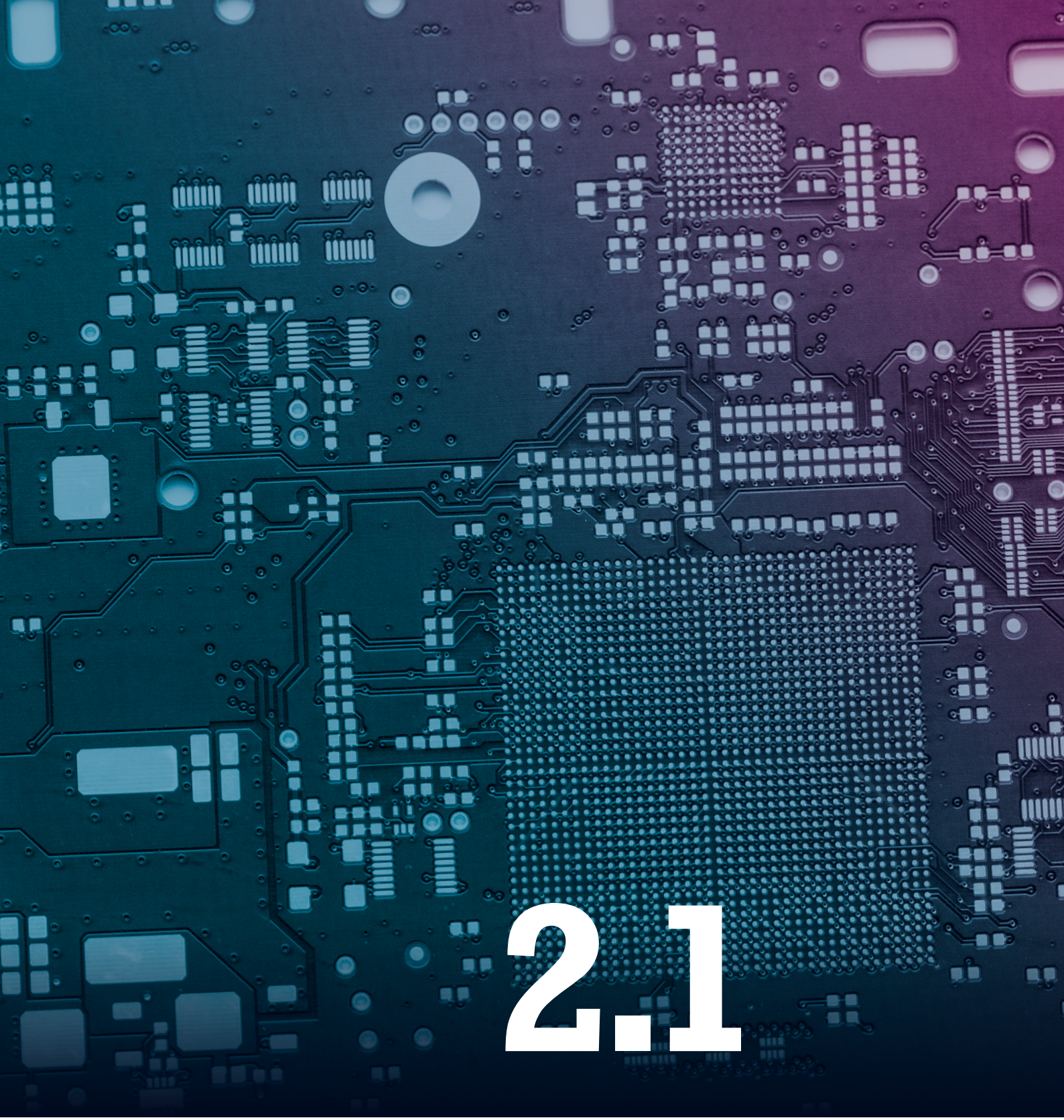
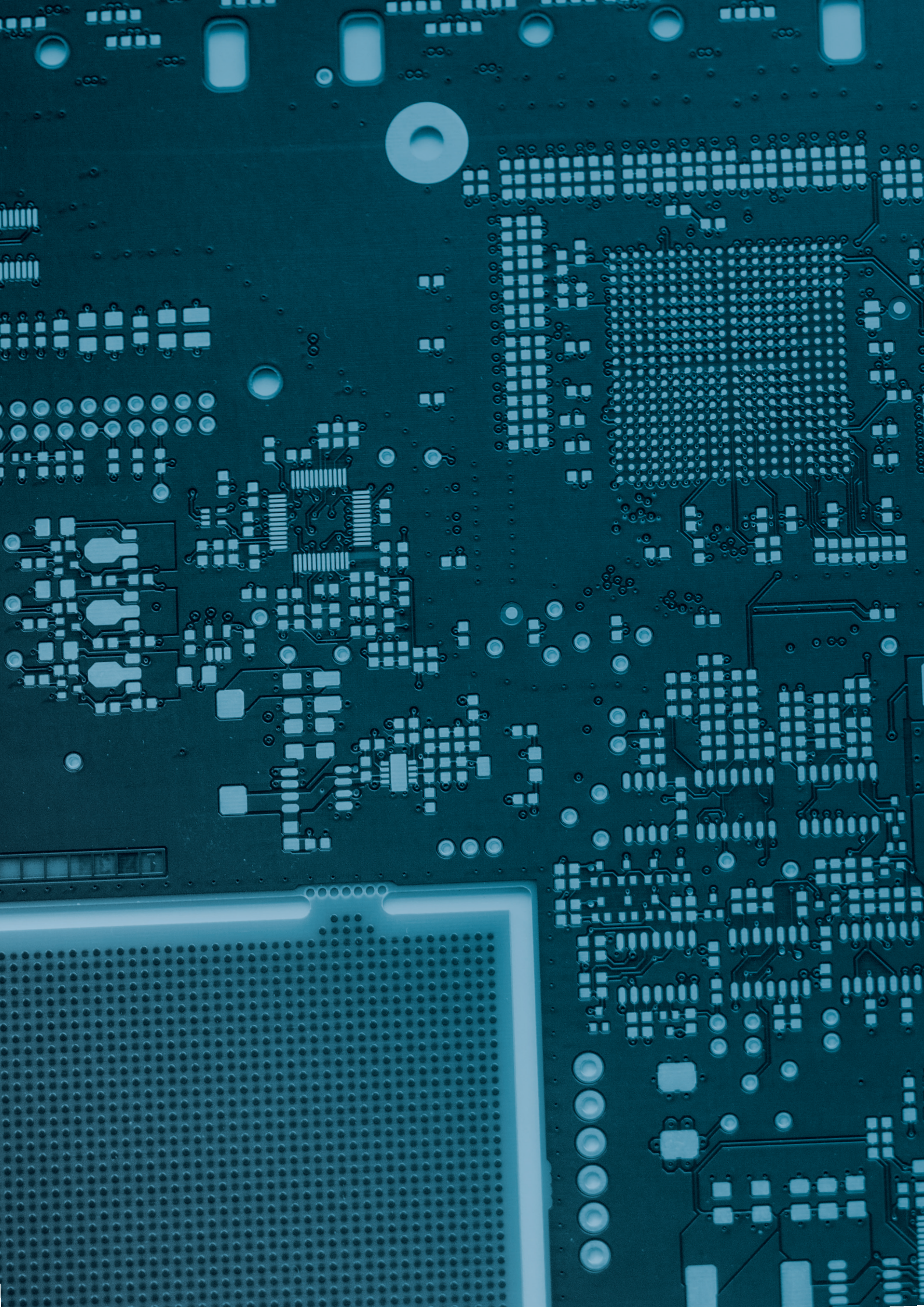


2.1

The Smarter Electronics Handbook

– focusing on the manufacturing documentation dialogue



About the handbook

The English version Smarter Electronic Handbook 2.1 has been produced within Smartare Elektroniksystem. which is an innovation program financed by Vinnova. Both the English and the Swedish versions of the Smarter Electronic Handbook 2.1 are further developments of the earlier Smarter Electronic Handbook 1.0 and 2.0, which were also strategic projects financed by Vinnova as part of the Smarter Electronic System program.

The working group that has contributed ”in-kind” to this handbook has consisted of participants from the companies and organisations below. The main part of this work was made prior to 2020.

In addition to working time, and competence, they have also contributed material in the form of text, illustrations and templates that have been used as input material for this handbook:

Participating companies and organizations

Svensk Elektronik Trade Association

Atlas Copco AB

EK Power Solutions AB

Eskilstuna Elektronikpartner AB

Frikab AB

Inission AB

Mycronic AB

NCAB Group Sweden AB

Note AB

Orbit One AB

Prevas AB

RISE IVF AB

Trimble AB

Veoneer AB

ÅF Digital Solutions AB

Disclaimer

The handbook, with templates and appendices, is based on collective knowledge and experience that the members of the working group have gained during their years in the electronic industry. Although we have reviewed the material carefully, we cannot guarantee that it is error-free. Therefore, whoever uses the contents of the handbook does so at their own risk. The working group declines all responsibility for errors or damage that may arise in areas covered by the handbook.

The handbook, appendices and templates can be downloaded from: www.svenskelektronik.se.

Copyright

This handbook may be freely used and disseminated as a whole. If images or parts of the text are used in other contexts, their origin must be stated.

Image sources

The following organisations have contributed illustrations:

EK Power Solutions AB

Eskilstuna ElektronikPartner AB

IPC

Mycronic AB

NCAB Group Sweden AB

Prevas AB

ÅF Digital Solutions AB

Revision description

This is the first English version of Smartare Elektronikhandboken. It has been translated from Smartare Elektronikhandboken 2.0, and updated to version 2.1

2026 September: Smarter Electronic Handbook in English 2.1 First issue.

Do you have comments and want to contribute to development of the handbook?

Please contact: info@svenskelektronik.com

Contents

1. Introduction 7

1.1 Product owner 9

1.2 Developer 10

1.3 Manufacturer 11

2. Before starting a new project 12

2.1 Industrialisation 12

2.2 Quality 13

2.3 Contract 13

2.4 Request for Quotation/RFQ 16

3. Product regulations 18

3.1 CE marking 18

3.2 Directives/Regulations and standards 18

4. Design 20

4.1 Design for Excellence/DfX 20

4.1.1 Design for Compliance/DfC 21

4.1.2 Design for Sourcing/DfS 24

4.1.3 Design for Manufacturing/Production, DfM/DfP 27

4.1.4 Rules of thumb for designing circuit boards and circuit board assemblies 31

4.1.5 Design for Test/DfT 36

4.1.6 Design for Failure/Worst mistakes 38

4.2 Schematics 39

4.3 Designing printed circuit boards 40

4.3.1 Requirements for printed circuit boards 40

4.3.2 CAD/layout 44

4.3.3 Via knowledge 44

4.3.4 Surface treatment 46

4.3.5 The cost of printed circuit boards (PCBs) 51

4.4 Development tools 53

5. Reliability 55

6. Before manufacturing 58

6.1 Electronic manufacturing described 59

6.2 Description of different tests 60

6.3 Documentation for electronic production 67

6.3.1 Documentation overview and revision management 68

6.3.2 Product description 69

6.3.3 Bill Of Material/BOM 69

6.3.4 Ordering printed circuit boards/PCB 70
6.3.5 Panel drawing 71

7. Manufacturing follow-up 74
7.1 Arrival inspection and verification 74
7.2 Producibility 75
7.3 Feedback 75
7.4 Change management 75

8. Templates, Reading Tips and Other Goodies 78

Symbols used in this handbook

Some important symbols to watch out for in the chapters you read (you have already seen some):

Chapter Chap. "2.3 Contract"
This is a reference to the chapter in the handbook where you can read more, in the Contract chapter in this case.

Template
When you find this symbol in a passage or chapter, there will be one or more templates you can use if you want to. As a help in including the relevant information in your documentation, we have prepared a number of templates that you can download and use freely. These templates are not hundred percent complete. I.e. If you have filled in all the boxes, that does not mean that you will not receive any questions, nor that nothing will be incorrect or missed, but they give you a good starting point. A list of templates is found in Chapter 8.

Checklist
When you find this symbol in a passage or chapter, there will be a check list you can use if you want to.

Reference
We have collected more comprehensive texts in appendices, or refer to the Internet, to prevent the handbook from becoming too heavy to read.

Tip
Here are some important tips.

Preface

Smarter electronic systems was a programme within the framework of the joint initiative by Vinnova, Formas, and The Swedish Energy Agency in strategic innovation. Its goal was to enable Swedish industry to achieve world-class sustainable development and competitiveness. As the agenda for the programme was being drafted, three main challenges emerged to ensure future requirements are met: leading-edge competence; competence supply; and an efficient value chain. Committees were formed to address each challenge, with work on this handbook was initiated by the Value Chain Committee.

The value chains involved in developing electronic products are complex, involving many different actors who make various contributions to the product until it reaches the market. To be innovative, competitive and produce reliable products, development, manufacturing and testing need to collaborate closely , as reliability and ease of manufacturing must be designed into the product. In particular, the interface between development and production has been found to be critical in determining the success of a product. Efficient collaboration enables lower production costs, faster time to market, and higher quality.

The Smarter Electronic Handbook has been created by representatives of companies and organisations who have contributed with their knowledge and experience.

The handbook focuses on what is needed to prepare the manufacturing documentation necessary to permit efficient knowledge transfer between the partners who collaborate in developing a product.

The first versions of the handbook: 1.0 and 2.0 (in Swedish), were highly appreciated. We have received many positive comments and suggestions, that we have considered in the latest update.

The present Smarter Electronic Handbook 2.1 is an update and an English translation of 2.0.

We hope you will find the handbook, with its templates, checklists and appendices useful in your daily work. It has been written by engineers for engineers, but we believe that non-specialists should also be able to find useful sections, as we have included the commercial and legal aspects of electronic projects.

– Please introduce your suppliers and customers to this handbook.

The handbook can be downloaded from:
www.svenskelektronik.se

Yours sincerely,
the Handbook Working Group

1. Introduction

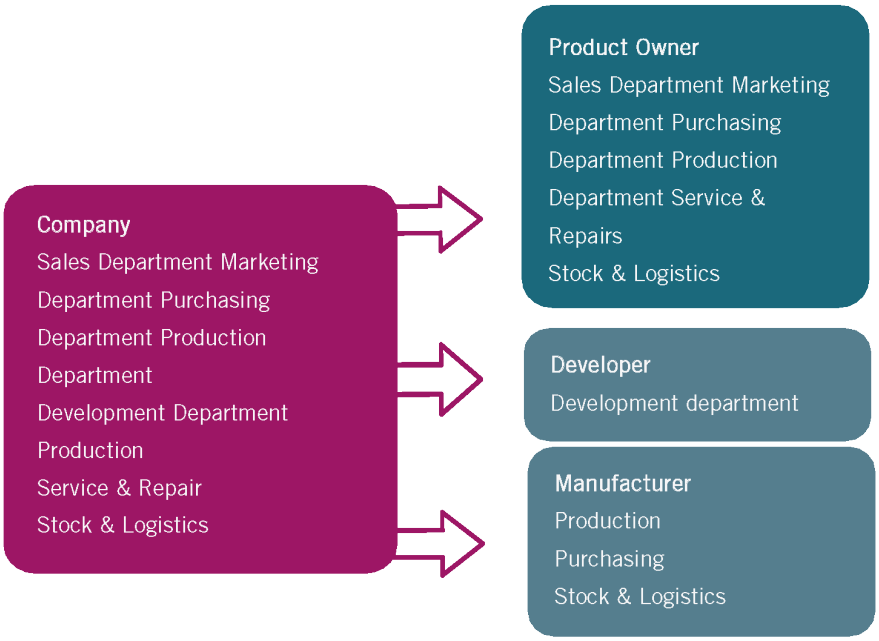
The Smarter Electronic Handbook 2.1 has been created to facilitate communication between stakeholders in the value chain who often participate in electronics projects that should result in a functioning printed circuit board assembly (PCBA) to be mounted in a product. This might sound simple, but experience has shown pitfalls in the process that cost time and money. The English handbook 2.1 is a translated and updated version of the handbook 2.0 from 2019, that in turn is a revised and supplemented version of the first handbook, which was issued in the autumn of 2017.

