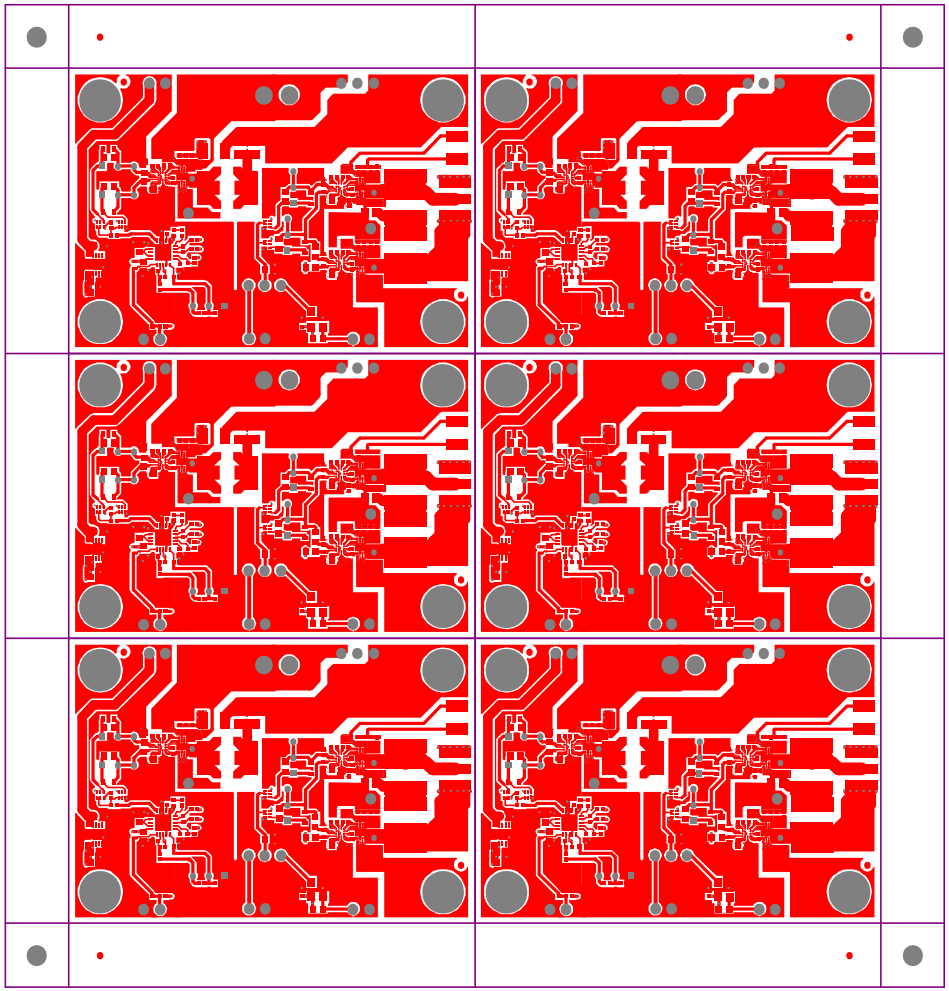
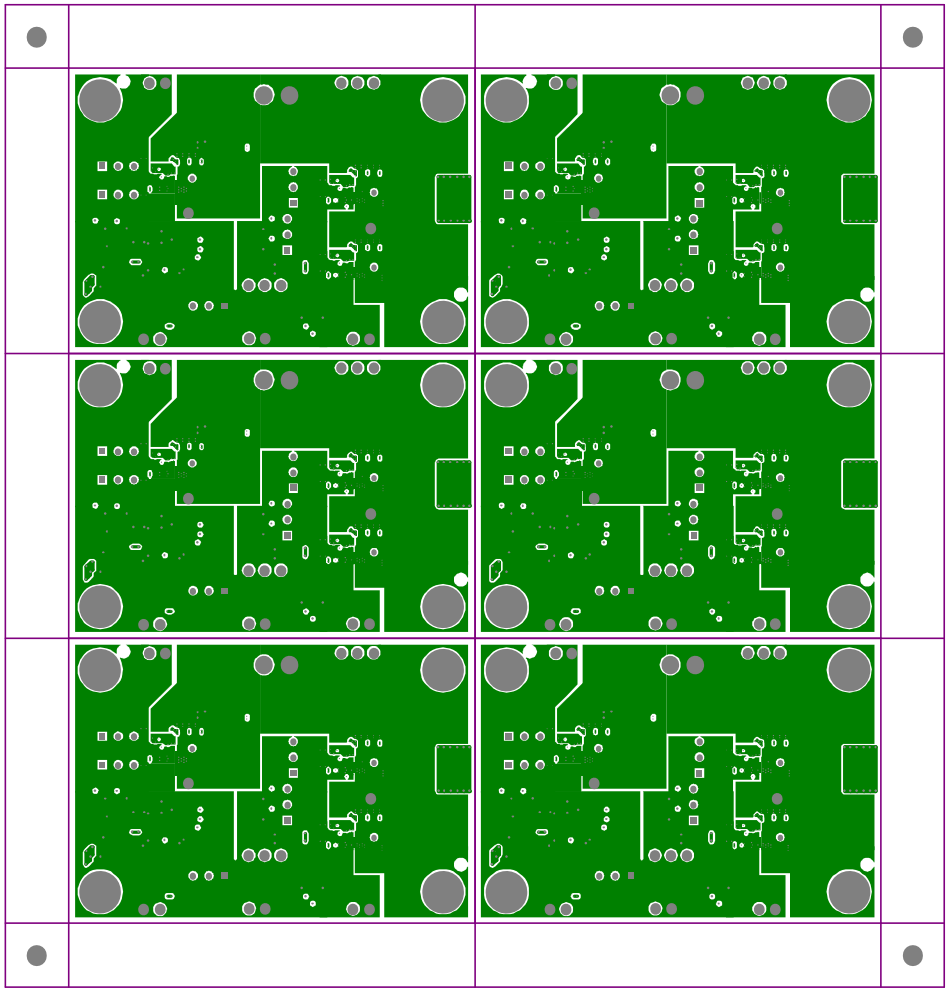


