

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.

7. If using a proto-board fab that plates all through holes, make sure all holes are defined as plated in the footprint.

FOOTPRINT FOR J-LINK 6-Pin Needle Adapter

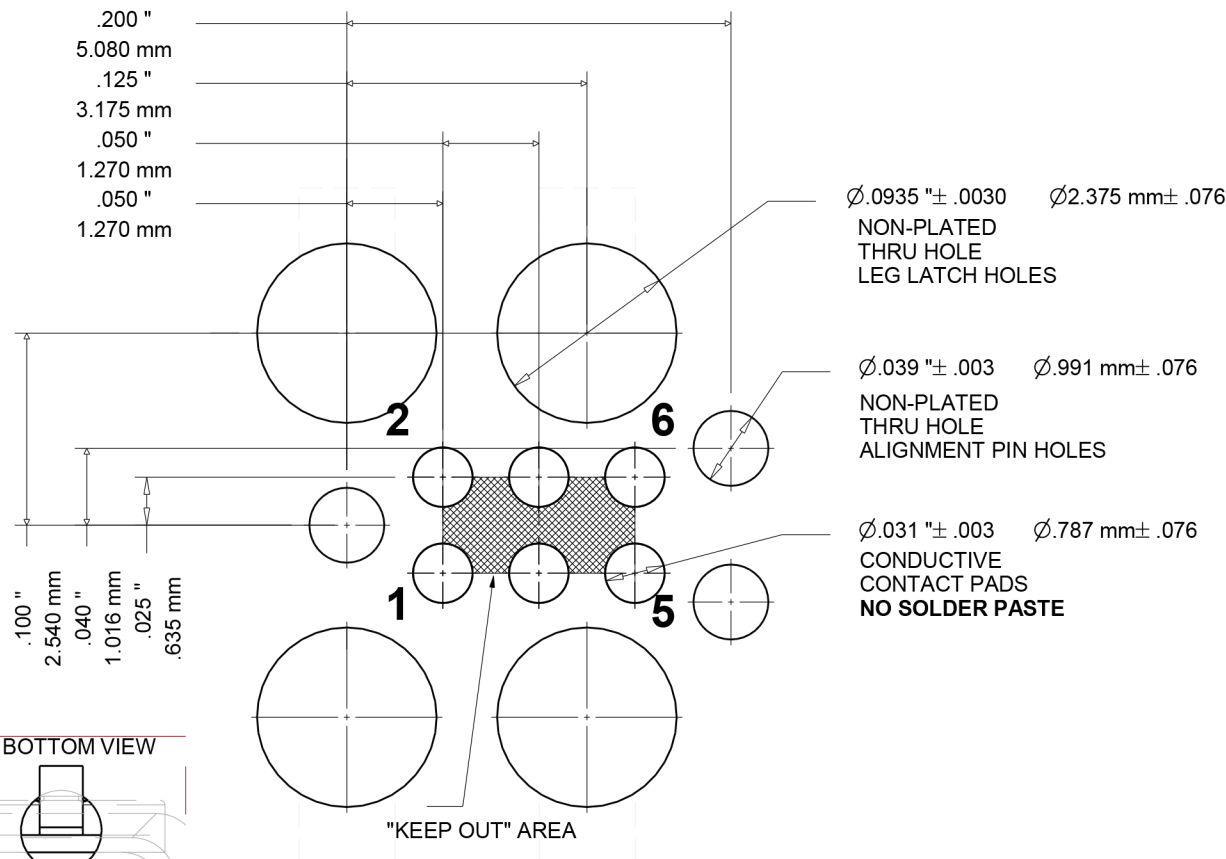
ZONE

REV

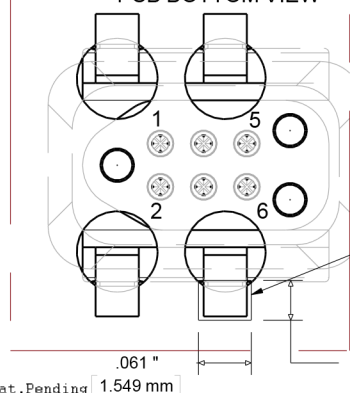
4/16/14

MS

Allow finger access from this side



PCB BOTTOM VIEW



Allow finger access from this side

ALLOW A .061 X .046
SQUARE AREA OF
CLEARANCE FOR
EACH FOOT

SIZE	FSCM NO.	DWG NO.	FOOTPRINT FOR 6-Pin Needle Adapter	REV
A				A
SCALE	www.segger.com		SHEET 1 Of 3	

