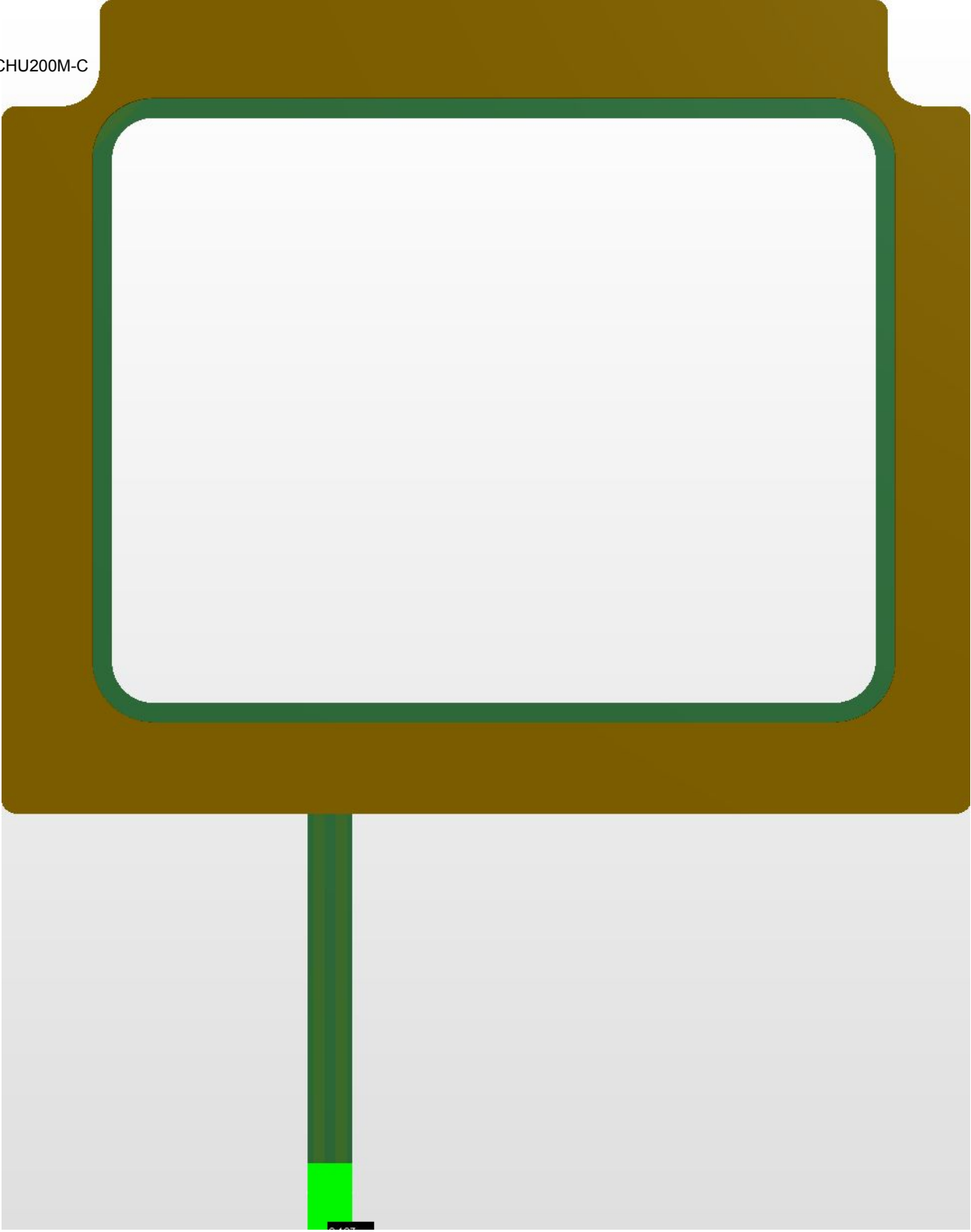