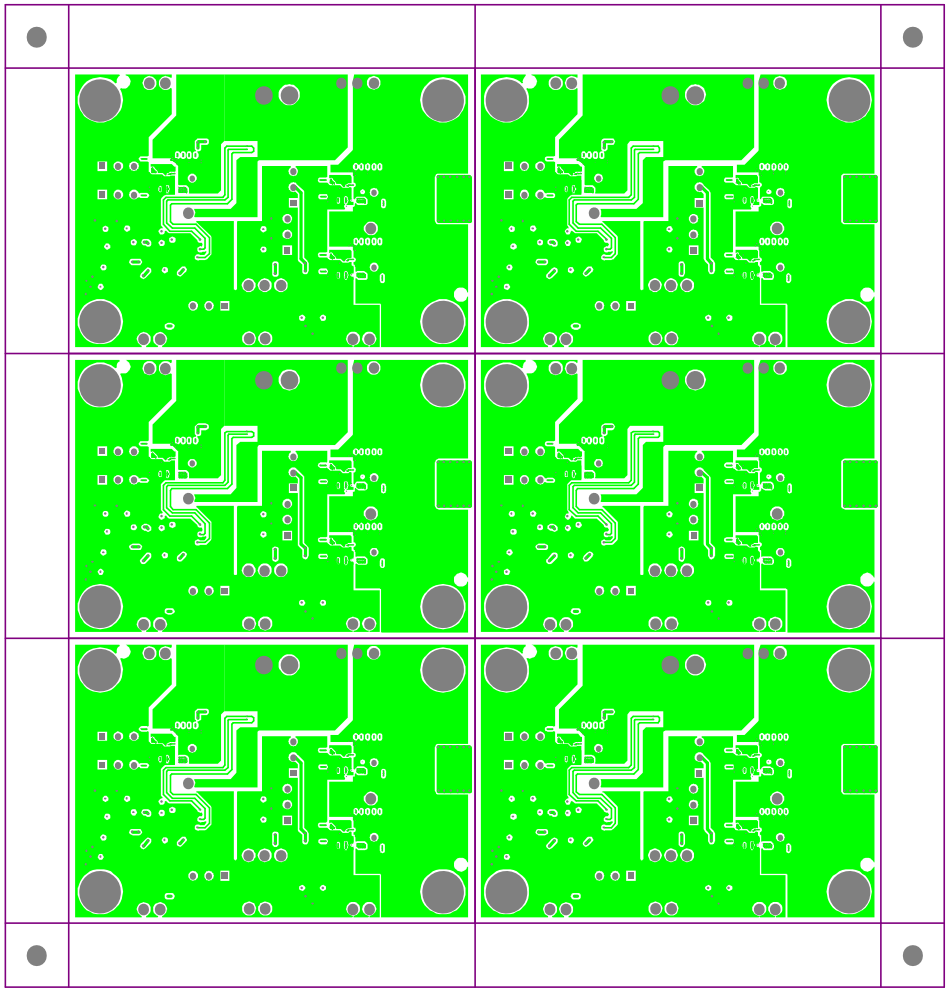
Top Layer
Board Outline

Efficient Power Conversion Corp. Proprietary and Confidential



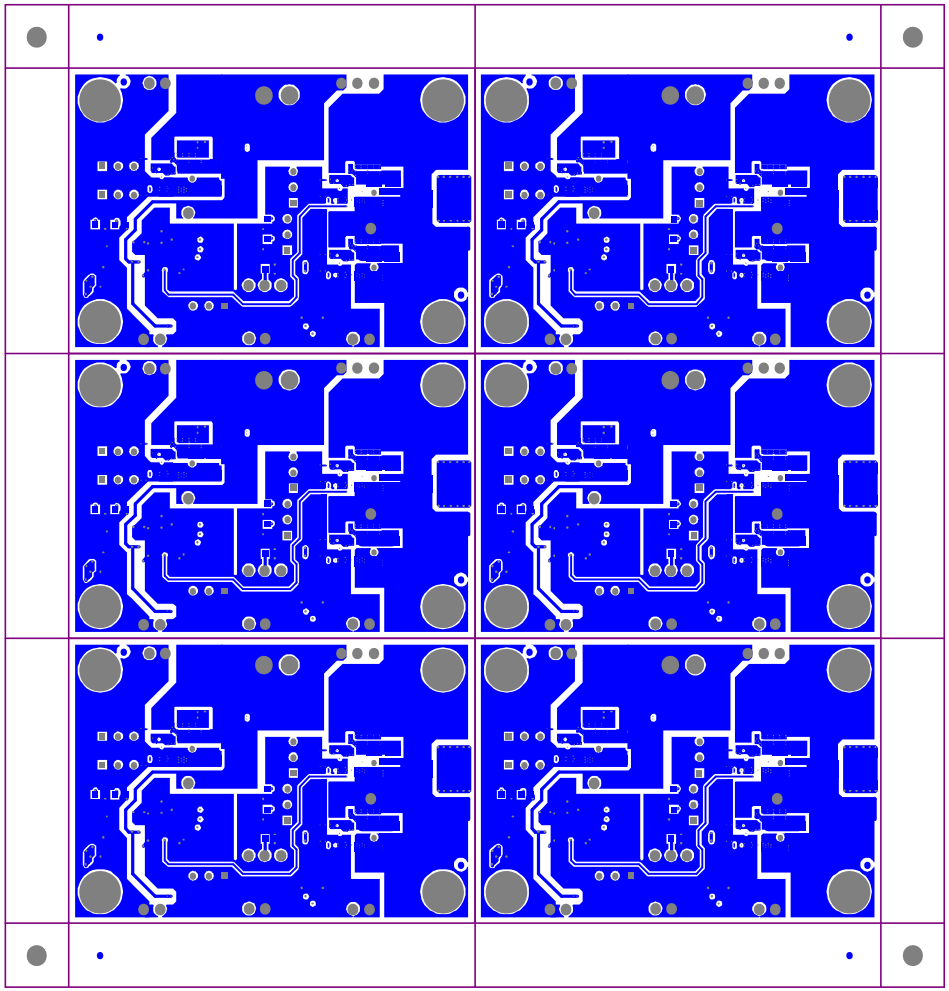
Mid-Layer 1
Board Outline

Efficient Power Conversion Corp. Proprietary and Confidential



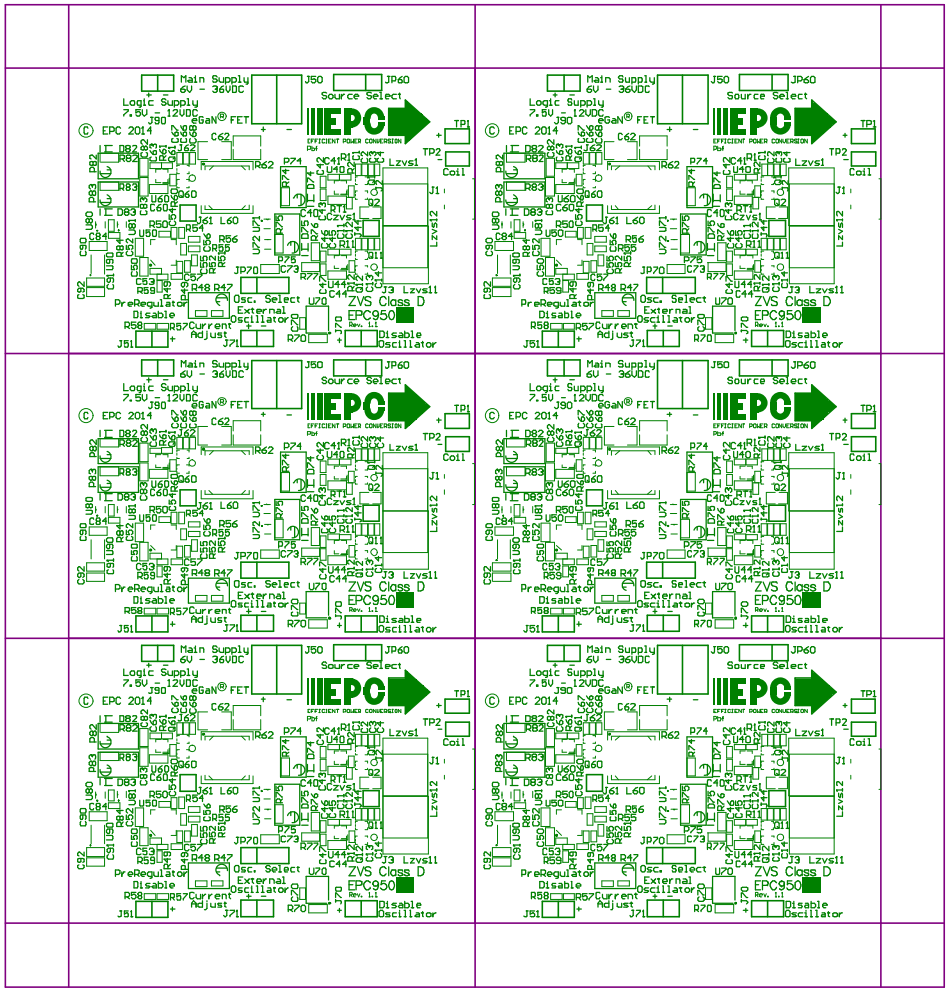