The main stakeholders in this handbook are product owners, developers and manufacturers (See below). Please note that we refer to companies, not people, i.e. a product owning company, a development company and an electronics manufacturing service company (EMS).

The handbook focuses on the process leading to the documentation that a manufacturer needs from the product owner and developer to be able to manufacture PCBAs that meet the product owner's requirements, but quotation templates and Feedback documents are also included. These documents are important for optimising interaction between the different actors.

Quotation templates can be useful for a manufacturer who wants to answer a request for quotation, but they can also be useful for the product owner's purchasing officer to indicate the type of answer they can expect from a request for quotation that they have issued to ensure that the right questions are asked.

Whereas the focus is on manufacturing PCBAs, we also refer to areas beyond PCBAs, where relevant. This handbook contains a range of information about various areas that need to be considered in preparing the documentation for a PCBA that can be successfully produced. For example, to make a quick proof of concept or laboratory mock-up to test whether an idea works at an early stage, it is not necessary to consider the entire handbook, as that device will not be marketed. But as soon as prototypes start to be developed, considerable time will be saved by considering the contents of the handbook.



Interaction and communication

Collaboration between the product owner, developer and manufacturer in developing a new product containing electronics is the focus of this handbook.

If there is no history of collaboration, it is even more important that everybody involved fully understands the expected function, cost and quality of the final product. A product owner will often want to make development and manufacturing competitive by requesting quotations from several manufacturers before choosing a collaboration partner. To make the quotations comparable, the information in the request for quotation needs to be as clear as possible. At the same time, it is advantageous for the product owner to be open to suggestions from their manufacturer as the product owner is unlikely to know everything about choice of materials, choice of components, production technology, testing, directives and standards etc. But if questions are welcome suggestions, it should be indicated clearly so that the receiver understands that they are expected to make proposals and do not miss the opportunity, or merely guess a solution.

Functional communication together with the right partner is the basis of successful collaboration.

Make sure that all communication takes place in such a way that mistakes, confusion and the impact of poor memory are minimised. The best way is to send information in writing (email) and to confirm verbal agreements by email afterwards.

It is good to check things over the telephone, but decisions should be confirmed in writing.

The commercial side should not be neglected in any project. Developing a product is mainly a question of technology. However, as the value chain often consists of many different companies, it is important to ensure that all contracts that relate to the handling of confidential information and commercial matters are signed early in the project. This ensures that companies can achieve a technically excellent product that is also good business for all parties involved. Only profitable companies can remain and shoulder their responsibility during the entire lifecycle of a product.

Chap. ”2.3 Contract”

Special terminology abounds in the electronic industry. Many old concepts, development tools, file formats etc, still used out of habit, are not only old-fashioned but even incorrect today. The fact that everything frequently turns out well in the end is because the receiver of the information ”knows” what the sender really means. But, to ensure correct communication and avoid misunderstandings, everyone should use designations that are as correct and comprehensible as possible. For this reason, we have compiled a vocabulary (in Swedish only) in that we have collected as many industry-specific terms as possible from those we have encountered while working on the handbook.

Read more in appendix ”Parlör”

Printed circuit board and printed circuit board assembly, PCB and PCBA, are examples of terms that are confused and used on the wrong occasions. PCB (Printed Circuit Board) is an unpopulated printed circuit board, whereas PCBA (Printed Circuit Board Assembly) is a printed circuit board with the components mounted. In this book we refer to both PCB/printed circuit board and PCBA/printed circuit board assembly.

1.1 Product owner

The goal of the project owner is to develop and produce a new product as quickly as possible at a reasonable cost so that it can be offered to new and existing customers. As product development costs money, the product owner wants a return on investment (ROI) as quickly as possible.

A product owner is defined in this handbook the company that owns the rights and liability for the product, including the electronics that will be produced. It is important to remember that the product owner is responsible for ensuring that sufficient and correct information is available for anyone who requires it later in the chain. This is to ensure that the product meets the specified expectations with the correct functionality, at the right time and with the right quality. The product owner is also responsible for ensuring that the product complies with all applicable directives and standards, and that it has the certifications required for the markets where the product will be sold.

Chap. ”3. Product regulations”

The first time a product owner works with a particular developer and manufacturer, the product owner should visit them both to optimise their understanding of how they work, their processes, instruments and machinery. The time that these visits take is well invested and will save much time further on.

The purchaser’s role at the product owner company has often changed, from purchasing for their own manufacturing to procuring development and manufacturing service. The role of the purchaser includes trying to create the best possible relationship between quality and price for a certain product or service. We hope that the handbook will make it easier to understand the fact that different developers and manufacturers differ in their ability to develop and manufacture various products. Their knowledge and experience, combined with their production equipment and choice of component suppliers, may differ.

Our hope is that purchasers at a product owner company will be able to utilise the handbook to get the best and most comparable offers possible. Possible answers to requests for quotation are shown in some of the **quotation templates.**

When it is time to order serial production, it is important to remember that an even, or at least predictable, production rate (predictable for the manufacturer via a monthly updated forecast, for example) means more efficient operation at the manufacturer’s, which helps reduce costs.

Investigate whether there are other services the manufacturer can offer, for example post-processing, packaging and logistics. These services can be very cost-effective.

Remember to sign suitable contracts early on. Using our templates will help save time.

Always place orders via email. This means that agreements are recorded. Verbal follow-ups are good. but remember to confirm any new agreements in writing after the conversation.

To utilise a manufacturer’s extensive knowledge of manufacturing and choice of input material and components, it might be useful to commission a pre-study relatively early in a project. In this way, the manufacturer can spend the time needed to do a good job, save time, and avoid problems.

1.2 Developer

Design work while developing new products entails responsibility in many areas. We only cover areas related to manufacturing of the PCBAs.

The developer referred to in the handbook may be either the product owner's own development department or a company on assignment from the product owner that designs the electronics to be produced by a manufacturer. If the developer and product owner are in the same company, contract formulation is simplified, but communication is just as important.

The goal is that the design achieved corresponds to the product owner's requirement as accurately as possible.

DfX, Design for Excellence, is a systematic product development methodology that optimizes a product's lifecycle. This means that during development work, the product is designed not only to be manufactured and tested as effectively as possible, but also to ensure that its quality, reliability and lifecycle will be as projected.

Chap."4.1 Design for Excellence/DfX"

The first time companies collaborate, much will be gained if the developer visits the intended manufacturer to better understand how they work, their processes and production equipment. The time that these visits take is well invested and will save time further on.

It is as important to specify everything that is pertinent to the design as it is not to over-specify. Unusual combinations of materials or components may have to be chosen that are difficult to obtain, for example, that makes the product difficult to manufacture or unnecessarily expensive. If areas remain unspecified to allow the manufacturer to suggest the most efficient alternatives, write this down clearly. Anything that might have to be guessed further down the chain entails the risk of mistakes, which costs time and money. If anything is uncertain, contact the manufacturer.

Gain an understanding of the way the intended manufacturer works, their processes and equipment, for example, by visiting the facility where the product will be manufactured.

Component manufacturers of ASICs, CPUs, power and RF components, PCBs etc can contribute considerable valuable information.

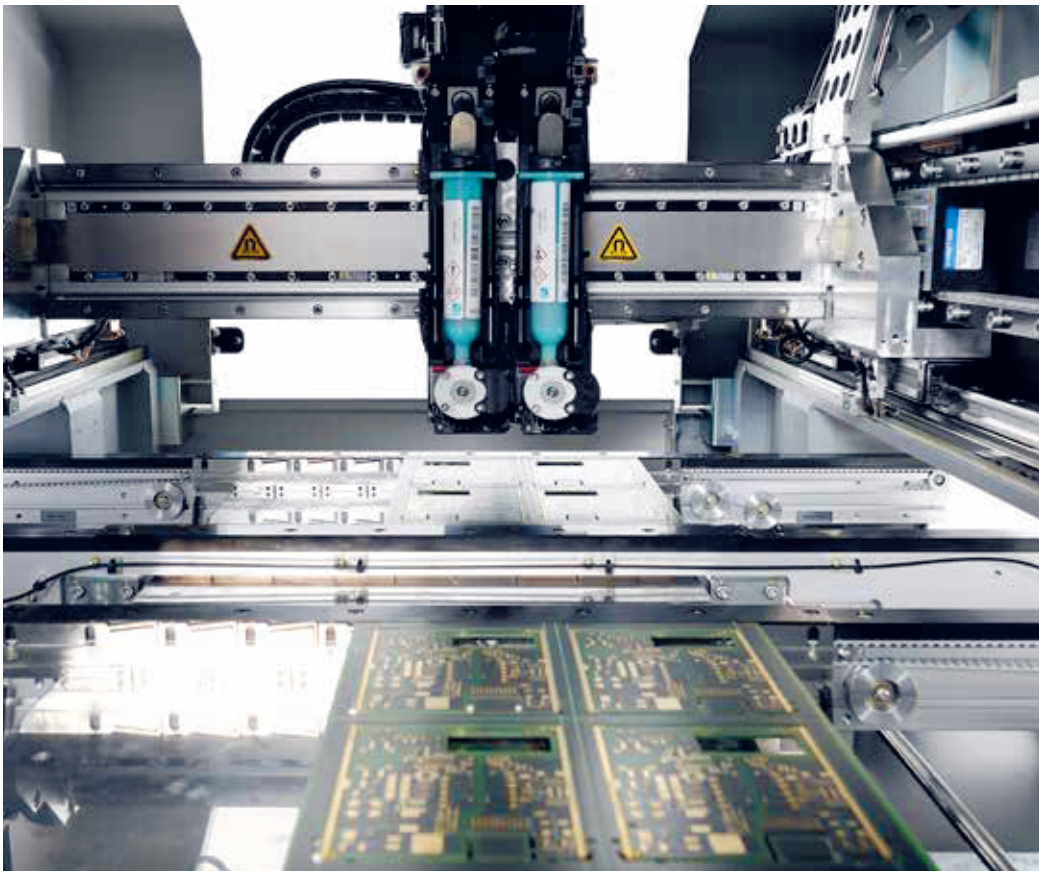
1.3 Manufacturer

The role of the manufacturer is to make PCBAs in accordance with the specifications that the product owner/developer includes with their order. When choosing a manufacturer, success factors include experience of similar products, geographic closeness to the customer, and matching the size of the company to the scope of the assignment. A manufacturer that meets these criteria can contribute considerable knowledge to the project. Many manufacturers can also offer further services that can add value to the entire business.

The handbook defines manufacturers as companies that make the electronics developed by the developer on assignment from the product owner, i.e. a PCBA manufacturer. Please note that reference is sometimes made to manufacturers of PCBs, that are then called PCB manufacturers.

Manufacturers of PCBs and populated PCBAs are normally different companies. Nowadays, the developer or the product owner is not always in contact with the PCB manufacturer. The manufacturer usually manages this contact. As the PCB is such an important component, it is considered in detail in the handbook.

It is important that documentation is as complete as possible to allow the manufacturer to work accurately and efficiently.



JETTING MACHINE ABOUT TO START SPRAYING SOLDER PASTE ON A PCB PANEL.

2. Before starting a new project

It is valuable to reflect on what needs to be in place for a project to be implemented smoothly, on time and within financial frameworks before the product design is completely set. The overall goal is for the final product to be producible and sold at an expected profit.

We will return to the background material for the actual production in later chapters:

Chap. 6.3 "Documentation for electronics production"

However, documentation only covers part of the interaction between parties. Time and money can be saved by doing things as accurately as possible from the start. Good documentation and clear agreements help avoid misunderstandings and unnecessary cost.

2.1 Industrialisation

Taking a design from working prototype to a circuit board optimised for manufacturing can take as much time as the entire development phase. We mention this process, industrialisation, so early in the handbook because it needs to be considered from the start, even if industrialisation usually occurs after functioning prototypes have been prepared and various functional and system tests carried out. This phase is often overlooked and the time required is often underestimated, that can lead to delays and higher costs than expected.

Much can be gained even before the first prototype is designed, by considering the points in the industrialisation phase, in particular that markets the product will be sold in and the standards it has to comply with.

The industrialisation process includes:

- Drafting a time plan for industrialisation.
- Determining the directives and standards that are applicable to the product and therefor have to be complied with.
- Reviewing the bill of material to optimise components for availability and lifecycle.
- Deciding that tests have to be performed on the circuit board and the finished product.
- Drafting a test specification.
- Planning for how the PCBA/product will be packed and transported.
- Planning product volumes , especially what is needed for each stage in industrialisation (prototypes, pre-series).
- Considering product variants. Planning for the way different variants will be handled in the production process and later in the logistics chain.

2.2 Quality

What is quality? It feels self-evident that to aim for good quality in products and services. Whereas the meaning of quality can vary, it essentially means that a product meets expectations. The handbook aims to help create documentation that will ensure, as far as possible, that a product obtains the desired quality.

The definition of quality. Quality is often used without defining what the word actually means. In standards, the usual meaning is that a product complies with the specified requirements in a specific standard. This does not imply any guarantee that the product will be reliable when used. Quality and reliability are often confused.

➡ **Read more: "Reliability assurance of soldered printed board assemblies" (Handbook published 2024)"**

IPC standards for soldered PCBAs are a good example of this: J-STD-001 and IPC-A-610. For the IPC, a quality solder joint only means that the board produced complies with the requirements in the standards. These are mainly visual requirements on solder joints. There is no requirement that the reliability of the solder joints is tested. For through-hole components where the solder joints are easy to inspect visually, it is often sufficient. However, for other components, such as QFN (Quad Flatpack No Leads) and BGA, it is both difficult to visually inspect the solder joints, and their reliability may be poor, even if they appear good. The IPC stipulates that it is not sufficient to comply with their requirements and standards to ensure that the product will be reliable in use. It is up to the user of the standards to assess whether the standards are relevant and sufficient to ensure that the product will be reliable or whether supplementary requirements need to added.

ISO9001 defines quality in a somewhat different manner. Quality is defined as complying with customer requirements, that means that the product has to be reliable in use: "Quality is the sum of all the properties of an object or occurrence that gives it the ability to satisfy specified or implied requirements".

2.3 Contract

Today, different companies often handle different stages in the value chain. Therefore, it is important, early in the project, to agree on whom is responsible for what. This is no less important when the product owner has part of the required development resources inhouse.

