

# Notes:

1. Allow clearance for finger squeeze access from the two sides shown.

2. No tracks or vias in the shaded "KEEP OUT" AREA.

3. No other track or signal within .02" of any contact pad.

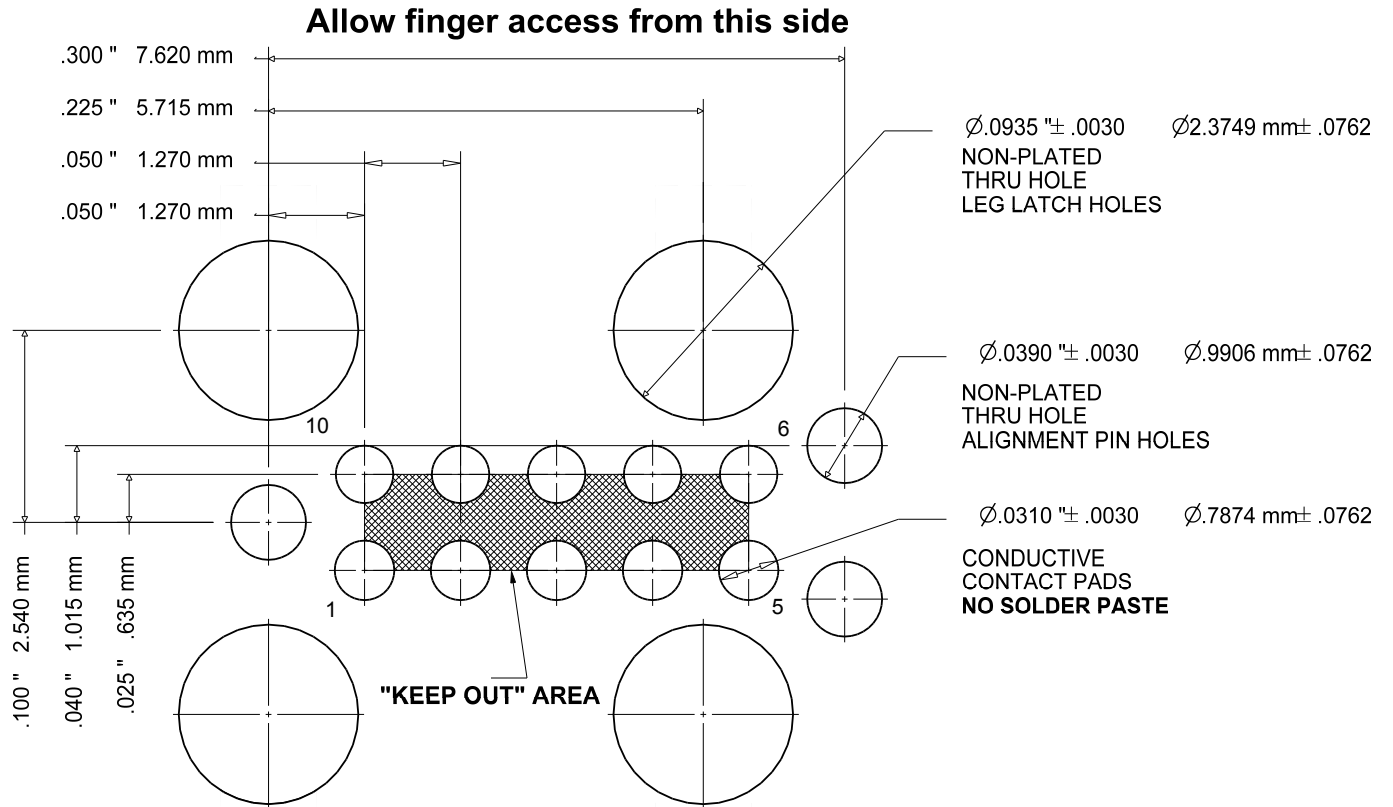
4. Do not allow solder paste on the contact pads. It is essential to eliminate any hole in the solder paste-mask (solder stencil) layer. If this is not possible make the contact pads thru-hole with a finished hole size of .008" or less.

5. Leg and alignment holes may be plated and used as vias but ensure minimum finished hole size meets specified tolerances.

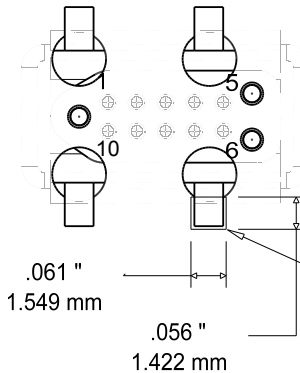
6. To prevent purchasing confusion, please specify DNL in your BOM.

