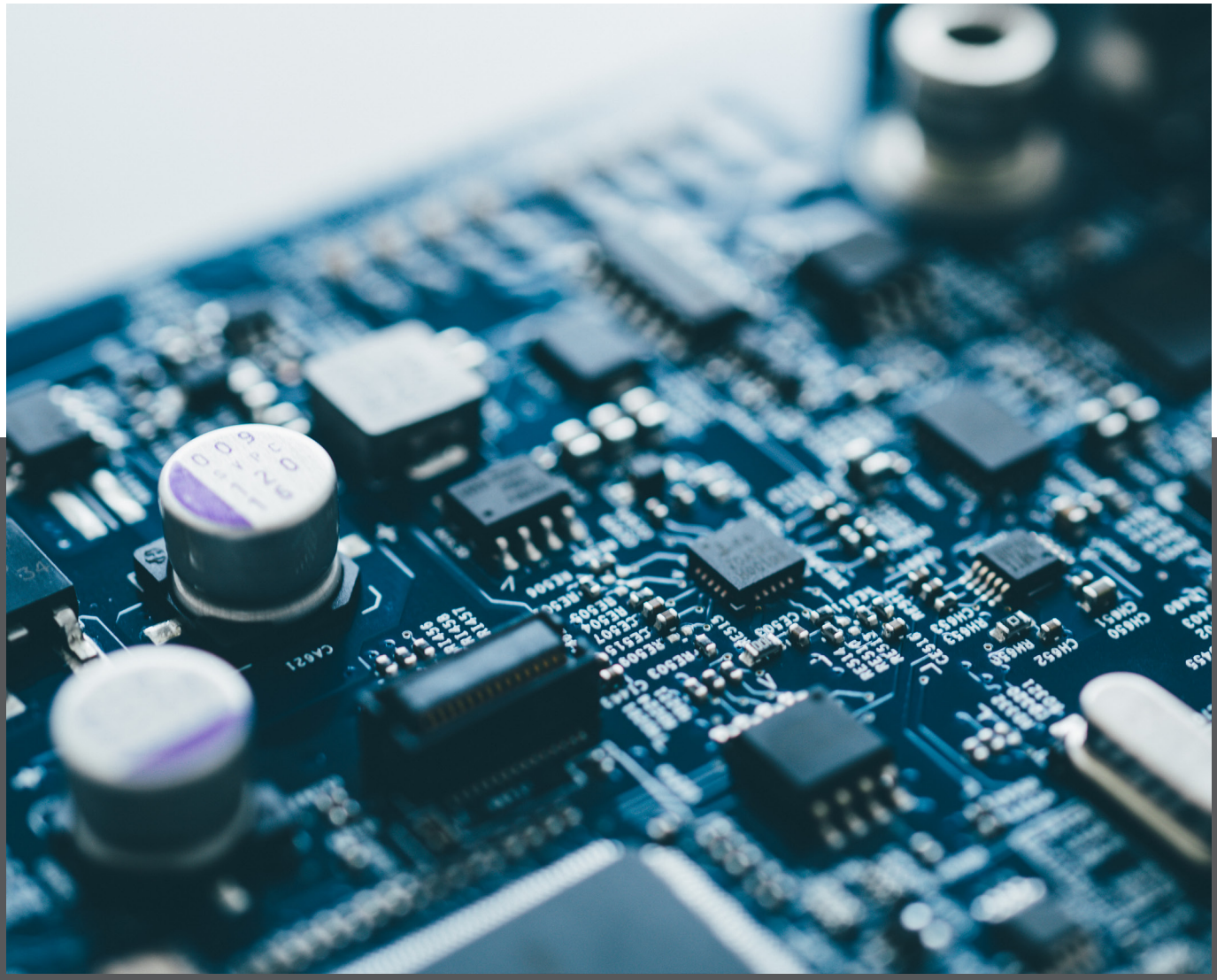
CIRCUITWISE

Bringing innovation to life

ELECTRONICS MANUFACTURING CHECKLIST



ABOUT US

Circuitwise is a contract electronics manufacturer specialising in complex electronics devices for companies that have a requirement for absolute reliability.

We manufacture devices across virtually every industry including:

Medical Military Mining Transport Gaming IOT Telecommunication

OUR CORE SERVICES INCLUDE

- **Printed Circuit Board (PCB) Assembly**
- **Turnkey Product Assembly**

PCBs are manufactured using both surface-mount technology and through-hole placement. We have a dedicated rapid prototyping area for quick turn around and providing fast feedback on manufacturability.

We assemble all the components into the final product and box it ready for shipping. In addition to the electronic parts, we can purchase all the enclosure and mechanical components or use customer-supplied components Free-in-Aid.