Commercial terms and responsibilities are best stipulated in a contract between the parties involved. Contracts are often considered complicated and difficult, but it is important to have them in place to be able to focus on developing the best possible product, which will be successful when it goes to market.

Contract templates have been prepared by various industrial organisations to facilitate contract writing. Those that best meet present-day requirements are referred to below in this chapter. They are well balanced, as they consider the buyers' and sellers' requirements, that accounts for their popularity as a point of departure. They provide the option, during negotiations with customers or suppliers, to edit the text and add appendices that modify the contract to exactly meet requirements. Contract templates are available in English and Swedish.

Confidentiality is often important when a new product is being developed. For this reason, it is common to sign a separate non-disclosure agreement (NDA) at the beginning before any detailed discussions begin.

The main purpose of the contract is to prevent the other party's information from being disclosed to third party(-ies). It is much simpler to agree on an NDA rather than entering into a full contract negotiation. In this way, the possibility of working can be discussed without formalising such collaboration in a commercial contract.

IP (intellectual property) and who owns it. It is not always self-evident that the product owner owns all rights, including patents, to the developed product, so ensure that this is stipulated in a contract.

Lead time includes how long it is/is expected to take, and how this is defined. This should be included in the contract. If the product owner demands short lead times, it is important to discuss how these can be achieved; for example, what happens if the lead time for subcomponents is considerably longer than the lead time requested for the finished product?

It may be prudent to use a forecast in that the product owner regularly updates their coming requirements for the following 12 months. Agreement also needs to be reached about how much the product owner can change the forecast and when the batch size is locked in. In forecast management, it is common to include a clause that stipulates who bears the cost of surplus components following changes to the product during its lifecycle, or of scrap material when a product is taken out of production.

Material authorisation. In some cases, a so-called material authorisation may be useful. This means, for example, that before serial production starts, the manufacturer can order components with long lead times, while the product owner completes tests or adjustments to the design. It is usually the product owner who gives the manufacturer a material authorisation, by that the product owner accepts full responsibility for components that the manufacturer purchases if it turns out later that they will not be used due to last-minute changes to the design.

Material authorisations can also be used so that the manufacturer can order larger quantities than necessary in the short term, in order to get better prices. Thereby the product owner guarantees that the manufacturer will not be holding surplus components if there are changes to the product or if production is suddenly cancelled.

Delivery terms are found in various forms in different parts of the supply chain. Use the latest version of the Incoterms delivery term list (see below) and agree about suitable delivery terms. During price discussions, it is important to clarify what is included, as the costs of freight, customs and document handling, packaging material etc can be additional. This is particularly important in relation to import/export outside the EU, where customs rules for different countries may differ. The term "Landed Cost", that is sometimes used, is not a delivery term in Incoterms, so avoid it if possible.

Logistics and packaging methods are areas that deserve some extra attention as considerable savings can be made if packaging methods and sizes suit all parties, for product size, handling and transport.

A new product or the next generation of a product will be much easier to produce if a company has already signed a contract with the customer/supplier. Then all that is needed is to add a supplement to the existing contract that specifies the new product and refers to terms previously agreed on.

Good contract templates to use:

- **SEAL2026 General Terms of Delivery (Svensk Elektronik Allmänna Leveransbestämmelser)** are suited for the delivery of electronic products and have been written by Svensk Elektronik
- **SEAL-K2026, Swedish Electronic General Delivery Terms for Consultant Contracts (Svensk Elektronik Allmänna Leverans-bestämmelser Konsultavtal)** are suited for delivering consulting services related to the design and development of electronic equipment. Contact Svensk Elektronik
- **NL17** is suited for the sales of technical products between Nordic companies, and has been written by Teknikföretagen. The contract can be downloaded free of charge from Teknikföretagen's website.
- **Orgalim** is suited for the sales of technical products globally and has been written by Orgalim, the European Engineering Industries Organisation. The contract can be downloaded for a fee from the Orgalim website.
- **Incoterms** details international delivery terms. Use the latest version, and avoid other terms or alternative delivery terms, otherwise there may be unintended costs and risks to bear. Always refer to Incoterms in commercial contracts. Incoterms are available from a number of organisations, such as Business Sweden and the ICC.

Remember to sign contracts as soon as possible. In the early stages of discussions in a new business relationship, it is advisable to sign an NDA and then sign a commercial contract later on before orders are placed.

Use SEAL/NL/Orgalim as the starting point and then add or delete paragraphs by using appendices to the original contract. This reduces the risk of unpleasant discussions later.

Always use modern delivery terms, that can be found in the latest issue of Incoterms.

Remember that lead time, packaging, and how logistics should work. These three issues are often forgotten and often cause problems when serial production is about to start.

2.4 Request for Quotation /RFQ

Save time and money by using a carefully written request for quotation (RFQ) containing all the pertinent documentation of the project. If quotations are requested from several suppliers, make sure to clearly explain what is expected, otherwise quotations will be difficult to compare.

An RFQ check-list is a useful reference document. This check-list refers to the chapters in the handbook, and the templates included are very helpful.

Chap. "8 Templates, reading tips and other goodies"

It is also worth mentioning that it may be useful to submit a request for information (RFI) before issuing an RFQ. The RFI should enquire about the goods and services that a company can provide. A request for proposal (RFP) is similar to an RFI, but contains more specific information about certain goods or services. RFIs and RFPs are mostly used in larger procurements

"Ask a silly question and you get a silly answer" is highly relevant in this context. A short and unspecific request for quotation can often lead to short and generally formulated answers. If there is uncertainty about whether the intended requests have been included, ask the potential supplier what they usually include. The quotation that comes back as the answer to an the RFQ might look more or less like one of the two quotation templates.

Comparable and good quotations. When requesting quotations from several suppliers, it is important to understand the differences between them. Different manufacturers may have different machinery and work routines, that means that they might not have the same ability to deliver the product.

A manufacturer that receives good manufacturing documentation, can likely offer more attractive pricing and product quality. Otherwise, the manufacturer has to make assumptions about unclear sections of the documentation. The manufacturer can also choose the components and production methods that produce the price and quality expected.

Minimum of documentation? Many times, a minimum amount of documentation can be sufficient, but what is a minimum? Start with the enclosed check-list and delete the sections that are not relevant. Retain the headings and make a note of sections that are not relevant, if any. This sends the right signal to the manufacturer and offers them the opportunity to ask questions.

The product description is possibly the most important document to append to a request for quotation, as it explains the context of the product. The product description helps the receiver by describing how the product will be used and in what environment.

Pre-study. A pre-study can be useful to understand how to optimise a product for production. During the pre-study, test or model various solutions based on the documentation provides and a desired target price. This helps obtaining the best balance between requirements and cost.

Formulate requirements as ”shall” or ”should be” to make the process simpler and quotations more accurate.

Visit the manufacturer selected to better understand how they work, their material handling, production, inspection, test, packing and final delivery procedures. There might be a small matter of adjusting the external dimensions of a PCB to simplify work in production, or in packing the products before delivery, that leads to cost savings.



THE ILLUSTRATION SHOWS A PRODUCTION LINE FOR A PCBA.

3. Product regulations

When an electronic product is marketed, it must comply with the legislation, requirements and standards of the country in that the product will be sold. This is the responsibility of the product owner. Within EU, these requirements are summarised in the CE-marking that all electric and electronic products must have. Other parts of the world have their own legislation, that must be complied with.

The handbook summarises the legislation for the single market in the EU. It is important to find out what applies to other countries when exporting to the rest of the world. For example, legislation that limits the use of certain chemicals in electronics in the EU, the RoHS-directive, is not identical to the legislation implemented in China.

3.1 CE marking

A product must be CE-marked to allow it to be marketed within the EU. The product owner must declare conformity with applicable Directives and regulations by means of an EU Declaration of Conformity (DoC).



"The New Legislative Framework", NLF is a package of measures that aim to improve market surveillance and boost the quality of conformity assessments. It was introduced in 2008 as Decision No 768/2008/EC. The directives/regulations stipulate general health and safety requirements (the so-called protection requirements). Member states may not have higher or lower requirements. Work on specifying the protection requirements has been assigned to the European standardisation organisations. This means that the requirements to be met for a product to be compliant with a directive are found in the standards developed. The use of harmonised standards (published in the Official Journal of the Commission) offers a presumption of compatibility.

Please note that the Commission has announced a review of the Framework during 2026.

3.2 Directives/ Regulations and standards

To determine the directives and standards that apply to a product, start by considering its specific area of use. Some products have their own standards, whereas for others, the generic environment in that the product is used, such as an industrial environment or a home environment, has to be considered.

Directives and regulations for electrical and electronic products are found in abundance. Here are some of the most common (websites of the main Swedish authorities in brackets):

- **RoHS:** Restriction of Hazardous Substances (www.kemi.se)
- **WEEE:** Waste Electric and Electronic Equipment (www.naturvardsverket.se)

- **ESPR:** Ecodesign for Sustainable products Regulation (www.energimyndigheten.se)
- **REACH:** Registration, Evaluation, Authorisation and Restriction of Chemicals (www.kemi.se)
- **EMCD:** Electromagnetic Compatibility Directive (www.elsakerhetsverket.se)
- **LVD:** Low Voltage Directive (www.elsakerhetsverket.se)
- **RED:** Radio Equipment Directive (www.pts.se)

It is important to note that RED and certain other directives such as MDR (Medical Device Regulation) have incorporated requirements for EMC and electrical safety. Vehicles and vehicle components have legal requirements for EMC compatibility in accordance with UN ECE R10.

The technical file shall enable the supervisory authority to assess how the product conforms with the requirements of the regulatory framework. It shall contain all relevant information for such assessment, such as design, risk analysis, testing, manufacture, and use of the equipment. The manufacturer (i.e. the product owner) or their authorized representative established in the European Union is required to maintain the technical documentation, normally for 10 years, after the last device has been placed on the market.

Regulatory changes

Regulations are often reviewed and changed. The product owner needs to remain updated and conform processes to comply. Such changes may be due to technological progress, environmental or geopolitical threats etc. The use of chemicals is one area where the companies must be very observant of changes. Other areas are AI and cyber security. Many new regulations have been introduced in recent years, at organisational and product levels.

For instance, Regulation (EU) 2024/2847 The EU Cyber Resilience Act (CRA) aims to improve the security of connected products in the EU market. New cyber security requirements have also been added to RED.

Standards are prepared to simplify collaboration in an industry. In addition to the harmonised standards that have to be complied with to obtain a presumption of conformity with the directives above, more or less widely disseminated standards exist in the electronic industry. For circuit board manufacturing. IPC standards are examples that have been developed by collecting ”best practice” in this area and have developed something that many companies work with or refer to. IPC are the world’s most widely used quality standards for electronic production. It is important to adopt a suitable level and refrain from specifying a higher standard than is required for the product. Please note that in June 2025, the IPC changed its name and is now known as the Global Electronics Association, but the standards still include ‘IPC’ in their names, e.g. IPC-A-610.

Read more in:
”Appendix ”EU Direktiv och förordningar”
”Handbook for Development of Cybersecure IoT Products”
”Handbok för effektivare framtagning av medicintekniska produkter” (Swedish).

Chap. ”4.1.1 Design for compliance”

4. Design

It is important to understand the many stages a printed circuit board assembly (PCBA) goes through before it is ready to be integrated into a product. If an early understanding is gained, it will ensure that the final product meets the necessary requirements.

This chapter outlines certain principles for designing a PCBA. However, the level of detail is limited due to the scope of the handbook. We describe the Design for Excellence concept and some typical stages in developing a PCBA. We have included as many tips as possible, to guide designers in their work. These tips are not necessarily the correct solution in all situations for all designs, but offer good general guidelines. Much attention has been given to the unmounted circuit board, the PCB, as it is an important component in determining the quality and reliability of the product.

Circuit board design is usually an iterative process that may begin with a previous design, with a brand-new idea, or a mix of both. The process passes through theoretical calculations and practical testing, and results in the documentation needed to produce the finished PCBA.

Chap. "6.3 Documentation for electronics production"

4.1 Design for Excellence /DfX

DfX concerns quality, reliability, producibility, service life, and compatibility with directives and regulations as early as possible during the electronics design phase. The concept is broken down into several sections in this chapter.

It is worth noting a chapter that is a humorous listing of faults and incorrect behaviour that most are well aware of, but that nevertheless occur in projects.

Chap. "4.1.6 Design for Failure/worst mistakes"

4.1.1 Design for Compliance/ DfC

Product compliance with regulations can be an expensive part of electronic design if it is the last phase of the development project. There is a major risk that the product does not comply with all requirements and that the test lab will return the product for modification. This generates heavy costs not only for the redesign and certification, but also because income is lost due to the delayed market launch.

Market access requires compliance

Products need a CE-marking to be sold in the EU. Corresponding markings exist in other parts of the world that must be considered, depending on the country in that the product is to be sold. In most parts of the world, products require some kind of product compliance, that must be met by designing products according to the respective regulations.

How does this affect the way design work?

The most important factor is that design requirements must be built in at the start of the development project. This saves considerable time, in hours and calendar time, and reduces development, certification, and product manufacturing costs.

One example of this is the traceability requirement in developing safety-critical units such as medical and functional safety equipment. In such cases, design requirements and decisions must be documented clearly, so that an external auditor can check not only the results, but also the process. This requirement is based on evidence that this is the best way to develop safety-critical and medical technology. Evidence is produced more easily by following a relevant harmonised standard.

➡ **Read more in appendix "Bibliography"**

Compile the requirements

Regulatory requirements are defined by the countries or regions where the unit will be sold. CE marking requirements are defined by the directives and regulations relevant to the product, and the harmonised standards that are related to them.

Define the market(s) where the product will be launched and summarise the requirements that have to be complied with. In this process, risk analyses are important. The development team must know the regulations that have to be complied with to make the product compatible. Finding these regulations can be difficult. There are often detailed technical requirements in the form of standards for each regulation. To determine that standards apply to a product, its use and function have to be defined. Many product types have their own standards.

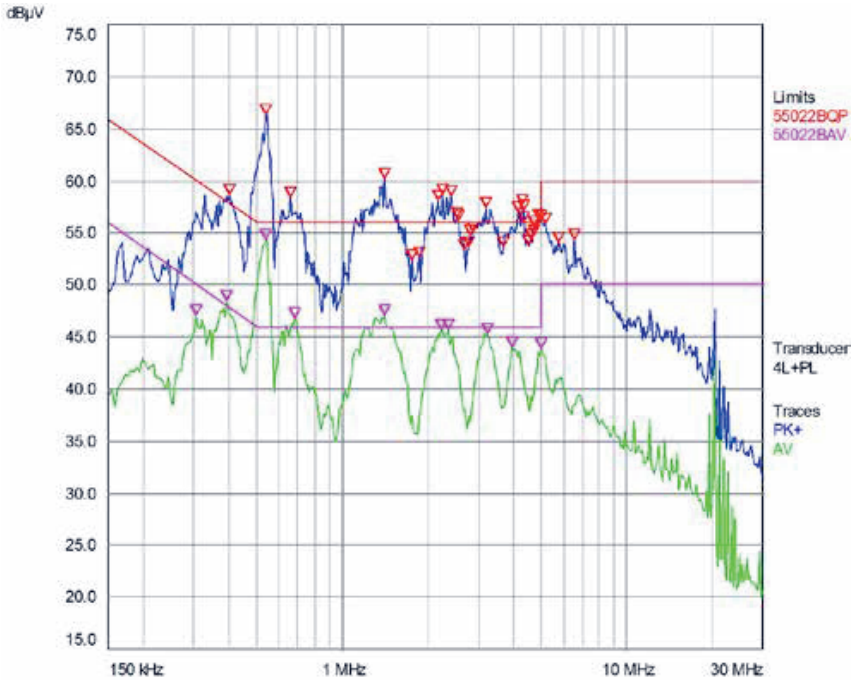
The development process

When all requirements are known and development can start, what is the best way to achieve product approval?

