



# 物料规格书

料号：040.02.0018		物料描述：PCB GRA-3000L-M1 Control A.1版 FR-4 2层 2oz 单板 147.5*145*1.6mm 1*2拼板 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：			

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. Test 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-6012  
5. PCB acceptance specification PC-A-600
- b. DfR 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 scoring 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 chisel-flare piece (inner layer) and auxiliary electroplate piece (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 "NPTH" please refer to

3. PARAMETERS

PCB Name: GRA-3000L-M1-Control	SPELL: TX2	# PCB Material: FR4	# TG: 150°
Silkscreen color: WHITE	Electrical tests: Yes	# Surface Finishing: HAL, Lead Free	# CTB: 175V
Brid 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 (金板通孔)	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
Silkscreen TOP	File: GTO	Positive
Silkscreen BOTTOM	File: GBO	Positive
Soldermask TOP	File: GTS	Negative
Soldermask BOTTOM	File: GBS	Negative
Drill Drawing Through	File: GD1	
Drill Guide Through	File: GD1	
NC DRILL	File: RoundHoles.TXT	DRILL
	File: SolderHoles.TXT	

Table 1: Definition of layers

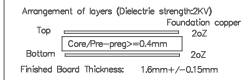


Figure 1: Arrangement of layers

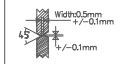
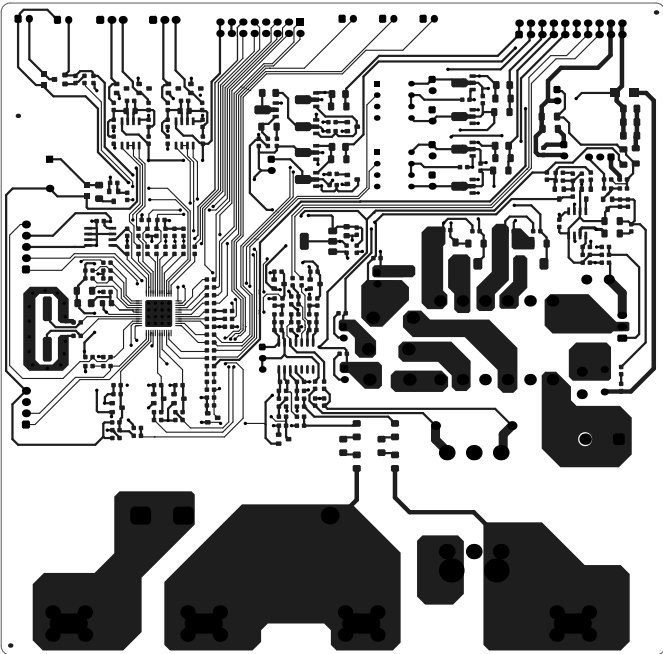


Figure 2: V-CUT



COMP TRACE

A.1	New	2022-12-22	GRA-3000L-M1-Control	ZONERGY
Rev	Modify No:	Date		
Technical Specification				
Designer				
Normalizer				
Approver			Item Code	
			040.02.0018	

1.GENERAL

a.Technical requirements are prioritized as follows:

- 1.Test 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-6012
- 5/PCB acceptance specification PC-A-600

b. D&B boards using drill data,d&B 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 scoring 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 chisel-flare piece(inner layer) and auxiliary electroplate piece(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"if"please refer to

3.PARAMETERS

PCB Name:GRA-3000L-M1-Control	SPELL: 1X2	# PCB Material:FR4	# TG>=150°
Silkscreen color: WHITE	Electrical test: Yes	# Surface Finishing:HAL Lead Free	# CTO>=175V
Brid 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 (金板通孔)	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:GDI	
Drill Guide Through	File:GGI	
NC DRILL	File-RoundHoles.TXT	DRILL
	File-SolHoles.TXT	

