



物料规格书

料号：07.02.0024		物料描述：PCB 离网逆变器 Granite 3000L-M1-MPPT A.1 FR-4 2层 2oZ 215*155mm 单板 RoHS			
最初使用机种：		RoHS 属性： <input checked="" type="checkbox"/> RoHS <input type="checkbox"/> 非 RoHS			
说明： <input checked="" type="checkbox"/> 电气规格 <input type="checkbox"/> 结构尺寸 <input type="checkbox"/> 辅料类 <input type="checkbox"/> 其他					
变更记录		版本	修改人	审核人	日期
改试产问题		V01-A.1	LJY		20230802
备注：					
制作人：李建英		审核人：廖永春			

注：在规格书里应尽量避免出现供应商的信息；
材料规格、尺寸等附页放在首页后；
变更需把变更记录写上，并写上 ECN 编号。

表单编号：ZONERGY-SJ-R-YF-003 版本：A/0
2023-08-07



A

B

C

D

E

Title: MPPT4860MB

Item Code

07.02.0024

ZONERGY

Rev: A.1

Dr: Ellen

Date: 2023/8/2

1

1

2

2

3

3

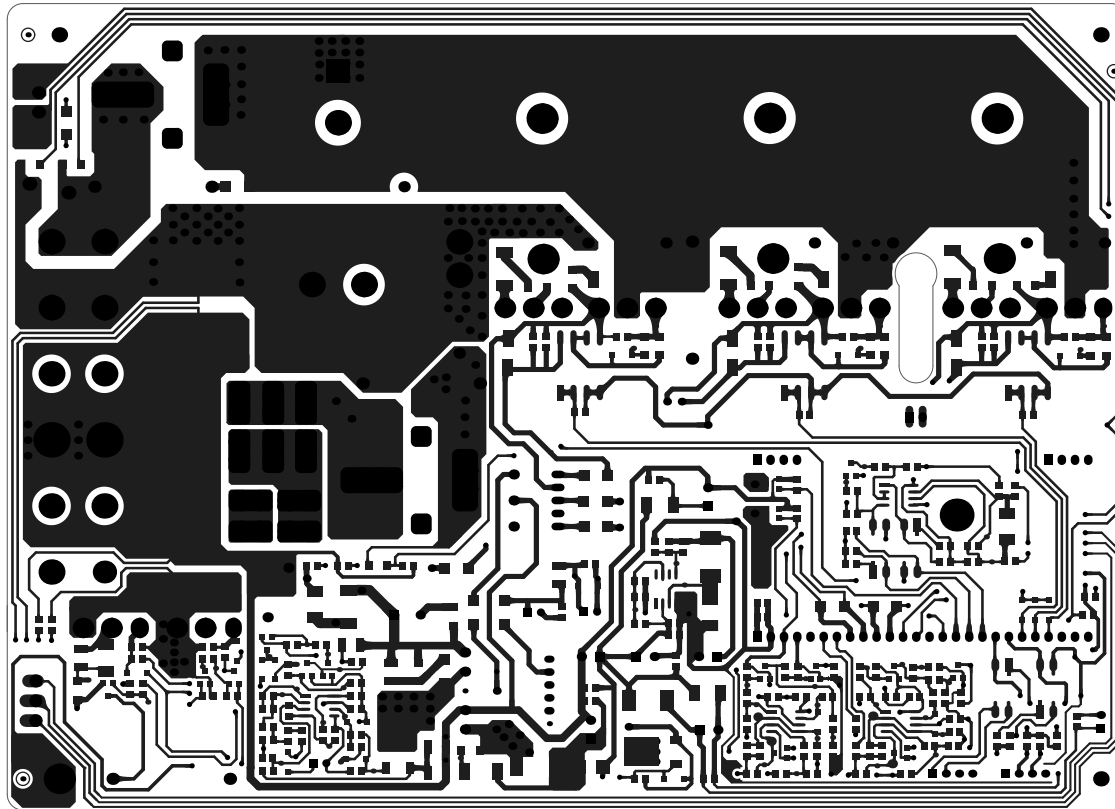
4

4

5

5

Top Layer



A

B

C

D

E

A

B

C

D

E

Title: MPPT4860MB

Item Code

07.02.0024