Major differences are found related to degrees of difficulty and complexity, depending on where the product is to be used, and the technical level of the design. High technology, safety-critical or medical devices have a very comprehensive requirement list. Complying with regulatory requirements is a large part of the product development project.

Some basic processes are common:

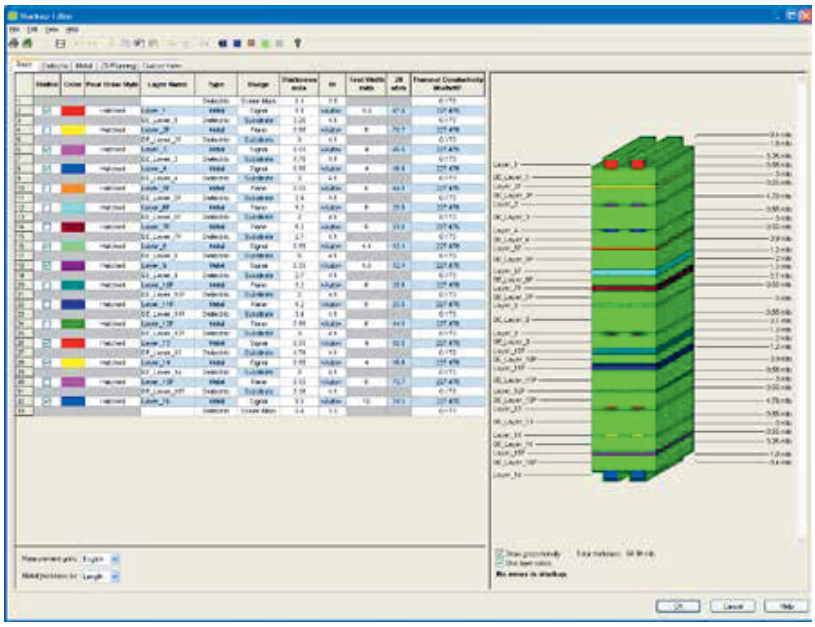
- Ensure that the development team has the required competence.
- Document decisions and design choices.
- Compile risk analyses in accordance with each body of regulations.
- Mitigate the risks found in the risk analyses by applying a relevant harmonised standard if possible. Otherwise, it may be laborious to prove that the risks have been mitigated.
- Hold minuted review meetings during the entire development project, where regulations are an item on the agenda, and include participants with competence in each area.
- Do tests and pre-compliance measurements/ tests (EMC and Electrical Safety) as early as possible, preferably starting with the first laboratory prototype.
- Choose components not only on the basis of function and cost, but also consider their chemical content and EMC properties.
- Download all component information to the technical file (it is not enough merely to refer to links to websites as information can change).
- Certificates for critical components in relation to a technical standard are often a requirement for obtaining third-party approval of electrical safety requirements. Critical components are defined as components which, if they fail, risk lowering the safety of an equipment to an unacceptable level.



EXAMPLE OF MEASUREMENT OF EQUIPMENT THAT DOES NOT COMPLY WITH EMC REQUIREMENTS

Technical matters that often affect EMC

- **Earthing philosophy**. Earthing is of decisive importance, both at circuit board level and equipment level.
- **The use of high-speed signals** is also decisive. Do not use signals with faster rise and fall times than necessary.
- **Cabling** should be shielded if possible. If cables are unshielded, filter all signals. Fix all internal cables mechanically.
- **Mechanical design**. Use metal enclosures if possible. Also remember to ensure the contact areas between the mechanical parts to avoid slits (a slit can function as an antenna). Prepare the layout so that shielding cages etc can be mounted if necessary.
- **Displays** often cause problems with EMC, mainly because of the large number of high-speed data signals and the fact that they are not easy to encapsulate. Displays require careful product design, mechanically and electrically.
- **Layer structure and number of layers**. The design of the PCB layers and the number of layers are very important, especially for more advanced and high-speed electronics. It can be beneficial to use extra layers for a product that will not be produced in large quantities to achieve goals faster. As signal integrity is also highly affected by layer design, it is important to communicate clearly with the PCB manufacturer to ensure that it is not modified (e.g., to reduce the cost of the board).
- **Layout rules**. Great care must be taken on layouts to ensure that components are located optimally, signals are routed to avoid unnecessary antenna effects and other interference phenomena, and filters and suppression components are included.



EXAMPLE OF LAYER STRUCTURE IN A CIRCUIT BOARD

Sometimes, legal requirements are not enough

All stipulated requirements must be complied with to allow a product to be launched in a market. However, please remember that legal aspects are sometimes not enough for the product to be functional. Always consider the context and the functional requirements applicable to a specific product as well. A good example of this is the EMC requirements for a product with built-in radio communication. Here, it is not unusual that a product complies with the legal requirements for EMC, but the built-in radio does not work well because of interference from inside the product.

Chemical regulations

Electronics are subject to regulations that deal with chemical content, so please be aware of this when choosing components and production methods. The substances regulated in the RoHS-directive are: mercury, cadmium, lead, hexavalent chromium, the flame retardants PBB and PBDE, and the phthalates DEHP, BBP, DBP and DIBP.

The REACH regulation contains, among other things, a candidate list of substances of very high concern (SVHC) that all actors in the value chain are required to declare if they are present in the product they supply. The list is updated twice a year.

➡ Read more in appendix ”EU Direktiv och Förordningar” 2026

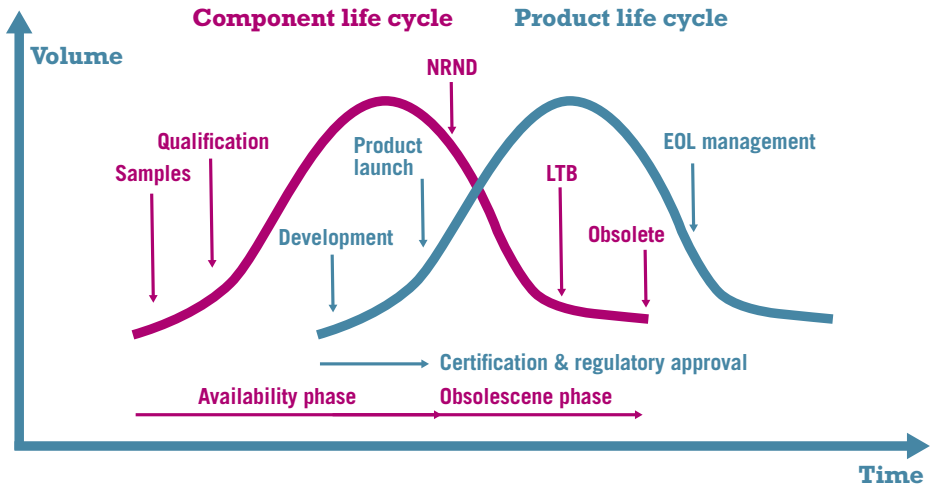
4.1.2 Design for Sourcing/DfS

Component choices that are made during product development are critical during the entire service life of the product. The function of the component is the most important aspect and is what the developer considers first.

However, it is also desirable to optimise the final product’s cost, quality, lead time, chemical content, and service life. Access to selected electrical components and other material, their whereabouts in their lifecycle, their lead time, second sources and their cost, all affect the maintenance cost of the product.

How to handle this. As it is difficult, probably impossible, to entirely avoid disruptions to the component supply, minimise the risks. Following up individual components that cannot easily be replaced by asking the component manufacturer is time-consuming and rarely done in practice. Often, the distributor provides general component information, that may be sufficient, except for the key components in the design.

It is also important to investigate the manufacturer’s preferred suppliers and the components they use as standard.



Comprehensive databases of electronic components are available nowadays. These contain more or less every component and their data, including lifecycle data, from most component manufacturers. Standard components such as capacitors and resistors, with common specifications, do not need much investigation. Focus on the other components that may have more unusual properties, such as integrated circuits, processors, memory chips and special discrete components. Companies that lack access to a database of their own can commission certain suppliers to go through a bill of material (BoM) and analyse the lifecycle data of the components. This makes it relatively easy to understand that components need replacing before the product verification progresses too far.

All too often, components selected during the development phase, and used for prototypes and certification, approach the end of their lifecycle when the final product reaches volume production.

NRND and Obsolete. It is not unusual for electronic designers to include components that are close to being phased out (Not Recommended for New Design, NRND) or are no longer manufactured. This is because components are often selected based on good experiences from previous designs, or because they are included in internal component databases in the company. As such component databases might be updated far too seldom, the designer is unaware about where a component is in its lifecycle.

New components. Newly launched components (engineering samples) can be very tempting, but depending on the component manufacturer's ramp-up plan, they can be difficult to purchase in sufficient volumes.

Unique components. If certain properties make selected components unique and mandatory for the function of the product, mark them clearly in the BoM to ensure that the manufacturer does not use a different, but similar, component (that might be an approved second source for other applications) instead.

The fact that the data sheets for two electronic components are similar does not mean they will always behave the same way in all use cases. For example, rise and fall times can differ between different brands of the "same" component, that can have fatal consequences in some designs.

See the section about bills of material: BoM

Another case of unique components is so-called design-in components, that is a common phenomenon when using strongly cost-driving components. This entails entering into an agreement with a component manufacturer during the development stage. They agree to sell a specific component, usually via a specific distributor, to a specific customer or project. This agreement is often reached between the developer/product owner and the component manufacturer, who then assigns a special price to this particular distributor. The distributor also provides technical support during the development phase. This procedure ensures that the component distributor is paid for the support they provide.

A variant of this is when the component manufacturer provides support themselves and then sets a special component price.

When issuing a request for quotation, it is important that the PCBA manufacturer is informed that certain components in the design are subject to a design-in agreement, so they can set a correct price for the board.

Second Source. Remember to try and find second source components as often as possible. This makes it easier to avoid long delivery times or production stops due to component shortages.

Chemical content. This also needs to be considered when selecting components. Certain commercially available components contain chemicals that are not permitted under RoHS, for example, or information must be provided about them under REACH.

Chap. "4.1.1 Design for Compliance"

End-user certificates may be required to allow certain advanced components to be used in a design. The product owner signs an End-User Certificate, that is issued by the component manufacturer, that contains information about what product the component is used for and the countries to that it will be sold. This is to prevent access to certain advanced components, such as signal processors, FPGAs etc, by countries under trade restrictions.

- Resistors and capacitors in chip format (surface-mounted components, SMD): Reduce the number of different values of standardised passive components. E12 or E24 series are usually sufficient for resistors.
- Component selection can cause quotations from different manufacturers to vary in price and lead time, as different manufacturers cooperate with different component suppliers. Always ask follow-up questions.
- Make a clear agreement with the manufacturer concerning who is responsible for purchasing and negotiating the prices of components. If a manufacturer has a special agreement with component manufacturers or distributors, it should benefit all parties.

Lifecycle Management entails sustaining a product during its entire life span. Please note that PCBAs included in a product may need to be produced many years after the product is taken off the market to guarantee access to spare parts. It is important to have a plan for spare parts supply and sustained service for customers.

During the production phase of a product series, lifecycle management is often a matter of monitoring when components reach their end of life (EOL), when new directives are issued or when existing directives are updated, that affect component use in different markets. If there are no approved second source components, it may become necessary to re-design.

Chap. "3. Product regulations"

An alternative when a component reaches its EOL is to buy stock before the Last Time Buy (LTB) date. Another alternative may be to buy components from the spot market later on. However, be prepared that there may not be a quality guarantee for these components. They need to be evaluated to ensure they are the correct components and that they work as intended. Initiate a dialogue about this with the manufacturer of the circuit board assembly if spot market parts become necessary.

4.1.3 Design for Manufacturing/ Production, DfM/DfP

Design for Manufacturing and Design for Production are interchangeable terms and mean that ease of manufacturing is taken into account during the design phase, and are also referred to as manufacturability or producibility. A printed circuit board will pass through many process stages before it becomes a finished assembly. This chapter provides a summary of these stages.

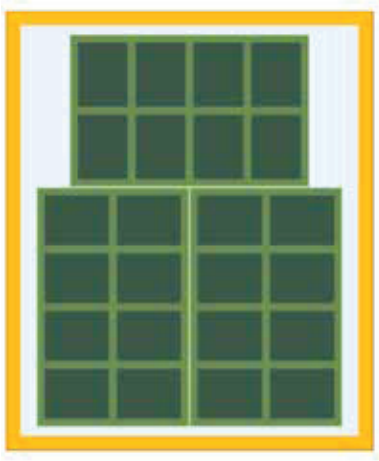
PCBs are normally supplied in panels whose size suits the manufacturer's machinery. It is important that both the developer and PCBA manufacturer communicate well with the PCB manufacturer, so that raw material usage can be optimised to benefit price and environmental impact. Please refer to the example below. When the PCBs have been mounted with components, the panels can be divided into separate PCBAs. Post-processing and testing can be done both before and after the panels are divided.

Chap. "4.1.4 Rules of thumb for design of printed circuit boards and PCBAs"

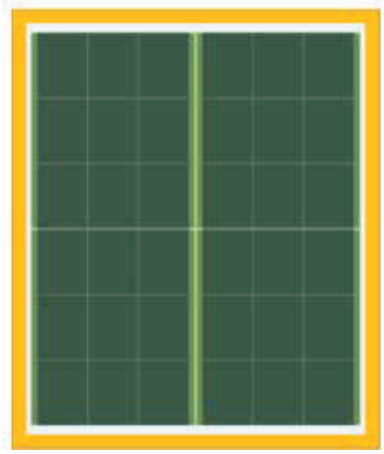
Chap. "6.3.5 Panel drafting"

SMT (Surface Mount Technology) Surface mounting components is usually the core process in today's electronics products. If nearly 100% of components are surface-mounted on a board, it is already a cost-effective product. The SMT process includes sub-processes such as dispensing and/ or screen printing of solder paste, component placement, reflow soldering and a number of inspection stages. Inspection can be SPI (Solder Paste Inspection), AOI (Automated Optical Inspection) and AXI (Automated Xray Inspection), that can be included inline or off-line.

Examples of how the same board size on different panel sizes affects the utilisation of PCB raw material:



A CIRCUIT BOARD OF 70X90 MM ON A CUSTOMER PANEL THAT IS 210X330 MM, THAT GIVES UTILISATION OF 48.9%.



A CIRCUIT BOARD OF 70X90 MM ON A CUSTOMER PANEL THAT IS 226X270 MM, THAT GIVES UTILISATION OF 73.3%.

