PCB LAYOUT NOTES:

1. No tracks or vias in the shaded "KEEP OUT" area.

2. No other track or signal within 0.020" of any contact pad.

3. Do not allow solder paste on the contact pads. It is essential to eliminate any hole in the solder paste-mask (solder stencil) layer. If this is not possible make the contact pads thru-hole with finished hole size of 0.012" or less.

4. To avoid ordering confusion, please specify DNL in your BOM - the PCB footprint IS NOT A PHYSICAL PART.

5. The TC2070-CLIP board can attach to the three alignment pins from underneath the PCB to temporarily retain the cable in place.

FOOTPRINT TO BE LABELED "TC2070-NL-FP" ON LAYOUT / BOM

***Use TC2070-FP (or TC2070-NL-FP) as the PCB "part" name – the cable must not appear on your BOM! Ensure the "part" in the PCB BOM is labelled as "Nothing to Purchase - Do Not Install" otherwise your CM’s will try to buy one cable per PCB assembly!***
Note:
Dimensions subject to change.

8" [203.2 mm]

5.6mm [0.22"]
PIN PENETRATION AT FULL COMPRESSION

Tag-Connect inserted into PCB for programming / debugging

PCB

[0.04"]
1.0mm (MAX ALLOWABLE COMPONENT HEIGHT UNDER CONNECTOR)

TC2070-IDC-NL
REV B

A
FSCM NO.
DWG NO.
SIZE
2:1
TC2070-IDC-NL
www.TAG-CONNECT.com
SHEET 2 OF 3
Notes:

1) It IS OK to have a conductive via (not filled, not soldermasked) in the center of any of the pads so long as the finished hole is 0.012” or smaller.

2) It's NOT OK to have a via between two pads without 0.02” between the via and a pad of a different signal.

3) It IS OK to have a small via centered between four pads.

Do not deposit solder paste on contact pads. View Solder Paste Mask (Solder Stencil) layer in pad-stack and eliminate any hole.