



# 物料规格书

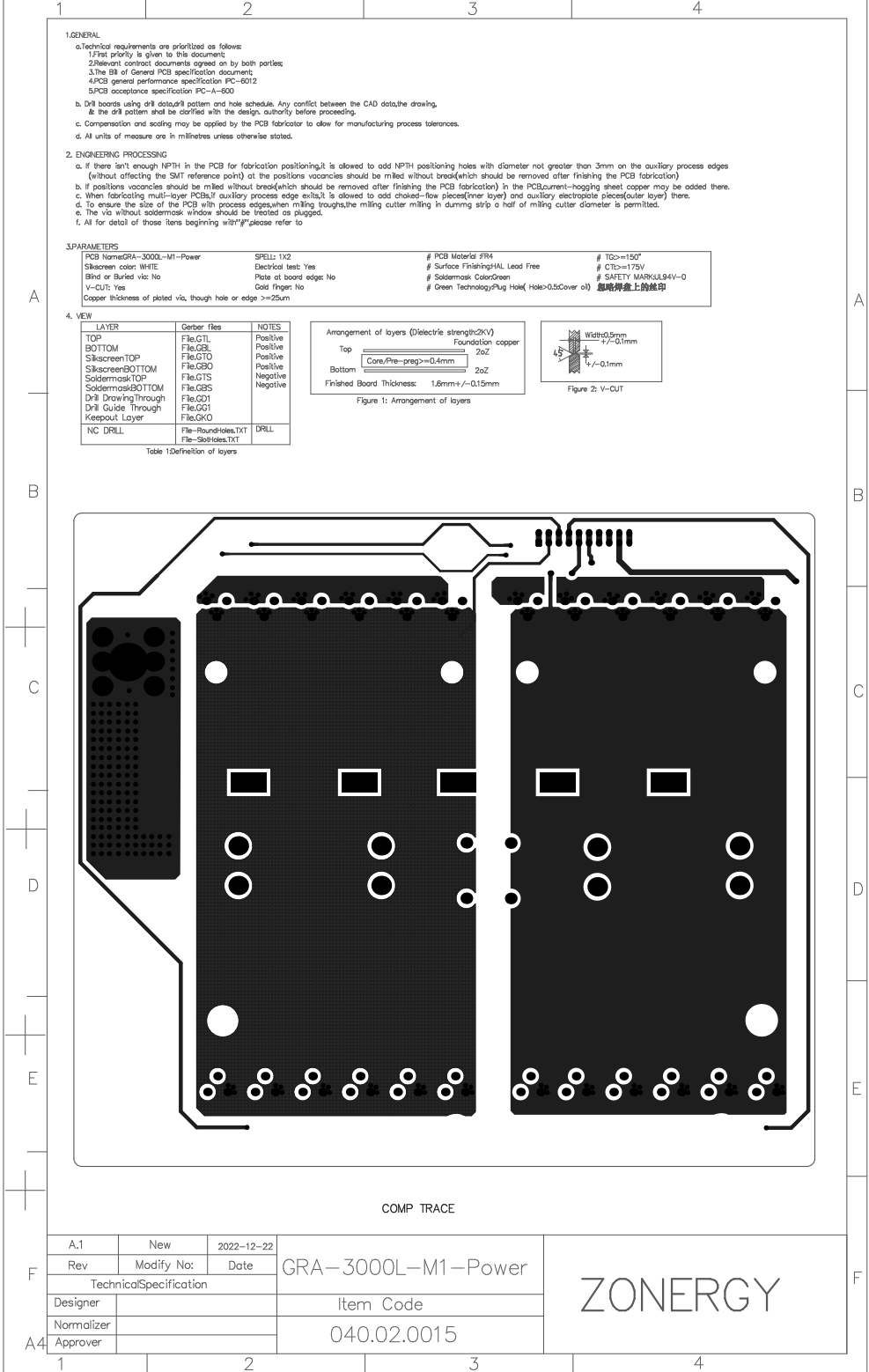
料号：040.02.0015		物料描述：PCB GRA-3000L-M1 Power (A.1) FR-4 2层 2oZ 187*165*1.6mm 单板 无铅喷锡 RoHS			
最初使用机种：		RoHS 属性： <input checked="" type="checkbox"/> RoHS <input type="checkbox"/> 非 RoHS			
说明： <input checked="" type="checkbox"/> 电气规格 <input type="checkbox"/> 结构尺寸 <input type="checkbox"/> 辅料类 <input type="checkbox"/> 其他					
变更记录		版本	修改人	审核人	日期
备注：					
制作人：李建英		审核人：万钢			