30+ YRS	EXPERIENCE SINCE 1988
8 MILLION	PCB ASSEMBLIES
40,000	CONTRACTS DELIVERED
1121	SATISFIED CUSTOMERS

Certified System

Certified to manufacture
ISO 13485
Medical Devices

Australian Owned & Operated

02 9836 4900 www.circuitwise.com.au sales@circuitwise.com.au

www.circuitwise.com.au

OUR SYSTEMS

Our facility at Norwest Business Park in Sydney houses the most advanced electronics manufacturing capability possible, including:

- Smart component storage
- Dedicated prototyping area
- PCB assembly lines and product testing/assembly areas, shown above
- Kanban storage facility

Our highly automated systems deliver information to workstations across the factory floor. The whole manufacturing area has full antistatic flooring.

OUR QUALITY

Developed over 15 years, our quality system is constantly refined to deliver:

- Reliability
- Traceability
- Continual improvement
- Regulatory compliance

Circuitwise is certified to both the ISO 13485:2016 quality standard for manufacturing medical devices and the ISO 9001 quality standard.

Our robust documentation processes ensure we deliver exactly what is ordered. Our engineering team creates a detailed plan for every customer product and the quality team reviews this before commencing production. Normally every board is built to IPC-A-610 Class 2 standard or Class 3 if required.

All documentation required for regulatory compliance is managed and stored in our system in perpetuity. We also institute advanced product quality planning (APQP) and production part approval processes (PPAP) and any other controls as required by the customer. We can issue certificates of compliance as required, including Underwriters Laboratories (UL) certificates.

TABLE OF CONTENTS

Time, Quality and Cost Factors

TABLE OF CONTENTS

DOWNLOAD THE CHECKLIST
HTTPS://WWW.CIRCUITWISE.COM.AU/PCB-CHECK-LIST

CIRCUITWISE
Electronics Manufacturing

DESIGN FOR MANUFACTURE

Following are our key recommendations for designers of PCB assemblies to optimise your design for manufacture:

01. PUT ALL SMT COMPONENTS ON ONE SIDE

A major cost component for electronics assembly is setting up the SMT machines before a run. Set up includes kitting up, verifying you have all the right components, programming the machines, and first article inspection. Putting all your SMT components on one side means this process only has to be done once. The same principle applies when putting the boards through the reflow ovens.

---

02. PUT THE HEAVY COMPONENTS ON ONE SIDE

If you do need a two-sided board, putting all your heavy components on just one side also smooths the production process. If heavy components are on both sides, then when the board goes through the reflow oven a second time, there is a risk heavy components on the bottom side can fall off. To prevent these components falling off, they need to be glued in place, which adds cost to the process.

---

03. MINIMISE THROUGH-HOLE COMPONENTS

Through-hole components are costly to insert and solder, as these are labour-intensive processes. Virtually all components are now available in SMT. Some designers are reluctant to move away from old proven designs that needlessly specify through-hole components. However, for large volumes, it is certainly worth considering a redesign. Note, some through-hole components can be used on the SMT lines.

---

04. LEAVE ADEQUATE SPACE FOR SELECTIVE SOLDERING

For boards with mixed SMT and through-hole components, selective soldering is often used where wave soldering is not economical. To optimise a design for selective soldering, make sure you leave adequate clearance for the solder fountain to reach the pins – the more space, the better. If other components are too close to the pins, it limits the options the manufacturer has for selective soldering of those components.

---

05. MINIMISE COMPONENT END-OF-LIFE RISKS

Aim to minimise the risk of components in your design hitting end-of-life (EOL). Select components from reputable suppliers with a stated commitment to the component. Risky components are those selected from suppliers that aren't well known or if there is only one supplier of the part. If a component is available from multiple suppliers, the risk is much lower. Circuitwise knows the trusted suppliers and is often given advance notice of any components going EOL.