NOTE:
Dimensions subject to change

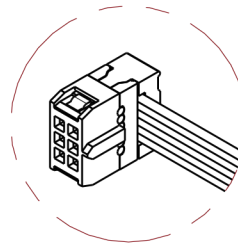
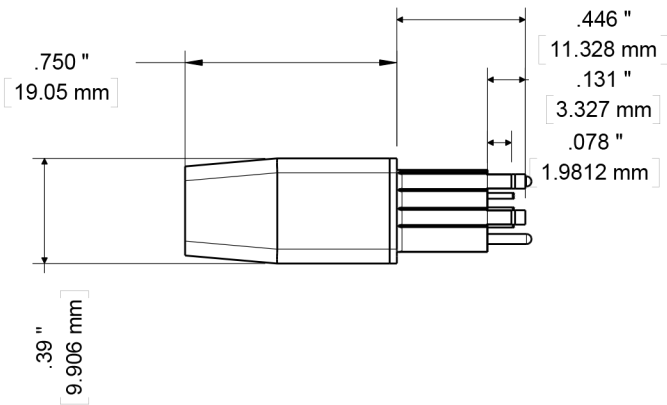
J-LINK 6-Pin Needle Adapter

ZONE

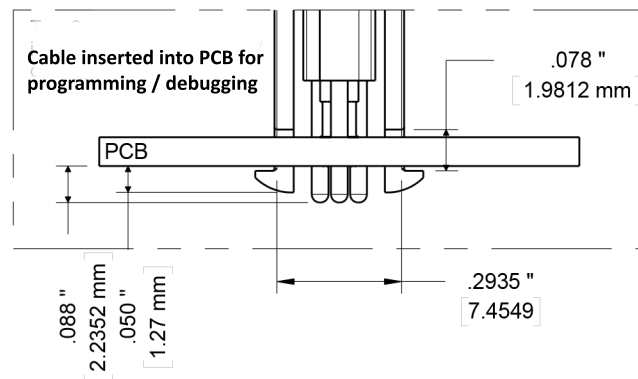
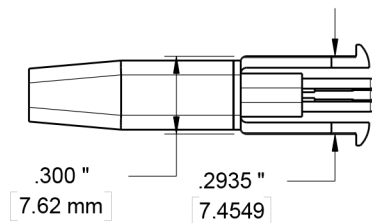
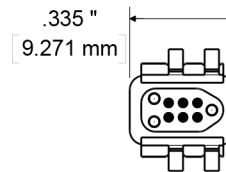
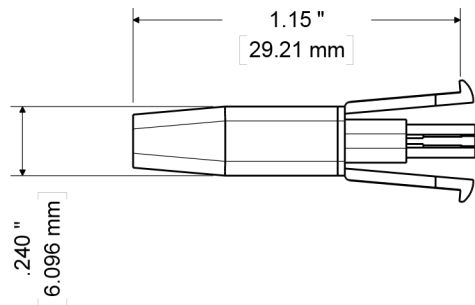
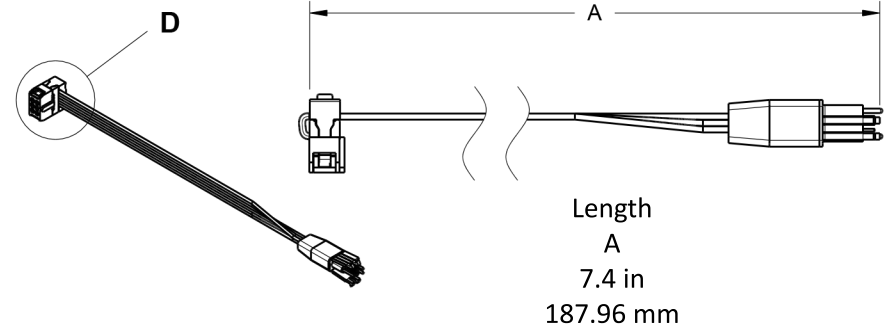
A