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

日期RQ

签章QZ



## 1.GENERAL

- a. Technical requirements are prioritized as follows:
  - 1) First priority is given to this document;
  - 2) General contract documents agreed on by both parties;
  - 3) The IIR of General PCB specification document;
  - 4) PCB general performance specification IPC-6012
  - 5) PCB acceptance specification IPC-A-600
- b. DRI based on the IIR data path pattern and hole schedule. Any conflict between the CAD data, the drawing, and the IIR 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 NPTB in the PCB fabrication positioning, it is allowed to add NPTB 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 broach (which should be removed after finishing the PCB fabrication).
- b. If positions vacancies should be milled without broach (which is allowed to add finished-flow piece) in the PCB auxiliary-hogging sheet, copper may be added there.
- c. When fabricating multi-layer PCB, auxiliary process edge extra is removed after choosing-flow piece (inner layer) and auxiliary electrode piece (outer layer) there.
- d. To ensure the size of the PCB with proper edges and inner through-hole, the milling cutter milling in during strip a half of milling cutter diameter is permitted.
- e. The via without soldermask window should be treated as plugger.
- f. All for detail of those items beginning with "if" please refer to

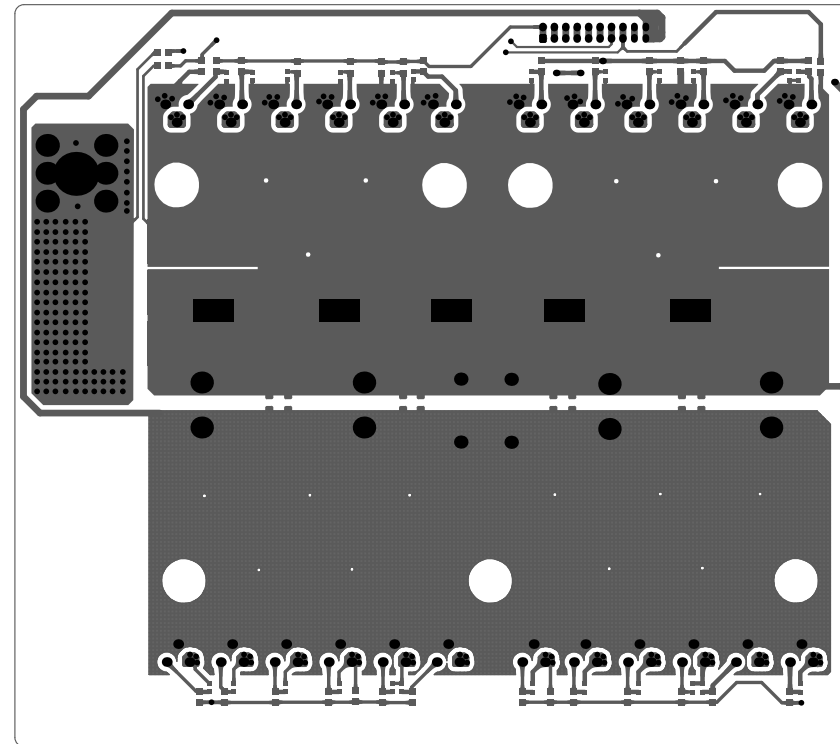
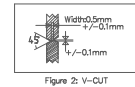
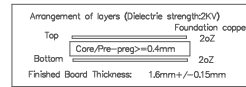
## 3PARAMETERS

PCB Name: GRA-3000L-M1-Power	SPELL: 1X2	# PCB Material: FR4	# TCo=150°
Silkscreen color: WHITE	Electrical test: Yes	# Surface Finishing: HAL Lead Free	# CTO=1.75V
Blind or Buried via: No	Plate at board edge: No	# Soldermask Color: Green	# SAFETY MARK: JLL94V-6
V-CUT: Yes	Gold finger: No	# Green Technology: Plug Hole (Hole>0.5, Cover oil)	超略牌上的烙印
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	
Keppout Layer	File.GKO	
NC DRILL	File-Round-Holes.TXT File-Slot-Holes.TXT	DRILL

