Manufacturing Electronics in China

2/8/2015 OFAE

I was asked if I had any experience in manufacturing electronics, particularly in the Pacific Rim, specifically China. Over the past 30 years, I have had my own manufacturing operations in Arizona & Mexico, with numerous products made by "CMs" (Contract Manufacturers) there, as well as Japan, Taiwan, South Korea and of course, China. To avoid writing a long book, I summarized the following information about what might be helpful for engaging a CM in China for your product.

This information is from my personal experiences and observations. The following is highly abbreviated with some very specific suggestions and answers. There could almost be a chapter for each bullet point. If you want more details or how a conclusion or recommendation came about (and what other options are available or were considered), just ask.

Here are some notes about Contract Manufacturing (CM) electronics in China in small volumes. These notes are greatly condensed using assumptions for the types of products and users on Tindie with PCB quantities of 100 ~ 1000 per order.

- Pacific Rim vendors will generally say "Yes" or "Yes we can" to any question about their capabilities. Not actually the case. If you ask "do you understand?" "Yes" can mean: "No", "Not completely", "I think so", "Mostly", "Yes, but exactly the opposite". Ask them to explain back to you, be prepared for some shocks.
- Any question or direction is understood literally at the simplest, easiest to do, level. Explain what you mean in detail & by physical example ("Show & Tell").
- Have a proficient English / Chinese interpreter <u>who understands the manufacturing process</u> (tools & procedures). Hong Kong and Taiwan are good places to look. A non-manufacturing experienced interpreter can make things worse than no interpreter.
- Looking for and getting the lowest price CM / Vendor can easily be very expensive in time and money. YOUR time & money.

Take or buy a small (<20mm D lens) 10x, lighted magnifier.

3 different types of items to manufacture:

1. PCB Assembly - PCB with all SMT & TH (Through-Hole) parts mounted
2. Wiring / Cables - Non-standard internal or external cables / wiring harnesses
3. Enclosed Product - PCBA, Connectors, Controls, User Interface, Labels in a case

Most small CM's are only proficient at only ONE of the above. The focus of these notes is on #1.

TH vs SMT. SMT is assumed to be done with machine, so less error. CMs will hand place small number of PCBs (<50~100), or some parts not on a tape & reel. TH is all hand placed and each pin hand soldered, but can be faster & cheaper. This includes TH parts on SMT PCBA (i.e. rotary encoder, connectors). Hand TH has potentially bad solder connections (i.e. PCBA doesn't always work, quits working after x weeks).

- Order enough to warrant set up of SMT Pick & Place machine. Put in writing "… All SMT parts to be Machine Placed…" and then verify after manufacturing.

Soldering SMT: Handling and Cleanliness is Vital!

- **Stencil** Starts with how solder is applied. Insist on metal stencil. Must use stencil machine with metal squeegee. Must be cleaned during use i.e. must not have any solder on bottom of stencil. Ask to see a stencil used a few weeks or months ago (intended to be used again). Look at walls (edges) of holes, must be smooth and clean (no small grains of solder anywhere). Metal stencil must be flat, have no dents or creases.
- **SMT Solder Paste** requirements:
  - Class III (smaller solder grains)
  - 3 or more ingredients: 97% Tin, 1% Silver, 2% Copper, (aka "SAC" for Sn Ag Cu)
  - Fresh i.e. only 1 label, complete with expiration date, <6 months
  - Refrigerated, & mixed before use
  - Preferably from Japan or Taiwan
  - Solder must NOT be left "lying around", open jar, used with dirty tools, mixed with different or old paste.

- SMT **Room** must be **Air Conditioned** (by refrigeration, no evaporative cooler) for a controlled 18-28c, dry environment.
- PCBs with solder paste must be **populated in < 1 hour** and **reflowed in < 1 hour**.
- Moving PCBs after mounting parts to reflow should be "in-line" with moving belts / chains. Handling by carrying PCBs and putting into "oven" creates badly soldered parts.
- **Inspection** during SMT operation with adequate light and mild magnification (3x):
  - PCB after Paste applied
    - All SMT pads should have equal <u>area</u> and <u>height</u> solder paste
  - PCB after parts are mounted
    - All parts should be straight,
    - All pins centered
    - All pins on top of solder paste (not pressed through to copper)
    - No loose or crooked parts
    - An occasional missing part is OK
  - PCB After Reflow Soldering
    - All parts are soldered straight
    - All parts are in the centers of the copper pads
    - No parts are standing on one pin (aka "tombstone")
    - No solder looks "grainy" or "sandy"
    - No solder on PCB where there are no pads. Looks like scattered sand (caused dirty stencil bottom)
    - No PCB burned / discolored edges
    - No PCB twist, bend or warp

SMT Bottom Line: Minimal Rework should be needed on an occasional PCB. If there are several people doing SMT rework, this is not the place. This indicates poor control of any or all of the above and many more issues.