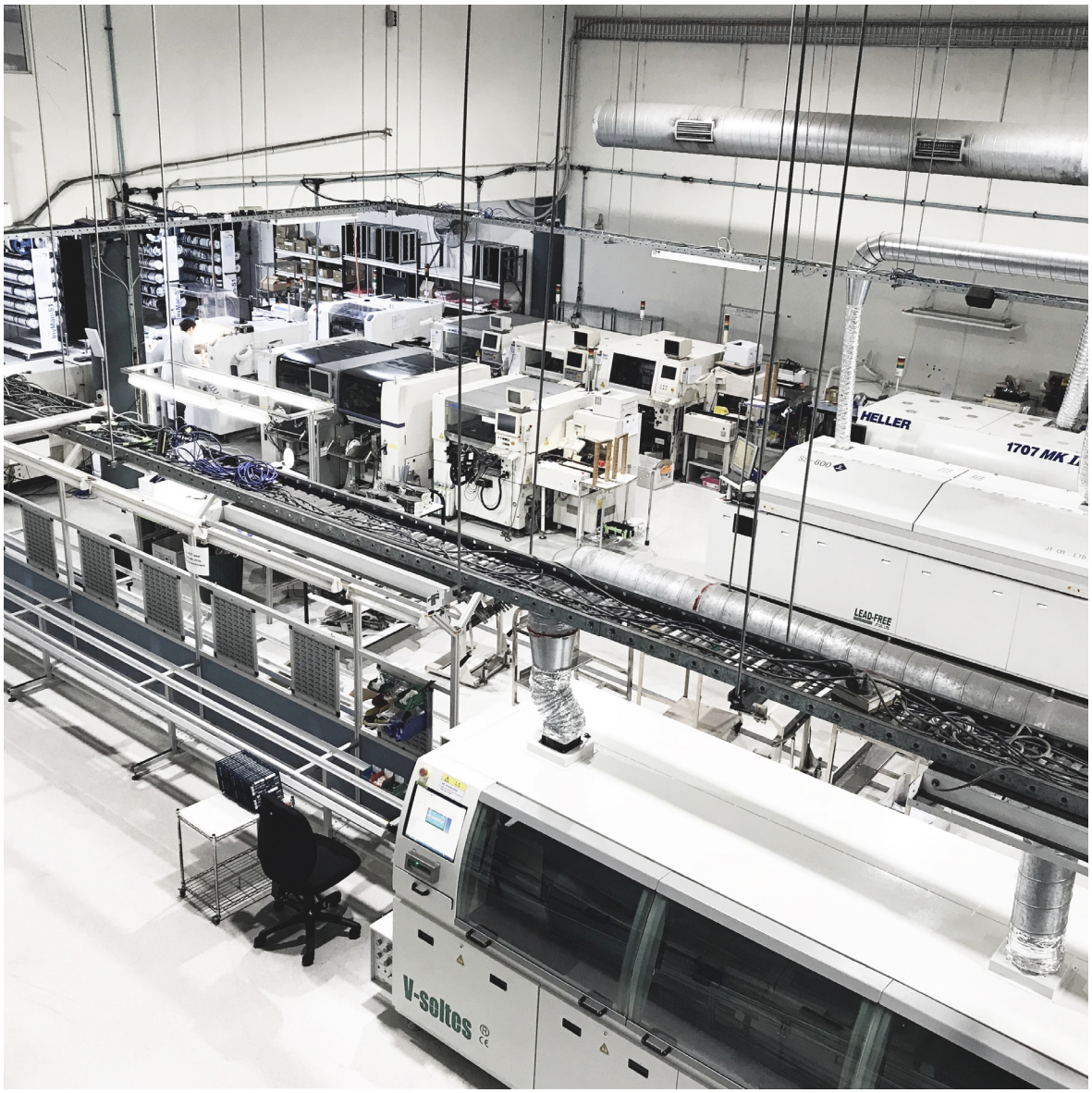
06. MINIMISE TESTING TIME

Many people treat testing as an after-thought but it is often the biggest bottleneck and one of the costliest parts of the process. Ensure your design is easily testable with probe points at appropriate circuit nodes and include as much self-testing into the product as possible. If the testing process is unavoidably long, consider testing on a panel or test multiple products at the same time.

---

07. OPTIMISE PRODUCT ASSEMBLY

Consider investing in a product design where the various elements can be easily snapped together without any need for screws. Screwing parts together takes time and can be a significant cost component. For example, industrial designers can create a casing with snap-fit forms that can be assembled in seconds, compared to minutes with more conventional loose fasteners. This approach makes sense for high volume products where the cost of designing the casing and manufacturing the tooling can be justified.



PRODUCTION MANAGEMENT

08. SPECIFY LEAD TIMES

Firstly, specify when you would like your quote by. The longer you give us to research procurement options, the better price we can give you. Similarly, your request for quote should include the quantity of PCB units required and the date by which you would like us to deliver them. Again, the longer the lead time, the better the price. If you demand a fast turn-around time, it forces us to go to just a few suppliers who can demand a premium.

09. IDENTIFY HIGH-CONFIDENCE COMPONENTS

During the proof-of-concept or prototyping phase, consider ordering enough parts for the full expected production run. Ordering just enough parts for a small number of prototypes can be expensive. If you are confident of the design and moving on to full production, it may be possible to order enough of the non-critical components for the full production run, only holding back parts that are most likely to change.