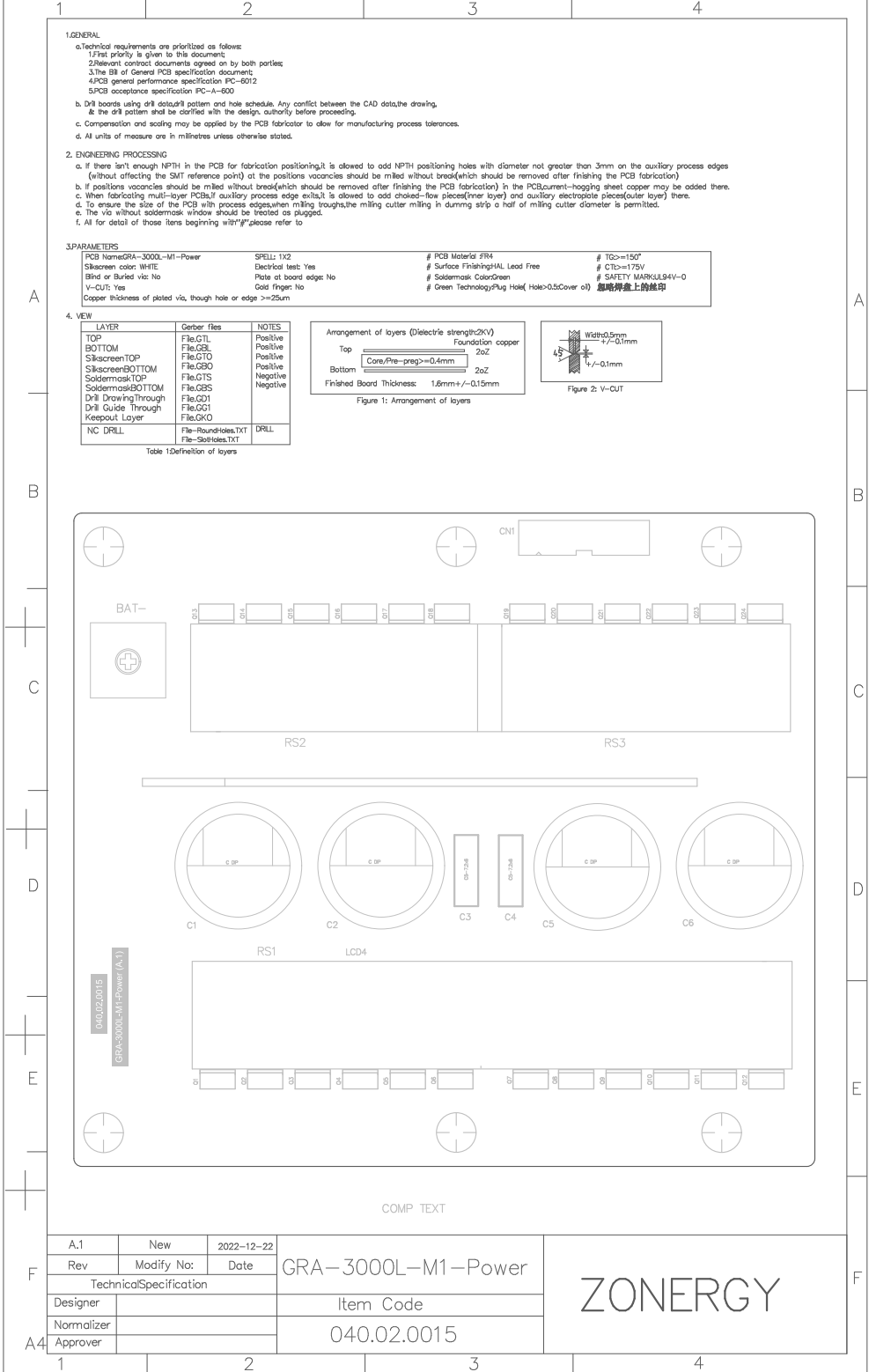
Table 1: Definition of layers



SOLDER TRACE

F	A.1	New	2022-12-22	GRA-3000L-M1-Power
	Rev	Modify No:	Date	
	Technical Specification			
	Designer			
A4	Normalizer			040.02.0015
	Approver			

ZONERGY



## 1. GENERAL

- a. Technical requirements are prioritized as follows:
  - 1. First priority is given to this document;
  - 2. General contract documents agreed on by both parties;
  - 3. The IIR of General PCB specification document;
  - 4. PCB general performance specification IPC-6012
  - 5. PCB acceptance specification IPC-A-600
- b. DRI based on IIR data path pattern and hole schedule. Any conflict between the CAD data, the drawing, and the IIR 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 NPTB in the PCB fabrication positioning, it is allowed to add NPTB 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 broach (which should be removed after finishing the PCB fabrication).
- b. If positions vacancies should be milled without broach (which is allowed to add finished-flow piece) in the PCB auxiliary-hogging sheet, copper may be added there.
- c. When fabricating multi-layer PCB, auxiliary process edge extra is removed after choosing-flow piece (inner layer) and auxiliary electrode piece (outer layer) there.
- d. To ensure the size of the PCB with proper edges and inner through-hole, the milling cutter milling in during strip a half of milling cutter diameter is permitted.
- e. The via without soldermask window should be treated as plugger.
- f. All for detail of those items beginning with "if" please refer to

