all the Copper Layers with GND Planes
2. ☑ Have at least one fully filled GND Layer below IPS2200
3. ☑ Use GND Vias to connect the Layers on every 1-2mm
4. ☑ Consider return Current Paths especially for VDD Supply
5. ☑ Only very short Tracks are allowed on GND Layer



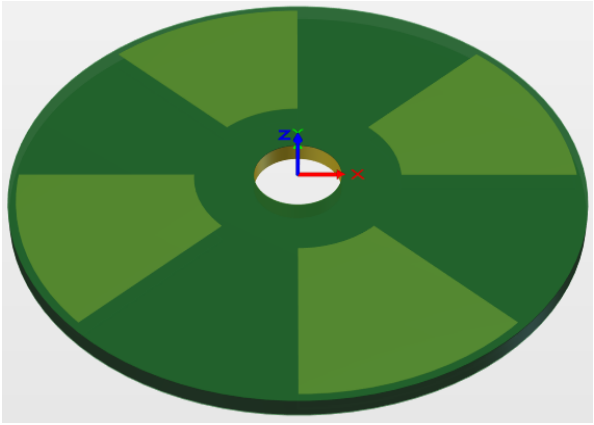
HOW TO DESIGN A PCB TARGET?

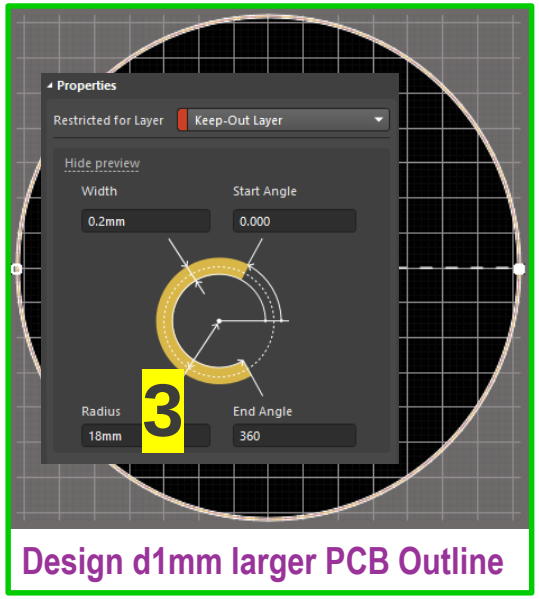


Open Target PcbDoc

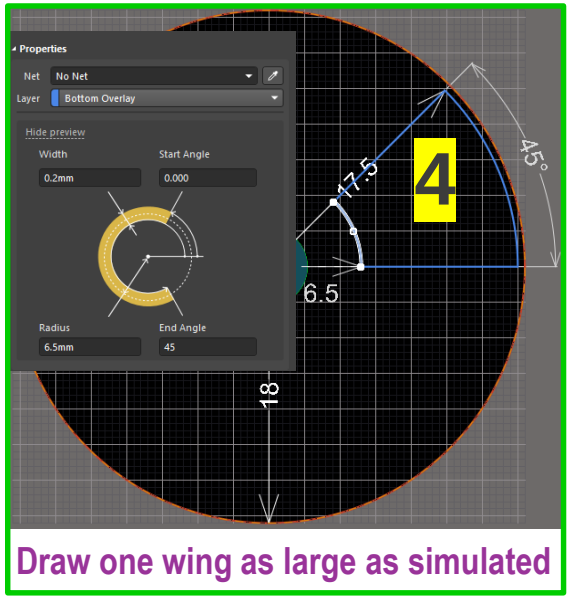