Table 1:Definition of layers

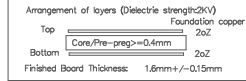


Figure 1: Arrangement of layers

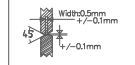
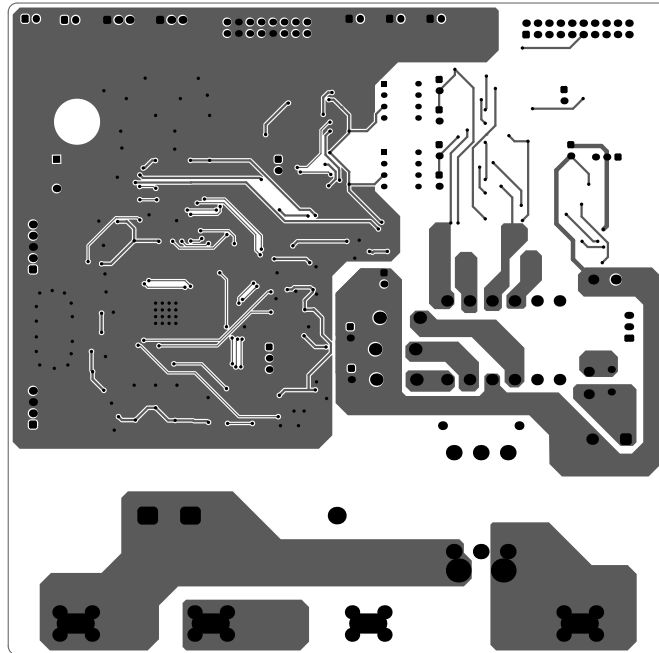


Figure 2: V-CUT



2022-12-22

A.1	New	2022-12-22	GRA-3000L-M1-Control
Rev	Modify No:	Date	
TechnicalSpecification			Item Code
Designer			
Normalizer			040.02.0018
Approver			

ZONERGY

1.GENERAL

- a. Technical requirements are prioritized as follows:  
1. Test 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-6012  
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 scoring 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 chisel-flare piece (inner layer) and auxiliary electroplate piece (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 "NPTH" please refer to

3. PARAMETERS

PCB Name: GRA-3000L-M1-Control	SPELL: TX2	# PCB Material: FR4	# TG: =150°
Silkscreen color: WHITE	Electrical test: Yes	# Surface Finishing: HAL Lead Free	# CTB: =175V
Bird 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 (金板孔)	是略焊盘上的丝印
Copper thickness of plated via, through hole or edge >=25um			

4. VIEW

LAYER	Gerber files	NOTES
TOP	File: GTL	Positive
BOTTOM	File: GBL	Positive
Silkscreen TOP	File: GTO	Positive
Silkscreen BOTTOM	File: GBO	Positive
Soldermask TOP	File: GTS	Negative
Soldermask BOTTOM	File: GBS	Negative
Drill Drawing Through	File: GD1	
Drill Guide Through	File: GD1	
NC DRILL	File: Roundholes.TXT	DRILL
	File: Sdrholes.TXT	

Table 1: Definition of layers

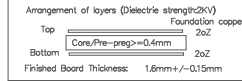


Figure 1: Arrangement of layers

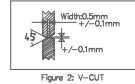
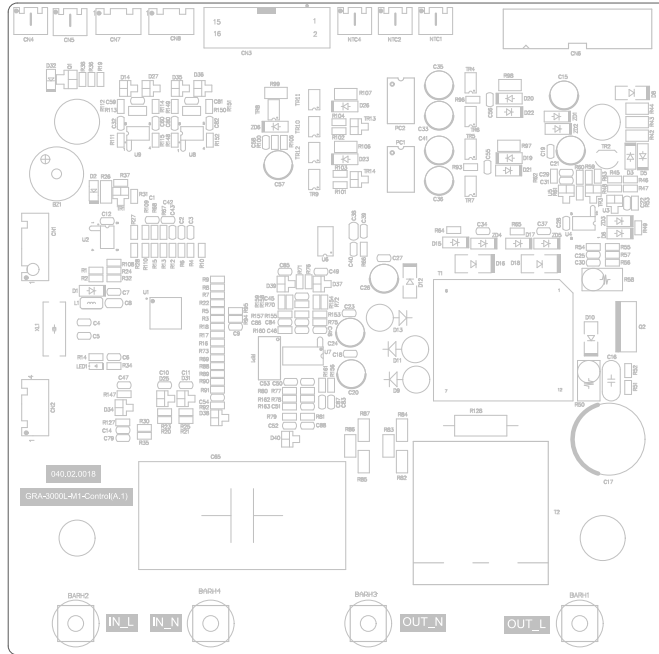