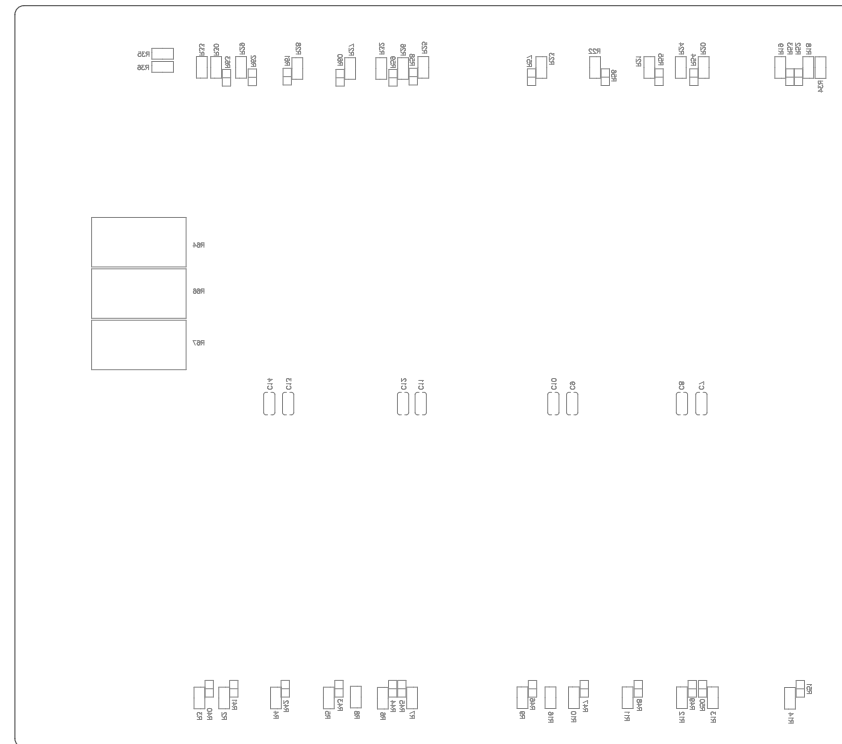
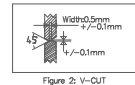
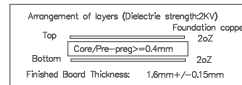
## 3PARAMETERS

PCB Name: GRA-3000L-M1-Power	SPELL: 1X2	# PCB Material: FR4	# TCO>=150V
Silkscreen color: WHITE	Electrical test: Yes	# Surface Finishing: HAL Lead Free	# CTD=175V
Blind or Buried via: No	Plate at board edges: No	# Soldermask Color: Green	# SAFETY MARK: UL94V-0
V-CUT: Yes	Gold finger: No	# Green Technology: Plug Hole (Hole>0.5Cover all)	# 越路焊盘上的丝印
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	
Keppout Layer	File.GKO	
NC DRILL	File-RoundHoles.TXT File-SlotHoles.TXT	DRILL

Table 1: Definition of layers



TEXT REORDER

F	A.1	New	2022-12-22	GRA-3000L-M1-Power	ZONERGY
	Rev	Modify No:	Date		
	Technical Specification			Item Code	
	Designer				
Normalizer					
A4	Approver			040.02.0015	

1

2

3

4

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 B1 of General PCB specification document;  
4.PCB general performance specification PC-6012  
5.PCB acceptance specification PC-A-600

b. D/4 boards using d/4 data,d/4 pattern and hole schedule. Any conflict between the CAD data,the drawing,  
& the d/4 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 (in/all) 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 durning 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:GRA-3000L-M1-Power

SPELL: 1X2

# PCB Material:FR4

# TG2=>150°

Silkscreen color: WHITE

Electrical test: Yes

# Surface Finishing:HAL Lead Free

# CT2=>175V

Blind or Buried via: No

Plate at board edge: No

# Soldermask Color:Green

# SAFETY MARK:UL94V-0

V-CUT: Yes

Gold finger: No

# Green Technology/Plug Hole( Hole>0.5Cover all)

忽略焊盘上的丝印

Copper thickness of plated via, through 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
D/4 Drawing Through	File:GD1	
D/4 Guide Through	File:GD1	
Keepout Layer	File:GKO	
NC DRILL	File-RoundHoles.TXT File-SlotHoles.TXT	DRILL