## PCB Footprint for J-LINK Needle Adapter

ZONE	A		12/2/10	MS
------	---	--	---------	----



PCB BOTTOM VIEW



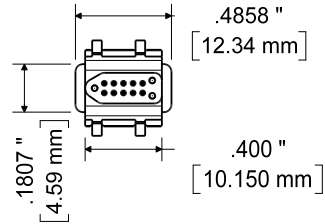
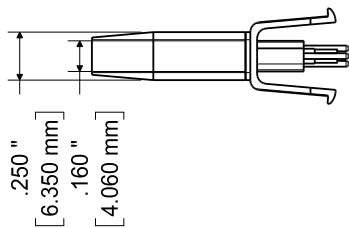
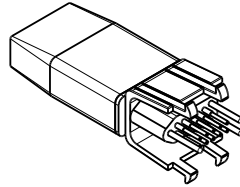
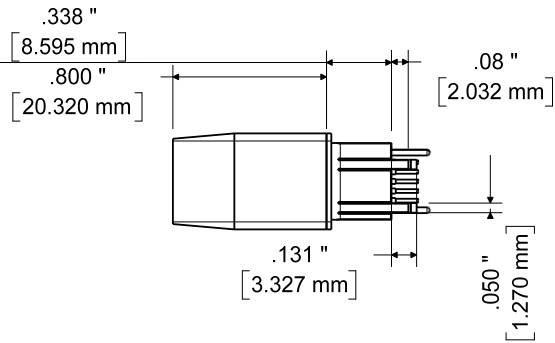
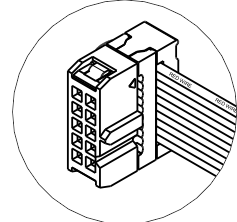
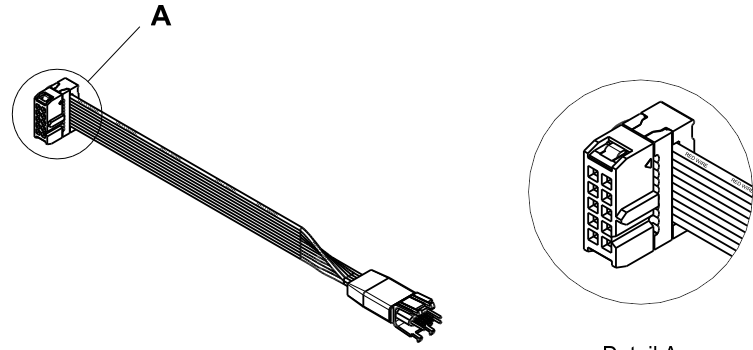
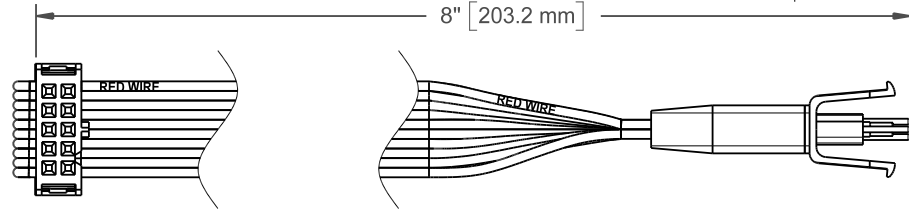
ALLOW A .061 x .056 SQUARE AREA OF CLEARANCE FOR EACH FOOT

Allow finger access from this side

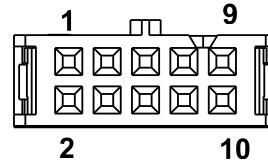
SIZE <b>A</b>	FSCM NO.	DWG NO.	REV <b>A</b>
SCALE 7:1	SHEET 1 of 3		

**NOTE:**  
Dimensions subject to change

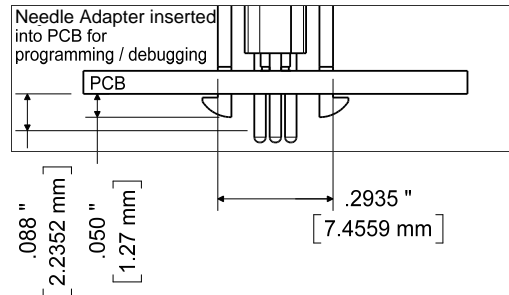
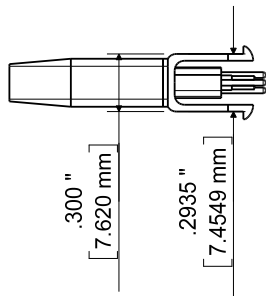
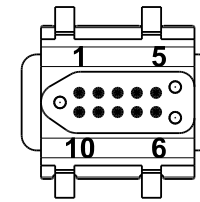
ZONE	A		12/2/10	MS
------	---	--	---------	----



