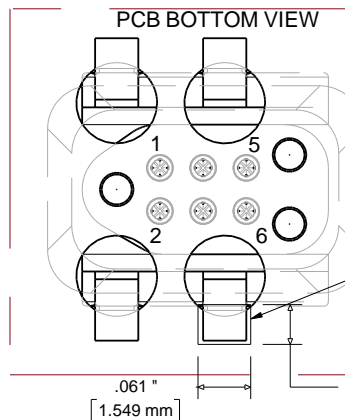
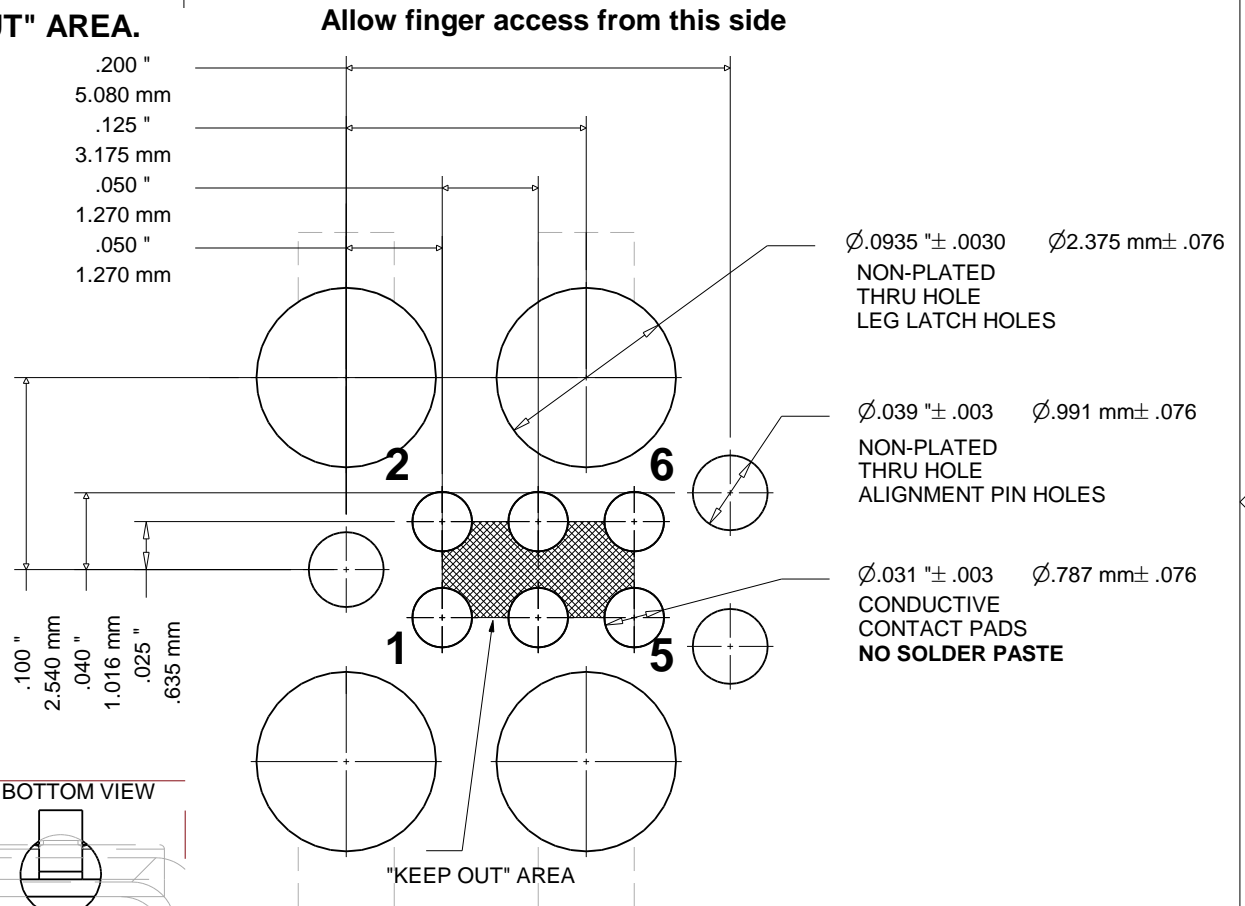


Notes:

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.