NOTE: The above are not "Military Spec", "Medical Grade", or any stringent SMT assembly requirements, just basic SMT manufacturing practices. Many CMs (not just small or medium sized ones) can't pass all of these simple, rudimentary requirements. Unless you enjoy inspecting and fixing your PCBs, or want failing, intermittent products and a lot of customer complaints, keep looking. You only need 1 good CM.

Manufacturing Tips:

- If you care at all about the performance of a part, any part, buy them yourself and **provide a** full or partial **reel*** to the CM. * A strip of tape is a poor substitute. Most SMT machines require at > 6"/ 150mm of "leader" and another >12" / 120mm of "cover" tape (though this can be "spliced" on).
  - If you would be OK with buying all your parts at a swap meet or on eBay, do nothing.
  - If your worry that the parts might be pulled from a discarded PCB, do nothing.
  - If the parts you specify are merely suggestions, do nothing.
  - If you don't care that the pins of the parts are stripped of, or have oxidized tin finishes, which will sooner or later cause product failure, do nothing.
  - If you think that specifying in the BOM, PO and contract that only parts from "Authorized Distributors" can be used will eliminate this issue, good luck.
  - Be aware that used ("re tinned") and counterfeit parts are sold on reels with "manufacturer" logos. Parts sourcing is multi-level and un-traceable. Even inexpensive parts such as transistors and diodes are faked. Even large Chinese Semiconductor companies (i.e. Semtech) have standard part numbers, but minimal or inadequate specifications and QC than the major Global Semiconductor companies.
  - Most chip resistors and capacitors are OK. Understand and specify the exact type of ceramic capacitor (MLCC) you need i.e. COG, X7R, X5R, X7S, Z5U, etc. Resistors are easy: Thick Film = More Noise, Thin Film = Less Noise. Use 1% resistors
  - Over specify electrolytic (aka "aluminum") capacitors, use the next standard higher voltage than the circuit needs; for longer life on values > 200 uF, use 105c types & use SMT not TH.

If after your product samples check out OK, be on-site and watch the <u>entire</u> process of making your product for the first time. Things "change". Stay alert, pay attention, and ask questions. If you don't <u>understand</u> what is supposed to be going on, take someone with you who does.

- If the CM has suppliers who provide things for your product (i.e. PCBs, wiring, brackets, cases, sub-assemblies), visit and look over their facilities and operations.

Inspection & DFM (Design For Testing)

- Make 3 "Golden Samples" for comparison
- Make instructions with annotated photos and drawings (I use MS Word for both)
- Every PCB Assembly and completed product must be inspected. Many CMs have an Electrical Tester ("Flying Probe" tester, MDA aka Manufacturing Defect Analyzer). Some have a VI (Visual Inspection Machine). These take time to set up and program for your PCB, so typically the quantity per build needs to be 500 or 1000 or more to make it practical. The machine tests are no better than the people who set them up and use them. Ask how long the person at this position has been doing this exact same job. Experience is critically important. Finally, there is a thorough visual inspection, comparing it to the "golden sample" by a very experienced person. This is the only inspection option if the machine testing is not available or practical.
- Testing (and programming of ICs) is vitally important. You need to provide a durable, easy to understand and quick to use ("easy" for the operator not you) fixture and related items for this purpose.
  - NOTE: The TESTERS at the CM have NO UNDERSTANDING of ELECTRONICS, NO UNDERSTANDING OF THE PARTS USED and NO UNDERSTANDING of the OPERATION or USE of YOUR PRODUCT. Make the instructions and fixture so that your Grandmother could use them to successfully test your product.

Not only will this save time and effort for you, but ensures you will only get working devices. It is also fair to the CM, because if it does not program or work, they get a second chance to fix it (clean contacts, remove excess solder or shorts, re-solder poor connections, etc.) before having to scrap it.

- Designing, making and verifying the test fixture / device is not trivial and may take as much time and effort as designing the PCB product. The alternative is sending the entire batch to you to program and do "clean-up" rework, with no recourse to the CM for any errors.