10. SPECIFY YOUR PRODUCTION ROADMAP

If you plan to manufacture regularly, consider using Circuitwise's Kanban system, whereby we can purchase all the materials in bulk but only do an assembly run as required. For example, we can order enough components for a production run of 1000 but only manufacture 250 at a time on a pre-set schedule or as required. Generally we quote annually, manufacture quarterly and deliver monthly. This approach lowers overall unit cost for customers that expect regular business but want to lower inventory costs.

11. DECIDE ON FIRST BOARD SACRIFICE

Considering ordering one extra PCB assembly to be sacrificed in order to optimise the reflow oven profile. An optimised reflow profile is useful when there is a mixture of large and small components or varying copper thickness, leading to uneven heat distribution and potential issues such as tombstoning. A sacrificial board is the first fully assembled board of a production run with sensors drilled into it. The sensors determine heat distribution during reflow and a program is used to determine the optimal temperature for each zone of the reflow oven.

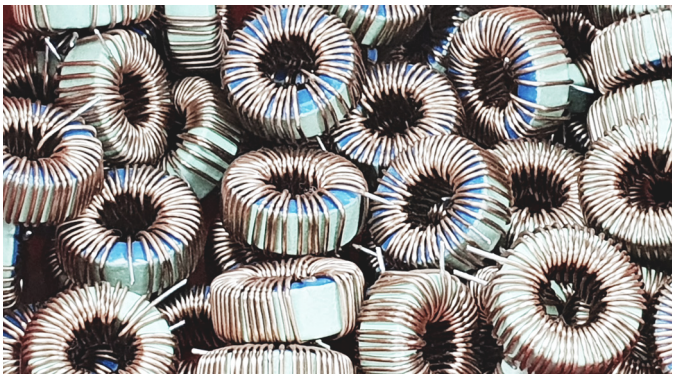
12. DETERMINE IF X-RAYING IS REQUIRED

Circuitwise owns an x-ray machine that is used for a variety of purposes. It can be used to verify joints, particular those that can't be visually inspected. For some designs, it is important that certain areas of the board are free of metallic solder and this is easily checked by X-ray. X-raying can also be used to check for counterfeit parts, which is an important consideration if a component has been declared end-of-life by a manufacturer. X-raying is not used routinely but consider if it is needed in your case.

13. ADVISE THE TESTING PROCESS EARLY

Define what the testing procedure will be and supply that when asking for quotes on PCB assembly. Many people focus on shaving cents off the unit cost of an assembled PCB and optimising an efficient manufacturable design, without realising that the testing process can be a significant percentage of the delivered cost and the main efficiency bottleneck. Even if you have not finalised the testing procedure, provide a description of what steps will be required and the estimated run times for each step, including setup, so that your quote will be more reflective of the actual delivered cost.

BILL OF MATERIALS (BOM)



14. MANAGE BOM REVISION NUMBERS

Please supply the BOM in excel format including all the information below. Clearly indicate the revision number of your BOM (e.g. using an X.X format). With change requests, attach a cover letter to your email indicating what the changes are, and update your BOM revision number, clearly marked at the start of the document. Do not request changes in the body of an email.

---

15. NAME THE MANUFACTURER AND THEIR PART NUMBER

For each component, either the manufacturer part number or supplier part number is a must. We prefer both as this ensures there is no possibility of misidentification. Avoid use of small, obscure or single source manufacturers which reduces our ability to get the best price and increases the risk of early end-of-life.

---

16. INCLUDE A DESCRIPTION OF THE PART

When supplying the manufacturer's part number, make sure you supply a description of the part. We use this to check if we have the right part and it minimises the chances of a typographical error leading to the wrong part order. Characteristics that are good to describe are properties such as function, colour, width, height, temperature range, package type, etc.

---

17. PROVIDE A SUPPLIER PART NUMBER

Circuitwise also likes to have a supplier part number. Suppliers are companies that distribute components of multiple manufacturers. Leading suppliers include Digi-Key, Element14 and Mouser. A supplier part number is a backup and ensures the part is a mainstream product.

---

18. CONSOLIDATE IDENTICAL COMPONENTS INTO A SINGLE LINE ITEM

Some CAD packages will output a BOM with a single line item for every component, even if there are 50 identical components in the BOM. BOM's supplied in this format force us to do the consolidation for you, in determining the quantities of each component required. Consolidation is an unnecessary step that increases the risk of error occurring. Aim to have your design package do the consolidation for you to reduce the risk of ordering the wrong number of parts.

---

19. DO NOT INCLUDE WASTAGE ESTIMATES

When ordering quantities for each part, you can assume we will account for any wastage in parts. Wastage occurs when loading new component reels into SMT machines, particularly for small parts. However, if you are supplying the parts as free-in-aid, be sure to account for wastage. For small components wastage is typically in the order of 2%.