ZONERGY

Rev: A.1

Dr: Ellen

Date: 2023/8/2

1

1

2

2

3

3

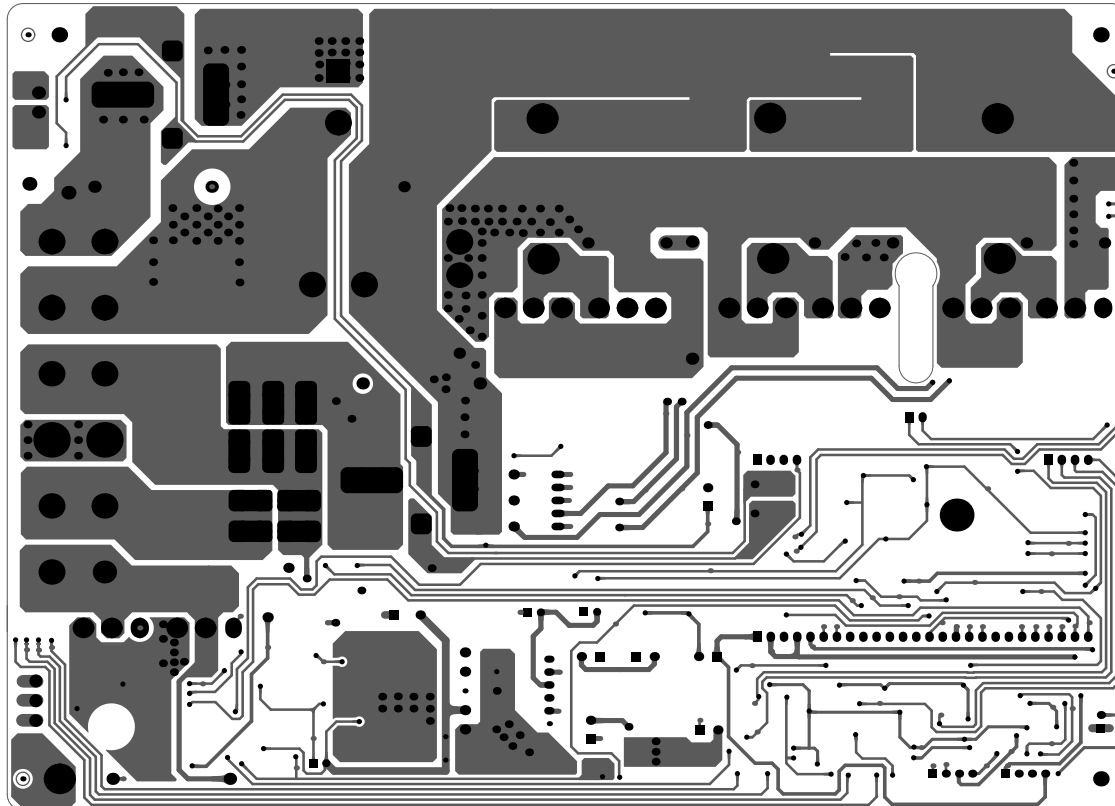
4

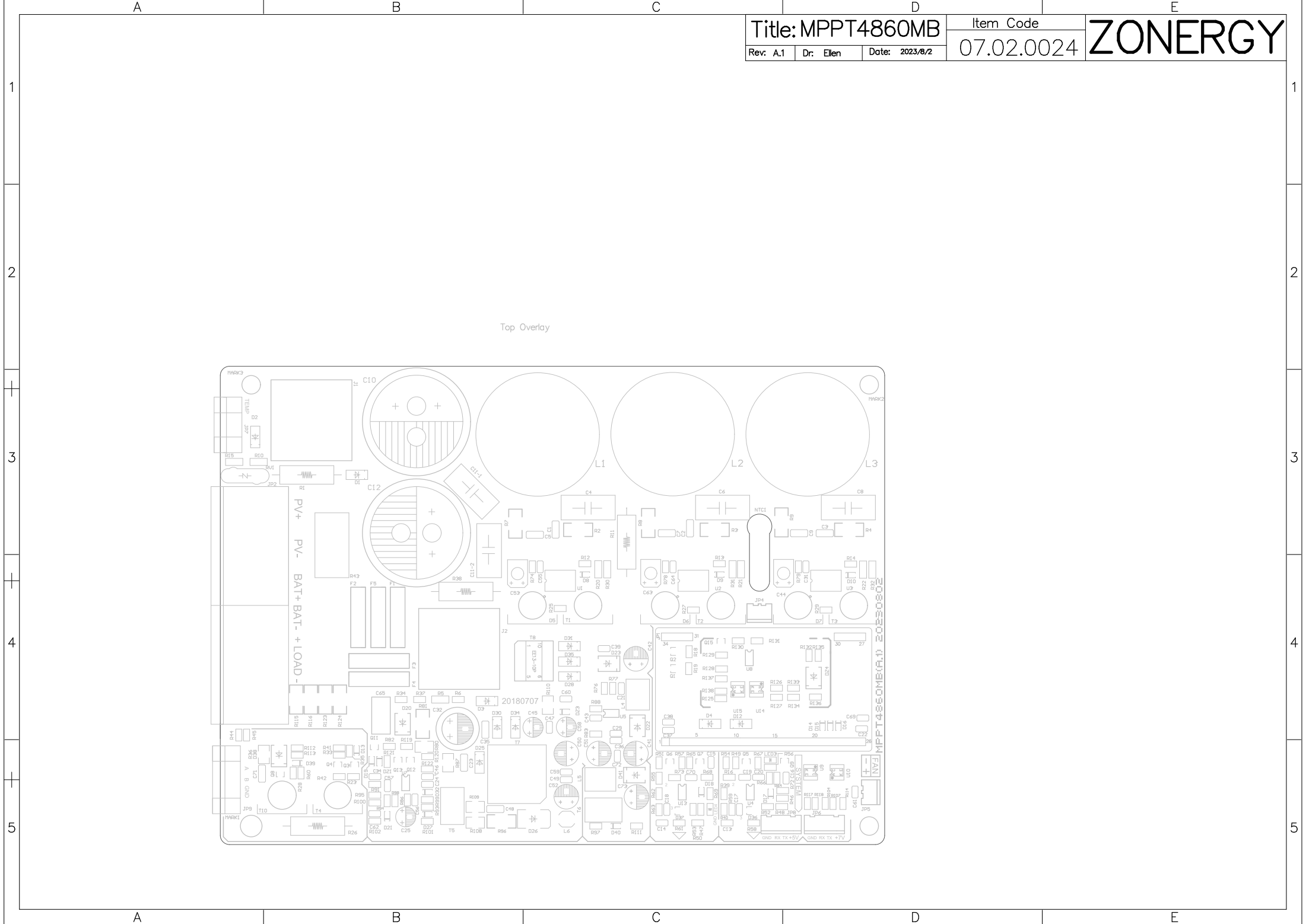
4

5

5

Bottom Layer





Title: MPPT4860MB

Item Code

ZONERGY

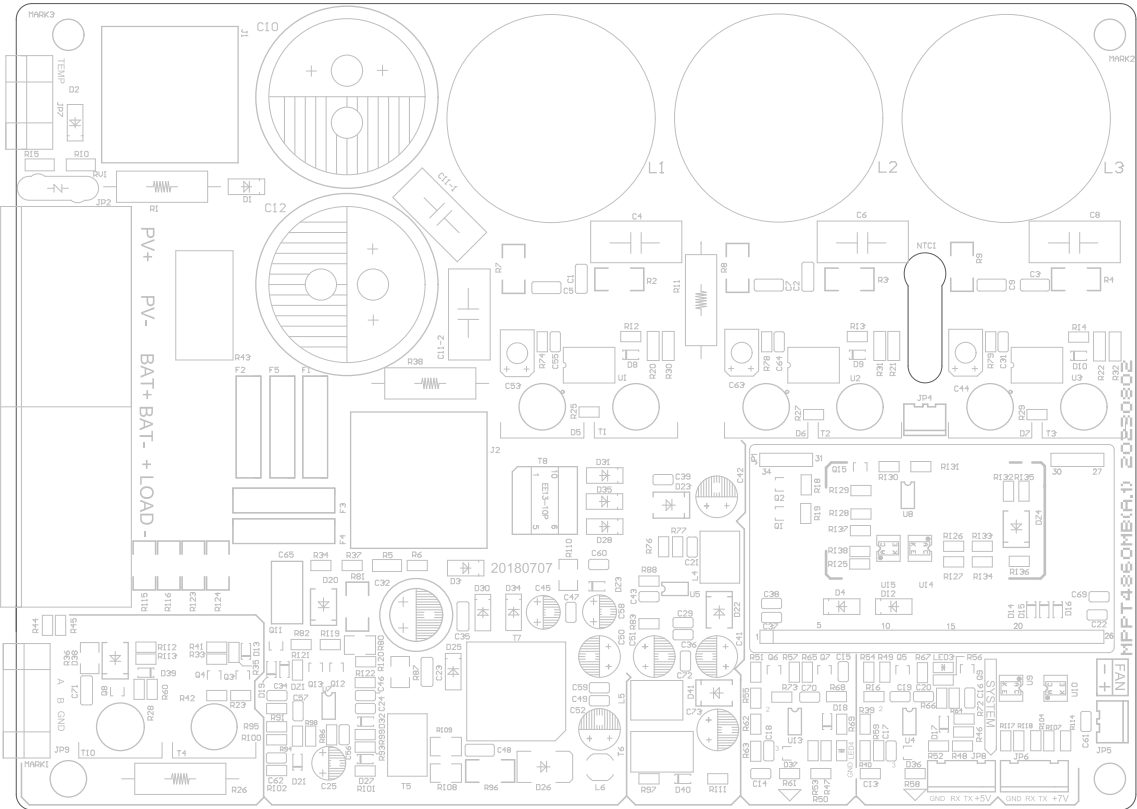