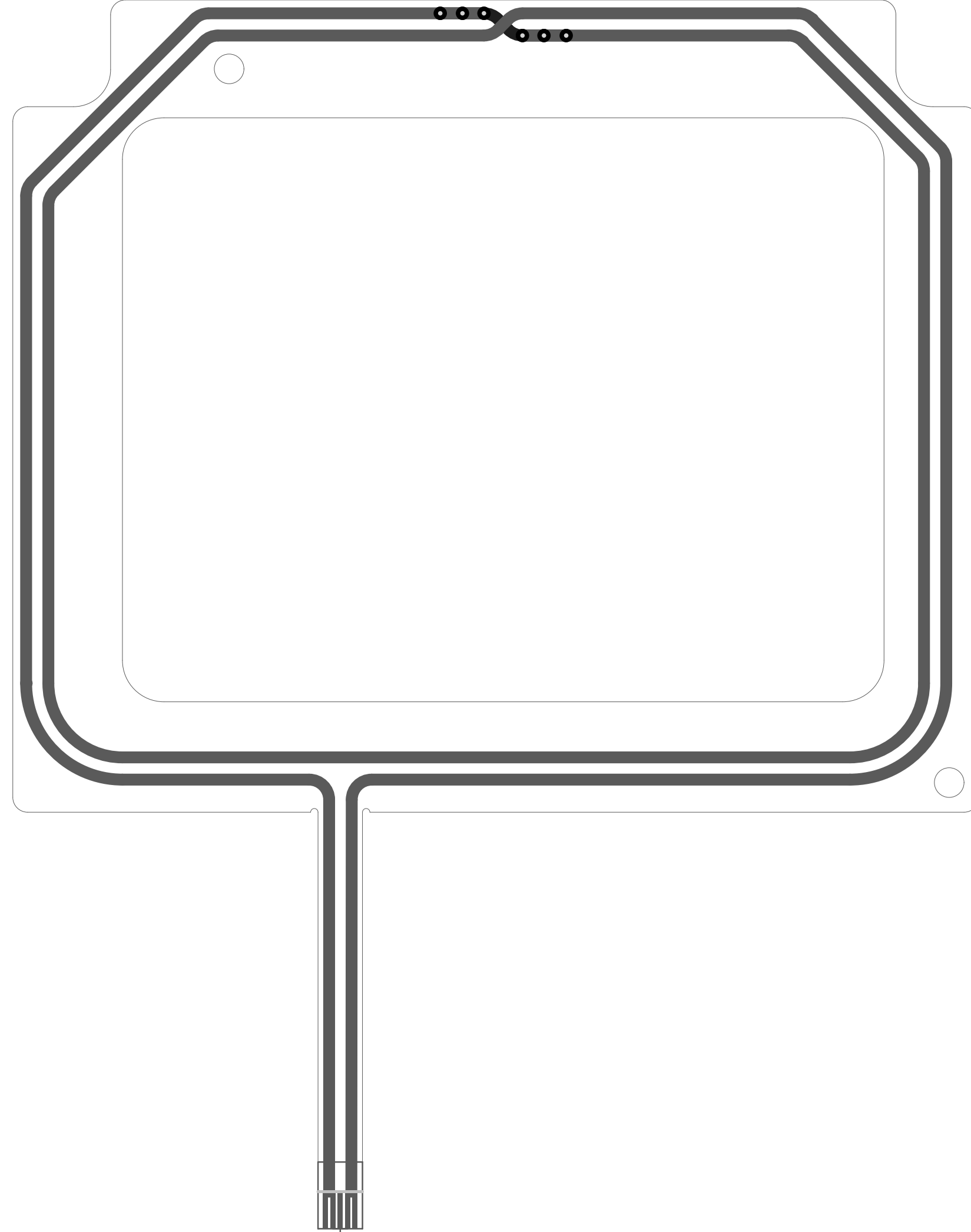