---

20. INCLUDE THE REFERENCE DESIGNATORS

In specifying the parts required, don't forget to include reference designators – the codes given in the legend of a PCB circuit schematic. We want to check the location of each component on a board, to determine if the location has any effect on how to place a component or which packaging format we should use for the components. For example, if we cannot place the component with SMT using a conventional reel, then some other method must be used.

---

21. SPECIFY THE CRITICALITY OF EACH PART

The most important part of a good BOM is to define the criticality of each part. Criticality relates to the tolerance for variation in the part specification and the care required in managing the part. High criticality means the part must be exactly supplied as specified – usually a specific part number from a specific manufacturer. Parts are either critical or open. If substitutions are permissible (open) for a component, we may be able to get a better price.

---

22. NOTE MSL RATINGS OF 3 OR MORE

The main care concern usually relates to the moisture sensitivity level (MSL) rating of the component. Some components will absorb moisture if left in ambient room conditions, with the risk of moisture expansion damage during manufacturing. For MSL ratings of 3 or above, Circuitwise must verify vendors to manage these components appropriately and we store them in a dry cabinet on arrival. Therefore, it is critical to alert us of any MSL 3+ components.

---

23. CHECK COMPONENTS MEET REGULATORY REQUIREMENTS

Occasionally, we have customers who have to replace components because they discover at a late stage they are not compliant with the regulatory requirements of the countries the product is being exported to. For example, in the European Union, products must comply with directives around the use of hazardous substance (RoHS) and conflict minerals which can be found in electronic components. Knowing what regulations you will need to adhere to and working closely with Circuitwise to source the right components will prevent headaches down the track.

---

24. CONDUCT AN INTERNAL REVIEW BEFORE SENDING

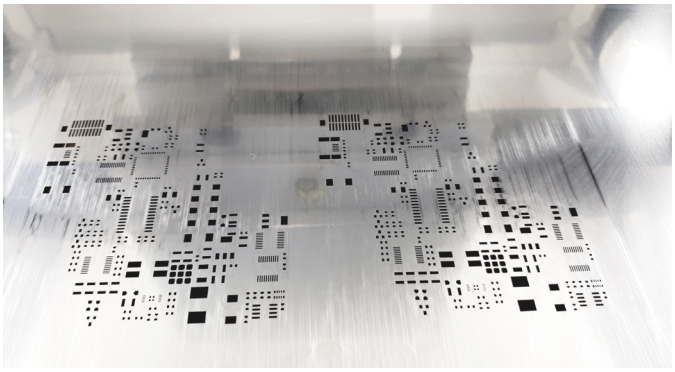
After being immersed in a product's design for months, it is easy to not see the final output with the same clarity as a fresh set of eyes. We recommend engaging another designer to conduct a final sanity check of all your documentation to check for any issues that have been missed. In particular, check component part numbers match the description and PCB footprint used.

---

25. MAINTAIN YOUR OWN MANUFACTURING PACKAGE

The BOM forms one part of a larger manufacturing package that serves other stakeholders, including suppliers of mechanical parts and regulatory authorities. The BOM should be maintained in the context of your wider design transfer documentation. This is particularly important for future product development. It is not uncommon for clients to ask Circuitwise for documentation from past products because they have lost their own files. We maintain records of everything sent to us in perpetuity but don't normally have access to the full design package.

GERBER FILES



26. INCLUDE A PCB FABRICATION INSTRUCTION

To ensure the trouble-free fabrication of the bare PCBs, it is not enough to supply Gerber files alone. Include a manufacturing instruction document with the information outlined in the points below. This document should be zipped with the released design documents and include a record of the specification for PCB fabrication. Including specification detail in email text is not recommended as this reduces design traceability.

---

27. DEFINE TRACE WIDTH AND SPACING

Specify if there are any restrictions on trace width and minimum spacing. PCB manufacturers may want to alter these dimensions to optimise manufacturing. The smaller the trace width and spacing requirements the more difficult it is to fabricate your PCB. Common minimum trace width and spacing is 0.15mm or higher and this will be affected by the copper thickness of your PCB layers. Indicate if impedance control is required for any trace and specify the impedance in relation to the ground plane. This will allow the PCB manufacturer to do a calculation and, if necessary, alter the track widths for the layer stack's tolerances.

---

28. SPECIFY THE PCB LAYER STACK

The PCB layer stack sets out the number of PCB layers, layer thickness, and material type. The stack is usually illustrated with a picture. At a minimum, specify the number of layers, layer order (e.g. top, mid-layer 1, mid-layer 2, bottom layer), and the distance between each layer. Other items that may be included are copper thickness, plating, and layer materials (FR4 S1000-2, high-Tg, etc). These properties affect the characteristic impedance of traces on the board. Impedance is related to the properties of the circuit board material (dielectric constant), distance to ground plane, and other factors that relate to the setup for PCB fabrication. The thermal properties of the PCB material also affect the expansion and contraction of the board during the assembly process which can place stress on vias or cause board delamination if incorrect material is specified.