Rev: A.1

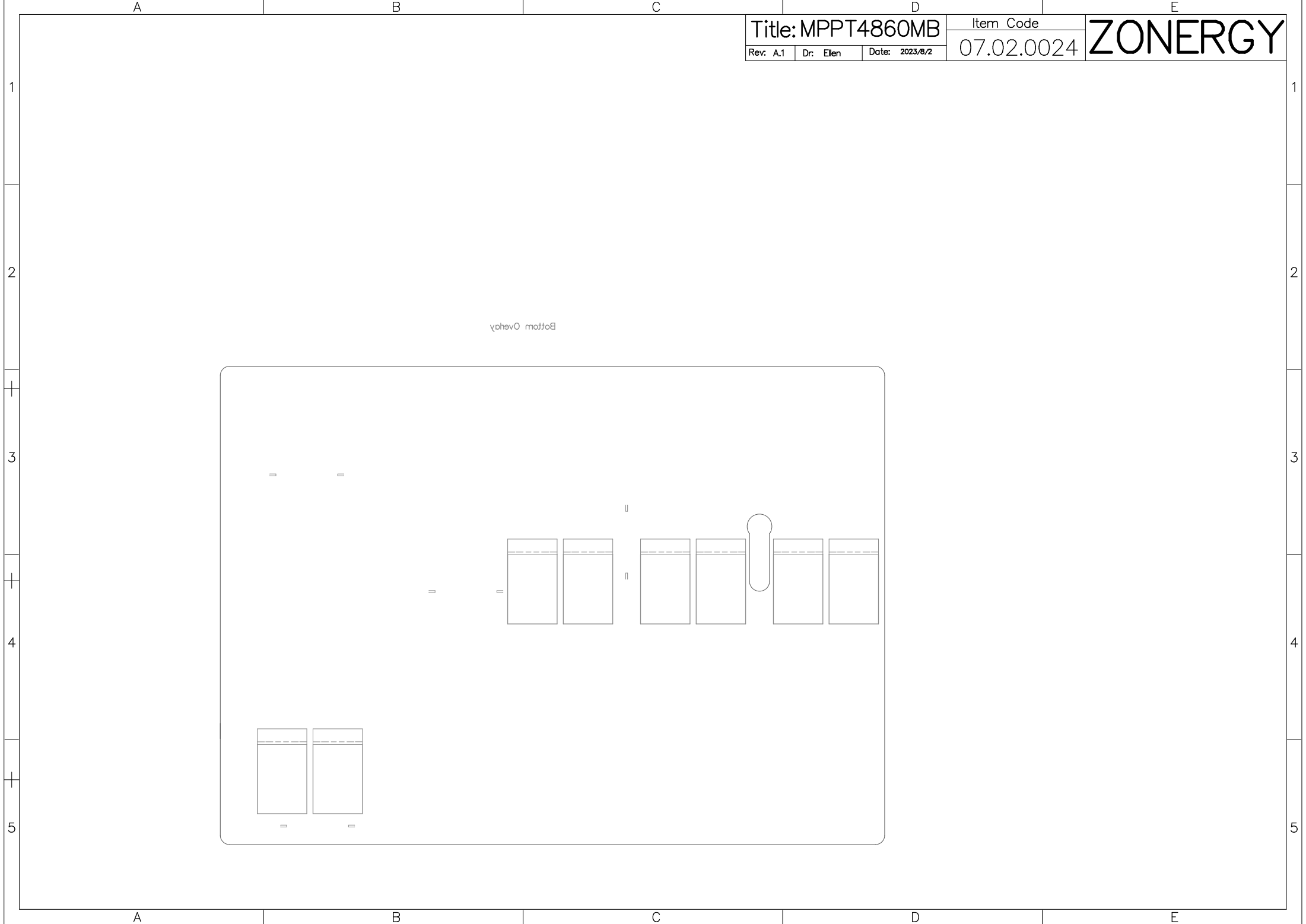
Dr: Ellen

Date: 2023/8/2

07.02.0024

Top Overlay





Title: MPPT4860MB

Item Code

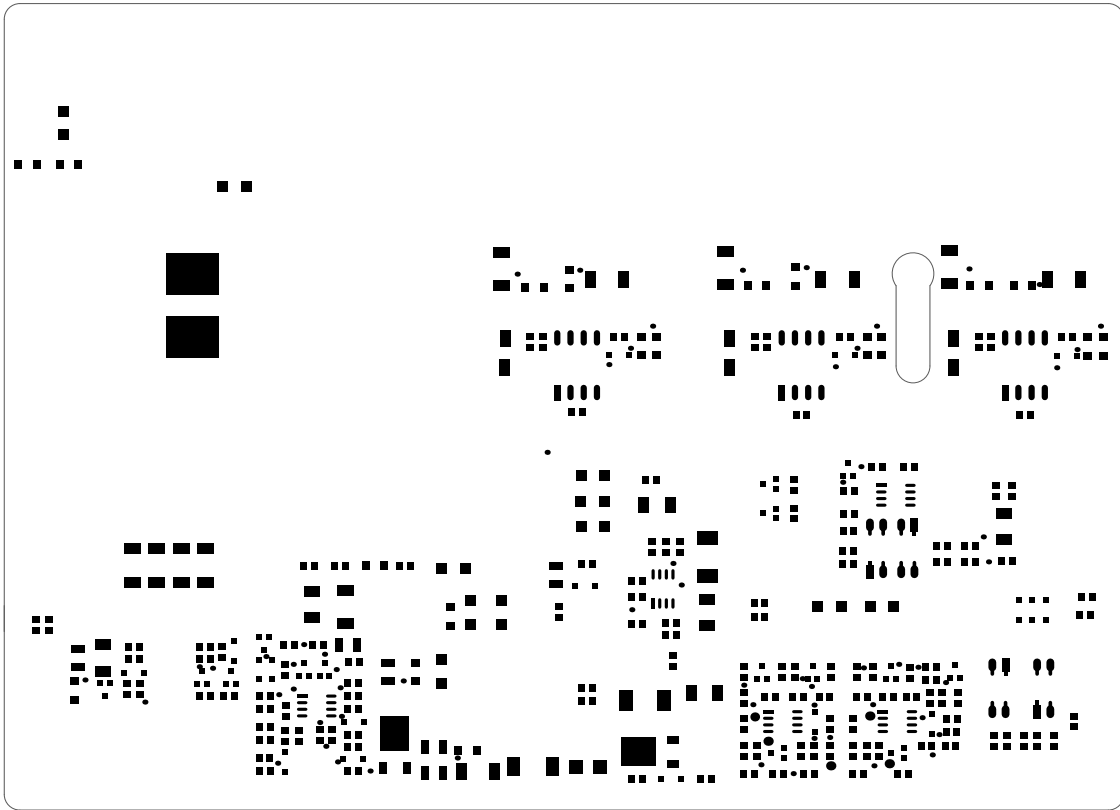
07.02.0024

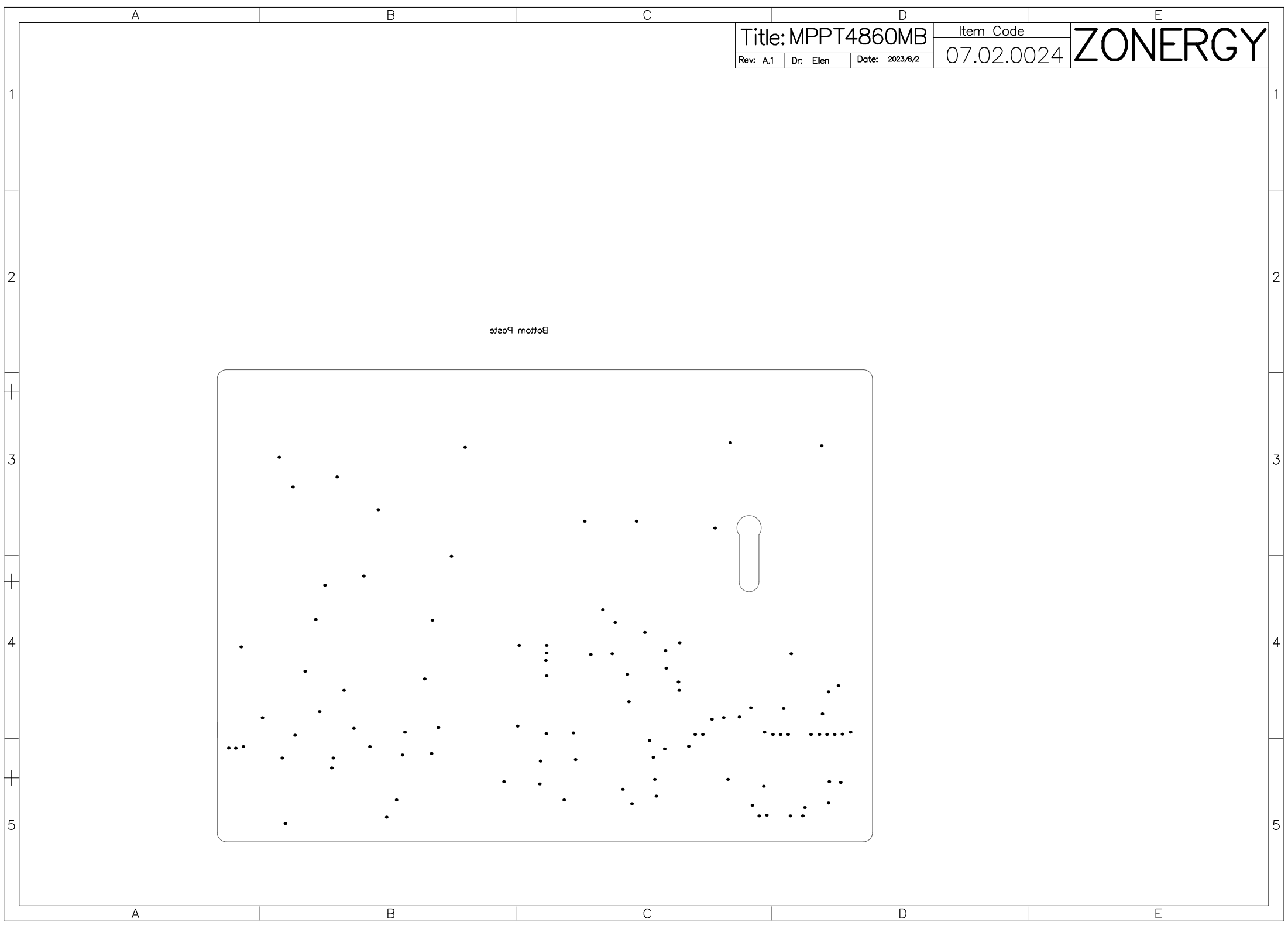
Rev: A.1

