

Kepler Communications double sided two layer PCB assembly process guide.

Generally there are two case assembly scenarios: (a) automated or semi-automated process or (b) manual with hand soldering of all components. Thus each way requires a separate set of tools and steps to take.

Basic production facility requires the following:

- A. ESD protected area with discharge plates and wrist wraps.
- B. Bright overhead lighting.
- C. Heavy weight workbench with flat even surface and ESD thermal-resistant pad.
- D. A magnifying glass and/or binocular microscope with local light source.
- E. Heavy duty soldering iron such as Ersa i-Con Nano (running 80 watts i-Tool) with 0102CDLF18L and/or 0102BDLF20 soldering tips.
- F. Controllable hot air gun such as Weller WHA 900 or equivalent.
- G. Hand tool set includes: curve-shaped, reverse-action tweezers, screwdrivers, hard and soft brushes, squeegee and so on.

Depending on production volume, machinery tools list might be as suggested:

- A. Benchtop SMT stencil such as SMTnet SPR-25 or Industrial automatic printer such as Yamaha YCP10;
- B. Manncorp MC-385 or similar pick and place machine;
- C. Vitronics XMP2 wave soldering oven or similar;

General expendables list:

- A. No-clean soldering paste.
- B. No-clean flux gel (syringe packaging preferred) and/or liquid flux (pen packaging).
- C. Flux-core soldering wires, up to 0.8mm thickness.
- D. Desoldering wick.
- E. Cramolin Flux-off or Isopropyl alcohol spray.
- F. Soft paper wipes.

In case (a) the list of steps as is following:

1. Setup machines to run the process, ensure vacuum pump and air compressors are running, load proper files, warm up the wave soldering oven and so on.
2. Check the solder paste for its temperature after extracting from storage.
3. Visually inspect the bare PCB for any defects such as coating mask delaminate, extra silk screen paint over pads and so on. Discard the board if quality standard is not met;

4.
 - 4.1. Run PCB through the Printer machine or
 - 4.2. Dispense solder paste over a metal frame SMT stencil using squeegee.
5. Visually inspect the pasted board to check for any inconsistencies. Sweep paste, wipe, clean with alcohol and repeat printing if necessary.
6.
 - 6.1. Insert pasted board into pick and place machine or
 - 6.2. Place SMT components with curve-shaped tweezers, paying attention to polarity and key pin designators.
7. Visually inspect for any serious misalignments or shifts of components. Fix with tweezers if necessary.
8. Run the PCB through the soldering oven. Let it cool down for a couple of minutes afterwards.
9. Visually inspect the soldered board.
 - 9.1. Remove soldering joints (shorts) with soldering iron, if any.
 - 9.2. Touch up with soldering iron any unsoldered component pads.
 - 9.3. Resolder shifted or turned over components with a hot air gun.
10. Repeat steps 2 to 9 for the bottom side if applicable.
11. Install through-hole components J1 through J4 and P1.
12. OPTIONAL: Apply little gel flux with on pins soft brush.
13. Solder through-hole components with soldering iron using proper thickness solder wire.
14. Visually inspect outcome PCB for any soldering imperfections. Fix if necessary.

In case (b) the list of steps as is following:

1. Visually inspect the bare PCB for any defects such as coating mask delaminate, extra silk screen paint over pads and so on. Discard the board if the quality standard is not met.
2. Touch up a single pad of each SMT component with soldering iron with a bit of solder.
3. Check for components polarity (if any) and solder SMT components to the pad.
4. Apply gel or liquid flux to all other SMT pads and solder using hand soldering iron with appropriate thickness solder wire.
5. Visually inspect the soldered board.
 - 5.1. Remove soldering joints (shorts), if any;
 - 5.2. Touch up with soldering iron unsoldered component pads, if any.
6. Repeat steps 2 to 5 for the bottom side if applicable.
7. Install through-hole components.
8. OPTIONAL: Apply little gel flux on pins with soft brush.
9. Solder pins using hand soldering iron with appropriate thickness solder wire.
10. Visually inspect outcome PCB for any soldering imperfections. Fix if necessary.
11. Remove excessive flux if required. Dissolve with Flux-off or Isopropyl alcohol and hard brush and clean up with paper wipes.

Note: depending on volume and quality required, some steps can be omitted or intensified.