PCB Footprint for J-LINK 6-Pin Needle Adapter

ZONE	REV	12/05/19	MS/NS
------	-----	----------	-------

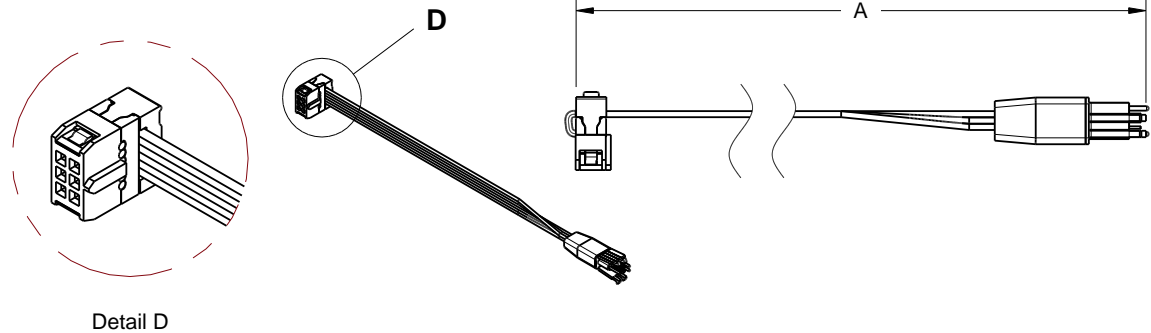
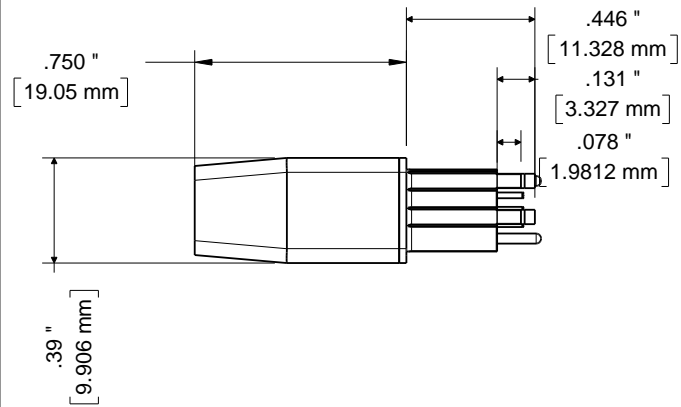


ALLOW A .061 X .046 SQUARE AREA OF CLEARANCE FOR EACH FOOT

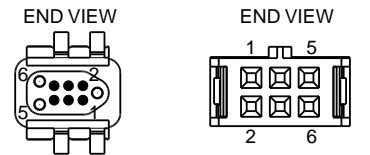
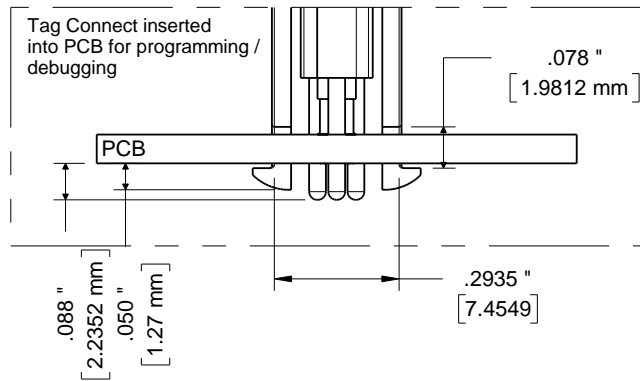
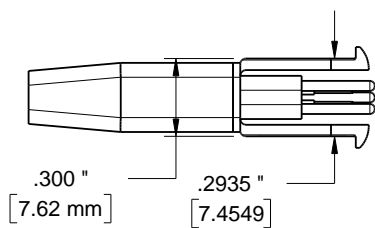
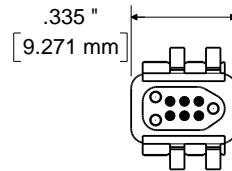
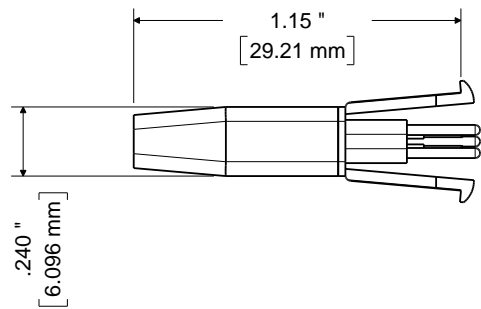
SIZE A	FSCM NO.	DWG NO.	REV B
SCALE	SHEET 1 of 3		