4/16/14

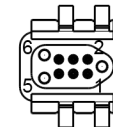
MS



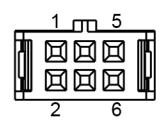
Detail D



END VIEW



END VIEW



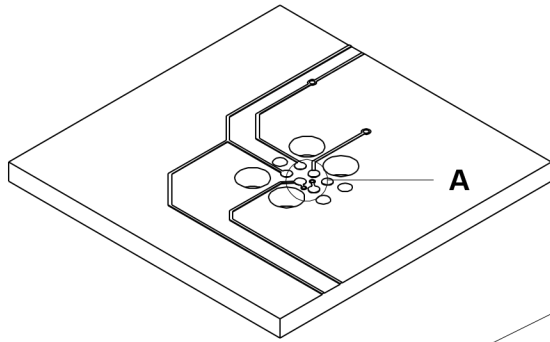
TAG 6 PIN

IDC 6pin

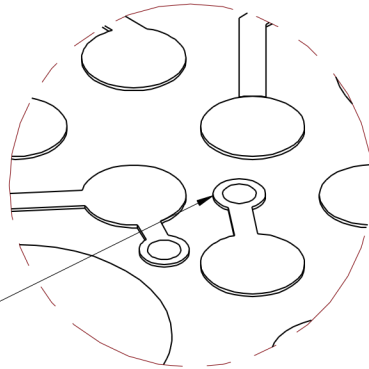
TAG 6 PIN	IDC 6pin
1	1
2	2
3	3
4	4
5	5
6	6

SIZE	FSCM NO.	DWG NO.	REV
A		6-Pin Needle Adapter	A
SCALE	www.segger.com SHEET 2 Of 3		

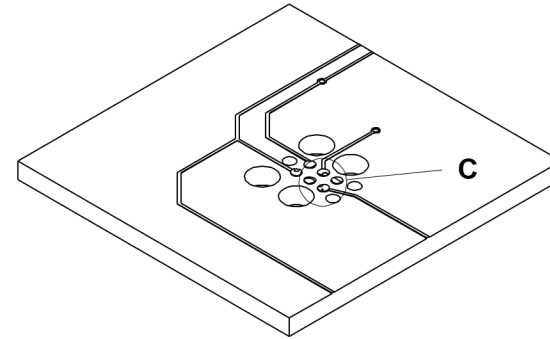
Layout Problems to Avoid



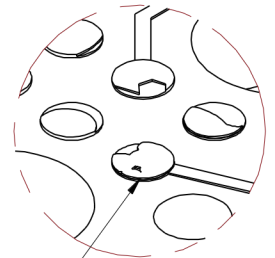
Vias between
contact pads



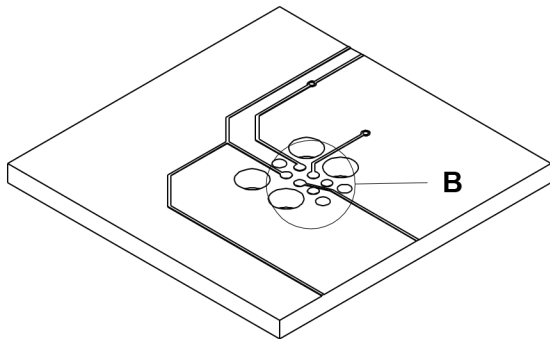
Detail A



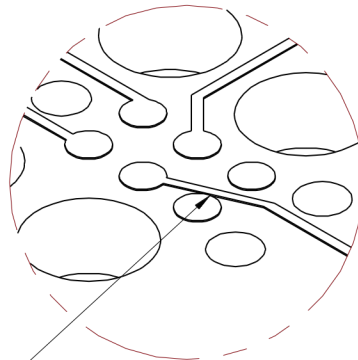
Do not deposit solder paste
on contact pads. View Solder
Paste Mask (Solder Stencil)
layer in pad-stack and eliminate
any hole.



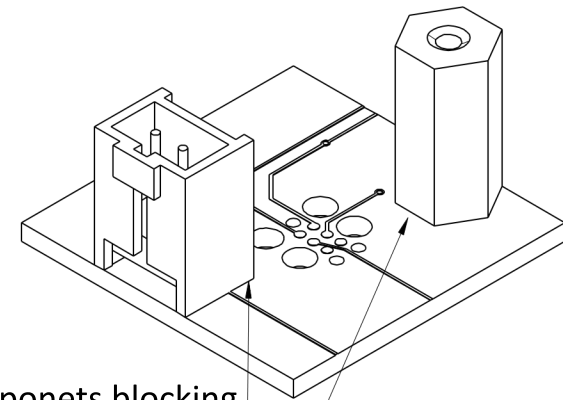
Detail C



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



Detail B



Large componets blocking
finger access to the cabel
leg clips