

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

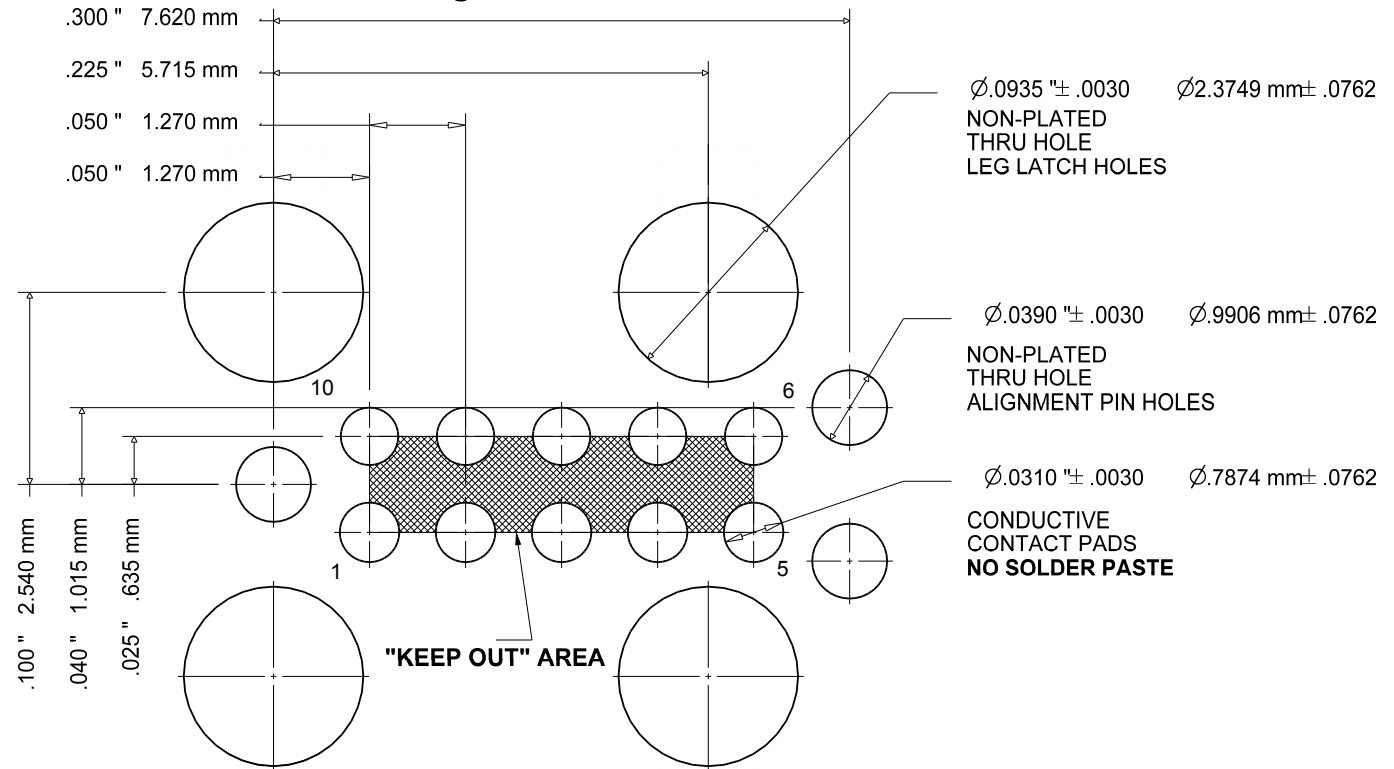
ZONE

A

12/2/10

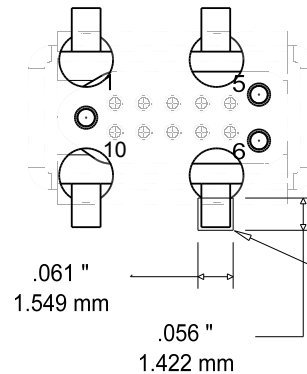
MS

Allow finger access from this side



Allow finger access from this side

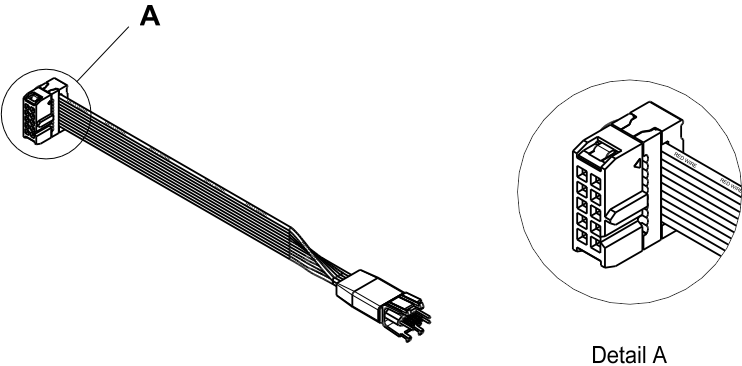