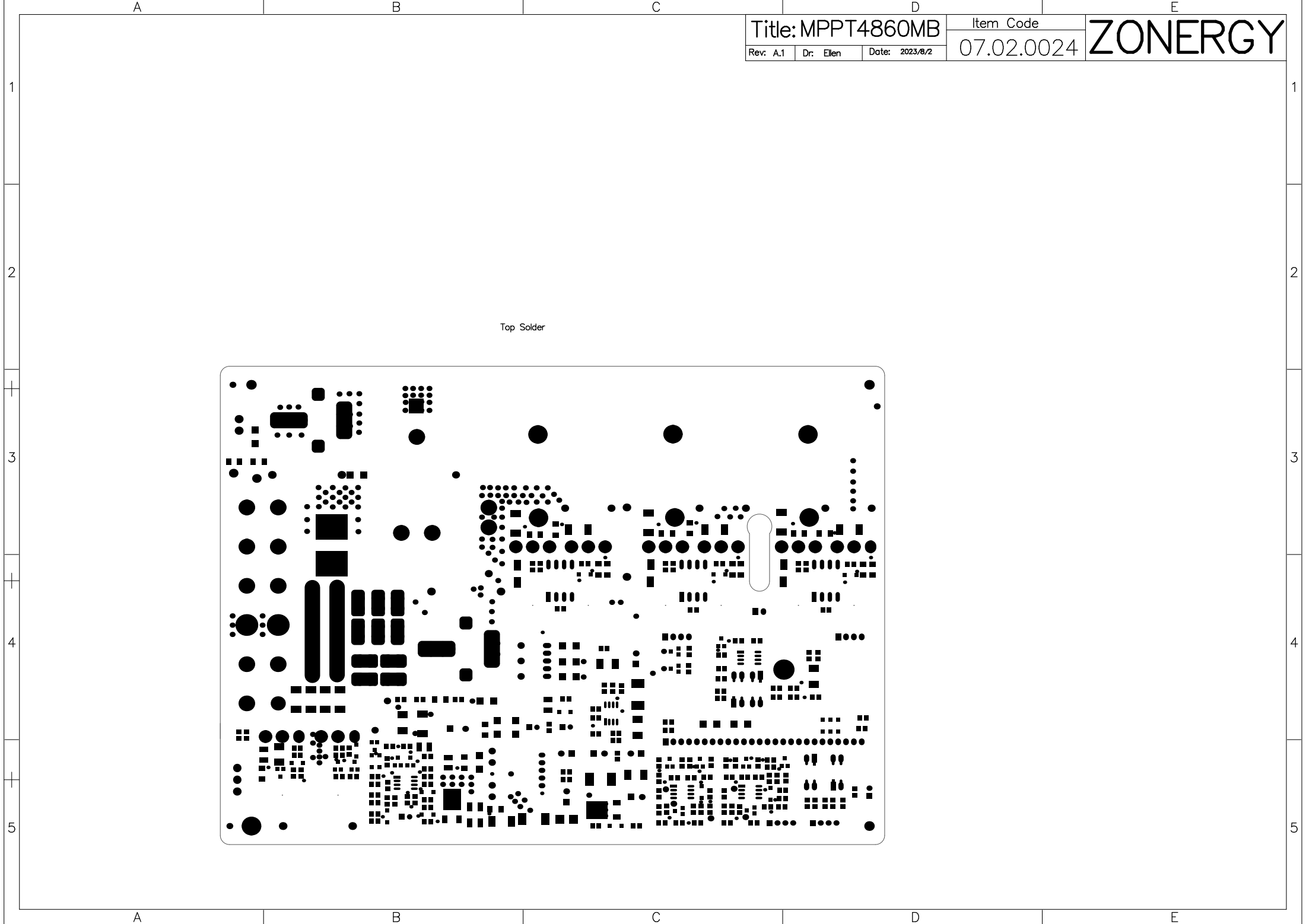
Dr: Ellen

Date: 2023/8/2

ZONERGY

A		B		C		D		E	
Title: MPPT4860MB						Item Code		ZONERGY	
Rev: A.1		Dr: Ellen		Date: 2023/8/2		07.02.0024			
1									1
2									2
3	Top Paste								3
4									4
5									5
A		B		C		D		E	