Mid-Layer 2
Board Outline

Efficient Power Conversion Corp. Proprietary and Confidential



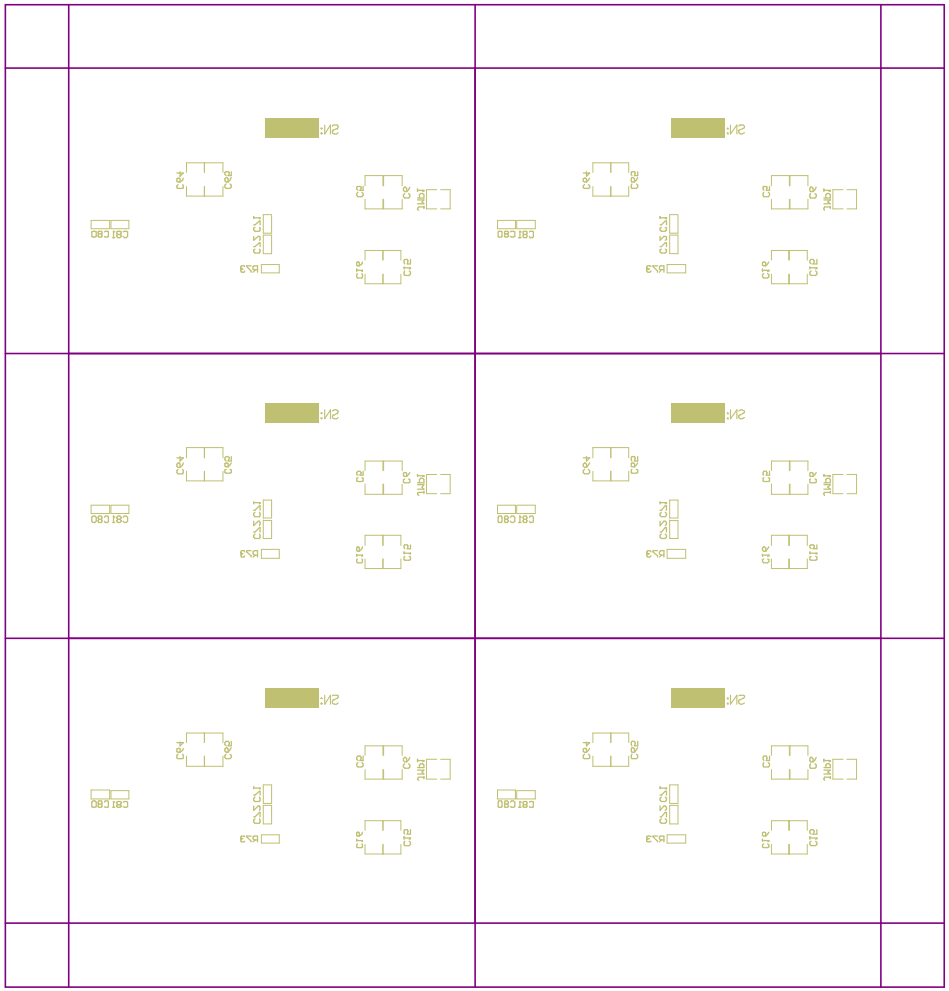
Bottom Layer
Board Outline

Efficient Power Conversion Corp. Proprietary and Confidential



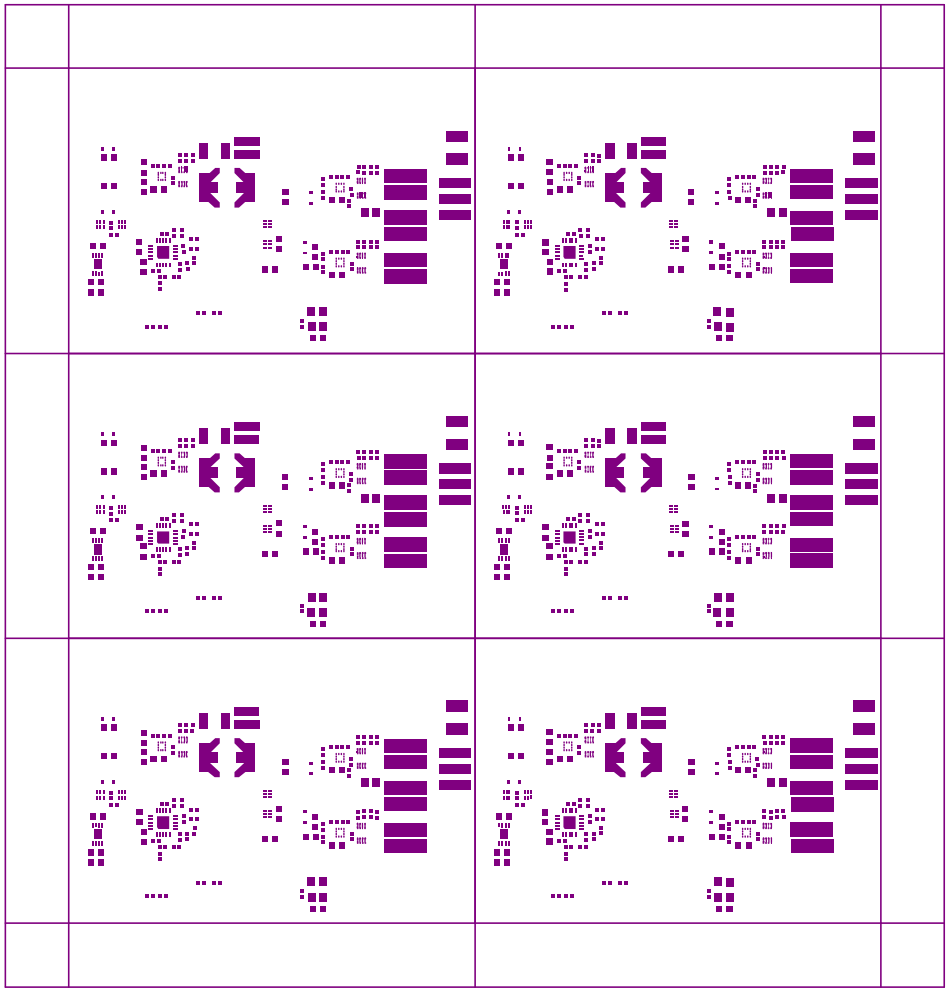
Top Silk