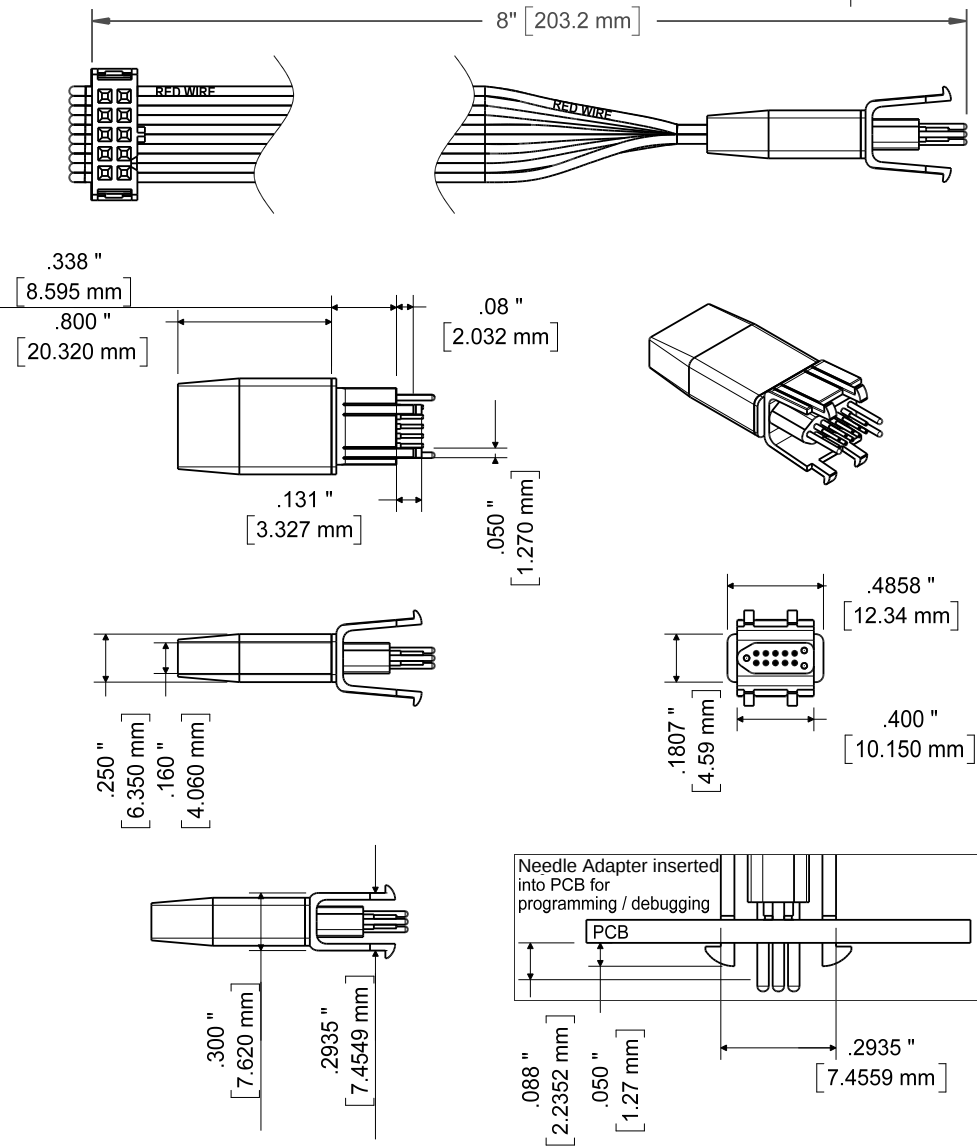
PCB BOTTOM VIEW



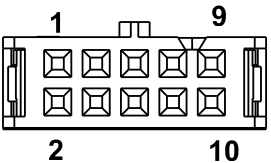
SIZE A	FSCM NO.	DWG NO.	REV A
SCALE 7:1		SHEET 1 Of 3	

NOTE:
Dimensions subject to change

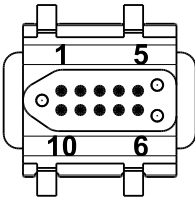
ZONE	A		12/2/10	MS
------	---	--	---------	----