Title: MPPT4860MB

Item Code

07.02.0024

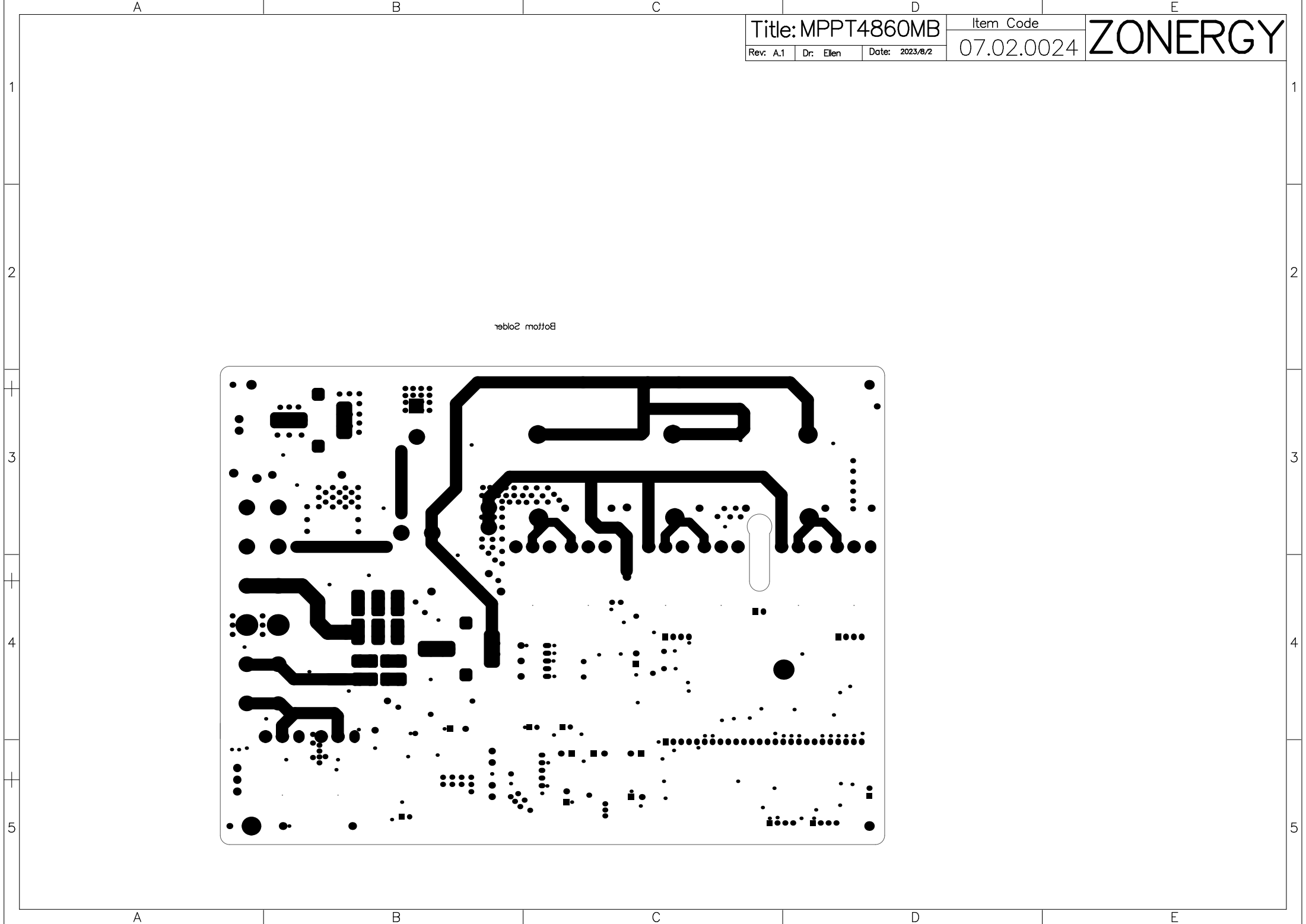
ZONERGY

Rev: A.1

Dr: Ellen

Date: 2023/8/2

Top Solder



Title: MPPT4860MB

Item Code

ZONERGY

Rev: A.1

Dr: Ellen

Date: 2023/8/2

07.02.0024

1.GENERAL

a.Technical requirements are prioritized as follows:

- 1.First priority is given to this document;
- 2.Relevant contract documents agreed on by both parties;
- 3.The Bill of General PCB specification document;
- 4.PCB general performance specification PC-6072
- 5.PCB acceptance specification PC-A-600

b. Drill boards using drill data/drill pattern and hole schedule. Any conflict between the CAD data,the drawing, & the drill pattern shall be clarified with the design, authority before proceeding.

c. Compensation and scaling may be applied by the PCB fabricator to allow for manufacturing process tolerances.

d. All units of measure are in millimetres unless otherwise stated.

2.ENGINEERING PROCESSING

- a. If there isn't enough NPTH in the PCB for fabrication positioning,it is allowed to add NPTH positioning holes with diameter not greater than 3mm on the auxiliary process edges (without affecting the SMT reference point) at the positions vacancies should be milled without break(which should be removed after finishing the PCB fabrication)
- b. If positions vacancies should be milled without break(which should be removed after finishing the PCB fabrication) in the PCB,current-hogging sheet copper may be added there.
- c. When fabricating multi-layer PCBs,if auxiliary process edge exists,it is allowed to add choked-flow pieces(inner layer) and auxiliary electroplate pieces(outer layer) there.
- d. To ensure the size of the PCB with process edges,when milling troughs,the milling cutter milling in dunnig strip a half of milling cutter diameter is permitted.
- e. The via without soldermask window should be treated as plugged.
- f. All for detail of those items beginning with"/",please refer to

3.PARAMETERS

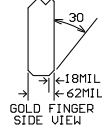
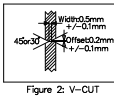
PCB Name: MPPT4860MB	SPELL: 1 x 1	# PCB Material :FR4	# TG>=150°
Silkscreen color: WHITE	Electrical test: Yes	# Surface Finishing:HAL Lead Free	# CTB>=175V
Blind or Buried via: No	Plate at board edge: No	# Soldermask Color:Green	# SAFETY MARK:UL94V-0
V-CUT: NO	Gold finger: No	# Green Technology:Plug Hole(VIA>0.5:Cover all)	忽略焊盘上的丝印
Copper thickness of plated via, though hole or edge >=25um			

4. VIEW

LAYER	Gerber files	NOTES
TOP	File:GTL	Positive
BOTTOM	File:GBL	Positive
SilkscreenTOP	File:GTO	Positive
SilkscreenBOTTOM	File:GBO	Positive
SoldermaskTOP	File:GTS	Negative
SoldermaskBOTTOM	File:GBS	Negative
Drill DrawingThrough	File:GD1	
Drill Guide Through	File:GG1	
NC DRILL	File-RoundHoles.TXT	DRILL
	File-SlotHoles.TXT	