Table 1:Definition of layers

Arrangement of layers (Dielectric strength:2KV)

Top Foundation copper  
2oz  
Core/Pre-preg=>0.4mm  
Bottom 2oz  
Finished Board Thickness: 1.6mm+/-0.15mm

Figure 1: Arrangement of layers

Figure 2: V-CUT

Width:0.5mm  
+/-0.1mm  
+/-0.1mm

COMP PASTE

A.1	New	2022-12-22	GRA-3000L-M1-Power	ZONERGY
Rev	Modify No:	Date		
TechnicalSpecification				
Designer				
Normalizer			Item Code	
Approver			040.02.0015	

## 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 IIR of General PCB specification document;
  4. General performance specification IPC-6012
  5. PCB acceptance specification IPC-A-600
- b. Drill boards using drill data/drill pattern and hole schedule. Any conflict between the CAD data, the drawing, and 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 NPHI in the PCB for fabrication positioning, it is allowed to add NPHI 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 PCB, auxiliary process edge exist, is allowed to add chined-low piece(nine layer) and auxiliary electrode piece(nine layer) there.
- d. To ensure the size of the PCB with process edges may not through the milling cutter milling in during strip a half of milling cutter diameter is permitted.
- e. The via without soldermask wing should be treated as plugged.
- f. At for detail of those items beginning with "if" please refer to

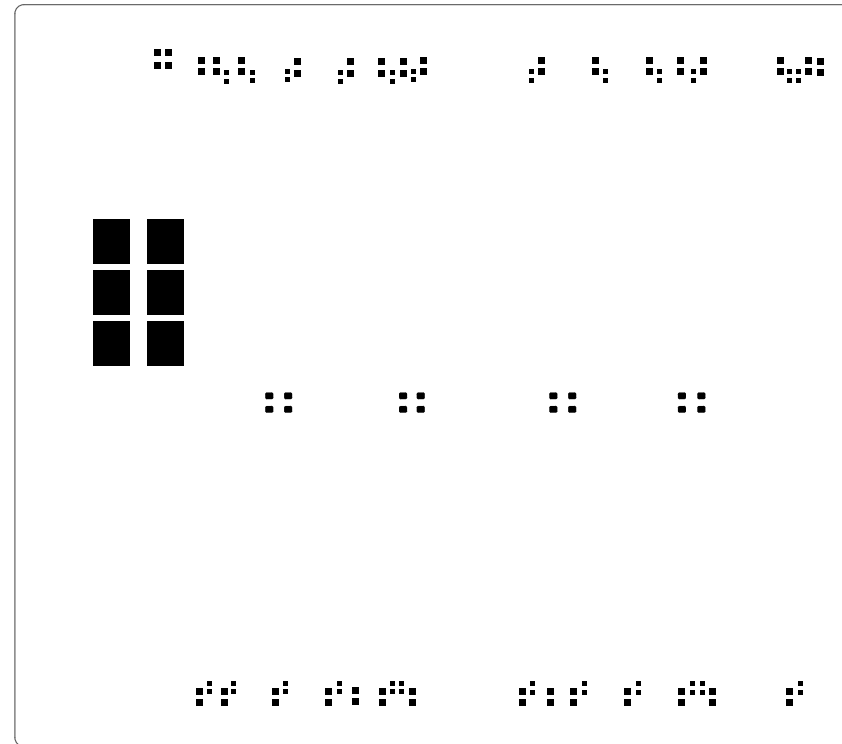
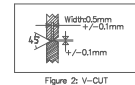
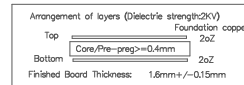
### 3PARAMETERS

PCB Name: GRA-3000L-M1-Power	SPELL: 1X2	# PCB Material #FR4	# TCO>=150°
Silkscreen color: WHITE	Color test: Yes	# Surface Finishing: HAL Lead Free	# TCO=175V
Blind or Buried via: No	Plate at board edge: No	# Soldermask Color: Green	# SAFETY MARK: UL94V-0
V-CUT: Yes	Gold finger: No	# Green Technology: Plug Hole (Hole>0.5*Cover 0)	加路焊盘上的丝印
Copper thickness of plated via, through 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	
Keppout Layer	File.GKO	
NC DRILL	File-Round-Holes.TXT File-Slot-Holes.TXT	DRILL