Figure 2: V-CUT



COMP TEXT

A.1	New	2022-12-22	GRA-3000L-M1-Control	ZONERGY
Rev	Modify No:	Date		
Technical Specification			Item Code	
Designer				
Normalizer			040.02.0018	
Approver				

1.GENERAL

a.Technical requirements are prioritized as follows:

- 1.Test 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-6012
- 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 scoring 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 chisel-flare piece (inner layer) and auxiliary electroplate piece (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 "if" please refer to

3.PARAMETERS

PCB Name: GRA-3000L-M1-Control	SPELL: TX2	# PCB Material: FR4	# TG: >=150°
Silkscreen color: WHITE	Electrical tests: Yes	# Surface Finishing: HAL Lead Free	# CTO: >=175V
Brid 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 (金板通孔)	是啥焊盘上的丝印
Copper thickness of plated via, through hole or edge >=25um			

4. VIEW

LAYER	Gerber files	NOTES
TOP	File: GTL	Positive
BOTTOM	File: GBL	Positive
Silkscreen TOP	File: GTO	Positive
Silkscreen BOTTOM	File: GBO	Positive
Soldermask TOP	File: GTS	Negative
Soldermask BOTTOM	File: GBS	Negative
Drill Drawing Through	File: GDI	
Drill Guide Through	File: GDI	
NC DRILL	File: Roundholes.TXT File: Sdrholes.TXT	DRILL

Table 1: Definition of layers

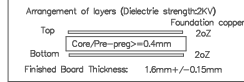


Figure 1: Arrangement of layers

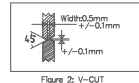
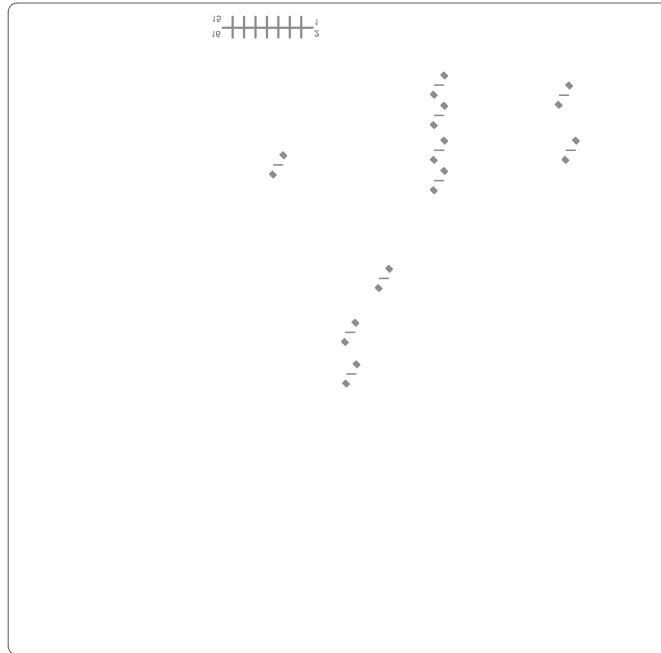


Figure 2: V-CUT



TEXT REVER

A.1	New	2022-12-22	GRA-3000L-M1-Control
Rev	Modify No:	Date	
Technical Specification			Item Code
Designer			
Normalizer			040.02.0018
Approver			

ZONERGY

1.GENERAL

- a. Technical requirements are prioritized as follows:  
1. Test 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-6012  
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 scoring 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 chisel-flare piece (inner layer) and auxiliary electroplate piece (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 "if" please refer to

3. PARAMETERS

PCB Name: GRA-3000L-M1-Control	SPELL: TX2	# PCB Material: FR4	# TG: =150°
Silkscreen color: WHITE	Electrical tests: Yes	# Surface Finishing: HAL, Lead Free	# CTB: =175V
Brid 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 (金板通孔)	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
Silkscreen TOP	File: GTO	Positive
Silkscreen BOTTOM	File: GBO	Positive
Soldermask TOP	File: GTS	Negative
Soldermask BOTTOM	File: GBS	Negative
Drill Drawing Through	File: GD1	
Drill Guide Through	File: GD1	
NC DRILL	File: Roundholes.TXT File: SdrHoles.TXT	DRILL

