

DECLARATION OF NATHANIEL E FRANK-WHITE

1. I am a Records Request Processor at the Internet Archive. I make this declaration of my own personal knowledge.

2. The Internet Archive is a website that provides access to a digital library of Internet sites and other cultural artifacts in digital form. Like a paper library, we provide free access to researchers, historians, scholars, and the general public. The Internet Archive has partnered with and receives support from various institutions, including the Library of Congress.

3. The Internet Archive has created a service known as the Wayback Machine. The Wayback Machine makes it possible to browse more than 450 billion pages stored in the Internet Archive's web archive. Visitors to the Wayback Machine can search archives by URL (i.e., a website address). If archived records for a URL are available, the visitor will be presented with a display of available dates. The visitor may select one of those dates, and begin browsing an archived version of the Web. Links on archived files in the Wayback Machine point to other archived files (whether HTML pages or other file types), if any are found for the URL indicated by a given link. For instance, the Wayback Machine is designed such that when a visitor clicks on a hyperlink on an archived page that points to another URL, the visitor will be served the archived file found for the hyperlink's URL with the closest available date to the initial file containing the hyperlink.

4. The archived data made viewable and browseable by the Wayback Machine is obtained by use of web archiving software that automatically stores copies of files available via the Internet, each file preserved as it existed at a particular point in time.

5. The Internet Archive assigns a URL on its site to the archived files in the format http://web.archive.org/web/[Year in yyyy][Month in mm][Day in dd][Time code in hh:mm:ss]/[Archived URL] aka an "extended URL". Thus, the extended URL http://web.archive.org/web/19970126045828/http://www.archive.org/ would be the URL for the record of the Internet Archive home page HTML file (http://www.archive.org/) archived on January 26, 1997 at 4:58 a.m. and 28 seconds (1997/01/26 at 04:58:28). The date indicated by an extended URL applies to a preserved instance of a file for a given URL, but not necessarily to any other files linked therein. Thus, in the case of a page constituted by a primary HTML file and other separate files (e.g., files with images, audio, multimedia, design elements, or other embedded content) linked within that primary HTML file, the primary HTML file and the other files will each have their own respective extended URLs and may not have been archived on the same dates.

6. Attached hereto as Exhibit A are true and accurate copies of screenshots of the Internet Archive's records of the archived files for the URLs and the dates specified in the attached coversheet of each printout.

INTERNET ARCHIVE
archive.org

7. I declare under penalty of perjury that the foregoing is true and correct.

DATE: Febuary 20, 2023

Nathaniel E Frank-White

Nathaniel E Frank-White

EXHIBIT A

http://web.archive.org/web/20130514125612/https:/electronics.stackexchange.com/questions/35625/soldering-pcbs-directly-together

INTERNET ARCHIVE WayBackMachine
https://electronics.stackexchange.com/questions/35625/soldering-pcbs-directly-together
Go
9 captures
14 May 2013 - 7 May 2021
APR MAY JUL
14
2012 2013 2014
About this capture

Electrical Engineering Stack Exchange is a question and answer site for electronics and electrical engineering professionals, students, and enthusiasts. It's 100% free, no registration required.
Tell me more ×

Soldering PCBs directly together

13

I am attempting to replace an old PLCC32 part that was directly soldered to the board with a new part of undecided form. We will definitely need an adapter as we have not been able to find a PLCC32 part that does what we need. I cannot use a PLCC adapter plug because there are also height restrictions. We are considering building a two-sided adapter board that has pads on the bottom side that match the PLCC32 layout on the current board, with the new layout on top. Theoretically, the adapter board would be soldered directly to the old board and the new chip on top of the adapter.

However, I have not seen any examples of soldering two PCBs directly together in this manner, which makes me think it is a likely to be a bad idea. Can anyone comment on this sort of custom adapter?

pcb soldering

share improve this question

asked Jul 12 '12 at 16:41
QuestionMan
391 2 9

TAGGED
pcb × 490
soldering × 219

ASKED 10 MONTHS AGO
VIEWED 823 TIMES
ACTIVE 9 MONTHS AGO

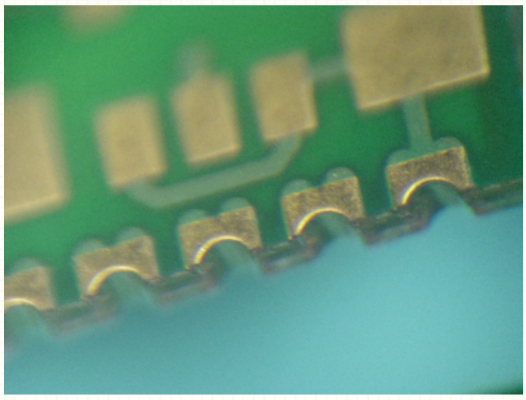
Help this community grow.
Follow us on twitter!