Table 1: Definition of layers



SOLDER PASTE

A.1	New	2022-12-22	GRA-3000L-M1-Power
Rev	Modify No:	Date	
Technical Specification			
Designer			
Normalizer			Item Code
Approver			040.02.0015

ZONERGY

## 1.GENERAL

- a. Technical requirements are prioritized as follows:
  - 1) First priority is given to this document;
  - 2) General contract documents agreed on by both parties;
  - 3) The IIR of General PCB specification document;
  - 4) PCB general performance specification IPC-6012
  - 5) PCB acceptance specification IPC-A-600
- b. DRI based on the IIR data path pattern and hole schedule. Any conflict between the CAD data, the drawing, and the IIR 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 NPTB in the PCB fabrication positioning, it is allowed to add NPTB 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 broach (which should be removed after finishing the PCB fabrication).
- b. If positions vacancies should be milled without broach (which is allowed to add finished-flow piece) in the PCB auxiliary-hogging sheet, copper may be added there.
- c. When fabricating multi-layer PCB, auxiliary process edge extra is removed after choosing-flow piece (inner layer) and auxiliary electrode piece (outer layer) there.
- d. To ensure the size of the PCB with proper edges and inner through-hole, the milling cutter milling in during strip a half of milling cutter diameter is permitted.
- e. The via without soldermask window should be treated as plugger.
- f. All for detail of those items beginning with "if" please refer to

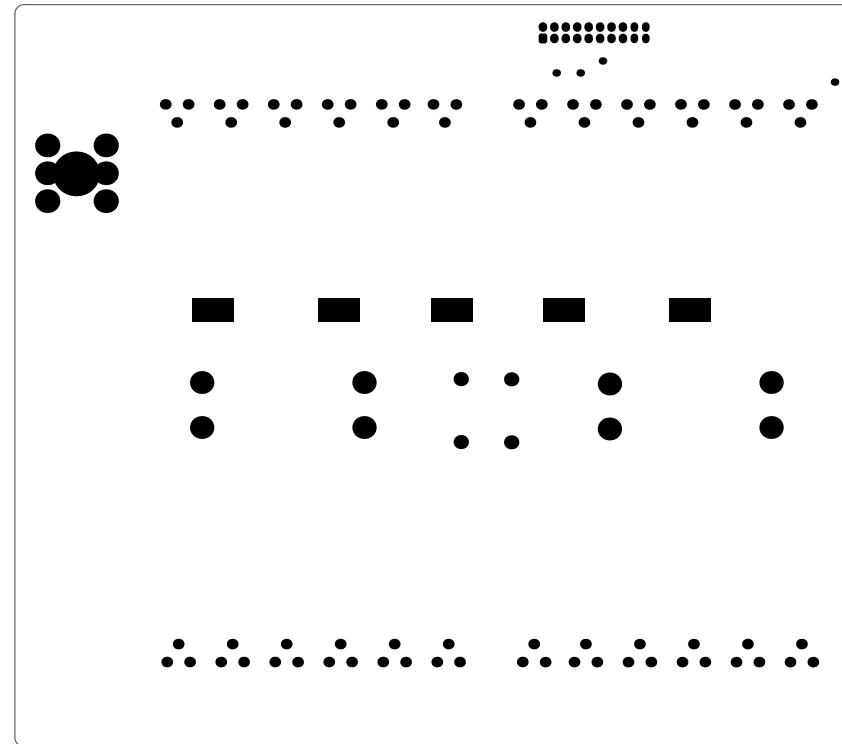
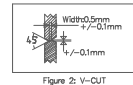
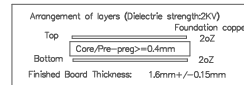
## 3PARAMETERS

PCB Name: GRA-3000L-M1-Power	SPELL: 1X2	# PCB Material #FR4	# TCO>=150°
Silkscreen color: WHITE	Electrical Test: Yes	# Surface Finishing: HAL Lead Free	# CTR>=175V
Blind or Buried via: No	Plate at board edge: No	# Soldermask Color: Green	# SAFETY MARK: UL94V-0
V-CUT: Yes	Gold finger: No	# Green Technology: Plug Hole (Hole>0.5: Cover all)	如路板上之綠印
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	
Keppout Layer	File.GKO	
NC DRILL	File-Round-Holes.TXT File-Slot-Holes.TXT	DRILL