Table 1: Definition of layers

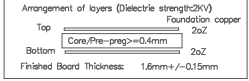


Figure 1: Arrangement of layers

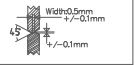
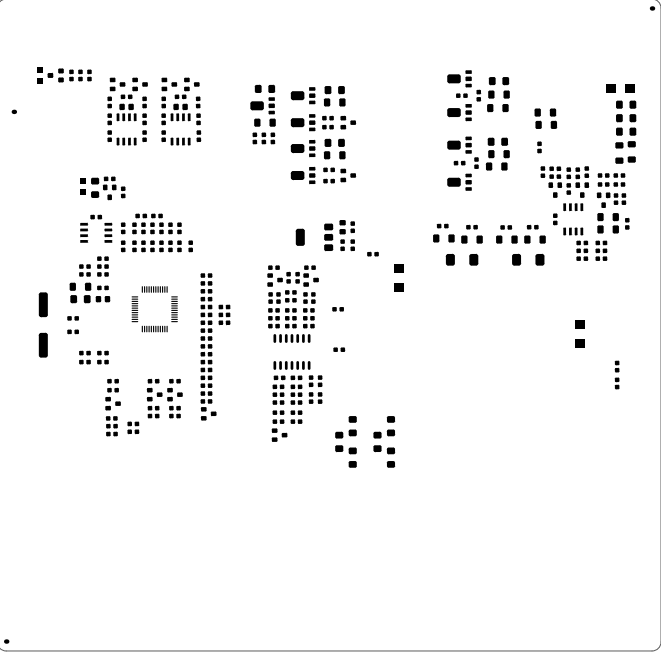


Figure 2: V-CUT



COMP PASTE

F	A.1	New	2022-12-22	GRA-3000L-M1-Control	ZONERGY
	Rev	Modify No:	Date		
	Technical Specification				
Designer			Item Code		
A4	Normalizer			040.02.0018	
	Approver				

## 1. GENERAL

- a. Technical requirements are prioritized as follows:
  - 1. First priority is given to this document;
  - 2. Relevant contract documents signed on by both parties;
  - 3. The Bill of General PCB specification document;
  - 4. PCB general performance specification PC-0012;
  - 5. PCB acceptance specification PC-A-600
- b. Drift occurs using drift detection pattern and schedule. Any conflict between the CAD data, the drawing, and the drift pattern shall be clarified with the design authority before proceeding.
- c. Compensation and scoring 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

- if there isn't enough NPTH in the PCB fabrication positioning 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 the vocanosis should be milled without bread(wire) should be removed after finishing the PCB fabrication)
- if positions vocanosis should be milled without bread(wire) should be removed after finishing the PCB fabrication) in the PCB,current-hogging shell copper may be added there.
- When fabricating multi-layer PCB auxiliary process edge is not allowed to add chiseled-flow pieces(inner layer) and auxiliary electrode pieces(outer layer) there.
- if the size of the PCB with process edge is larger than 100mm, the milling cutter milling in durning strip a half of milling cutter diameter is permitted.
- the via without soldermask window should be treated as plug.
- for all detail of those items beginning with "if" please refer to

## 3PARAMETERS

PCB Name:GRA-3000L-M1-Control	SPELL: 1X2	# PCB Material #FR4	# TCO>=150°
Silkscreen color: WHITE	Electrical test: Yes	# Surface Finishing:HAL Lead Free	# CTI>=175V
Blind or Buried via: No	Plote at board edge: No	# Soldermask Color:Green	# SAFETY MARK:UL94V-0
V-CUT: Yes	Gold finger: No	# Green Technology:Plug Hole [金板穿孔]	加姆焊盘上的丝印
Copper thickness of plated via, though hole or edge >=25um			

4. VIEW ☐

LAYER	Gerber files	NOTES
TOP	File:CTL	Positive
BOTTOM	File:GBL	Positive
SilkscreenTOP	File:GTO	Positive
SilkscreenBOTTOM	File:GTS	Positive
SoldermaskTOP	File:GBO	Negative
SoldermaskBOTTOM	File:GBS	Negative
Drill DrawingThrough	File:GD1	
Drill Guide Through	File:GD1	
NC DRILL	File--Round-holes.TXT File--Slot-holes.TXT	DRILL