4 Answers

active **oldest** votes

15

No problem. I had to look for a picture that illustrates the technique:



You make a PCB with plated through holes on the PLCC's pads, so at a 1.27 mm pitch, and mill the four sides so that you get the half holes like in the picture. These are easily solderable on the old PLCC footprint, it's an often used technique, called *castellation*.

A picture of a complete board:



Love this site?
Get the **weekly newsletter!**
- Top questions and answers
- Important announcements
- Unanswered questions

Sign up for the newsletter
see an example newsletter

RELATED

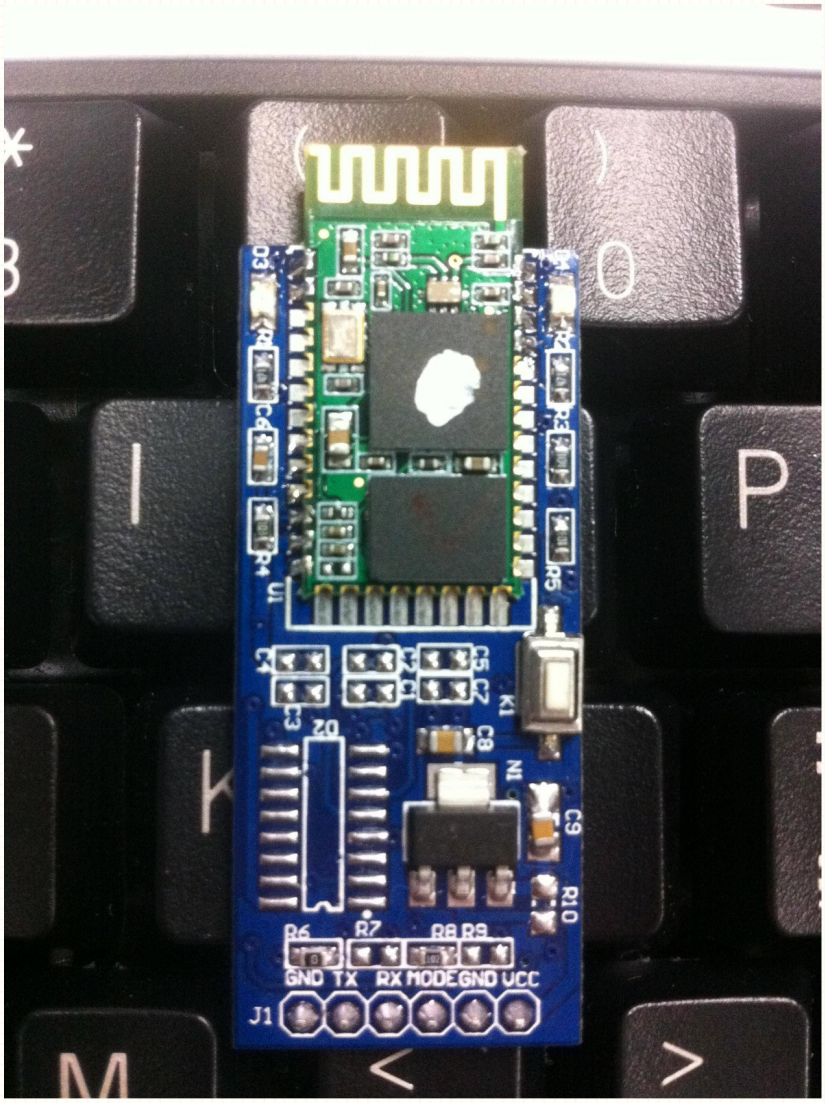
45 Are solder fumes bad for me?
6 How do I keep PCBs cool when soldering?
5 Solder very close together contacts
-5 ARM processor on a PCB
7 How do you remove the buzzer from this PCB whose solder side is inaccessible?
6 Fast approaches for Soldering / PCB assembly
1 How do I dead bug solder this SOT-23 to my PCB?
9 Why would large portions of traces be completely covered in solder
3 Using alligator leads as heat sink for soldering/desoldering PCBs
2 Eagle compatible autorouter

to solder components (except vias) only on bottom layer of double sided PCB

and another one:



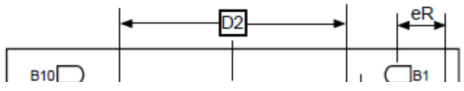
or this one from a question posted 1 minute ago:

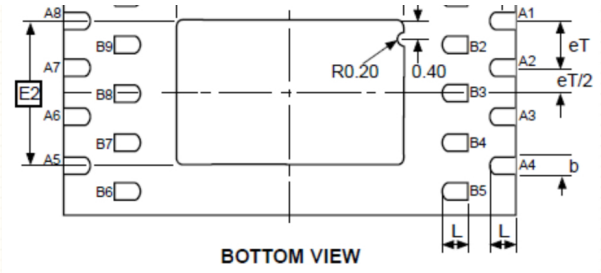


You get the idea.

You'll have to find a part which fits inside this small PCB, but given the miniaturization of the last years that may not be a problem.

edit 2012-07-15
QuestionMan suggested to make the PCB a bit larger so that the PLCC's solder pads are under it. For BGAs the solder balls are also under the IC, but that's solid solder balls, not paste, and I don't know how solder paste will behave when squeezed between two PCBs. But today I bumped into this IC package:





It's the "Staggered Dual-row MicroLeadFrame® Package (MLF)" of the ATMega8HVD, and it has pins under the IC as well. This is 3.5 mm x 6.5 mm, and weighs a lot less than the small PCB. That may be important, because thanks to the low weight capillary forces of the molten solder paste can pull the IC to its exact position. I'm not sure if that will also be the case for that PCB, and then positioning may be a problem.

share improve this answer

edited Jul 31 '12 at 17:00

answered Jul 12 '12 at 17:13

stevenvh
85.5k 5 143 266

Can you comment on any specific instructions to the board house for the 'half pads' along the edge? It must be significantly different from the plated through hole process. I can't think of how I would even represent this in my ECAD (Altium). – Jason Jul 12 '12 at 17:21

2 @Jason - I think that you draw them like any regular hole on a PCB which is a bit larger, and pass an outline drawing which cuts the holes in half to the PCB shop. You'll have to instruct the DRC to ignore that the holes are overlapping the PCB's edge. Milling of course, no V-cut :-). I don't know if the plating is something special. – stevenvh Jul 12 '12 at 17:25

What about expanding over the original chip layout with SMD pads, e.g. without the castellations? SMD pads on the bottom so the top layout can be larger than the original pads. – QuestionMan Jul 12 '12 at 18:19

@QuestionMan - I don't know if they do this with reflow solder paste. BGAs use this technique, since the BGA carrier is actually a thin PCB, but then you would have to find some solder balls and a way to attach them. Also a well controlled reflow oven. – stevenvh Jul 12 '12 at 18:27

@stevenvh I asked my board house about this and they basically confirmed what you have said (outline through holes and inform in fab notes). Their term for this is 'half holes'. – Jason Jul 12 '12 at 19:15

show 1 more comment

3

I'd consider a Ball Grid Array (BGA) IC package to be close to an example of that. It comes with solder-balls preplaced on the "component" PCB. Assembly is tricky, usually done via automated placement and hot air, frequently with preheat from below too. In your case you presumably would only have contacts around the periphery so inspection would be a bit easier. However you probably won't have the preformed solder balls. You might look at rework solutions for re-balling BGAs.

There is also some similarity to a QFN package, which is usually soldered by depositing paste with a stencil and then using a similar external area heat source, however you won't have the metalization up the edge thickness which many QFN's have to aid filleting (and incidentally give you a limited ability to do rework with an extremely fine-tip iron)

If your PCB house will do it, the plated through holes cut in half by the board outline idea seen on some recent chip-carrier modules might be an interesting idea, as that would give you metalization up the thickness. I think you might have a fair shot of soldering that on with an iron or an air pencil.

share improve this answer

answered Jul 12 '12 at 17:09

Chris Stratton
8,812 12 27

6

Soldering a small PCB flat onto a larger PCB is possible. In fact, that's how many of the embedded radio models are mounted (example, example). The pad can be on the edge of the board (via cut to form a half-cylinder*). Or, the SMT pads are either directly underneath.