Board Outline

Efficient Power Conversion Corp. Proprietary and Confidential



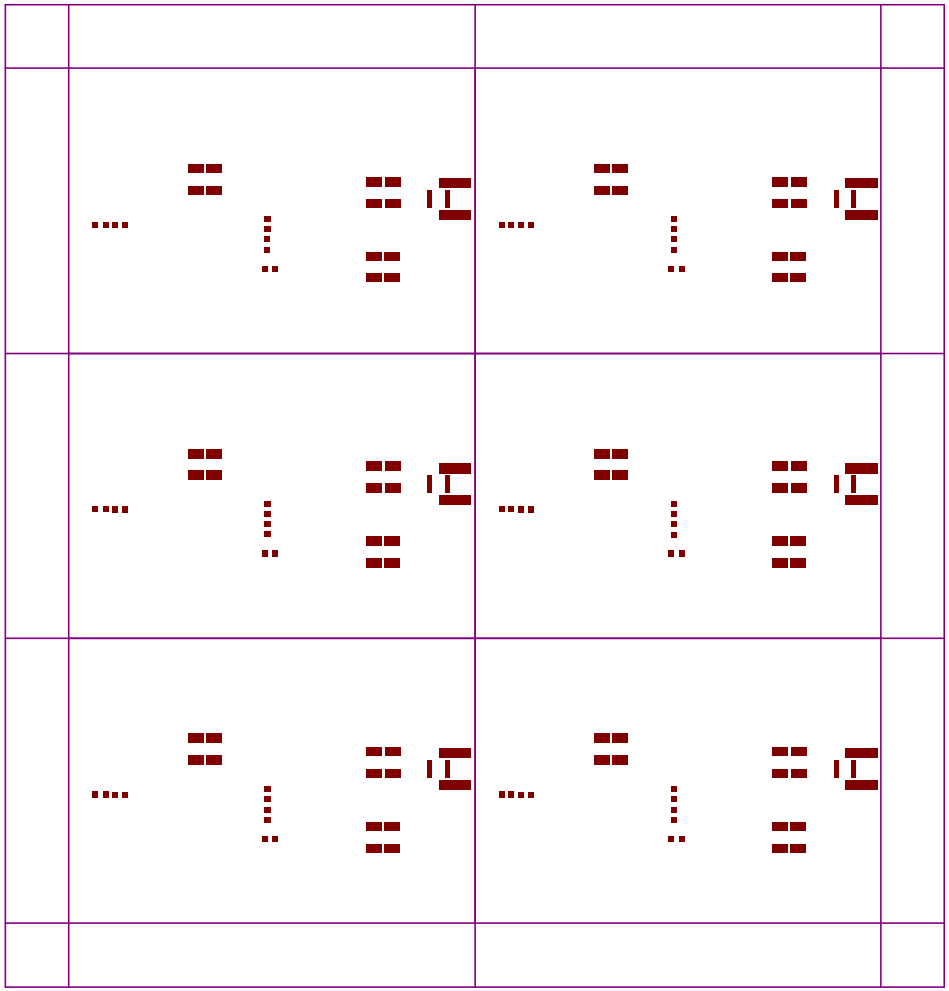
Bottom Silk
Board Outline

Efficient Power Conversion Corp. Proprietary and Confidential



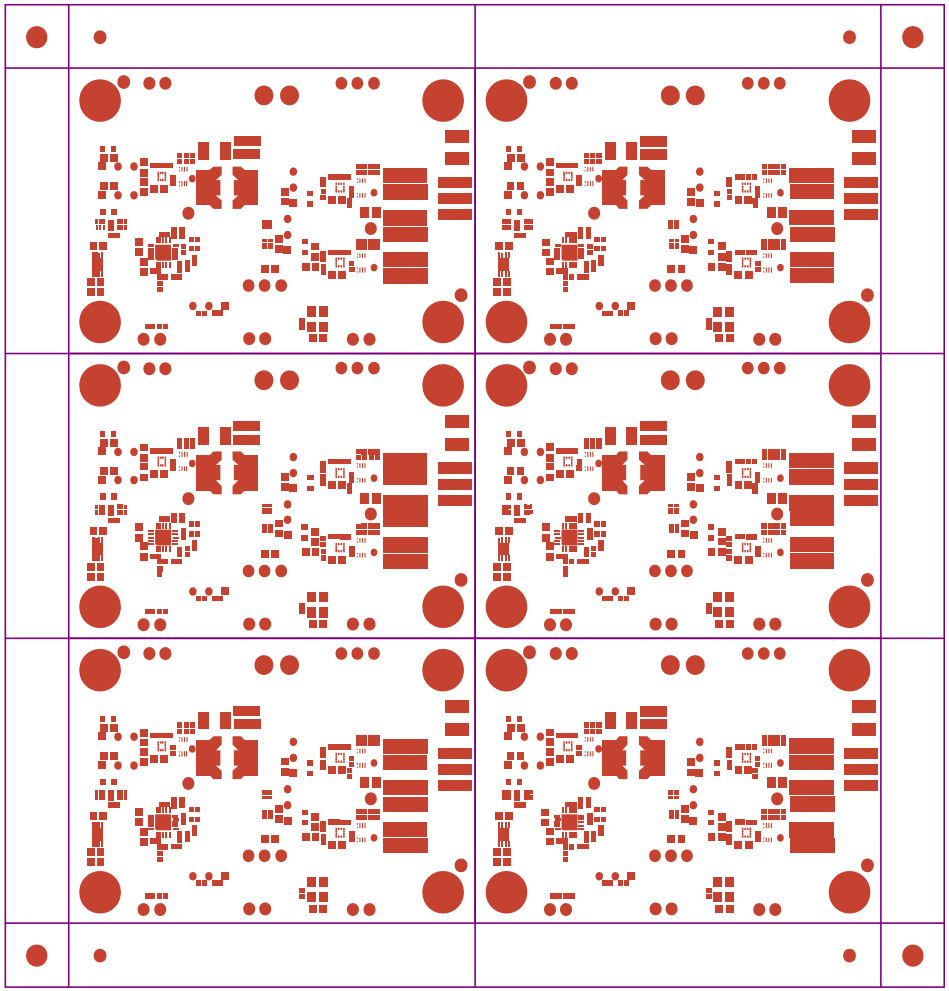
Top Paste
Board Outline