Design Tips:

Make your schematic first. Find or make your "parts" i.e. schematic symbol and PCB "footprint" / "decal" / "pad array". When choosing parts, think about manufacturing. Can this be put on the PCB with a machine instead of hand soldered?

Is there **protection** for:

- ESD (Electro Static Discharge)? Remember that the built in ESD diodes in most ICs are there for handling in manufacture, not actual product use.
- Reverse Polarity?
- Excess current in or out?
- Transient pulses (from inductive loads, relays, solenoids, motors, speakers. Remember a straight wire a few feet long is a pretty big inductor to pulse edges measured in nanoseconds).
  NOTE: ESD Diodes are for 1 us pulses, TVS Diodes are for > 50 ms pulses. ESD Diodes will not protect against inductive pulses!
- Be careful using MOV (Metal Oxide Varistor) and PPTC (Polymeric Positive Temperature Coefficient, aka Polyfuse, Polyswitch) devices for protection. Both deteriorate capability with each use. MOVs loose surge capability with each transient they suppress, PPTCs increase in resistance with each activation. In as few as 10 activations (protection events) they can render your product useless.
- All CMOS ICs (especially microcontrollers PL arrays) need a pull up or pull down on most of the digital pins. While most uCs have internal, programmable pull-up or pull-down resistors, they are <u>not active until after</u> the part "boots-up". Adding a 100k resistor may be needed if the state of the pin during "boot-up" is important.

PCB Design

- If at all possible, make it a RECTANGLE. Much cheaper and easier to assemble than beautifully curvy routed PCBs. If your end product with the PCB inside needs to be curvy, put a rectangular PCB in a curvy enclosure.
- If it is small, plan on an array / panel of 2x6, 4 x 5 …etc. PCBs on a single large PCB. These can be manufactured faster and cheaper than individual PCBs. They are scored with a groove to snap apart into individual PCBs after manufacturing. Make sure there is a margin > 2mm near the edges from any parts, do not literally "snap" them apart. Gently bend back & forth instead. Parts near the snap lines are subject to a momentary high stress which can crack the parts (especially MLCC caps, crystals and thin, large, fine pitched uCs)
- After visiting the CM's favorite PCB supplier, ask the CM what the maximum panel size (& component height) they can handle. Ask if they need a "manufacturing" or "tooling" edge for production. This is often a breakaway 10-15mm added edge on two sides of the PCB with holes and fiducials for their equipment.
- Place 2-3 fiducials 1mm or greater D with a hole in the solder mask 2x larger near the opposite corners of EACH PCB. This fiducial should have no solder mask and no solder paste.
- China has a unique PCB finish: Tin Spray. This is thinner & flatter than the lousy HASL surface. Use this unless you have very fine pitch parts such as micro No lead (i.e. QFN) or micro BGA or LGA. In these cases use MacDermid Silver or ENIG (Electroless Nickel Immersion Gold) finish. The silver has a short shelf life and can tarnish but is cheaper and makes a better solder joint.

- Use LPI (Liquid Photo Imageble) solder mask in the color of your choice (green is usually cheapest).
- Solder mask should be between each pad / pin of ICs, even fine pitch Quad Flat packages. Some PCB vendors just put a rectangle of no solder mask on pin sets with nothing between the pins to prevent shorting or later incursion of the dreaded "Tin Whisker"
- Standard PCB Material is 2 or 4 sides, 35 um (1 oz.) copper layers, 18um (½ oz.) holes. 70 um is generally available. Thicker copper is custom
- Cheapest is 2 layer, 4 allows Ground & Power planes, more layers not only cost more, but require higher skill & experience levels to design and manufacture.
- The PCB raw material is covered in copper. You already paid for it, so use it! Learn to use copper pours for current and shielding purposes.
- USE MILLIMETERS & MICROMETERS! Forget "thousands", "thou", "mils", and fractions of inches.
- The smallest hole (including vias) should be ≥ 0.5mm (0.020"). Most Chinese PCB vendors have difficulty making holes with smooth, plate-able holes any smaller (most PCB vendors in US, Japan, Korea, and Europe have no problem with much smaller holes).
- Keep traces & spaces to ≥0.25mm (0.010"), especially with thicker copper layers.
- If the PCB vendor can't do the above specs, find another.
- Provide Gerber RS-274X files for all layers, including solder mask and solder paste / stencil layers. Insist that the PCB vendor use them (they like to "make up" their own!).
- The smallest practical size for "Silkscreen Text" is 1mm high by 0.15mm thick. This may result in some barely readable text / numbers from many PCB vendors. 1.5mm high x 0.15 – 0.20mm thick should look good from most PCB vendors.
- 1608 metric / 603 resistors and capacitors should be no problem. 1005 metric / 402 or smaller parts disqualify the vast majority of PCB and CM vendors. Make sure the one you choose can do the finest pitch and largest parts in your design. Ask to see examples from older PCBs they made.
- CURRENT RULES! Make all current paths as Wide, Thick and Short as possible, even if it is only 50 mA DC. The hidden issue is that no one actually measures dynamic current / current spikes which cause a lot of performance issues, as well as interfere with everything else (even other circuits on the PCB). If there is a switching voltage regulator IC or Module on you PCB, find the app note for PCB layout. Linear Tech and TI (including National) have suggested layouts on SMT switching regulator IC Data Sheets and Application Notes explaining them. Study them! A 6 mm trace can make the difference whether you have an unstable, oscillating, regulator, or a "quiet" low ripple workhorse regulator that will outlive your product. Or just duplicate the data sheet /application note example layout exactly!