Table 1: Definition of layers



A.1	New	2022-12-22	GRA-3000L-M1-Power
Rev	Modify No:	Date	
Technical Specification			
Designer	Item Code		
Normalizer	040.02.0015		
Approver			

ZONERGY



## 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 IIR of General PCB specification document;
  4. General performance specification IPC-6012
  5. PCB acceptance specification IPC-A-600
- b. Drill boards using drill data/drill pattern and hole schedule. Any conflict between the CAD data, the drawing, and 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 NPHI in the PCB for fabrication positioning, it is allowed to add NPHI 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 PCB, auxiliary process edge exist, is allowed to add chined-low piece(nine layer) and auxiliary electrode piece(nine layer) there.
- d. To ensure the size of the PCB with process edges meet the requirements, the milling machine cutting in during strip a half of milling cutter diameter is permitted.
- e. The via without soldermask window should be treated as plugged.
- f. As for detail of those items beginning with "if" please refer to

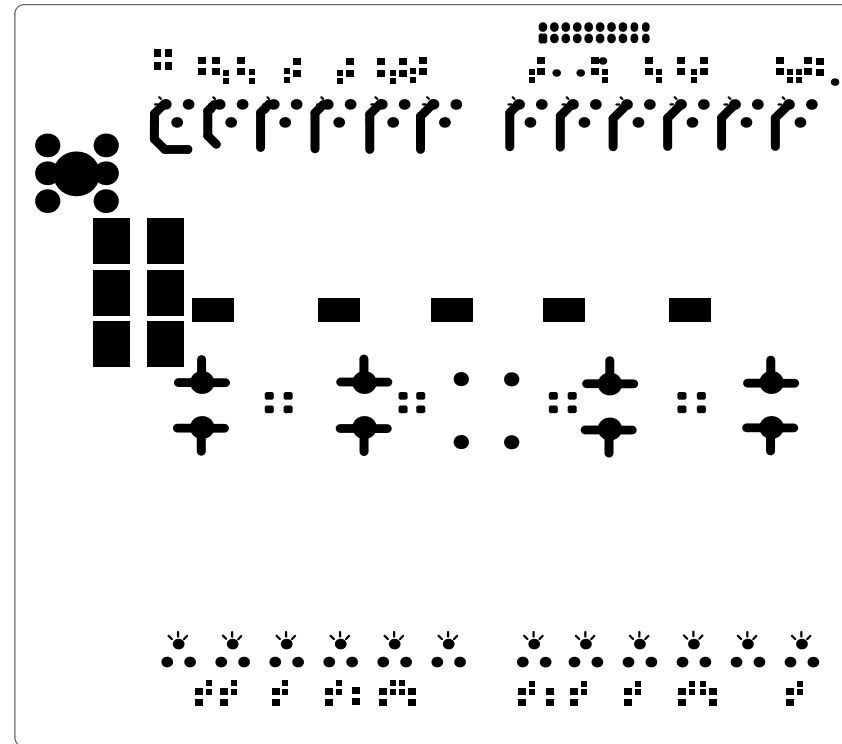
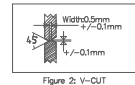
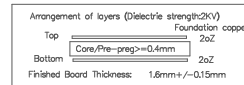
## 3PARAMETERS

PCB Name: GRA-3000L-M1-Power	SPELL: 1X2	# PCB Material: FR4	# TCo=150"
Silkscreen color: WHITE	Electrical Test: Yes	# Surface Finishing: HAL Lead Free	# CTO=175V
Blind or Buried via: No	Plate at board edge: No	# Soldermask Color: Green	# SAFETY MARK: UL94V-0
V-CUT: Yes	Gold Finger: No	# Green Technology: Plug Hole (Hole>0.5Cover 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	
Keppout Layer	File.GKO	
NC DRILL	File-Round-Holes.TXT File-Slot-Holes.TXT	DRILL

Table 1: Definition of layers



SOLDER MASK

A.1	New	2022-12-22	GRA-3000L-M1-Power
Rev	Modify No:	Date	
Technical Specification			
Designer			
Normalizer			Item Code
Approver			040.02.0015