NOTE:
Dimensions subject to change

ZONE	A		5/17/11	MS
------	---	--	---------	----



Detail D

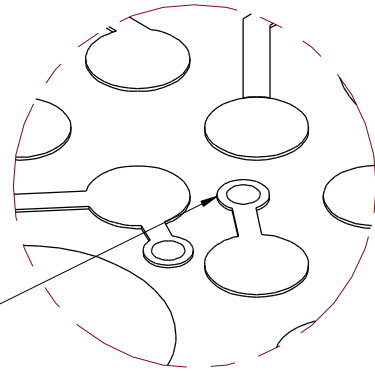
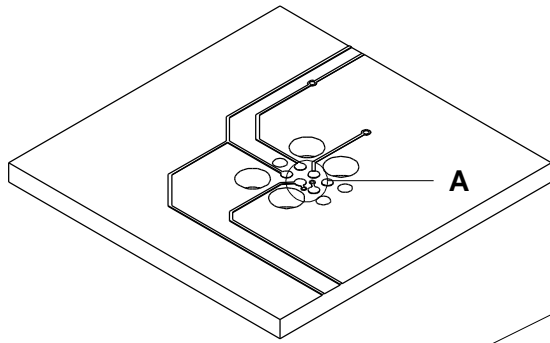


TAG 6 PIN	IDC 6pin
1	1
2	2
3	3
4	4
5	5
6	6

PART NUMBER	A	
	IN	MM
TC2030-IDC	8.8	225
TC2030-IDC-10	10.5	265

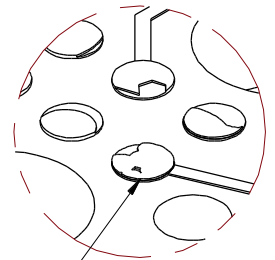
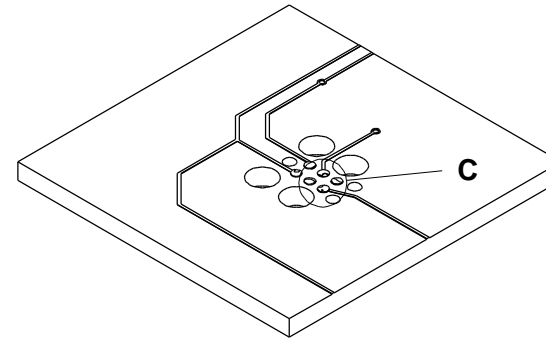
SIZE A	FSCM NO.	DWG NO.	REV B
SCALE	SHEET 2 Of 3		

Layout Problems to Avoid



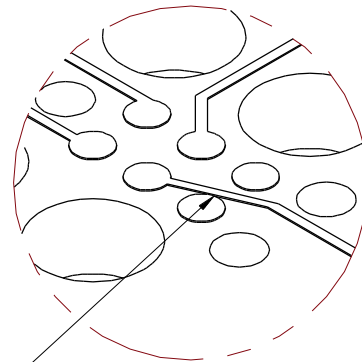
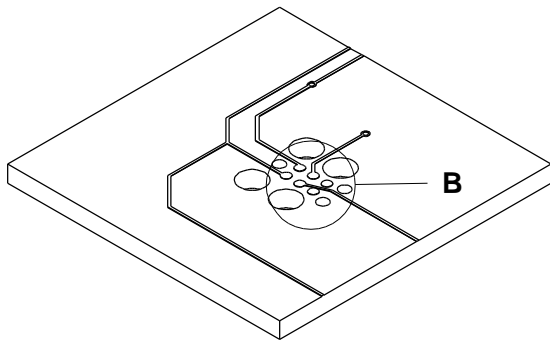
Vias between contact pads

Detail A



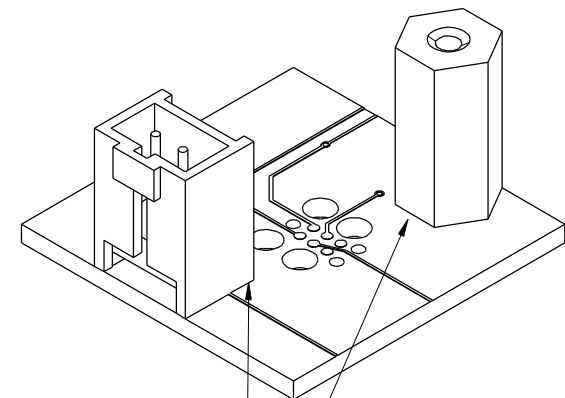
Detail C

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

Detail B



Large components blocking finger access to the Needle Adapter leg clips