Reference points or alignment marks on printed circuit boards are standard requirements for an optimised surface mount process. Not all production equipment supports all variants of reference points, so please consult the manufacturer for tips. The example below displays two recommended variants. Reference points are used to centre the boards in many different processes (like screen printing and inspection). Please note that the reference points should be on both sides of a double-sided printed circuit board, even if components are mounted on one side only.

Include two or three reference points per board, not four. If references are placed four symmetrically, the board can be turned in any orientation by mistake.



RECOMMENDED DESIGN OF REFERENCE POINTS.

Solder paste application is achieved either by using a laser cut or etched plate, where solder paste is screen printed onto the printed circuit board, or direct dispensing (jetting).

Adhesive can be applied in later stages, after the solder paste has been applied.

A large number of parameters are involved in the screen-printing process, including pressure, speed, squeegee design and cleaning interval, that are critical to obtain a fully acceptable result.

Component placement (Pick and place) is usually done in a surface mount machine that picks up and places components using nozzles (vacuum). Placement accuracy and repeatability vary depending on the equipment used, and should be selected and matched with the complexity of the product. Some components may require custom nozzles because of their size, shape or weight.

Bare dies, such as "flip chips", are mounted directly on the PCB with their contact surface facing down, after that the wafer is soldered directly onto the PCB. You can Wafers can also be placed with their contact surface facing upwards on a PCB. However, this requires a completely different process, that takes place after the SMT. Please refer to the Chip-on-Board section below.

Soldering of SMT components is done by reflow, in a hot air oven (reflow) or a vapour zone oven (condensation soldering).

During reflow, the temperature of each heat zone, the speed of the conveyor belt and oven power are some of the critical parameters that create fully acceptable solder joints.

SMT processes are normally quality controlled in a number of inspections such as SPI, AOI, AXI and XRay. These are described in:

Chap. "6.2 Description of different tests".

THT (Through-Hole Technology), i.e. inserting component leads through holes, is still used even if it is difficult to automate. It is still used because certain components cannot be converted to surface mounting. Through-hole components should, however, be avoided if possible as handling is more complicated and might even require manual assembly. THT mounting includes a number of processes such as preparation, masking, pre-assembly, insertion and soldering. Inspection can be done by automated optical inspection (AOI) or by manual inspection.

Preparation means cutting and shaping component leads. Techniques ranging from manual shaping and clipping to the use of fully automated machinery are available.

Simple preparation machinery merely cuts off taped components for direct application to the printed circuit board. More advanced machinery can shape the component leads, so that components are mounted at a certain distance above the printed circuit board or lying down, an alternative to ready-made spacers. Component leads may need special shaping when unique components and designs have been developed.

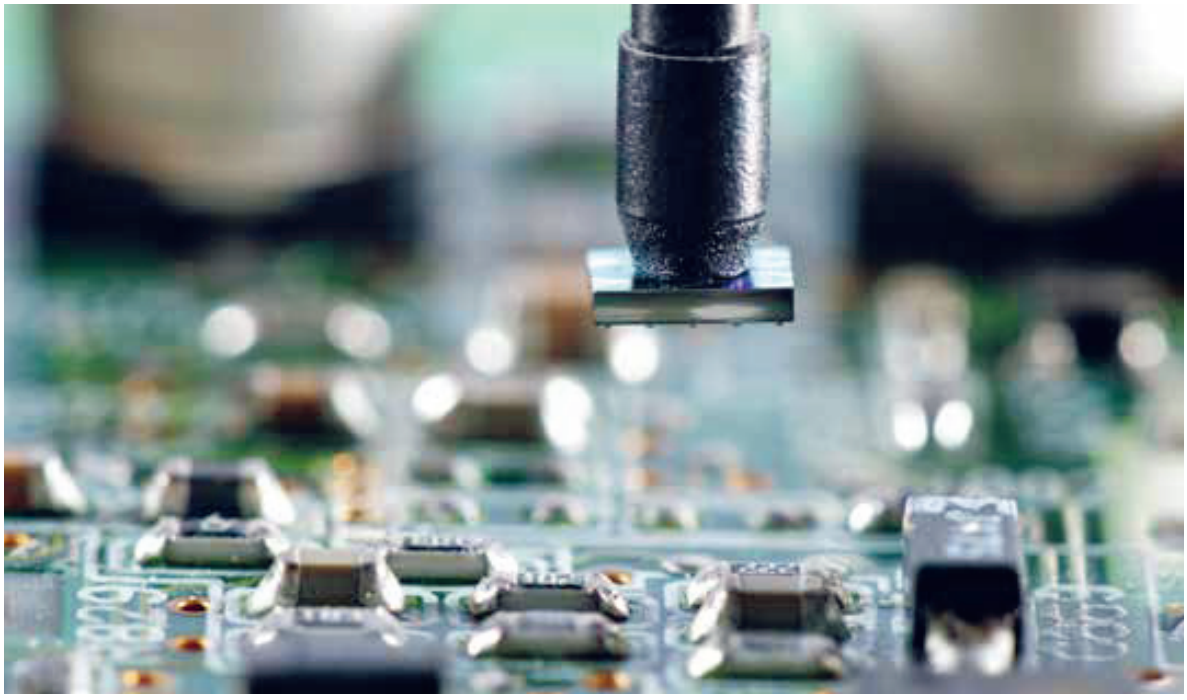
Pre-assembly is done on snap-in components and components that require fasteners.

Assembly of the THT components to be soldered then follows. Most THT assembly in Sweden is done manually unless unique serial production systems are developed. Automatic placement machines were previously used but these have been phased out as most components have become surface-mounted.

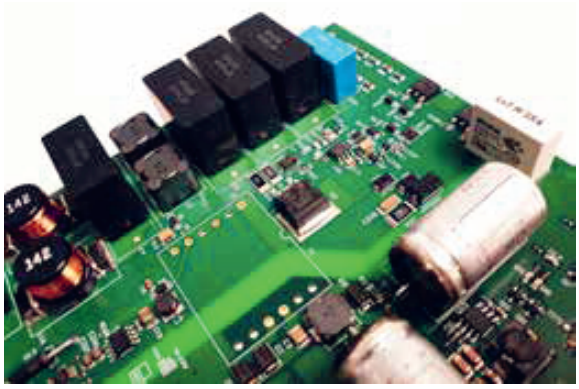
Soldering of through-hole components requires wave soldering, selective soldering or hand soldering.

Wave soldering is done in an open frame or in a selective soldering frame where only the areas to be soldered are opened up. The flux, heat profile, speed, inclination and design of the wave are important parameters in the soldering process. It is common to use nitrogen in the soldering process (nitrogen hood) to create an inert environment that prevents oxidation during soldering (enhanced process window). No-clean flux is the most commonly used flux product as it eliminates the need to clean the PCBA after soldering. Selective soldering machinery is specially designed to solder double-sided mounted boards where both through-hole and SMD components are used.

Temperature, choice of tip, experience and training are crucial in hand soldering. Many components today require soldering to be done under a microscope.



SURFACE MOUNT ASSEMBLY MACHINE THAT PLACES SMT COMPONENTS WITH VACUUM TOOLS.



A CIRCUIT BOARD ASSEMBLY WITH A MIX OF SURFACE AND THROUGH-HOLE COMPONENTS.

In addition to THT and SMT processes, several other processes are used together with circuit board assembly, such as panel division, washing, chip-on-board, underfill, coating and press fit.

No-clean flux does not generally need washing, but some applications have tougher requirements for cleanliness and must therefore be washed. Washing can also be needed before coating. A number of different washing methods are available, where the simplest, but the least, effective method is washing with deionised water. More advanced washing methods include chemicals and range from industrial dishwashers to more advanced equipment.

Chip-on-board entails mounting a wafer directly on the printed circuit board and then using wire bonding. The component is frequently encapsulated in a glob top compound. The process requires a cleanroom and includes the same technologies as manufacturing of integrated circuits. Please refer to the "Bibliography" appendix for more information.

Underfill is often used to mechanically reinforce BGA circuits by adding an underfill compound beneath the circuit. Side fill can be used in a similar manner with components that are mounted flat on the printed circuit board, such as QFN.

Press fit is used for certain high-speed connectors and other components instead of soldering. The component leads are shaped to be pressed into mounting holes and should be pressed with controlled force onto the printed circuit board. Tools or fixtures are needed to fix the component on the top side and to support the underside.

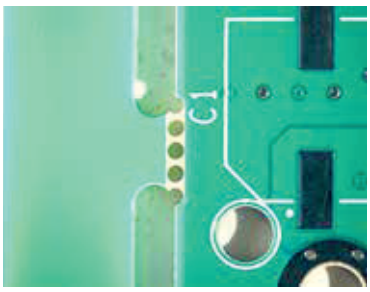
Conformal coating and potting are some of the ways a circuit board can be protected from moisture etc.

Coating can be applied manually by dipping, brushing or spraying. Application can also be automated by using a robot.

These methods can sometimes have an undesired impact on solder reliability:

➡ **Read more: Reliability Assurance of Soldered Printed Board Assemblies Handbook. 2024**

Panel division at breakaway tabs should be done with a milling machine, despite the name. Punches or hand tools can be used in exceptional cases and with great care. Panel division at V-cuts requires a special knife blade splitter. It is important to choose the correct tool and method for dividing panels so as to avoid cracks in solder joints or components.



EXAMPLE OF BREAKAWAY TAB.



EXAMPLE OF V-CUT.

So-called MELF components (Metal Electrode Leadless Face) should be avoided if possible as they are likely to roll about on the circuit board during the production process. Their solder joints are also smaller and thus less mechanically stable than other surface mount joints.

If there is room for surface mounted components with larger packages, they may be preferred to smaller ones, as they are easier to mount and easier to inspect after soldering. Talk to the manufacturer about their preferences.

IPCA610 contains useful information about electronic assembly.

4.1.4 Rules of thumb for designing circuit boards and circuit board assemblies

This chapter follows the previous chapter, Design for Manufacturing, closely. It is intended to be used as a check list, with rules of thumb for designing PCBs and PCBAs to facilitate producibility and ensure quality. These rules of thumb are based on long experience of electronic production, and most people should be able to use them as a starting point for their own designs. However, it is important that each person considers whether these rules of thumb give the correct design for the functions of their own project.

Solder paste stencil thickness should mainly be determined by the reliability of the solder joint. Experience has shown that thickness is important in this respect. The flatness specification for components with many terminations is typically 100 µm. The recommended thickness for the solder paste stencil thus becomes 125 µm instead of the usual thickness of 100 µm, also for:

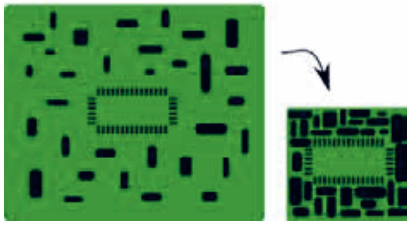
- QFN packages
- Ceramic components down to and including 1005 (0402 Imperial)
- Oscillators (these often need enlarged paste screen holes, to increase the distance somewhat to the circuit board)

Where there are openings in the solder paste stencil:

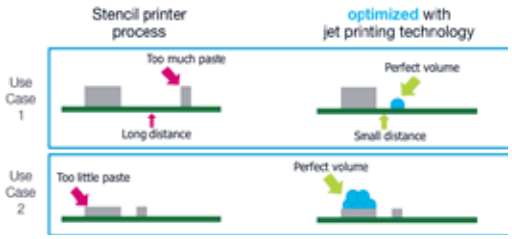
- Openings must not be larger than 3.5 mm, otherwise the squeegee will remove too much solder paste
- The corners of the openings in the stencil should be rounded, with a recommended radius of 125 µm, to allow the solder paste to release better from the stencil
- There are methods (so-called nano-coating) for surface-treating the stencil to make the wall openings smoother and improve release
- For BGA, the paste openings should be larger than the solder tabs. This provides increased distance between the components and the printed circuit board, that makes it easier to wash the printed circuit board if necessary. A square with the same side as the diameter of the solder pad should be used.

Jet printing or ”jetting” is an alternative to using a solder paste stencil. Jet printing uses a technology similar to the way an ink jet printer works. Solder paste is ejected onto the printed circuit board by piezo technology or compressed air. The advantage of this technology is that it is completely software controlled, that permits rapid iteration of new designs, optimisation and correcting ”on-the-fly” in production, and rapid switches between different jobs.

In addition to the speed of switching between jobs, this technology solves a common problem, i.e., that many components need different thicknesses of solder paste to optimise the solder joints. The thicknesses of a solder stencil is often a compromise between different components, as the ideal is to have only one thickness of solder paste stencil.



Jet printing avoids this type of problem as each individual solder pad can be given an appropriate volume, and the design of the circuit board can be optimised to become smaller and more compact without needing to consider the limitations of the solder paste stencil. The volume on each solder pad comprises the minimum number of impulses needed to achieve the target volume at the same time as the size and positioning of the solder pads are considered.

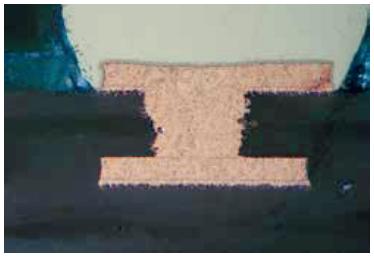


Jet printing improves flexibility at the expense of time, compared with solder paste stencils, but in high-volume production it can be advantageous to combine a solder paste stencil with jet printing. The combination of both technologies provides the high speed offered by solder paste stencils plus the flexibility and accurate volume of the jet printer.

Vias in solder pads should be avoided so as not to reduce the reliability of the solder joints.

- When there are through-going or blind vias, solder is sucked down into the via hole and ends up causing insufficient solder on the surface.
- When there are through-going vias, the solder can cause an unevenness that causes problems on the opposite side, for example if a heat sink or print solder paste is desired in a second soldering operation.
- Macro vias cause air inclusions that expand and cause a potential risk during the soldering process.

Macro vias in soldered surfaces should therefore be filled up with conductive material, such as copper.



- Vias in thermal solder surfaces are not recommended and should be minimised if they can not be plugged, i.e. filled up with epoxy or similar in accordance with IPC-4761 method VI or VII. On the other hand, it is a good idea to site the vias outside the actual solder pads, protected by solder mask, if there is room.

Solder pads

- BGA solder pads on PCBs should be as large as the BGA package if they are not defined by the solder mask. If they are copper defined, the diameter should be reduced to 85%. This applies if they are defined by the solder mask on the BGA package, that they generally are. If they are defined by copper on the BGA package, they should be defined similarly on the PCB, using the same diameter as the BGA package. It is unsuitable to mix solder-mask-defined and copper- defined solder pads on the same BGA package.
- For chip components that are 1608 (0603 Imperial) or smaller, the copper connections to the solder pads should have the same thermal mass. It is recommended to use "thermal relief" solder pads if the connections are on the same plane to eliminate the risk of the "tombstone" effect.
- To reduce the risk of over-soldering, it is recommended to have solder mask between solder pads on QPFs without reducing the size of the solder pads. The recommended minimum width is 0.08 mm but depends on the supplier. Please consult the intended circuit board supplier.

- It is not recommended that solder mask come up on the solder pads (with the exception of solder-mask-defined solder pads) although this is permitted by IPC on all solid surfaces except BTA solder pads. Solder mask openings should be 100µm (50µm per side) larger than the actual solder pads because of manufacturing tolerances.

