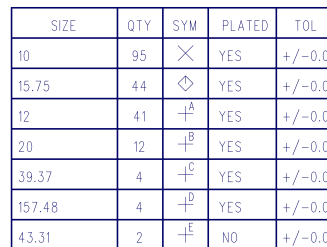


MINIMUM SPACING: 0.1778 mm (7 mils)  
MINIMUM TRACE: 0.1778 mm (7 mils)  
SMALLEST VIA: 0.3 mm drill in 0.6 mm pad  
(11.81 mil drill in 23.62 mil pad)

- 1) Many pads have intentional undersize solder mask openings/oversize pads for physical strength.
- 2) Make top and bottom solder masks exactly as shown. Do \_NOT\_ "open-up" the solder mask layers. This design uses "solder-mask-defined-pads".
- 3) Silkscreened Legend (white) must be clean and clear. Silkscreen legend must have good registration. Silkscreen is intentionally placed over bare copper. Do not clip silkscreen to solder mask layer.
- 4) Edges of board must be smooth. Use router. Do not use "score and break".
- 5) Board must be exactly like:  
CFA10075\_0v2\_PCB\_Preview.pdf



 **CrystalFontz**  
12412 East Saltese Avenue  
Spokane Valley, WA 99216 USA  
Tel: (509) 892-1200 <http://www.crystalfontz.com>  
Copyright © 2018 CrystalFontz America, Inc.

|          |  |           |  |            |  |     |  |
|----------|--|-----------|--|------------|--|-----|--|
| PROJECT  |  | CFA-10075 |  | VERSION    |  | 0v2 |  |
| PCB      |  | REVISION  |  | DATE       |  |     |  |
| PCB10075 |  | 0v2       |  | 2018-12-06 |  |     |  |

LAYER  
Drill Drawing/Notes

FILE NAME  
C-CFA10075\_0v2\_DD.pho



Edge is  
not plated

Copper goes  
all the way  
to the edge