Delete unwanted drawings

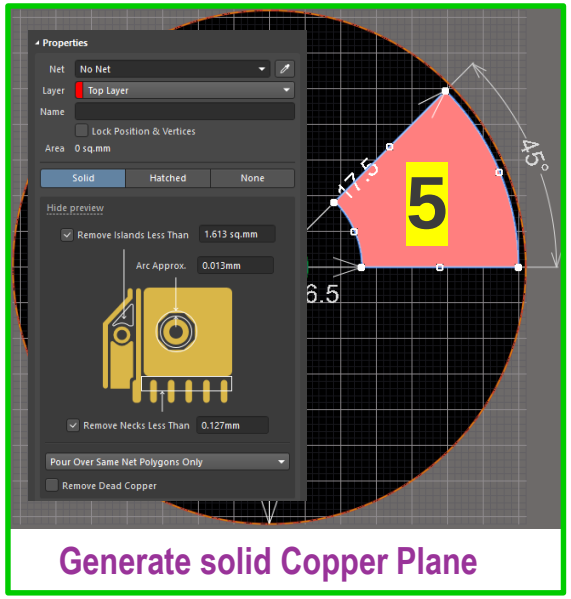




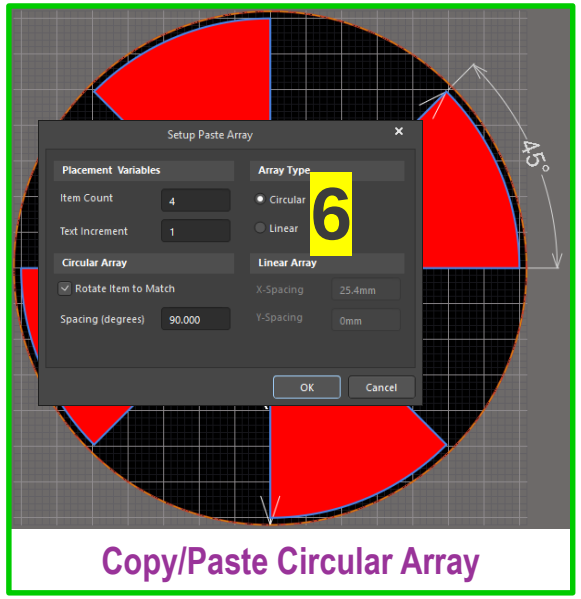
Design d1mm larger PCB Outline



Draw one wing as large as simulated



Generate solid Copper Plane

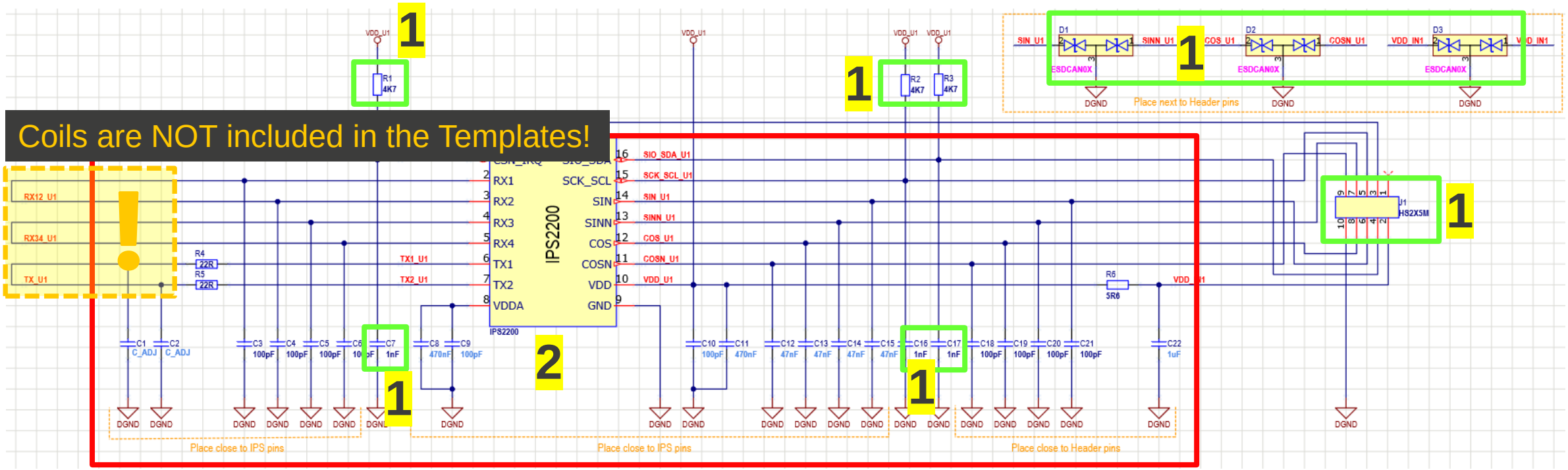


Copy/Paste Circular Array

IPS2200 SCHEMATIC TEMPLATES

1 Remove Components if unused or change them freely to your needs!

2 We advise NOT to change the rest of the schematic!



THANK YOU FOR YOUR ATTENTION!