Efficient Power Conversion Corp. Proprietary and Confidential



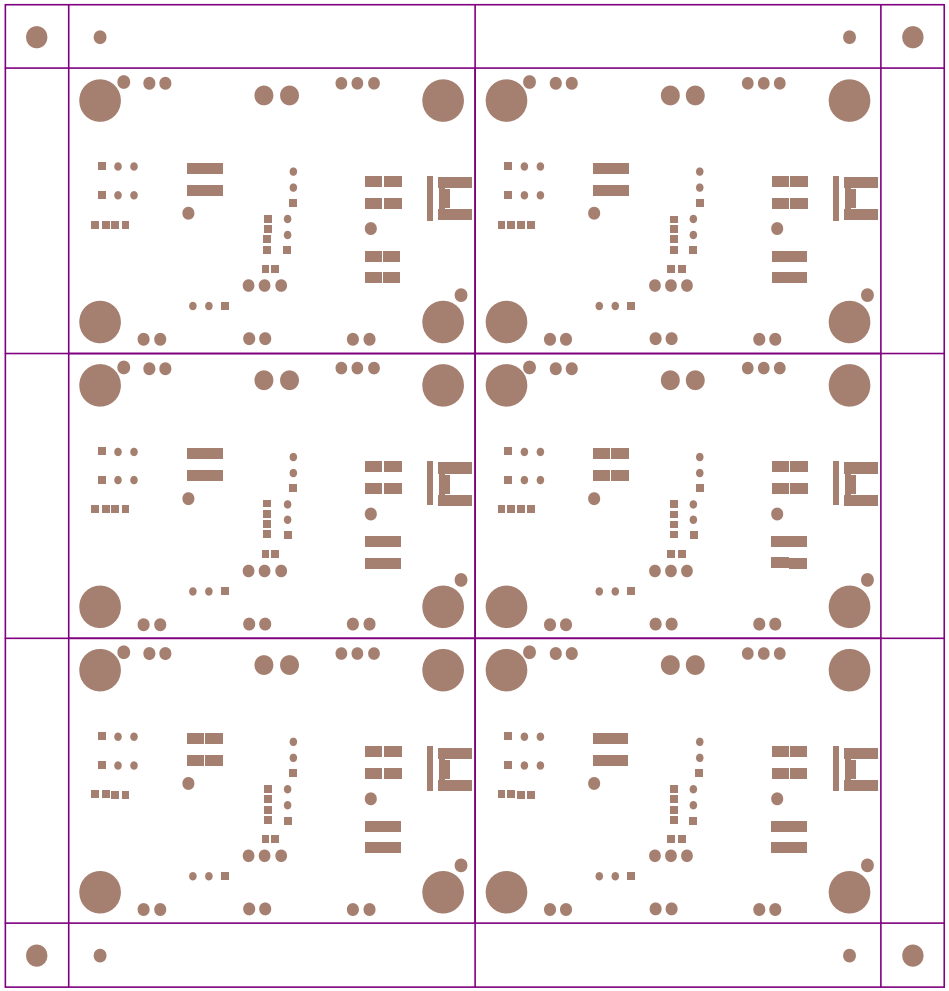
Bottom Paste
Board Outline

Efficient Power Conversion Corp. Proprietary and Confidential



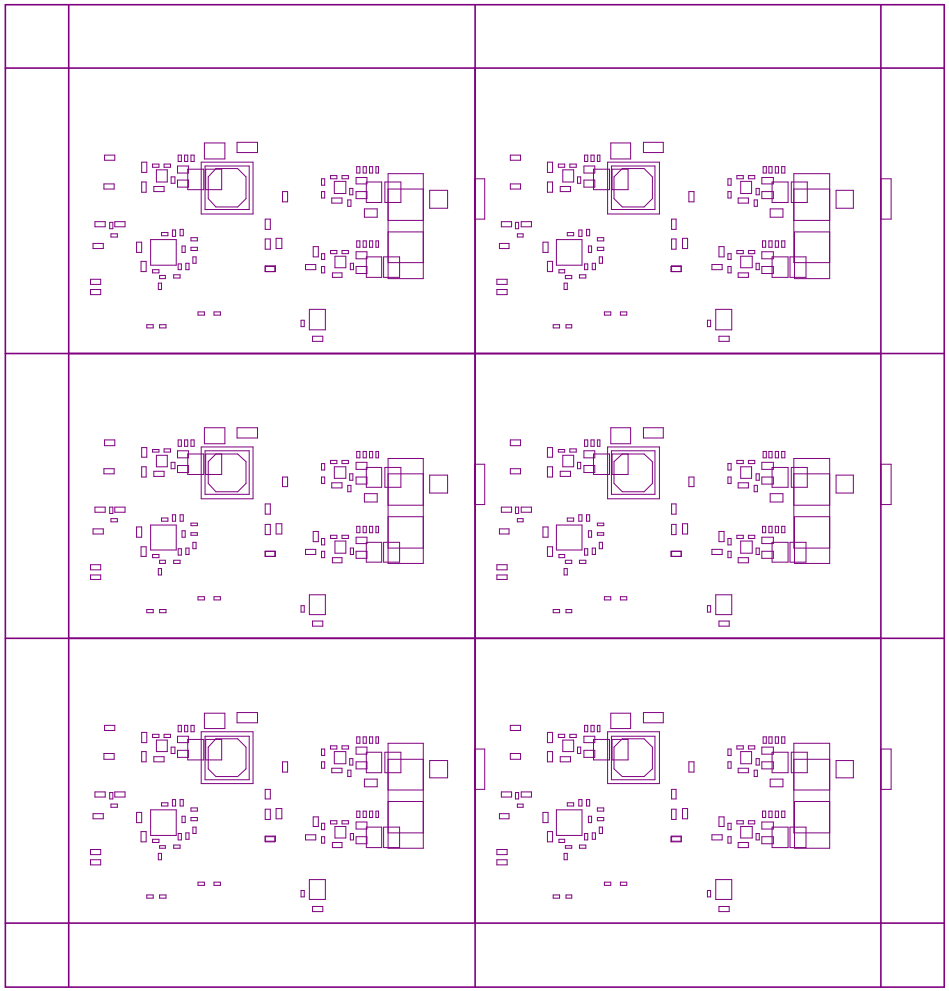
Top Mask
Board Outline

Efficient Power Conversion Corp. Proprietary and Confidential