END VIEW  
2x5 0.1" PITCH  
IDC CONNECTOR

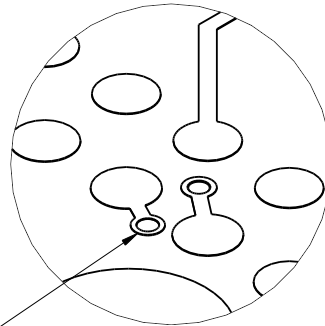
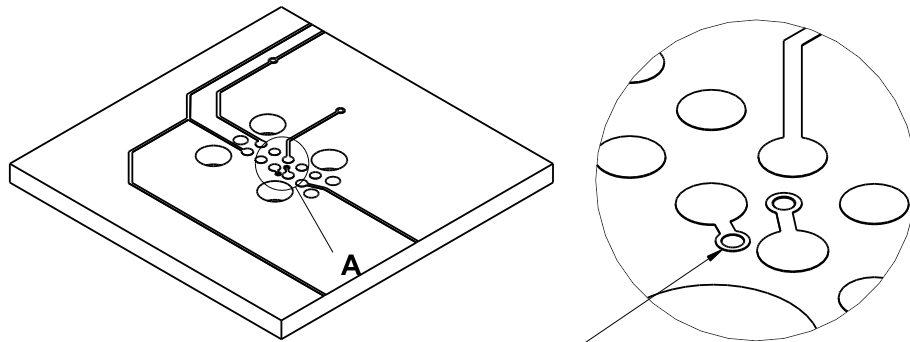


END VIEW  
10 PIN  
CONNECTOR



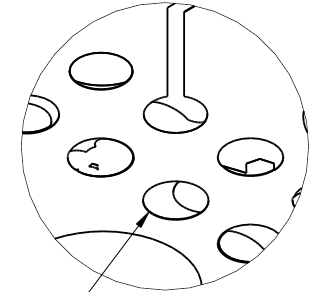
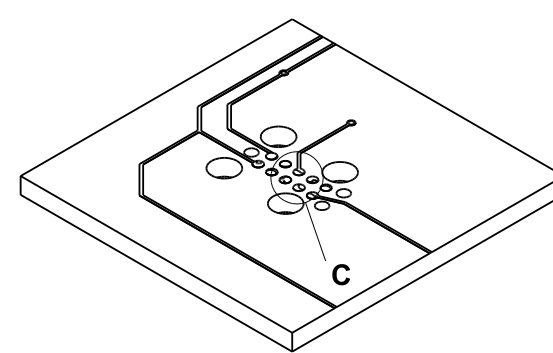
SIZE <b>A</b>	FSCM NO.	DWG NO.	REV <b>A</b>
SCALE		SHEET 2 Of 3	

# Layout Problems to Avoid



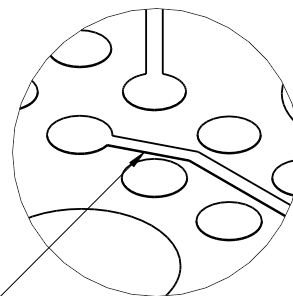
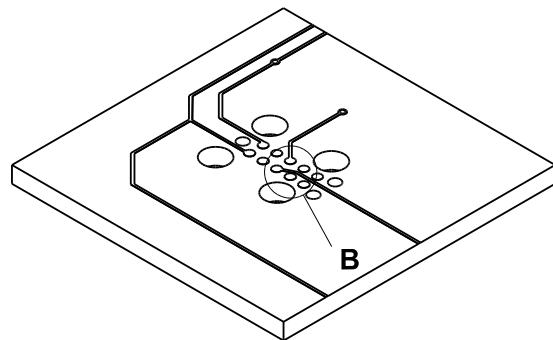
Detail A

Vias between contact pads



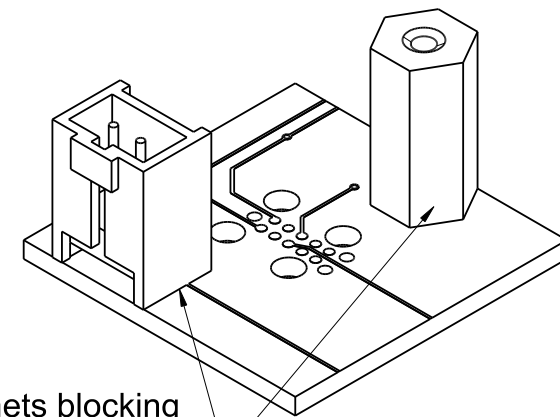
Detail C

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



Detail B

Traces less than .02" away from contact pad or in "KEEP OUT" area



Large componets blocking finger access to the Needle-Adapter's leg tabs.

# Mouser Electronics

Authorized Distributor

Click to View Pricing, Inventory, Delivery & Lifecycle Information:

[Segger Microcontroller:](#)

[8.06.05](#)