---

29. SPECIFY MINIMUM/MAXIMUM HOLE SIZE

Specify the minimum and maximum hole size. A good default signal via size uses a 0.6mm pad with a 0.3mm hole. If you intend to use small via size below 0.2mm the maximum thickness of the associated PCB layers may need to be considered. CAD packages will generate drill files that specify the size and number of each hole type used and whether the hole is plated or unplated. It is a good idea to review all unplated holes to make sure they are intended to be unplated. Buried and blind vias should only be used when necessary as they will increase the board cost and fabrication time.

---

30. SPECIFY MINIMUM ANNULAR RING SIZE

CAD packages default to an ideal design. In the real-world, tolerance for manufacturing variance needs to be included. When drilling holes there may be a slight offset from the centre of the pad introduced during PCB fabrication. To allow for this offset a minimum annular ring size is specified. A good default minimum annular ring size is 0.15mm which can relate to a suggested via size of 0.6mm pad and 0.3mm hole size (maybe include a picture showing pad size, hole size, and annular ring).

---

31. SPECIFY SURFACE FINISH

Don't forget to specify the surface finish. Hot air levelling (HASL) is the most commonly used surface. HASL is low cost, re-workable, has a long shelf-life and is suitable for lead-free soldering. However, HASL surfaces are not as flat as other options, which may be preferred for particular components. Other surfaces include immersion gold (ENIG), hard gold, immersion silver, immersion tin, nickel palladium and organic protective film.

---

32. SPECIFY ANY SPECIAL PROCESSES

Special processes for PCB fabrication include gold finger plating, via-in-pad, or via plugging. If special via processes are required, state these in the manufacturing instructions with details such as location and technique to be used. If your board has gold fingers, specify if they need to be hard-gold plated, or normal electro-less plated. Identify if edge plating is required. If a product will be removed and inserted many times, then you need to increase the gold thickness to tolerate the wear and specify a harder gold finish.

---

33. SPECIFY THE SOLDER MASK AND OVERLAY COLOUR

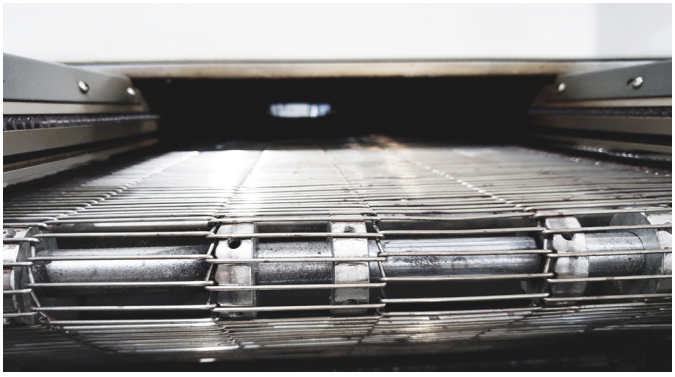
The standard solder mask colour on a bare PCB is green. If you want any other colour (e.g. red, blue, purple, white, yellow, etc), please specify. Solder mask or resist is so-called because it stops solder adhering to the board in areas where it is not wanted. It helps protect the board and prevents short circuits between traces. Overlay or silkscreen colour is usually white or black. This is the colour of component outlines and text such as designators on the PCB.

---

34. ADD IDENTIFICATION DETAILS

If a board arrives ready for assembly without any identifying markers, it can increase the risk of misidentification. To help us overcome this risk, add a PCB part number and name to either an overlay or top or bottom electrical layer. It is a good idea to include a revision number as well.

ASSEMBLY INSTRUCTIONS



35. INDICATE IF LEADED SOLDER IS REQUIRED

We use lead-free SN100C or SAC305 solder as a standard. Indicate if leaded solder is required. Lead solder has been largely phased out due to environmental concerns. However, in some circumstances, leaded solder is still required. Leaded solder has higher performance characteristics in harsh environments, including resistance to cracking of soldered joints due to its eutectic property.

36. NOTE MANUAL SOLDERING REQUIREMENTS

Note any components that need to be manually soldered. Manual soldering is usually required if a component cannot withstand the high temperatures of reflow ovens or wave soldering. Examples of temperature sensitive components include some special types of reed switches, aluminium and tantalum ceramic capacitors, crystals, oscillators and fuses. Some components such as fans and certain types of large connectors are also not amenable to SMT assembly. For such components, high temperatures can induce immediate failure or reduce their long-term reliability.