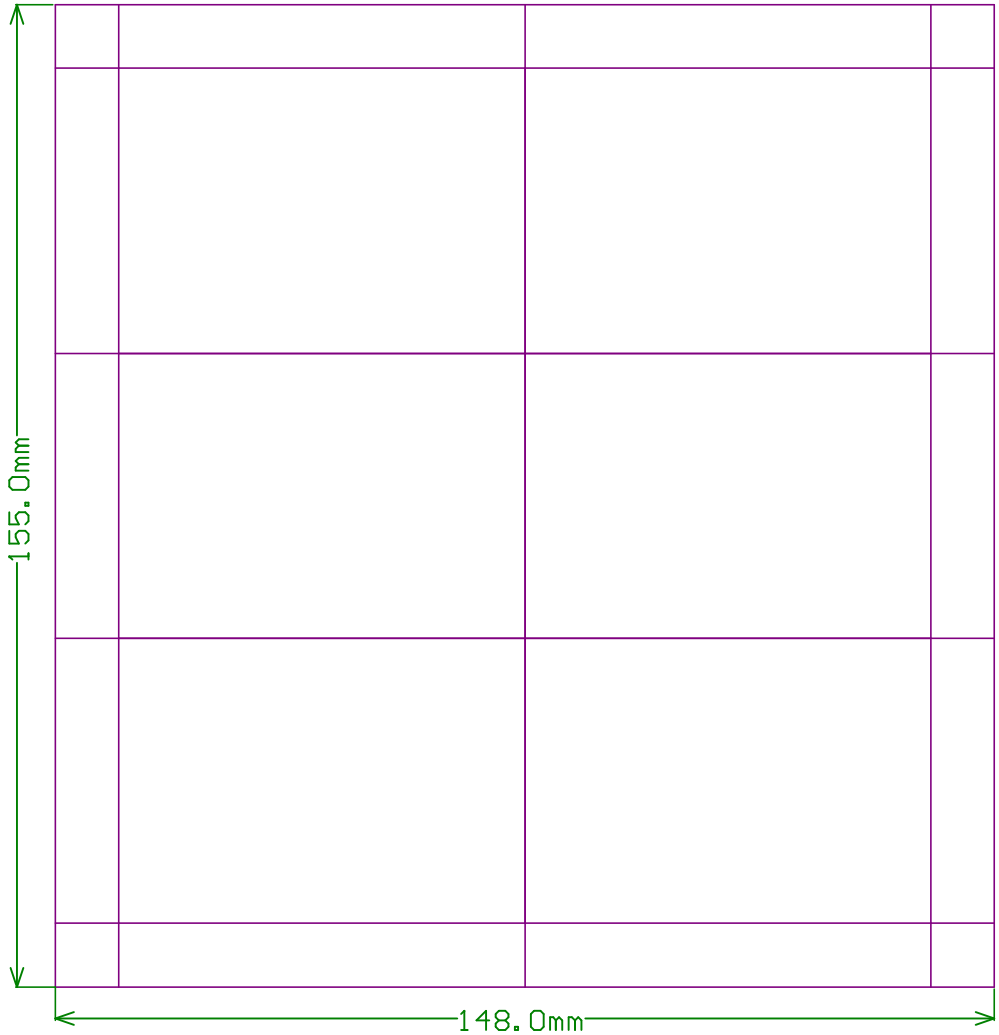
Bottom Mask
Board Outline

Efficient Power Conversion Corp. Proprietary and Confidential



Component Outline
Board Outline

Efficient Power Conversion Corp. Proprietary and Confidential



Notes, unless otherwise specified:

1. Material, Copper Clad Laminate sheet in accordance with IPC-4101.
   Total Board thickness to be 0.062 inches with torelarance to be +/-10%
   Material: Use FR370-HR Substrate
2. All holes to be located by pad centers & be within .003 inches dia. of true position.
   Layer to layer registration to be within 0.004 inches
3. All holes to be Copper (Cu) Plated thru per IPC-2221, 0.001 inches min. thickness
4. Solder Mask over bare Copper, both sides, using Taiyu 4000HFX L.P.I or
   equiv., color gren per IPC-SM-840, class T, RoHS Compliant
   Solder mask created 1:1, DO NOT enlarge.
   LDI mask must be used to register tolerance.
5. Finish, all exposed Copper, both sides, after solder mask, to be
   Electroless Nickle Immersion Gold (Ni thickness typical 75 microinches
   and Au 3-5 micro inches) in accordance with IPC-4552
6. Silkscreen both sides using white permanent ink.
7. Fabriacate using Gerber artwork.
8. Acceptability of PCB Fabrication, except as specified in this drawing,
   must comply to IPC-A-600, class 2.
9. Any changes to the design must be approved by EPC in writing

RoHS
Compliant
2002/95/EC

Layer Stack Up Details

Copper Layer Name	Substrate Layer Name	Thickness
Top Layer		2oz
	Core	5mils
Mid-Layer 1		2oz
	Prepreg	45mils
Mid-Layer 2		2oz
	Core	5mils
Bottom Layer		2oz

ZVS Class D Wireless Amp.

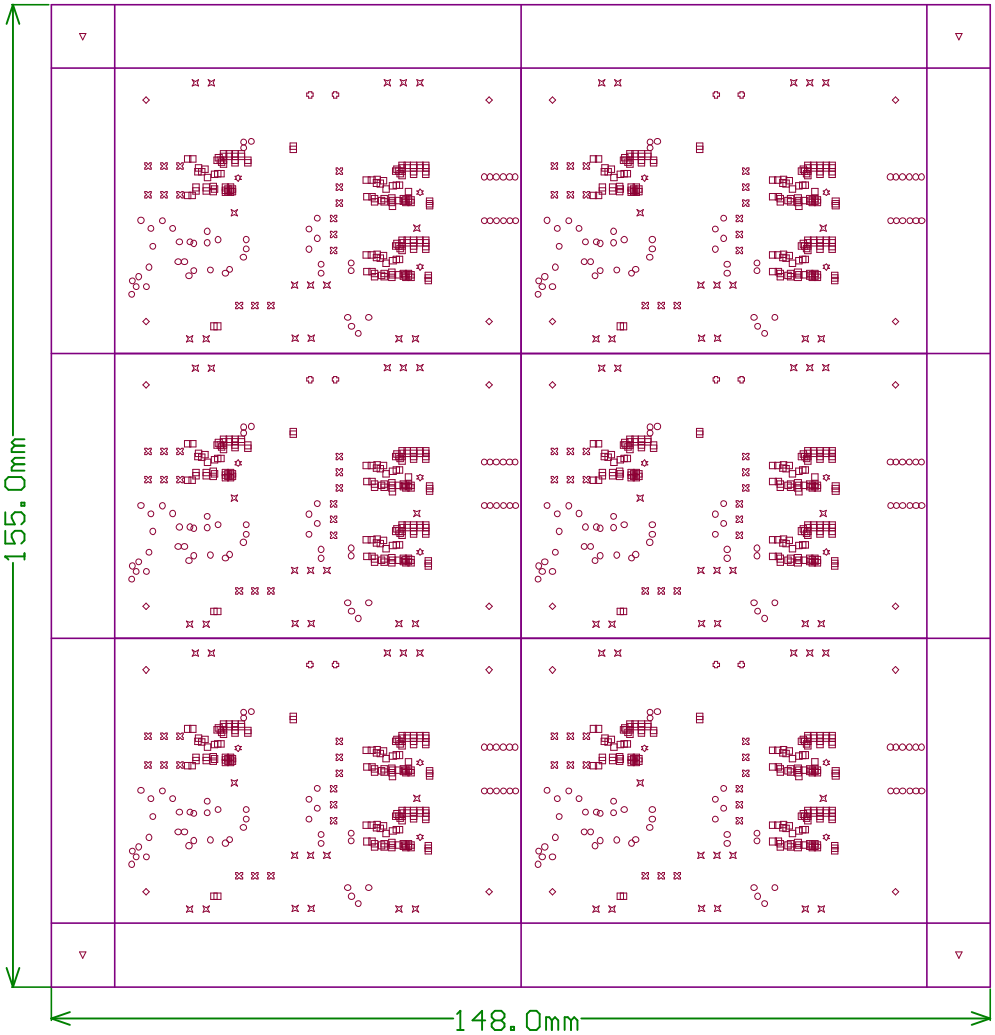
Date: May 14, 2014
Board #: EPC9507_r1_1_Panel
Scale: FULL
Sheet: 1of 1

Efficient Power Conversion
909 N. Sepulveda Blvd. Ste 230
El Segundo, CA 90245, U.S.A.
Tel: (310) 615 0279

Copyright Efficient Power Conversion Corp. 2014

Board Outline
Fab

Efficient Power Conversion Corp. Proprietary and Confidential



Notes, unless otherwise specified:

1. Material, Copper Clad Laminate sheet in accordance with IPC-4101.
   Total Board thickness to be 0.062 inches with torelarance to be +/-10%
   Material: Use FR370-HR Substrate
2. All holes to be located by pad centers & be within .003 inches dia. of true position.
   Layer to layer registration to be within 0.004 inches
3. All holes to be Copper (Cu) Plated thru per IPC-2221, 0.001 inches min. thickness
4. Solder Mask over bare Copper, both sides, using Taiyu 4000HFX L.P.I or
   equiv., color gren per IPC-SM-840, class T, RoHS Compliant
   Solder mask created 1:1, DO NOT enlarge.
   LDI mask must be used to register tolerance.
5. Finish, all exposed Copper, both sides, after solder mask, to be
   Electroless Nickle Immersion Gold (Ni thickness typical 75 microinches
   and Au 3-5 micro inches) in accordance with IPC-4552
6. Silkscreen both sides using white permanent ink.
7. Fabriacate using Gerber artwork.
8. Acceptability of PCB Fabrication, except as specified in this drawing,
   must comply to IPC-A-600, class 2.
9. Any changes to the design must be approved by EPC in writing

RoHS
Compliant
2002/95/EC

Layer Stack Up Details

Copper Layer Name	Substrate Layer Name	Thickness
Top Layer		2oz
	Core	5mils
Mid-Layer 1		2oz
	Prepreg	45mils
Mid-Layer 2		2oz
	Core	5mils
Bottom Layer		2oz

ZVS Class D Wireless Amp.