END VIEW
2x5 0.1" PITCH
IDC CONNECTOR

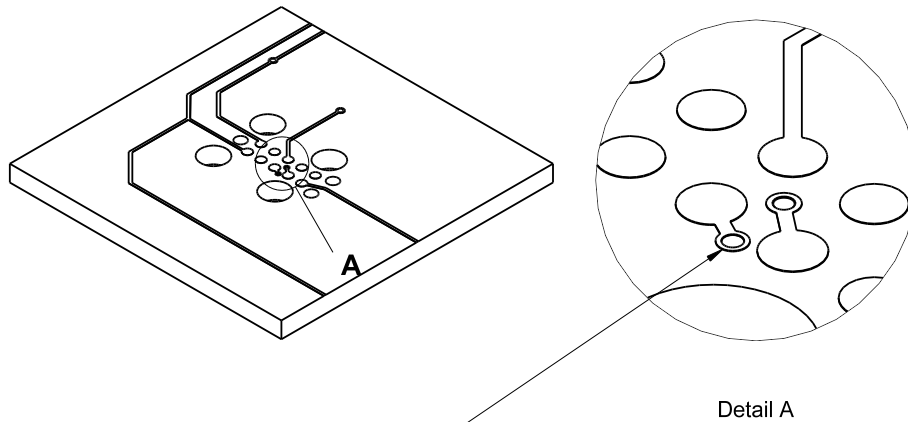


END VIEW
10 PIN
CONNECTOR

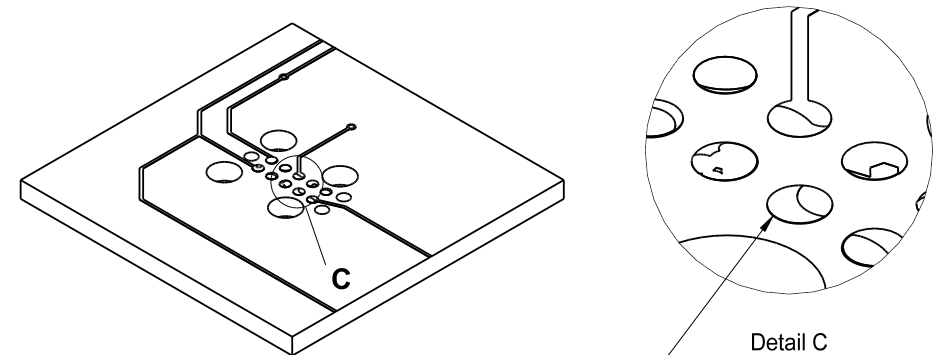


SIZE A	FSCM NO.	DWG NO.	REV A
SCALE		SHEET 2 Of 3	

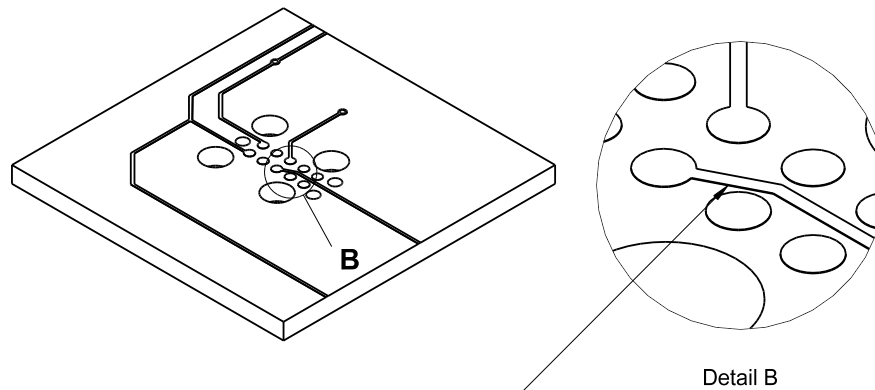
Layout Problems to Avoid



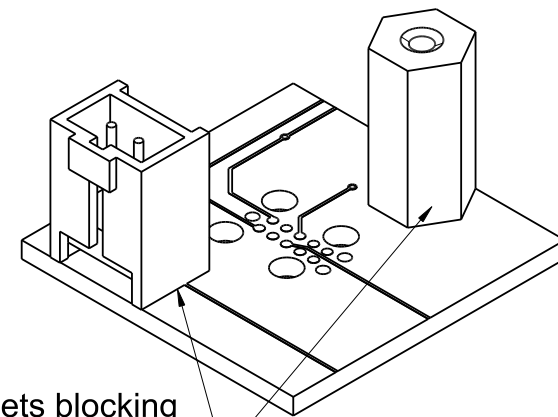
Vias between
contact pads



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



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.