DFM (Design For Manufacture)

When designing a PCB, it should be easy to manufacture. The parts should not be too close together or cause interference with the SMT and reflow soldering machines. Visualize the process of the CM manufacturer, which is quite different from your hand building the prototypes. Cruise YouTube and CM web sites to see how it is actually done, imagining your PCB and parts in the video.

When visiting some CM facilities, think of how you could make your design easier and more "fool-proof". Take your prototype and ask what they would suggest changing to make it better or easier to make & test.

Scams

The world is full of people who portray themselves as "Consultants", "Facilitators", "Agents", and "Coordinators" who want to help you connect with the right CM to get the best quality, and delivery at the lowest price for your product. While there are a few legitimate persons who do exactly that, most are just people who speak some English and know "about" some people in the CM / electronics business. Their

primary, and usually ONLY, goal is to facilitate your money into their pockets. Some of these folks and companies may have an office near wherever you are. Most likely:

A. You are too small and inexperienced to use the few legitimate ones.
B. The others will only waste your time and take your money

The above is true the world over, but seems to be 1000 times more so in China. The best recommendation is to find a CM on the internet (Alibaba comes to mind) and then start corresponding. If there is a reasonable response, write that you will visit them for an inspection, but would like some information first: photos of every aspect of the operation , equipment lists, number of production workers, the experience of the SMT operators (Solder prep, stenciling, Pick & Place, Reflow), the experience of: production supervisors, test operators, technicians and rework specialist. Get the names of the above people and a photo if possible. Try to get referrals of the CM's existing customers (may not be possible) or clear, hi-res photos of the items they have made (with covers off, or just the PCB assembly). Why say you will visit for an inspection? Because all of the information could be totally false and actually be of other facilities and people copied from the internet. When you actually plan a visit, mention you would like to meet all of these people and ask them a few questions (through a translator).

Another avenue to look into is Hong Kong. It is only a ferryboat ride from the manufacturing in Shenzhen, and has a lot of English speaking people familiar with technology. It's a good place to look for an interpreter. Taiwan also has some English speaking people who are knowledgeable about electronic manufacturing and particularly CMs in China. This is because most Taiwanese companies have a manufacturing subsidiary, partner, or CM in China.

Yes, there are some really unconscionable, evil people who just want to get your money, and you will likely meet one or two, but there are only a few. They tend to speak English too well, dress sharply, always smiling and joking and want be your buddy. Most of the world calls them "con-men" (confidence crooks).

The Chinese (or any other people) are NOT evil, cheating or immoral. They are just trying to make a living, are desperate to get business, and often act out of fear or ignorance. They will tell you what they think you want to hear because they are afraid of not getting the business and not having an income.

Tools

Free or "dirt cheap" can be very expensive! As much frustrating time as you may have invested in trying to figure out the "global standard", free PCB tool Eagle, you still can't make a great PCB. People spend more money on cooking items & coffee brewing accessories than the tool that builds the underpinning foundation of their electronic design: the PCB. Over the past 30+ years, I have used them all. I put up with PADS at work every day. Previously it was ORCAD, which is even worse. I make no money or have any relationship with these guys (other than being a registered customer), but I highly recommend that you try **Number One Systems' Easy-PC**. For $500 to <$1000 it will make your life easier and a lot more productive than all of the free or cheap PCB software tools in existence. http://www.numberone.com/easypc.asp

If you can spend more about $3500, plus an annual licensing fee, get Altium.