* see also photo in stevenh's answer. Such feature is called *castellation* (thanks, The Photon).

Look also at Aries Correct-a-Chip adapters. Some of them (like this one) go from one SMT footprint to another SMT. There are also companies that specialize in making custom adapters. adapters-Plus, for example.

share improve this answer

edited Jul 12 '12 at 17:38

answered Jul 12 '12 at 17:12

Nick Alexeev
6,250 1 10 32

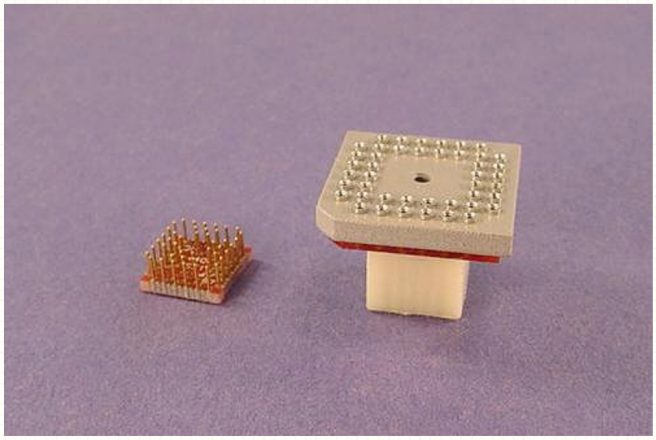
4 "castellation"? – The Photon Jul 12 '12 at 17:23

@ThePhoton "castellation" Yes, that's it, thank you! – Nick Alexeev Jul 12 '12 at 17:37

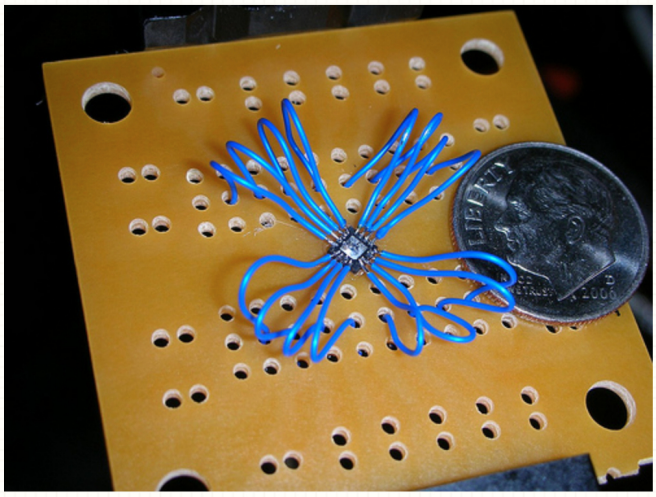
@ThePhoton - You could have said that sooner!! I've tried about every possible keyword on images.google.com!! Dammit ;-) – stevenvh Jul 12 '12 at 17:37

5

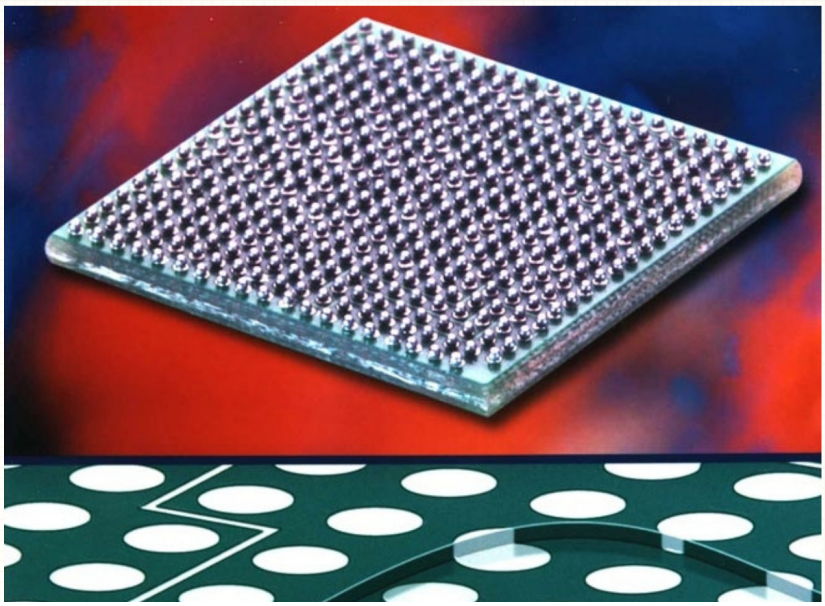
They make adapters for just about every footprint to any other footprint. And if it's not made, there are companies that will make one custom for you. But they are usually pretty expensive and, as you mentioned, tall.