MINIMUM ARTWORK FEATURE SIZE

Track to track:	0.7	mm	0.028	inches
Track to pad:	0.15	mm	0.006	inches
Track width:	0.8	mm	0.031	inches
Drill Hole Size:	0.3	mm	0.012	inches

BOARD TYPE

- ☐ Single Layer, Flex
- ☒ Double layer, Flex
- ☐ Multi Layer,
- ☐ HDI Multi Layer
- ☐ Embedded PCB Resistor

VIA CONSTRUCTION

- ☐ Blind Vias
- ☐ Buried Vias
- ☐ HDI Micro Vias
- ☐ Stacked HDI Micro Vias

GENERAL NOTES:

1. Finished Board Thickness and Layer Stackup shown in View 'A'
2. All gerber files are in RS274X Metric 4:4 format with embedded apertures.
3. Board shall be fabricated as per IPC-6012B guidelines (unless otherwise stated).
4. ~~Vias are to be epoxy filled and Tented (unless otherwise stated)~~
5. All hole sizes described in this Document are Finished Hole Sizes (FHS).
6. No silkscreen allowed on exposed lands or in holes.
7. Finished boards shall not have nicks, scratches, voids, poor plating or mis-drilled holes
8. The PCB is to be 100% Electrically Tested.
9. No copper to be added or removed from the design, except that which is stated above.
- 10.
- 11.
- 12.
- 13.

MATERIALS

PCB / Panel Size: 65.2x82.9mm
PCBs Per Panel:
Laminate : Polyimide Dielectric $\epsilon_r = 3.5$

Thickness: 0.3mm +/- 10% After Plating
Flammability : UL94-V0 at FPC Tail

PROFILE METHOD

- ☐ Routing, 2mm Rout Path
- ☐ V-Score 30 Deg

ACCEPTABILITY

- ☐ IPC-3-300
- ☒ IPC-A-600 Class 2
- ☐ IPC-A-600 Class 3
- ☒ UL Approval

SURFACE MOUNT

- ☐ Single Sided
- ☐ Double Sided

COVER LAYER:

POLYIMIDE FILM 25um
 ■ Top Coverlay
 ■ Bottom Coverlay

SCREEN PRINT OVERLAYS:

- ☐ Top Overlay (White)
☒ Bottom Overlay (White)
- TRACK FINISH:**
- ☐ Hot air levelled
☒ Immersion Gold over Nickel (1u")
☐ OSP (Organic Solder Preservatives)
☐ Silver Immersion (6-20u")
☐ Hard Gold

- Rolled-annealed (RA)
 □ Half Hard Electro-deposited (ED)

PRODUCTION FILE LIST

	DESCRIPTION	FILE
↓ ⇕ LAMINATION SEQUENCE	■ TOP FR4 Stiffener	GM9
	■ TOP PI Stiffener	GM10
	■ TOP FLEX COPPER LAYER	GTL
	■ BOTTOM FLEX COPPER LAYER	GBL
	■ BOTTOM COVERLAY OPENING	GBS
	■ BOTTOM OVERLAY	GB0
	□	
	□	
	□	
	□	
⇕ ⇕	■ BOARD OUTLINE	GM1
	□ ROUTE OUTLINE	GM2
	□ V-SCORE OUTLINE	GM3
	□ PASTE SHIM OUTLINE	GM14
	□ TOP PASTE MASK	GTP
	□ BOTTOM PASTE MASK	GBP
	■ APERTURE FILE	APR
	■ DRILL DRAWING	GDx
	■ DRILL GUIDE	GGx
	■ NC DRILL – TOOL FILE	DRL
	■ NC DRILL – TEXT FILE	TXx
	■ NC DRILL – BINARY FILE	DRx

VIEW 'A' : LAMINATION STACK-UP STRUCTURE

0.30mm

Layer :

- FR4 Stiffener (GM9)
- PI Stiffener (GM10)
- Top Coverlay
- Double-Sided Base Lay
- Bottom Coverlay
- Bottom Overlay (White)

Thickness :

- 0.5mm
- PI + AD
- 25um PI + AD
- 1oz RA Cu + 25um PI + 1oz RA Cu
- 25um PI + AD

NOTES:

1. Finished PCB thickness tolerance +/- 10%
2. Finished PCB board outline tolerance +/- 0.1mm

CONTROLLED IMPEDENCE

None

DRILL TABLE

Symbol	Count	Hole Size	Plated	Hole Type
O	6	11.81mil (0.300mm)	PTH	Round
	6 Total			