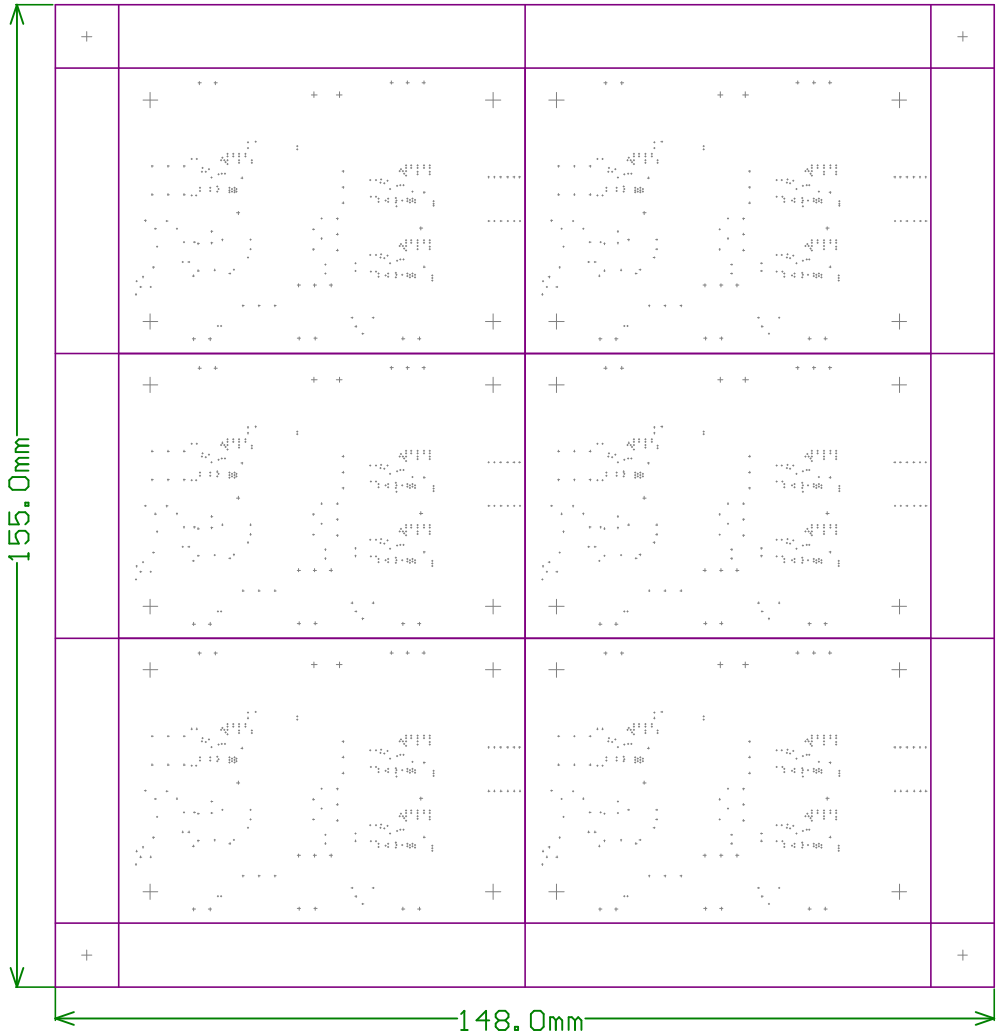
Date: May 14, 2014
Board #: EPC9507_r1_1_Panel
Scale: FULL
Sheet: 1of 1

Efficient Power Conversion
909 N. Sepulveda Blvd. Ste 230
El Segundo, CA 90245, U.S.A.
Tel: (310) 615 0279

Copyright Efficient Power Conversion Corp. 2014

Drill Drawing
Board Outline
Fab

Efficient Power Conversion Corp. Proprietary and Confidential



Notes, unless otherwise specified:

1. Material, Copper Clad Laminate sheet in accordance with IPC-4101.
   Total Board thickness to be 0.062 inches with torelarance to be +/-10%
   Material: Use FR370-HR Substrate
2. All holes to be located by pad centers & be within .003 inches dia. of true position.
   Layer to layer registration to be within 0.004 inches
3. All holes to be Copper (Cu) Plated thru per IPC-2221, 0.001 inches min. thickness
4. Solder Mask over bare Copper, both sides, using Taiyu 4000HFX L.P.I or
   equiv., color gren per IPC-SM-840, class T, RoHS Compliant
   Solder mask created 1:1, DO NOT enlarge.
   LDI mask must be used to register tolerance.
5. Finish, all exposed Copper, both sides, after solder mask, to be
   Electroless Nickle Immersion Gold (Ni thickness typical 75 microinches
   and Au 3-5 micro inches) in accordance with IPC-4552
6. Silkscreen both sides using white permanent ink.
7. Fabriacate using Gerber artwork.
8. Acceptability of PCB Fabrication, except as specified in this drawing,
   must comply to IPC-A-600, class 2.
9. Any changes to the design must be approved by EPC in writing

RoHS
Compliant
2002/95/EC

Layer Stack Up Details

Copper Layer Name	Substrate Layer Name	Thickness
Top Layer		2oz
	Core	5mils
Mid-Layer 1		2oz
	Prepreg	45mils
Mid-Layer 2		2oz
	Core	5mils
Bottom Layer		2oz

ZVS Class D Wireless Amp.

Date: May 14, 2014
Board #: EPC9507_r1_1_Panel
Scale: FULL
Sheet: 1of 1

Efficient Power Conversion
909 N. Sepulveda Blvd. Ste 230
El Segundo, CA 90245, U.S.A.
Tel: (310) 615 0279

Copyright Efficient Power Conversion Corp. 2014

Drill Guide
Board Outline
Fab

Efficient Power Conversion Corp. Proprietary and Confidential