Component assembly

Your various rules of thumb should be considered to facilitate production of PCBAs.

- Components older than 12 months should not be used in serial production unless they have been stored correctly. Correct storage means an inert environment such as a vacuum or nitrogen.
- When pin-in-paste technology is used, the pins should not protrude more than 0.5 mm past the printed circuit board. The solder paste could otherwise form a drop, which, in the worst case, could fall off in the oven and damage it. The result would also be insufficient solder paste in the solder joint.
- In double- sided assembly, large differences in the number of components on the top and underside (>20%) should be avoided. Cost is reduced because the installation machine can more easily optimise installation.
- In double-sided installation, ensure there are margins between in-hole-mounted component leads and surface-mounted components to facilitate hand soldering, selective soldering or a selective wave soldering panel. Requirements from the manufacturers are usually about 4 mm.
- Components should be rotated at 0° or 90° to facilitate inspection.
- Packages that rest against the printed circuit board, such as QFN, should not be mounted on the side that is to be wave- or selective-soldered. QFN packages must be protected from the solder wave. The flux that is sprayed before selective soldering can creep in underneath the package and cause electrochemical migration (ECM).
- High impedance circuits should not be located on the side of the printed circuit board that will be selective-soldered. Flux can be corrosive and cause electrochemical migration (ECM) or change the impedance of sensitive circuits.
- High (>5 mm), heavy and through-hole components such as inductors and electrolytic capacitors should be located on only one side of the printed circuit board to facilitate assembly. Heavy and through-hole pin-in-paste components must be located on the same side to prevent them falling off during reflow soldering. Please remember that the oven may have limited space below.
- Components that absorb a large amount of x-ray radiation (e.g. high-density components) should not be mounted in double-sided assembly so that they overlap components mounted in the second phase. No components may be located underneath a BGA so that they conceal the view of the solder joints on a BGA because the X-ray field of view is shadowed. Minimise the number of small components (suppression) that conceal solder joints.
- Some components (such as connectors) must be located with the component body outside the edge of the PCB. This is not permissible unless the component is fixed mechanically or if the component interferes with the conveyor belt on the assembly line. Discuss and design the panel in collaboration with the manufacturer to avoid interference with production of the PCB.
- Avoid locating components too near the edge. A good rule of thumb is at least 2 mm from the edge of the printed circuit board. Components near the edge are more often subjected to stress and can interfere with the assembly line conveyor.
- Avoid placing ceramic surface-mount components close to break-off tabs and V-cuts. When panels are divided along break-off tabs or V-cuts, small components can easily be destroyed by microcracks when the circuit board is flexed by the breaking movement.

Locating a copper plane right beside the break-off tab helps keep the laminate together during division.

- Leave sufficient space for mechanical components, installation holes, heat sinks, connectors, fixing brackets, marking and access for screwdrivers, assembly fixtures for press fit components, riveting tools etc during manufacture.
- It is advisable to consult the manufacturer before starting CAD work on the product.

Selective soldering

Lead-free selective soldering is a process that entails a considerable risk of over-soldering. Press fit is a better choice in many cases in terms of price and quality. If selective soldering has to be used, several issues need to be considered.

- Spacing for selective soldering must be at least 2.54 mm, and leads should protrude 0.5 – 1.5 mm through the PDB to avoid of short circuiting.
- In selective soldering, the hole diameter must be the maximum diameter of the leg (plus tolerance) +0.4 mm to allow capillary forces to work well.
- In selective soldering, the teardrop ”annular ring” should be 0.5 mm. If soldered manually, the solder pads have to be of a certain size to avoid breakage. Temperature shock during soldering destroys the solder pads if they do not have sufficient room for expansion along the Z axis.
- In selective soldering, a 5 mm boundary area should be used to prevent flux from being sprayed onto SMT components. Inactivated flux can cause ECM. In addition, small surface mounted components can be pulled along by the melted solder paste.
- In selective soldering, the distance between the component and the PCB must be at least 0.3 mm because otherwise the flux will have difficulty outgassing.
- Use round leads for selective soldering as they have a wider process window than leads with a rectangular cross-section.
- In selective soldering, use thermal relief at the junction to the copper plane to achieve even heating of the plated hole and ensure full upwards flow of the solder, which facilitates inspection.
- In selective soldering, the distance between the plastic alignment pins on a connector and the leads to be soldered should not be less than 4 mm. Alignment pins can be deformed and balls of solder can be formed that loosen later on.

Conformal coating

- An application sometimes need protection by so-called ”conformal coating”, often when the circuit board assembly needs to meet environmental requirements such as moisture and dust in its application. Different polymeric resins can be used, such as acrylic, silicone, urethane.
- The coating has a maximum temperature tolerance. Some components on a circuit board assembly can become hotter than this, especially if the circuit board assembly is encapsulated.
- The electrical properties of the coating can affect the function of the circuit, e.g. RF circuits.
- Coating can disturb the mechanical installation of the printed circuit board assembly in its enclosure. For this reason, define a boundary area for lacquering printed circuit board assemblies that will be mounted in mechanical devices.
- The most common method of applying the coating is robot spraying down onto the circuit board assembly through a nozzle. The designer should choose component locations with care to avoid spraying components that should not be sprayed. To ensure that the coating is applied correctly,

a drawing should be made that describes the areas and components that need coating. Masking is also a solution but should be avoided if possible because it is expensive.

- Because adhesion between coating and solder mask can vary between different brands, consult the chosen printed circuit board manufacturer for the best results. Practical tests may be necessary to verify the final result.
- Components such as buzzers, connectors etc have to be covered during the process to avoid their becoming filled with resin. There must be access area for installing and removing protective tape etc.
- Try to retain the card in its production panel during lacquering as this facilitates handling.

Research evidence shows that coating can make lead-free solder joints less reliable.

➡ **Read more in: appendix "Reliability Assurance of Soldered Printed Board Assemblies Handbook (2024)"**

➡ **Appendix "Bibliography" Chap.**

Underfilling of BGAs

- One method for reducing stress on the solder joints of the BGA is to secure the component with glue after the soldering process, especially if the product is subjected to heavy vibration or impacts. The glue is applied with a dispenser and creeps in underneath the component due to capillary force.
- If underfilling needs to be used, a certain distance is needed between the BGA and adjacent components for access to apply the glue and so that the glue can creep in the correct direction. Consult the manufacturer to avoid problems.

Some BGAs do not have a full matrix of solder balls. This can cause problems with air pockets because capillary force is not sufficient to fill out underneath the entire BGA. If the circuit board has not been washed or if there is flux residue underneath the BGA after washing, the flux residue can also cause air pockets.

General recommendations

- A design review of producibility (DfM/DfX) should be done jointly by the developer/PCB manufacturer/PCBA manufacturer. The manufacturer will need to approve discrepancies that cannot be rectified through new documentation. Signed notes from this meeting should be appended to the order documentation.
- Circuit board material for rigid PCBs should be referred to the IPC-4101 standard. Unless there is a special reason, avoid specifying a particular make. A sample requirement specification might be: "FR4 material in accordance with IPC-4101/99"
- Parameters that affect the choice of base material include the number of layers, board thickness, copper thickness, all structures, number of solder processes, operating environment, solder/solder-free soldering process.
- It is advisable to consult the circuit board manufacturer about the choice of material.
- Avoid more stricter requirements than ±0.15 mm on the external dimensions of the printed circuit board.

Patterns on printed circuit boards

- Copper coated areas should be balanced within 20% to avoid the risk of bowing and twisting. This applies to each layer and the entire layer design.
- For non-plated through-holes, ensure the solder mask opening is at least 0.2 mm larger than the hole diameter to avoid residue.
- The minimum size for through-going drilled vias that does not increase cost is 0.25 mm after plating (applies to board thickness of up to 1.6 mm).
- The minimum size of a laser-drilled via that does not increase cost is 0.1 mm.

- The minimum recommended trace width/distance between traces is 100µm/100µm, that does not increase cost. If the base copper thickness exceeds 18µm (1/2 oz), ask the PCB manufacturer for recommendations for suitable trace width/spacing between copper areas.
- If narrower trace widths/distances between traces are needed, consult the PCB manufacturer.
- All connections to vias and solder pads for a BGA or press fit should have collar reinforcement ("teardrops") if there is room. Teardrops reduce the risk of fracture indications and increase the registration tolerance of drills. Boards designed to comply with IPC class 2 should have teardrop reinforcements.
- The recommended teardrop size should be at least 0.3 larger than the diameter of the hole.
- For laser drilled holes, the teardrop should be at least 0.2 mm greater than the diameter of the hole.

4.1.5 Design for Test/DfT

Consider "designing for testing" to achieve the desired quality in the boards that are to be produced. Different production test methods place different demands on the design to allow it to be implemented. Doing so from the outset saves costs as it is expensive to modify designs to suit various tests when a printed circuit board is fully developed.

Conduct a test coverage analysis of the design to determine suitable test methods and build in facilities to make it easy to do the required tests. Moreover, configuration to suit the intended test interface is also important. This includes guide holes, test points, sight marks, test connections etc. In addition, check whether the printed circuit board will be tested in a panel or as a separate PCB when the panels have been divided.

Chap. "6.2 Description of different tests"

The concepts of test and verification are sometimes used interchangeably. However, verification generally means the work that the developer does to ensure that the finished design complies with the stated requirements.

This can include functional verification or a wider verification, such as environmental or EMC requirements.

Testing generally refers what the manufacturer does with manufactured products to ensure that the manufacturing process, printed circuit board assembly, and components function correctly.

Tests are done either on all manufactured products/PCBAs or on samples only. At this stage, the product owner has to agree with the manufacturers about what should be done and who should pay for any specific test hardware and software. It is quite common that all manufactured PCBAs are tested to begin with, but transitions to testing samples when manufacturing process has become sufficiently stable and the number of the faults is low.

Traceability is an important quality parameter. Agree about how test results should be saved so that it is possible to go back and see trends or find individual units that might develop faults when in use. The latter requires that every circuit board assembly is assigned a unique serial number during manufacture, that is also registered during testing.

Who should develop the tests and test tools?
This can be done by the product owner, developer, manufacturer or any external party that specialises in test development. Ensure agreement about this early in the project, as test tools are associated with extra costs.

Who owns the test tools and test software?
And who is responsible for maintenance and future upgrades? These are important questions that should be agreed before serial production starts.

Read more in Appendix "Bibliography"

Test tips

In some cases, tests can be conducted ”without extra cost” when other tasks are done. An example is when the manufacturer downloads test software to a printed circuit board assembly. This enables automatic checks on the function of certain parts the PCBA.

Visit the intended manufacturer and inspect their processes and production. This makes it easier to develop a product that can be produced rationally.

If ICT testing is to be used, ensure that there is sufficient space between SMD/THT solder pads and ICT points, so that the test pins are not contaminated by flux residue.

Configure to suit the intended test interface, e.g. sight marks, guide holes, test pads, test connectors etc, and whether the PCBA will be tested in the panel or separately.

Who owns and who maintains test equipment? The product owner or the manufacturer? Ensure that this is clearly stipulated in the contract.

If possible, choose junctions and connectors in test equipment that can be connected several times, or include specially designed wear parts that are replaced regularly before they cause problems in production.

If a PCBA needs conformal coating, design the test so it can done without disassembling the production panel. This facilitates testing and handling of the lacquered printed circuit board assembly.

4.1.6 Design for Failure/ Worst mistakes

Nobody wants to design a bad product. But too often product launches are delayed, or even worse; the product does not work as intended when used by the customer. So why is that? Below is a list of common mistakes that have been gathered overmany years of experience. They are not listed in any specific order.

How to lose time in product development:

✖ Never reuse a functioning design.

✖ Never read the component manufacturer's data sheet.

✖ Never waste time in technical discussions and above all, do not review each other's work. Designers, layout experts and product owners work separately. Nobody reviews what the others do, so that all misapprehensions are discovered at the end of the project.

✖ Do not involve mechanical designers, purchasers, production engineers or anybody who knows about EMC before the design is finished.

✖ Start working on a production test strategy once the design is finished.

✖ Check regulatory requirements applicable to the product just before product launch, when it turns out that the product must be CE marked.

How to create technical problems that are difficult to solve:

✖ Do not bother with decoupling capacitors or mount them far away from the component they decouple, while omitting the earth plane.

✖ Make all traces on the circuit board really thin, especially the conductors that carry a lot of current or the conductors that earth a component that handles high frequencies.

✖ Do not bother about layer structure on the circuit board drawings and let the circuit board manufacturers do what they want. This can be reinforced by changing the PCB manufacturer now and then, so that the circuit board structure changes from time to time.

✖ Impedance matching sounds difficult, so ignore it.

✖ Keep the number of layers in the PCB to an absolute minimum, leading to generously perforated voltage planes and earth planes. Or do not bother at all about a homogeneous earth plane.

✖ Make the switching node in the DC/DC circuit design as large as possible and preferably surround sensitive, high impedance components with this node.

How to make product maintenance more difficult:

✖ Choose components that are at the end of their lifecycle.

✖ Choose file names with no connection to revision numbers, so that each file has to be opened to find its revision.

✖ Change revision numbers only when major changes are made; minor changes can be managed by email to the manufacturer.

✖ Make sure that it is impossible to track the printed circuit board item at hand. Avoid etching the part number into the copper.

Connector special:

✖ Place connectors very close to each other, so the cables cannot be connected.

✖ Marking pin number one in silk screen printing is cheating. It is more fun to let operators guess.

✖ Only inspect contact pin layout on the circuit board. The other side of the cable connection is somebody else's problem.

✖ Choose the same connector for several completely different functions to make it easier for the user to connect it wrongly. At best, the design will burn up altogether, that eliminates troublesome returns.

4.2 Schematics

A Schematic, (also called circuit schematic or diagram) is a graphic representation of an electrical circuit. The schematic shows the components included and the connection between them by means of standardised symbolic representations.

The schematic is used for the design and maintenance of electrical and electronic equipment. The manufacturer does not really need the electrical schematics to make the circuit board assembly, but it is necessary if the manufacturer is responsible for devising production tests or faults tracing.

CAD programs are used to draft schematics. Please refer to:

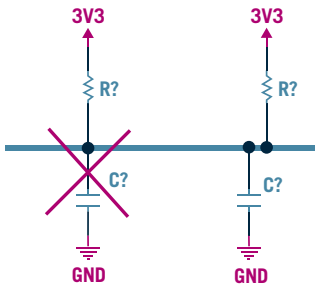
Chap. ”4.3.2 CAD and layout”
Chap. ”4.4 Development tools”

The same tool is used to create the graphical illustration of the circuit and to design the printed circuit board. The choice of CAD tool and format of the files saved must be compatible with the tools used when layout work on the printed circuit board is conducted.

A schematic should contain instructions to the PCB designer who does the layout work. But the electronic designer can and should normally participate in layout work, because they know what they have been thinking during the design work and can thus contribute important information. It is important that the electronic designer reviews the design documentation and schematics with the PCB designer. Discuss difficulties, risks and expectations at an early stage.