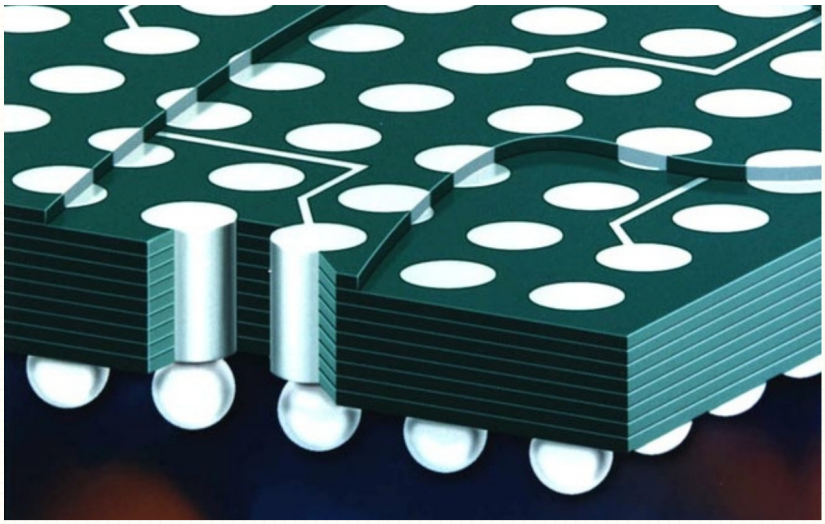


Another option is deadbugging the chip. But looking at your other question, you have a production run of ~70K units. So this solution would seem impractical. The chances of a wire being placed incorrectly or a solder joint not holding (especially if subjected to vibration) are probably too great on a run that size. And when you factor in technician time, it is also pretty expensive.



They do make BGA adapters so something that is more solid than deadbugging and shorter than a normal adapter is possible. In order to accept another PLCC32, the board would proabbyl need to be bigger than the original PLCC32 footprint and soldered using solder paste on the original pads and a reflow oven like a BGA component would be. Then the new PLCC32 would be soldered on the adapter's pads. Again, expensive.





Your best bet would be to consider using a new chip with a smaller footprint. Then having a small board made up that is the size of a PLCC32 with similar pins. I've seen something similar for 8051 ICEs. I couldn't find a good picture though.

For a production run of the size you're talking about. I would at least price out respinning the board. Comparing to the cost of a custom adapter plus technician time to install, the respin may be cheaper in the long run.

share improve this answer

answered Jul 12 '12 at 17:19
embedded.kyle
4,803 1 7 25

Your Answer

B I " { }

Sign up or log in

Sign up using Google

Sign up using Facebook

Sign up using Stack Exchange

Post as a guest

Name

Email

Post Your Answer

By posting your answer, you agree to the privacy policy and terms of service.

Not the answer you're looking for? Browse other questions tagged pcb soldering or ask your own question.

question feed

Stack Overflow
Server Fault
Super User
Web Applications
Ask Ubuntu
Webmasters
Game Development
TeX - LaTeX

Programmers
Unix & Linux
Ask Different (Apple)
WordPress Answers
Geographic Information Systems
Electrical Engineering
Android Enthusiasts
IT Security

Database Administrators
Drupal Answers
SharePoint
User Experience
Mathematica
more (14)

Photography
Science Fiction & Fantasy
Seasoned Advice (cooking)
Home Improvement
more (13)

English Language & Usage
Skeptics
Mi Yodeya (Judaism)
Travel
Arqade (gaming)
Bicycles
Role-playing Games
more (22)

Mathematics
Cross Validated (stats)
Theoretical Computer Science
Physics
more (7)

Stack Apps
Meta Stack Overflow
Area 51
Stack Overflow Careers

site design / logo © 2013 stack exchange inc; user contributions licensed under cc-wiki with attribution required
rev 2013.5.14.688

SOME RIGHTS