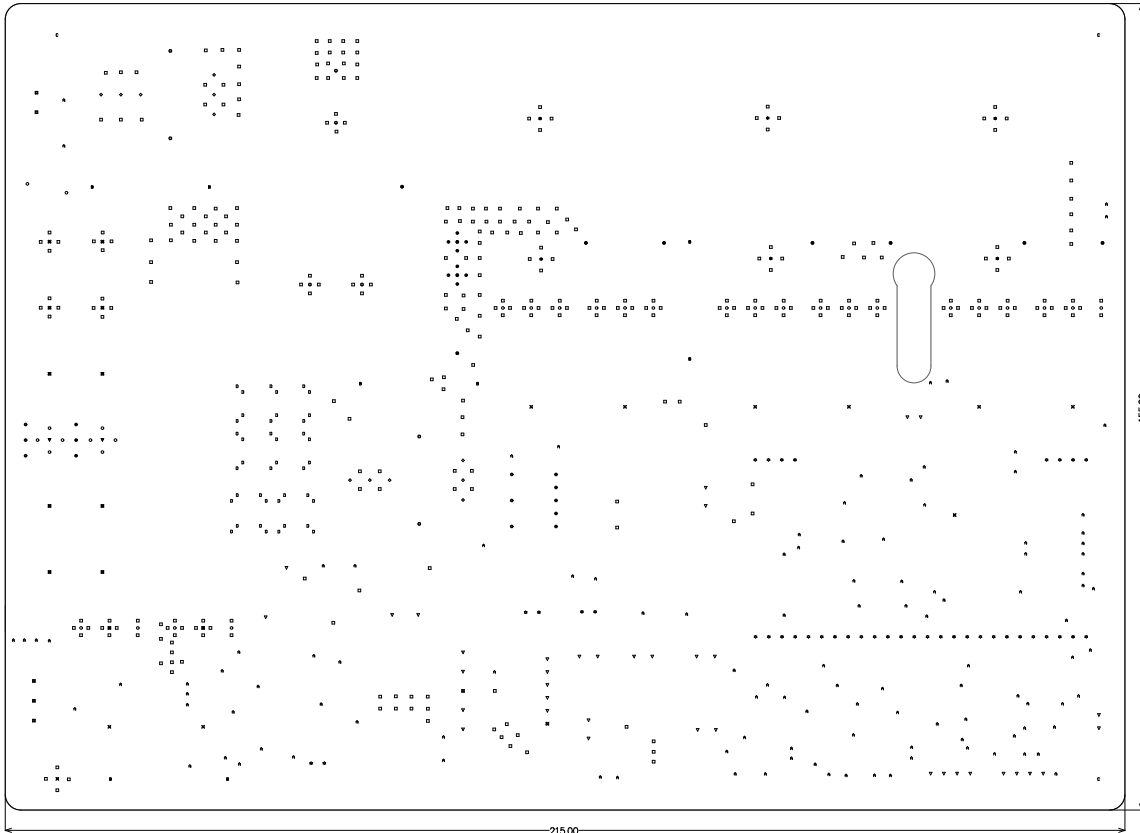
Table 1:Definition of layers

Arrangement of layers (Dielectric strength:2KV)
Foundation copper
Top
Core/Pre-preg>=0.2mm
Bottom
Finished Board Thickness: 1.6mm+/-0.15mm



1. 板内所有镀层及PAD公差±10%.
2. 外形公差±0.1mm.
3. 孔径公差: PTH孔±0.08mm, NPTH孔±0.05mm, VIA及槽±0.1mm.
4. 其他未标注均遵照IPC-208标准制造.

Drill Drawing



Title: MPPT4860MB

Item Code

07.02.0024

ZONERGY

Rev: A.1

Dr: Ellen

Date: 2023/8/2

Symbol	Count	Hole Size	Plated	Hole Type	Via/Pad
⊗	2	0.90mm (35.43mil)	NPTH	Round	Pad
▽	2	2.00mm (78.74mil)	PTH	Round	Pad
✕	2	3.20mm (125.98mil)	PTH	Round	Pad
C	3	3.20mm (125.98mil)	NPTH	Round	Pad
■	5	1.30mm (51.18mil)	PTH	Round	Pad
*	6	3.70mm (145.67mil)	PTH	Round	Pad
B	8	1.10mm (43.31mil)	PTH	Round	Pad
⊙	8	2.20mm (86.61mil)	PTH	Round	Pad
✕	8	8.00mm (314.96mil)	NPTH	Round	Pad
⊕	10	1.20mm (47.24mil)	PTH	Round	Pad
⊗	12	1.80mm (70.87mil)	PTH	Round	Pad
◇	12	2.50mm (98.43mil)	PTH	Round	Pad
⊙	22	1.00mm (39.37mil)	PTH	Round	Pad
⊙	32	1.50mm (59.06mil)	PTH	Round	Pad
▽	37	0.90mm (35.43mil)	PTH	Round	Pad
D	40	0.50mm (19.69mil)	PTH	Slot	Pad
*	40	0.80mm (31.50mil)	PTH	Round	Pad
A	117	0.50mm (19.69mil)	PTH	Round	Via
□	308	0.76mm (30.00mil)	PTH	Round	Pad
674 Total					

Slot definitions : Routed Path Length = Calculated from tool start centre position to tool end centre position.
Hole Length = Routed Path Length + Tool Size = Slot length as defined in the PCB layout