The way that the schematics is drawn is a matter of taste and is often affected by the program used, but below are some tips that might be worth considering:

- The primary purpose of the schematic is to enable understanding of the electrical function.
- If the schematic comprises several pages, try to group functions so that each page has a short heading.
- As far as possible, signals should travel from left to right.
- Do not pack the pages too tight.
- Restrict the schematics pages to A3 and A4, landscape format, so they are legible on printouts and full screen.
- The flow in a schematic should correspond to the function. For example, put suppression capacitors in the schematics close to the pins that they are to decouple.
- On components that have replaceable pins, such as GPIO (General-Purpose Input/Output) on processors, choose pins that suit the layout rather than what feels natural in the schematics.
- Choose signal names that indicate what the signal is and do not use multiple names that are confusingly similar but different. For example, avoid using both ”+5V” and ”5V”.
- Write information in the schematic that facilitates review. For example, the current levels that blocks or outputs are dimensioned for.
- Mark isolation barriers and dangerous voltages if they exist.
- Structure the numbering of components in a stringent manner. For example, let all component numbers start with 100 on page 1 and 200 on page 2 etc. The advantage here is that the BoM and layout automatically contain information about the function block to that they belong and where they can be found in the schematic.
- Avoid four-way crosses as they be confused with two conductors that cross without being connected.



Simulation

Because prototype manufacture is often a considerable cost in the project, avoid unnecessary updates and modifications to the printed circuit board late in the development work. One way to reduce the risk of incorrect design is to use simulation.

SPICE (Simulation Program with Integrated Circuit Emphasis) is an industry standard for simulation tools, in that every component has a SPICE model that is provided by the component manufacturer.

The SPICE tool can be a more or less integrated part of the CAD software.

4.3 Designing printed circuit boards

The PCB is the physical implementation of the electric circuits in the electronic designer's schematic and is one of the most important components in the product.

The design of the PCB should efficiently solve the electrical function, but an almost infinite number of combinations of materials and production methods are available. The electronic designer sometimes designs the PCB, but this is often done by a PCB designer who is an expert user of PCB-design CAD systems. A good introduction and close contact between both are needed so that the PCB will comply with requirements.

Different PCB properties require an active choice of input material and manufacturing method when the PCBs are produced. Please note that different PCB manufacturers use different processes and not all are equally suitable to produce a specific type of PCB, that may affect price and lead time. Please refer to:

Chap."4.3.5 Costs of printed circuit boards"
Chap. "6.3.4 Ordering of printed circuit boards/PCB"

The different combinations of properties give the finished printed circuit board assembly (PCBA) various properties and performance. Examples of properties are isolation distance, trace width, impedance matching at higher frequencies, tolerance of higher temperatures, vibration tolerance, signal integrity etc.

4.3.1 Requirements for printed circuit boards

Quality standard/Acceptance level. It is best to refer to an established quality standard that most PCB manufacturers are aware of and that their quality/process control/inspection departments already work with.

The IPC 6010 series is one standard that the entire industry follows, from suppliers of input material to producers of the actual PCB.

Avoid using the old Danish standard, PERFAG, for example, that contains more or less the same elements as IPC 6010, but is not as widely disseminated.

The advantage of a standard is that the acceptance level of approved/not approved is good.

If the standard is deemed insufficient, use a supplementary requirement or what the IPC defines as AABUS (As Agreed Between User and Supplier).

IPC also allow some leeway for certain modifications, mending and improvement of PCBs. It is important to understand what this means. Changes beyond this must be specified in the printed circuit board specification

Below are two examples:

- **Collar reinforcement "teardrops" at via pads** IPC-6012 allows the circuit board manufacturer to add teardrops in this case if they are not found in the layout files. This permission applies to all class I and II boards. If this is not permitted for some reason, it must be mentioned in the circuit board specification.
- **Non-functional pads (NFP) on interior layers.** In this case, IPC-6016 does not allow the circuit board supplier to delete these pads without customer acceptance.

However, there are so many advantages to deleting these, both electrically and related to production, that it is advisable either to design without NFPs or accept that they are deleted from the circuit board specification.

	IPC Class I	IPC Class II	IPC Class III
Production category	Consumer electronics	Dedicated electronics	Electronics with extra high reliability
Product examples	Personal computers, cellphones, tablets, TV, DVD, satellite receivers, radio receivers.	Industrial computers for installation, automobile electronics, industrial control electronics, medical equipment, some avionics, some military electronics.	Medical systems, space electronics, defence electronics with stringent requirements for function even when the products are subjected to vibration, thermal shocks etc.
Expected service life	Limited service life	Longer service life	Long service life
Expected reliability	Lower reliability acceptable	Continuous operation, uninterrupted service desirable, but not critical	Continual operation or performance is critical; no stoppages can be accepted

THE STANDARD DEFINES THREE CATEGORIES OF BOARDS.

Test requirement/declaration. Even if the IPC has a comprehensive list of tests that have to be done on the boards, the products may require further testing. Examples of such include AOI verification of the inner and outer layers, impedance verification, Hi-Pot tests, special cleanliness tests, or environmental tests.

These must be mentioned in the requirement specification. In addition, automatic or manual tests may be desirable to ensure the final quality.

If automatic board testing is to be included, it may be necessary to add specific test points for this. Please refer to:

Chap. "6.2 Description of different tests"

Regulation requirements also affect the circuit board. Please refer to:

Chap. "3 Product regulations"

Design (CAD):

- Part number
- Revision
- Regulation markings, CE etc
- Recycling information, such as WEEE

Circuit board manufacturer:

- Production week and year
- Factory logotype
- If applicable, CTI/UL/RoHS etc.

This requires that there is space for the information on the PCB and that marking is still visible when all the components have been mounted. Clear instructions may be needed about where and how marking should be done. Note that it can be difficult to get space for full marking on small boards.

Requirements for base material specifications. The base material should be selected with care to ensure that it is suitable for the application and the final product complies with the quality and service life required.

IPC 4100 is a good specification for standard laminate intended for rigid boards as it contains a number of material grades and other properties that can be expected from them.

As most PCBs are exposed to heating cycles, it is advisable to study parameters such as:

- **Tg = glass transition temperature**. The Tg value is the temperature when the material changes from being stiff to becoming more elastic and flexible. Material properties are considerably degraded when the temperature exceeds this value. When the Tg value is exceeded, it mostly affects expansion along the Z axis (thickness), that increases the load on plating in vias and holes.
- **Td = Degradation temperature.** The Td value refers to material degradation and is defined as when the material loses 5% of its weight. This temperature is not an absolute value, but is measured during a temperature rise (10° C /minute) in an oxygen-free environment. This value is particularly important for boards that have to withstand many soldering processes.
- **CTE = Coefficient of expansion.** CTE is a measure of the amount the card expands during heating. It is normally specified as PPM/degree centigrade before and after Tg. It can also be specified as a percentage across a specific temperature range (normally 50 – 260° C). In a lead-free process, Z-axis expansion is a critical parameter. An incorrect choice can lead to broken plating on vias and holes, as well as delamination.
- **T260 = Time for delamination at 260°C.** The time taken for the base material to delaminate at 260° C.
- **T288 = Time for delamination at 288°C.** The time taken for the base material to delaminate at 288°C.

All these parameters are important to ensure that the PCB will withstand soldering operations during manufacture and the temperature variations that the board will be subjected to in its intended application.

It is strongly recommended to choose a material category from IPC-4101 instead of specifying a unique material. The category in the specification would then be specified as "Material in accordance with IPC-4101/126", for example where /126 is the category (in this case it is a high Tg material intended for lead-free soldering with a low CTE).

Even if FR4 materials intended for lead-free soldering are more heat tolerant than older versions of FR4, they are much more brittle due to the different hardening system. In a demanding environment, cracks can easily occur in the material, that causes problems.

The product owner is responsible for discovering whether the material is sufficiently good and ensures the intended service life, even if they are advised by a circuit board manufacturer in discussions and in making decisions.

Copper thickness/plating is an old definition of copper thickness that expresses the thickness in ounce /ft². The usual thickness is used are:

1/3 oz = 12 µm

1/2 oz = 18 µm

1 oz = 35 µm

2 oz = 70 µm

3 oz = 105 µm

The best way is to specify the basic foil thickness of the inner and outer layers and then use the excellent tables found in both IPC-6012 and IPC-A-600. This shows the copper thickness that can be expected on the plated layers (normally outer layer) and on the etched-only layers (normally the inner layer). The outer layer thickness then commonly becomes approximately 20µm thicker with plating in accordance with IPC class II and about 25µm thicker with plating in accordance with IPC class III, so the basic copper plus copper plating is usually specified as the final copper thickness or "finished copper".

The copper thickness that can be regarded as standard and that normally works for most printed circuit boards is 35µm on the inner layer and 18µm on the outer layer.

Many grades of copper foil are available and are normally defined in accordance with IPC-4562.

On standard boards, it is often unnecessary to specify the properties of the foil. On boards with high signal speeds (>5 GHz) and longer conductors, it may be necessary to specify the copper file required to minimise signal losses.

Information about any discussions that might have taken place among developer/manufacturer/PCB manufacturer about specific solutions during development work must be included in the quotation documentation.

Read more in "Reliability Assurance of Soldered Printed Board Assemblies Handbook (2024)".

4.3.2 CAD/layout

Whoever designs or does the layout of the PCB normally compiles the manufacturing documentation.

Such documentation must consider the factory's capability and applicable tolerances in accordance with demands specified in standards to obtain robust and reliable PCBAs. It is absolutely critical that the documentation is unambiguous, so that the manufacturer can produce the PCBs without delay and the involvement of the product owner. Please refer to:

Chap. "6.3.4 Ordering of printed circuit boards/PCB"

The following are guidelines that the person laying out the PCB will have to require from the electronic designer to ensure a robust and reliable PCB.

- Define current capacity and impedance matching for critical conductors at an early stage. This affects parameters such as trace width, isolation distance, copper thickness, choice of material and number of layers. At this point, it is important to consider the tolerances allowed by the standard for trace width and copper thickness to ensure sufficient current-carrying capacity for the function.
- If the component manufacturer has layout recommendations in the data sheet or "application notes", these should be followed as far as possible.
- Define critical sections on the board, such as where earth and voltage planes need to be as intact and free from other conductors as possible.
- Understand the operating environment for the application, that can affect the choice of material, choice of surface treatment, and coverage of via holes.
- Investigate whether testing is needed that requires special test points to be added to the layout.

The PCB designer must also ensure symmetry in card structure for the position of layers, dielectric separation, foil thickness, and copper balance.

4.3.3 Via knowledge

This chapter is about via holes, that are intended to electrically connect different layers in the PCB.

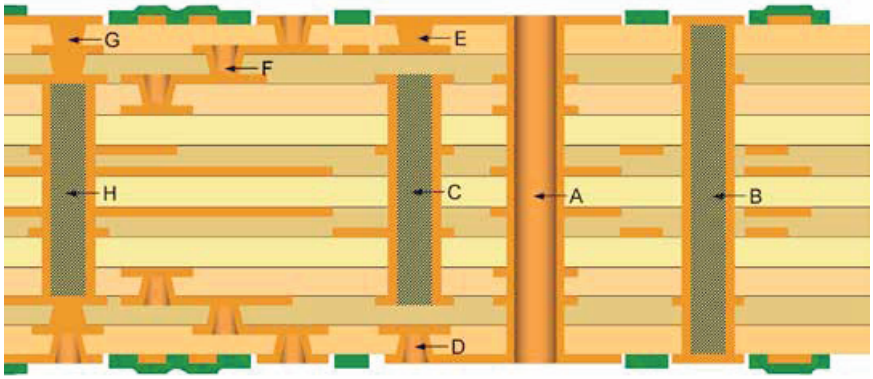
Some examples of applications:

- join conductors on different signal layers
- create earth connections or current connections between one or more inner layers
- heat conduction
- screening of sensitive areas

Ordinary through-going vias are used in simpler boards because they are easiest to make and thus the cheapest. PCBs that have many layers, are densely populated or where unbroken current planes are critical require other kinds of vias, as described below. Using different kinds of vias on the board requires a number of process stages, that means that the cost of the PCB increases.

Via holes can be open or plugged with different materials such as solder mask, epoxy or electrical/thermal conductive compound.
A dialogue with the PCB manufacturer will help decide the via hole structures that can be combined on a board and that are cost-effective at the same time.

Examples of various kinds of vias:



Type A: Through-going via holes

As the name implies, this is a via hole that goes straight through the board. For best reliability, via holes of this type should be left either completely open or completely closed. It is not enough to just close these via holes in the solder mask and then expect fully acceptable coverage/plugging.

Plugging should meet the demands specified in IPC-4716 type VI, either with solder mask or epoxy resin. Consult a PCB supplier about hole sizes that can be plugged as different manufacturers' capability can vary.

Type B: Through-going via holes with plugging and over-plating

As the name implies, this is a via hole that goes straight through the board and is then plugged and plated over with copper.

Use this type of hole when via holes are located directly in solder pads, test points or are so close to solder pads that the solder will be drained from the solder surface into the via hole. Plugging with epoxy resin and plating over is subject to the requirements in IPC-4761 type VII. The thickness of the over-plating is subject to the requirements in IPC-6012.

Consult the PCB manufacturer about the hole sizes that can be plugged, that varies among manufacturers. The method increases the cost compared with holes in type A, especially on boards with few layers.

Kap. "4.3.5 Kostnader för mönsterkort"

Type C: Buried via holes with epoxy filling.

These via holes create an electrical connection between inner layers and are only found inside the board. Filling with epoxy resin is subject to strict requirements, as air inclusion can result in delamination at worst. Symmetry between layers is a critical factor, as in all circuit board design. This means that the buried via holes must be located symmetrically to the centre of the board to permit the flatness that IPC requires.

However, buried vias have several advantages, such as releasing space on the outer layers for component installation, better EMC properties etc.

Type D: Blind holes, micro-vias

This is a via hole that goes from an outer layer and down into, but not through, the PCB. The most common type of blind hole is the micro-via.

In the latest definition of IPC-T-50M, the micro-via is defined as a blind structure with a maximum "aspect ratio" (the relationship between the hole's diameter and its depth) of 1:1, which is connected to a pad at a maximum depth of 0.25 mm, measured from the surface. A typical hole diameter is 0.1 mm, and the distance to an adjoining layer is 0.1 mm.

The holes are normally laser-drilled, but traditional drilling in CNC machines with extremely good positioning on the Z axis also occurs.

Type E: Blind holes, micro-vias with copper filling

When micro-vias are located in solder pads for components, these should be copper- filled to optimise soldering and minimise voids in the solder joints.

This process is necessary for BGA circuits with micro-vias in the solder pads and other components with small "footprints". Micro-vias are sometimes filled with epoxy and plated over with copper. This process is more difficult to control compared with copper- filled micro-vias, however, and also gives inferior electrical and thermal conduction compared with copper-filled micro-vias.

Type F: "Staggered vias", blind/buried micro-vias

"Staggered vias" are stacked and aligned in a staircase pattern.

In this case, it is not necessary to fill the inner layer macro-vias in a separate process, as resin from lamination fills the vias. This method is recommended in IPC-2226 on boards used in demanding environments, such as in the automotive industry.

A disadvantage is that this type of hole interrupts the flat layers somewhat more than stacked macro-vias because of the side displacement required. The primary area of application for this kind of hole is to transmit signals from BGA components with insufficient space for traditional via holes.