Table 1: Definition of layers

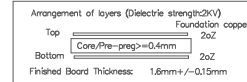


Figure 1: Arrangement of layers

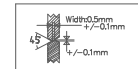


Figure 2: V-CUT

SOLDER PASTE

F	A.1	New	2022-12-22	GRA-3000L-M1-Control	ZONERGY
	Rev	Modify No:	Date		
	Technical Specification				
	Designer			Item Code	
	Normalizer			040.02.0018	
A.4	Approver				

1.GENERAL

a.Technical requirements are prioritized as follows:

- 1.Test 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-6012
- 5/PCB acceptance specification PC-A-600

b. D&B boards using drill data,d&B 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

- 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)
- 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.
- When fabricating multi-layer PCBs,if auxiliary process edge exist,it is allowed to add chisel-flow piece(inner layer) and auxiliary electroplate piece(outer layer) there.
- 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.
- The via without soldermask window should be treated as plugged.
- All for detail of those items beginning with"if"please refer to

3.PARAMETERS

PCB Name:GRA-3000L-M1-Control	SPELL: TX2	# PCB Material:FR4	# TG>=150°
Silkscreen color: WHITE	Electrical test: Yes	# Surface Finishing:HAL Lead Free	# CTO>=175V
Brid 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 (金板通孔)	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:GDI	
Drill Guide Through	File:GGI	
NC DRILL	File-RoundHoles.TXT File-SlotHoles.TXT	DRILL

Table 1:Definition of layers

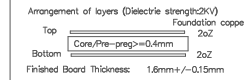


Figure 1: Arrangement of layers

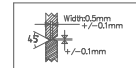
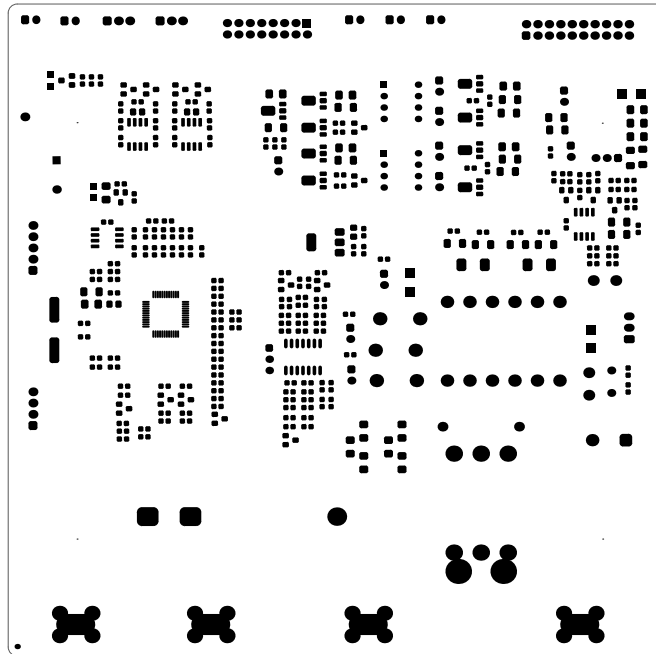


Figure 2: V-CUT



COMP MASK

A.1	New	2022-12-22	GRA-3000L-M1-Control
Rev	Modify No:	Date	
TechnicalSpecification			Item Code
Designer			
Normalizer			040.02.0018
Approver			

ZONERGY

1.GENERAL

a.Technical requirements are prioritized as follows:

- 1.Test 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-6012
- 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 chisel-flare piece (inner layer) and auxiliary electroplate piece (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 "if" please refer to

3.PARAMETERS

PCB Name: GRA-3000L-M1-Control	SPELL: TX2	# PCB Material: FR4	# TG: =150°
Silkscreen color: WHITE	Electrical tests: Yes	# Surface Finishing: HAL Lead Free	# CTB: =175V
Brid 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 (金板通孔)	# 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
Silkscreen TOP	File: GTO	Positive
Silkscreen BOTTOM	File: GBO	Positive
Soldermask TOP	File: GTS	Negative
Soldermask BOTTOM	File: GBS	Negative
Drill Drawing Through	File: GD1	
Drill Guide Through	File: GD1	
NC DRILL	File: RoundHoles.TXT File: SolderHoles.TXT	DRILL