37. SPECIFY IF CLASS 3 QUALITY CONTROL IS REQUIRED

Circuitwise manufactures all assemblies at a level that allows them to pass the IPC-A-610 Class 2 standard for acceptability of electronic assemblies. While we manufacture both class two and three products with controlled processes and environment, the difference is that we inspect class three products to assure the quality level. Class three standard is usually only required in safety critical environments such as medical, mining, defence and aerospace – which have regulations requiring special quality inspections on every assembly.

38. INCLUDE A PHOTO FOR COMPLEX ASSEMBLY

Sometimes PCB assemblies have an unusual or complex set of components. For example, some large through-hole leads may be bent to fit large components in a casing. For these kinds of assemblies please include a photo of what the final PCB assembly should look like. If the product has not been assembled before, most CAD packages can generate 2D or 3D views.

39. SPECIFY CLEANING REQUIREMENTS

Tell us if cleaning is required and, if so, the type of cleaning. Circuitwise uses a no-clean flux assembly process, and there is no need to clean the boards. However, in some cases, there may be specific requirements for the cleanliness of a PCB (both visual and ionic contamination level). Cleaning options include simple brushing, vapour degreasing with an ultra-sonic final rinse bath and the use of an aqueous washing machine. Carefully note if your design includes components that should not be cleaned using some or all these methods.

40. INDICATE WHAT PROTECTION IS REQUIRED

Customers that require cleaning often also require a conformal coating – a film that protects the boards from environmental factors including corrosion, dust, humidity and high temperatures. Boards that are conformally coated are washed and dried before-hand to ensure there is no bacteria trapped under the coating. Our facilities include a PVA automatic coating machine, a manual spray booth, and dip coating. Another option is potting, which waterproofs the final product.

---

41. INCLUDE THE PROGRAMMING AND TEST PROCEDURES

Supply any firmware that needs to be loaded, along with installation instructions and any equipment. Your procedure should be a detailed step-by-step description. Equipment needs to be supplied by the customer including any PCs and test jigs. If you have a maintenance and/or calibration procedure please also provide this information. Circuitwise prefers to do any testing in-house so we can check the performance of the PCB assembly we are delivering.

---

42. NOTE ANY REPORTING REQUIREMENTS

Tell us if you need a written certification, including the laboratory or standards body requiring it. Often customers require reports or certificates to be supplied to quality auditors such as an ISO certification body or Underwriters Laboratories (UL). These reports can include both process qualification and product qualification. Process reports include records of environmental conditions of virtually any parameter during the manufacturing process, such as the temperatures in the reflow ovens through to humidity levels at which components were stored. Product reports cover the result of tests, including automated optical inspection reports and x-ray results.

---

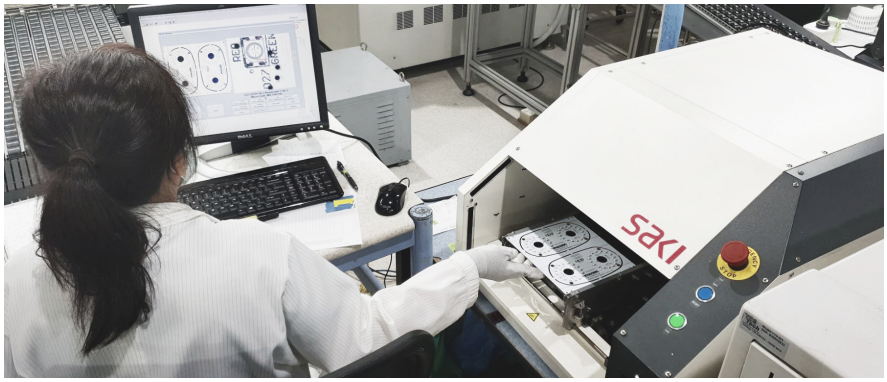
43. INCLUDE INSTRUCTIONS FOR FINAL ASSEMBLY

If you wish to have the final product assembled by Circuitwise, please provide detailed step-by-step instructions. Ideally, the instructions should include photos for each step so there is no ambiguity on the correct procedure. Include an estimate of how long each step should take, so we can quote accurately. If any final testing of the assembled product is required, please specify the testing procedure.

---

44. INCLUDE PACKAGING INSTRUCTIONS

For PCB assembly-only contracts, the boards are placed in ESD bags and shipped in boxes with dividers to securely hold them during transport. If any additional labels are required on the bags such as serial numbers, please specify. For finished product packaging, please specify the boxing procedures, again with photos to illustrate each step. Specify how long each step should take. Advise if any records need to be taken of what is placed in each box. Provide instructions on how to ship the boxed products in bulk.