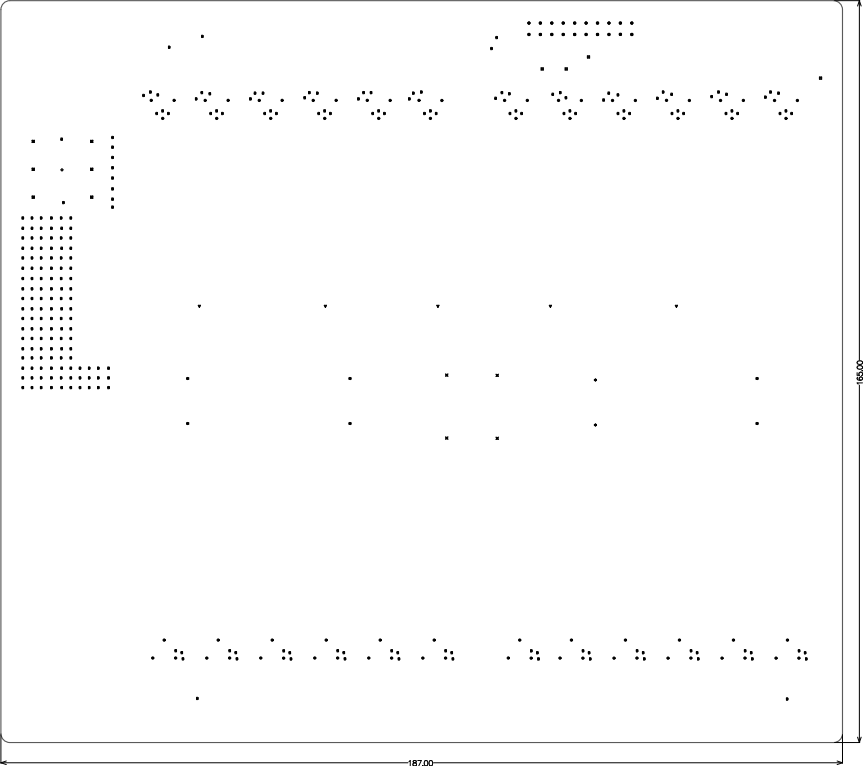
ZONERGY



Symbol	Count	Hole Size	Plated	Hole Type	Via/Pad
□	4	0.700mm (27.56mil)	PTH	Round	Pad
⊖	244	0.762mm (30.00mil)	PTH	Round	Via
⊕	20	1.000mm (39.37mil)	PTH	Round	Pad
✕	4	1.200mm (47.24mil)	PTH	Round	Pad
○	72	1.300mm (51.18mil)	PTH	Round	Pad
▽	5	1.500mm (59.06mil)	PTH	Slot	Pad
★	8	2.000mm (78.74mil)	PTH	Round	Pad
✳	6	3.200mm (125.98mil)	PTH	Round	Pad
◇	1	6.000mm (236.22mil)	PTH	Round	Pad
364 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

PCB SIDE	2 SIDE	COPPER-CLAD	2oz
PCB THICKNESS	1,6 +/-0,15mm	PCB MATERIAL	FR-4 HAL
SAFETY	UL94V-0	PTH	> 0.8mil



A.1	New	2022-12-22	GRA-3000L-M1-Power
Rev	Modify No:	Date	
Technical Specification			
Designer			
Normalizer			Item Code
Approver			040.02.0015

ZONERGY

1

2

3

4

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 B1 of General PCB specification document  
4.PCB general performance specification PC-6012  
5.PCB acceptance specification PC-A-600

b.Dr 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 exist,it is allowed to add choked-flow pieces(over 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 durning 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:GRA-3000L-M1-Power

SPELL: 1X2

# PCB Material:FR4

# TG2=>150°

Silkscreen color: WHITE

Electrical test: Yes

# Surface Finishing:HAL Lead Free

# CT2=>175V

Blind or Buried via: No

Plate at board edge: No

# Soldermask Color:Green

# SAFETY MARK:UL94V-0

V-CUT: Yes

Gold finger: No

# Green Technology:Plug Hole( Hole>0.5Cover all)

忽略焊盘上的丝印

Copper thickness of plated via, through 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:GD1	
Keepout Layer	File:GKO	
NC DRILL	File-RoundHoles.TXT File-SlotHoles.TXT	DRILL

Arrangement of layers (Dielectric strength:2KV)

Top Foundation copper  
2oz  
Bottom Core/Pre-preg=>0.4mm  
2oz  
Finished Board Thickness: 1.6mm+/-0.15mm

Figure 1: Arrangement of layers

Figure 2: V-CUT

Width:0.5mm  
+/-0.1mm  
V/-0.1mm

Table 1:Definition of layers

Drill Guide

A.1	New	2022-12-22	GRA-3000L-M1-Power	ZONERGY
Rev	Modify No:	Date		
TechnicalSpecification				
Designer				
Normalizer			Item Code	
Approver			040.02.0015	

1

2

3

4