Table 1: Definition of layers

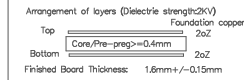


Figure 1: Arrangement of layers

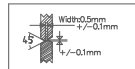
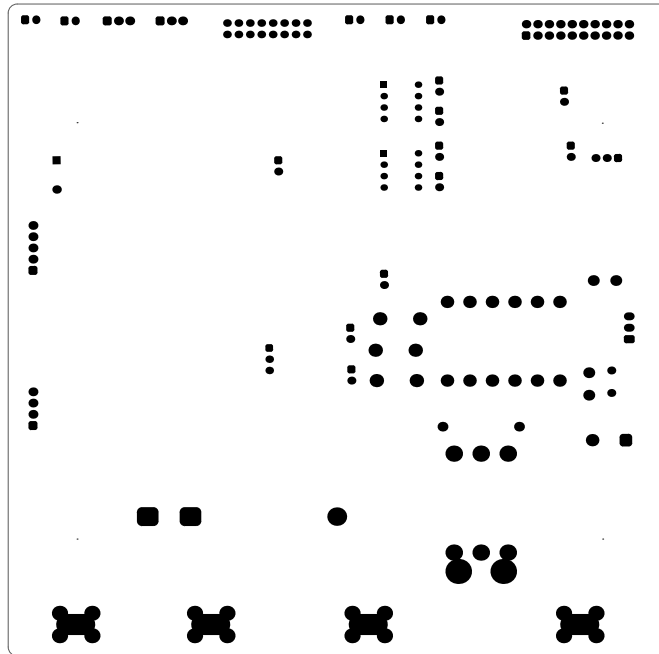