CHECKLIST

DESIGN FOR MANUFACTURE

- [ ] **1.** PUT ALL SMT COMPONENTS ON ONE SIDE — Page 6
- [ ] **2.** PUT THE HEAVY COMPONENTS ON ONE SIDE — Page 6
- [ ] **3.** MINIMISE THROUGH-HOLE COMPONENTS — Page 6
- [ ] **4.** LEAVE ADEQUATE SPACE FOR SELECTIVE SOLDERING — Page 6
- [ ] **5.** MINIMISE COMPONENT END-OF-LIFE RISKS — Page 6
- [ ] **6.** MINIMISE TESTING TIME — Page 7
- [ ] **7.** OPTIMISE PRODUCT ASSEMBLY — Page 7

PRODUCTION MANAGEMENT

- [ ] **8.** SPECIFY LEAD TIMES — Page 8
- [ ] **9.** IDENTIFY HIGH-CONFIDENCE COMPONENTS — Page 8
- [ ] **10.** SPECIFY YOUR PRODUCTION ROADMAP — Page 8
- [ ] **11.** DECIDE ON FIRST BOARD SACRIFICE — Page 8
- [ ] **12.** DETERMINE IF X-RAYING IS REQUIRED — Page 8
- [ ] **13.** ADVISE THE TESTING PROCESS EARLY — Page 9

CHECKLIST

BILL OF MATERIALS (BOM)

☐	14.	MANAGE BOM REVISION NUMBERS	Page 9
☐	15.	NAME OF THE MANUFACTURER AND THEIR PART NUMBER	Page 9
☐	16.	INCLUDE A DESCRIPTION OF THE PART	Page 9
☐	17.	PROVIDE A SUPPLIER PART NUMBER	Page 9
☐	18.	CONSOLIDATE IDENTICAL COMPONENTS INTO A SINGLE LINE ITEM	Page 10
☐	19.	DO NOT INCLUDE WASTAGE ESTIMATES	Page 10
☐	20.	INCLUDE THE REFERENCE DESIGNATOR(S) FOR EACH COMPONENT	Page 10
☐	21.	SPECIFY THE CRITICALITY OF EACH PART	Page 10
☐	22.	NOTE MSL RATINGS OF 3 OR MORE	Page 10
☐	23.	ENSURE YOUR COMPONENTS MEET REGULATORY REQUIREMENTS	Page 10
☐	24.	CONDUCT AN INTERNAL REVIEW BEFORE SENDING	Page 11
☐	25.	MAINTAIN YOUR OWN RECORD OF THE BOM	Page 11

CHECKLIST

GERBER FILES

- [ ] **26.** INCLUDE A PCB FABRICATION INSTRUCTION — Page 12
- [ ] **27.** DEFINE TRACE WIDTH AND SPACING — Page 12
- [ ] **28.** SPECIFY THE PCB LAYER STACK — Page 12
- [ ] **29.** SPECIFY MINIMUM/MAXIMUM HOLE SIZE — Page 12
- [ ] **30.** SPECIFY MINIMUM ANNULAR RING SIZE — Page 13
- [ ] **31.** SPECIFY SURFACE FINISH — Page 13
- [ ] **32.** SPECIFY ANY SPECIAL PROCESSES — Page 13
- [ ] **33.** SPECIFY THE SOLDER MASK AND OVERLAY COLOUR — Page 13
- [ ] **34.** ADD IDENTIFICATION DETAILS — Page 13

CHECKLIST

ASSEMBLY INSTRUCTIONS

- [ ] **35.** **INDICATE IF LEADED SOLDER IS REQUIRED** Page 14
- [ ] **36.** **NOTE MANUAL SOLDERING REQUIREMENTS** Page 14
- [ ] **37.** **SPECIFY IF CLASS 3 QUALITY CONTROL IS REQUIRED** Page 14
- [ ] **38.** **INCLUDE A PHOTO FOR COMPLEX ASSEMBLY** Page 14
- [ ] **39.** **SPECIFY CLEANING REQUIREMENTS** Page 14
- [ ] **40.** **INDICATE WHAT PROTECTION IS REQUIRED** Page 15
- [ ] **41.** **DEFINE THE PROGRAMMING AND TEST PROCEDURES** Page 15
- [ ] **42.** **NOTE ANY REPORTING REQUIREMENTS** Page 15
- [ ] **43.** **INCLUDE INSTRUCTIONS FOR FINAL ASSEMBLY** Page 15
- [ ] **44.** **INCLUDE PACKAGING INSTRUCTIONS** Page 15

CIRCUITWISE

Electronics Manufacturing

02 9836 4900
sales@circuitwise.com.au
www.circuitwise.com.au