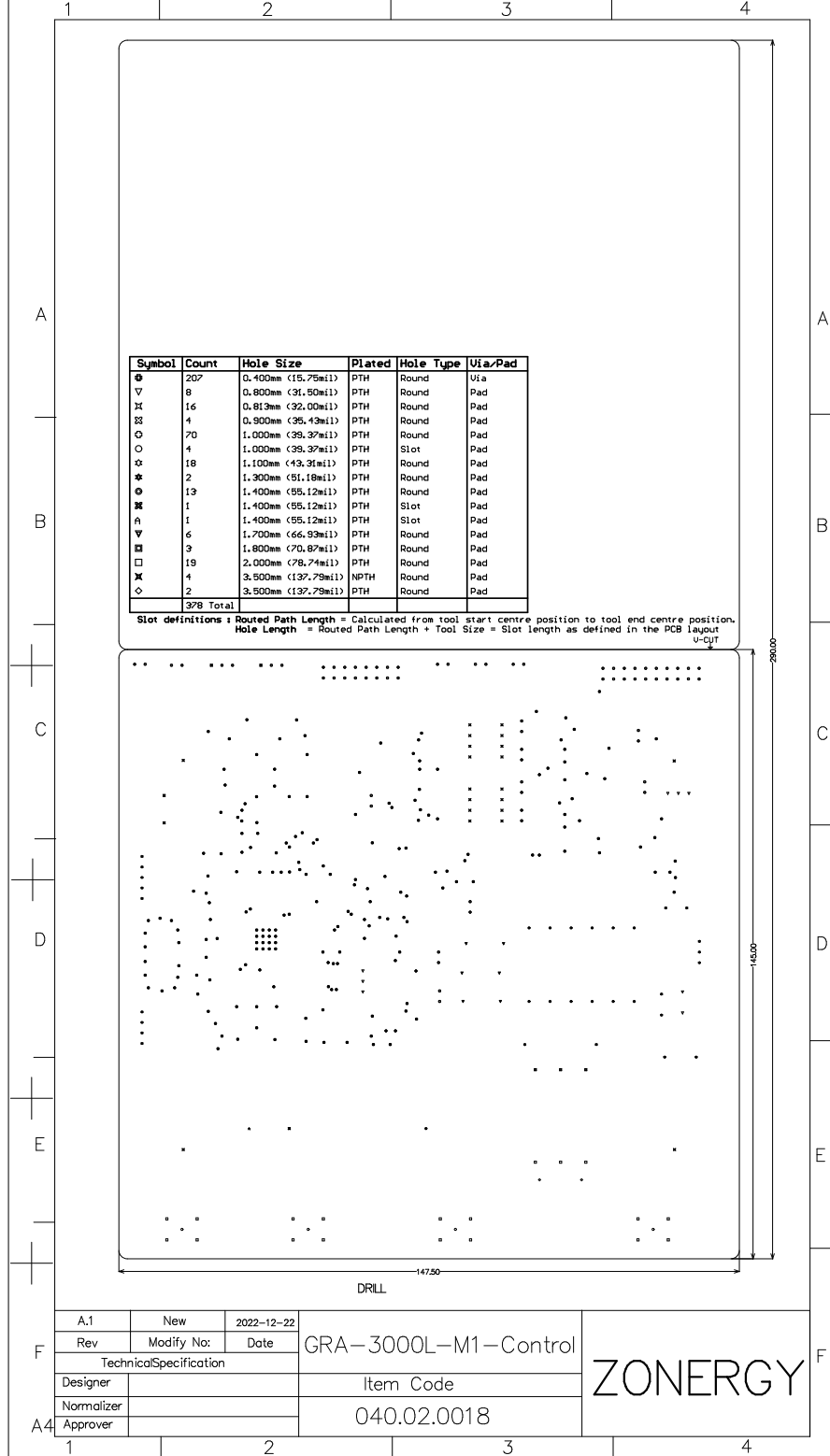
Figure 2: V-CUT



K2AM RECL102

A.1	New	2022-12-22	GRA-3000L-M1-Control
Rev	Modify No:	Date	
Technical Specification			Item Code
Designer			
Normalizer			040.02.0018
Approver			

ZONERGY



1.GENERAL

- a. Technical requirements are prioritized as follows:  
1. Test 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-6012  
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 chisel-flow piece (inner layer) and auxiliary electrode piece (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 "if" please refer to

3. PARAMETERS

PCB Name: GRA-3000L-M1-Control	SPELL: TX2	# PCB Material: FR4	# TG: =150°
Silkscreen color: WHITE	Electrical test: Yes	# Surface Finishing: HAL Lead Free	# CTB: =175V
Buried 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 (金板通孔)	是略焊盘上的丝印
Copper thickness of plated via, through hole or edge >=25um			

4. VIEW

LAYER	Gerber files	NOTES
TOP	File: GTL	Positive
BOTTOM	File: GBL	Positive
Silkscreen TOP	File: GTO	Positive
Silkscreen BOTTOM	File: GBO	Positive
Soldermask TOP	File: GTS	Negative
Soldermask BOTTOM	File: GBS	Negative
Drill Drawing Through	File: GDI	
Drill Guide Through	File: GDI	
NC DRILL	File: RoundHoles.TXT File: SdrHoles.TXT	DRILL

Table 1: Definition of layers

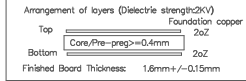


Figure 1: Arrangement of layers

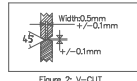
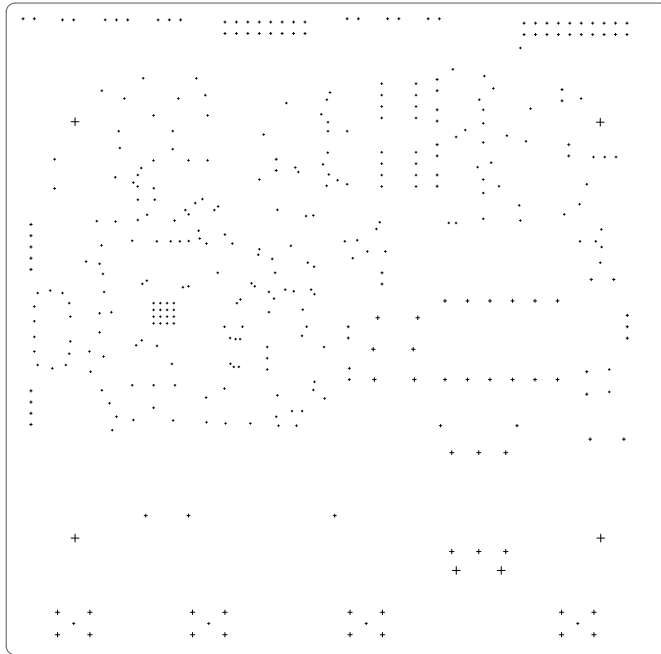


Figure 2: V-CUT



Drill Guide

A.1	New	2022-12-22	GRA-3000L-M1-Control	ZONERGY
Rev	Modify No:	Date		
Technical Specification			Item Code	
Designer				
Normalizer			040.02.0018	
Approver				