Type G: Stacked via holes, blind/buried micro-vias

Stacked via holes located on top of each other are used mainly where minimising the size of the PCB is an important parameter.

The manufacturing process is more difficult to control than for "staggered vias", which makes stacked vias less suitable to use in demanding environments. The advantage, however, is that this type of via hole does not break up flat layers as much as staggered micro-vias.

The primary area of application for this kind of hole is to transmit signals from BGA components with insufficient space for traditional via holes.

Type H: Buried over-plated via holes with stacked micro-vias ("VIPPO")

This type of hole is often called "VIPPO" (Via-In-Pad Plated Over), that means that micro-vias are located above a buried via hole that has been plugged and plated over.

The advantage is that it is easier to reach further down into the PCB and still use components with small "pitch". The manufacturing process is highly demanding and requires strict process control, primarily related to plugging and plating over the buried via holes. This design is less suitable for boards used in demanding operating requirements.

"Failed" vias

The images on the next page display via holes where the process/requirement specification/ understanding has failed in some way. This is to provide insights into problems that can occur that can lead to serious quality failure at worst. Component holes are normally filled with solder during the soldering process, which ensures good environmental protection.

Environmental protection for via holes is often neglected. Requirement specification is very important here to avoid problems short-term and long-term issues such as contamination, corrosion migration etc.

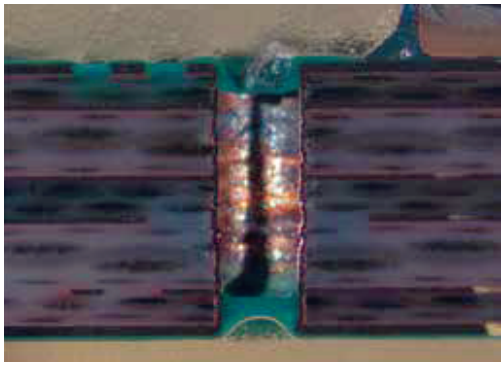
4.3.4 Surface treatment

Many options are available. All service treatments have different advantages, disadvantages and price ranges.

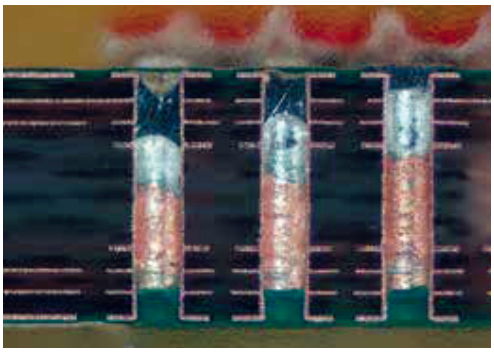
It is advisable to consult the PCB supplier and/or the person doing the layout CAD work to select the most suitable alternative on the basis of the specifications received.

IPC-6012 has general requirements related to surface treatment. Please refer to the IPC-4550 series for more specific requirements related to thickness, chemical tin grade, chemical silver and ENIG.

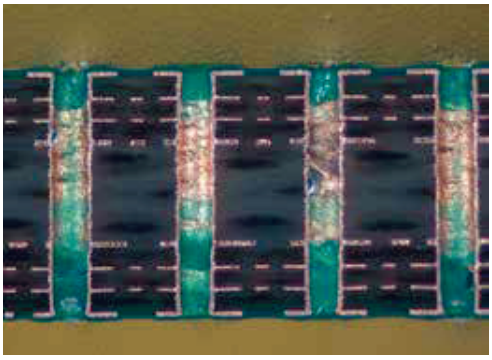
There are no thickness requirements for SnPb, lead-free HASL and OSP, but IPC-6012 requires that surface coatings shall be solderable. The standard also contains references to the way solderability should be tested. There are also requirements for HASL, which the service coating should be cover fully, but no criteria for how this should be assessed.



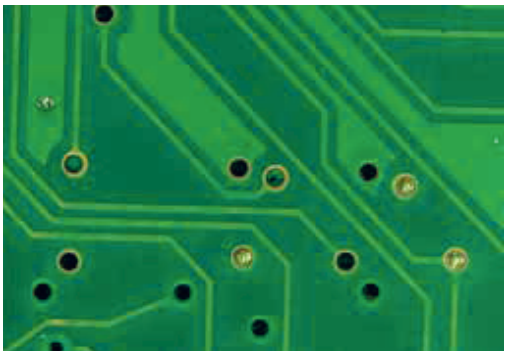
1. The specification has failed here because the board lacks openings in the solder mask (”tenting”) for the via hole in the image. The result is that the hole wall has insufficient corrosion protection. During heating, the air bubble will expand and cause strain and cracks. This via should have either been plugged in accordance with IPC-4761 type VI-b or left completely open on both sides.



2. This board lacks openings in the solder mask for the via holes on one side and only has a small opening on the other side. The board has been specified with HASL for surface treatment, which means that the solder is neither able to fill the hole nor can be blown out because the via hole is blocked on one side with solder mask. Insufficient corrosion protection is also a risk here. The air bubble will expand on heating and may eject small balls of solder that might cause short-circuiting. The IPC does not recommend any type of single-sided coverage/plugging. These vias should have been plugged in accordance with IPC-4761 type VI-b or left completely open on both sides.



3. The specification has failed here because the board lacks openings in the solder mask (”tenting”) for the via holes in the image. Thus, the solder mask has forced its way into the hole and has partly covered/plugged the hole. Small amounts of solder residue have also managed to find their way down into the hole, so process chemicals may likely be encapsulated in the holes These layers should have been plugged in accordance with IPC-4761 type VI-b or left completely open on both sides.



4. Yet another attempt at ”tenting” the via holes. This means that varying amounts of solder mask have found their way down into the holes and have partly covered/plugged them. Balls of solder have also stuck in some holes. The air bubble will expand on heating and may eject small balls of solder that might cause short-circuiting. These vias should have been plugged in accordance with IPC-4761 type VI-b or left completely open on both sides.

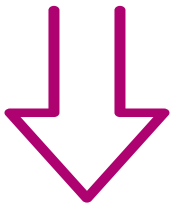
A list of the advantages and disadvantages of the most common commercially available surface treatments:

HASL - tin/lead hot tinning

Typical thickness 1-40µm shelf life 12 months



1. Very good solderability
2. Low cost
3. Permits wide process windows
4. Tried and tested surface treatment
5. Multiple soldering operations



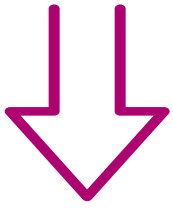
1. Difference in thickness/surface structure between large and small solder pads
2. Not suitable for <508 µm (20 mil, Imperial) pitch, SMD & BGA
3. Risk of bridging between small isolated areas
4. Low surface isolation resistance because of flux residue from hot tinning
5. Not ideal for HDI products

LF HASL - Lead-free hot tinning

Typical thickness 1-40µm shelf life 12 months



1. Very good solderability
2. Low cost
3. Permits a wide process window
4. Tried and tested surface treatment
5. Multiple soldering operations



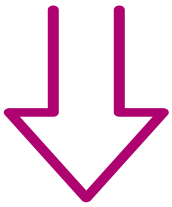
1. Difference in thickness/surface structure between large and small solder pads
2. High process temperature 260-270°C
3. Not suitable for <508 µm (20 mil, Imperial) pitch, SMD & BGA
4. Risk of bridging between small isolated areas
5. Low surface isolation resistance because of flux residue from hot tinning
6. Not ideal for HDI products

ENIG – chemical gold

Typical thickness 3-6 µm Nickel/0.05 - 0.125 Gold Shelf life 12 months



1. Excellent smoothness
2. Suitable for small conductors/BGA/small components
3. Tested process
4. Good solderability
5. Can be bonded (AL wedge)



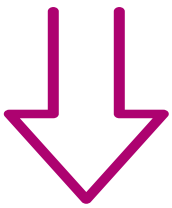
1. Expensive surface treatment
2. ”Black pad” problems (especially for BGA)
3. Aggressive to solder mask – large solder mask bridges are preferable
4. Avoid solder mask defined BGA design
5. Do not plug/cover one side

Chemical tin

Typical thickness ≥1.0µm shelf life: 6 months



1. Chemical surface treatment = excellent smoothness
2. Suitable for finer conductors/BGA/smaller components
3. Average high cost for lead-free surface coating
4. Suitable for Press Fit
5. Good solderability



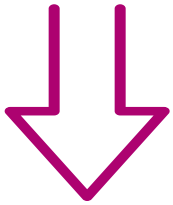
1. Very sensitive to handling – gloves must be used
2. Risk of whiskers – dendrite tin growths
3. Aggressive to solder mask – solder mask bridges must be ≥127 µm (5 mil, imperial)
4. Baking before soldering can have a negative effect on solderability
5. Reduced solderability during multiple soldering operations
6. Pull-off solder mask is not recommended
7. Never plug or cover on one side only

Chemical silver

Typical thickness 0.12 – 0.40 µm. shelf life: 6 months



1. Chemical surface treatment = excellent smoothness
2. Good for small conductors/BGA/small components
3. Average high cost for lead-free surface coating
4. Can be reworked
5. Medium good shelf life if well packaged



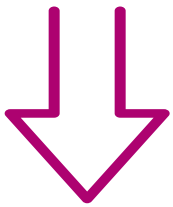
1. Requires special handling/sensitive to scratches – gloves must be used
2. Requires a special packaging routine. If the package is opened and not all boards are used at once, the package must be resealed quickly
3. The time between multiple soldering operations must be kept short
4. Removable solder masks are not recommended
5. Never plug or cover on one side

OSP (Organic Solderability Preservative)

Typical thickness 0.12 – 0.40 µm. shelf life: 6 months



1. Excellent smoothness
2. Good for small conductors/BGA/small components
3. Low cost
4. Can be reworked
5. Clean, environmentally compatible process



1. Very sensitive to handling. Gloves must be used and scratches avoided
2. The time between multiple soldering operations must be kept short
3. Not suitable where multiple soldering operations are required
4. Restricted shelf life. Does not suit certain types of transport or long storage
5. Very difficult to inspect visually
6. Surface treatment may be negatively affected if incorrectly printed solder paste is washed off
7. Baking before soldering can negatively affect solderability

➡ **Read more: "Reliability Assurance of Soldered Printed Board Assemblies Handbook (2024)"**

4.3.5 The cost of printed circuit boards (PCBs)

A number of different factors affect the price of PCBs. This chapter endeavours to summarise factors that have a decisive effect on the final price.

Lead time

One of the primary cost drivers is lead time, or, conversely, the lack of reasonable lead time, which always leads to price increases because the PCB manufacturer will have to accelerate production. At worst, the expected lead time can be so short that only a single PCB manufacturer can meet the delivery, and this will be expensive. The expected delivery time is normally the most critical factor when making prototypes, and because the number of PCBs is often small, the price may not be so important. Try to plan for 7–10 days lead time for simple prototype boards and 12–15 days for more complex prototype boards.

Quality requirements and UL approval

The quality and reliability requirements for a PCB are directly related to its cost. For example, if there is a requirement for production and qualification in accordance with IPC class 3/3A and/or MIL-31032, an ADCAP & ESA, the price may increase considerably because it drastically reduces the number of approved factories that can manufacture the PCB and do the increased number of required tests.

UL requirements may affect cost, mainly because the design ends up outside the framework of the factory's approval. This especially applies to flex-rigid boards, hybrid boards with mixed materials and, to some extent, boards made from RF material.

It is advisable to find out the demands that must be applied to the PCB to comply with the requirements for the final product. For example, find out what final product category applies to UL approval.

Many times, it might be sufficient for the raw material to comply with a flammability classification, such as V1 or V0.

Communication between product owner, developer and manufacturer is critical here.

Choice of material and technology level

Choice of material and technology level affect the cost of the PCB during the entire service life of the product. Therefore, it is important to make correct decisions at the beginning of the development project.

The choice of material does not affect cost very much (see table below) as long as it is standard material. However, the choice of special material may affect the price by several hundred percent. A close dialogue with the PCB manufacturer is important because there are often equivalent materials at lower cost.

Other choices that have to be made and that affect costs are:

- Thickness
- Number of layers
- Copper thickness
- Trace width
- Single sided or double-sided assembly
- Number of the via holes between layers and whether these are hidden or not
- Hole size and number of different hole sizes
- Impedance requirements
- Isolation requirements
- Surface treatment

The table below indicates how much the different factors will affect the cost of the PCB. The table is based on an order quantity equivalent to 10 m2 and a nominal lead time of 3–5 weeks.

This table should only be seen as an indication because every PCB must be analysed to calculate a correct price. Parameters will also change over time when processes and technology evolve.

Number of layers	2	4	6	8	10	12	14	16
Cost index	= 1.00	* 1.0	* 2.0	* 2.5	* 3.0	* 4.0	* 5.0	* 6.0
Material FR-4.0 Tg130	0%	0%	0%	0%	0%	0%	0%	0%
Material FR-4.0 Tg150	5%	5%	3%	0%	0%	0%	0%	0%
Material FR-4.0 Tg170	20%	15%	12%	10%	6%	6%	4%	3%
ENIG	25%	18%	13%	9%	7%	6%	5%	4%
Chemical silver	14%	10%	7%	5%	4%	3%	3%	2%
Chemical tin	14%	10%	7%	5%	4%	3%	3%	2%
Lead HASL	0%	0%	0%	0%	0%	0%	0%	0%
Lead-free HASL	0%	0%	0%	0%	0%	0%	0%	0%
OSP	-4%	-3%	0%	0%	0%	0%	0%	0%
All layers with 70µm base foil	20%	29%	30%	29%	29%	29%	28%	27%
All layers with 105µm base foil	40%	57%	60%	59%	58%	59%	55%	54%
Solder mask plugging	4%	3%	0%	0%	0%	0%	0%	0%
Plugging in accordance with IPC 4761 VI (epoxy)	19%	13%	9%	7%	5%	5%	4%	3%
Plugging in accordance with IPC 4761 VII (epoxy)	56%	40%	28%	21%	16%	14%	11%	9%
Board thickness 1.0 mm	-6%	-6%	-6%	-5%	-4%	N/A	N/A	N/A
Board thickness 2.0 mm	13%	9%	6%	5%	0%	0%	0%	0%
Board thickness 2.4 mm	25%	18%	13%	-9%	0%	0%	0%	0%
Removable solder mask	6%	4%	3%	2%	2%	2%	1%	1%
Solder mask colours other than green	6%	4%	3%	2%	2%	2%	1%	1%

TABLE OF PRICE-AFFECTING FACTORS

The cost index is based on the following parameters:

1.	Board thickness	1.60 mm
2.	Copper thickness	18µm or 35µm base foil
3.	Solder mask colour	Green
4.	Minimum conduct width	0.125 mm
5.	Minimum isolation distance	0.125 mm
6.	Minimum hole size	0.300 mm
7.	Surface treatment	HASL
8.	Quality in accordance with	IPC 6012 class 2

Price calculation example

The basis for this example is a PCB that has two layers and complies with the eight basic parameters above. It does not contain any of the cost-driving factors in the table above. To begin, we assume that this PCB will cost SEK 10/board.

The new printed circuit board will have six layers, it will be 2.00 mm thick, it will be produced in a medium Tg material, and will have ENIG surface treatment.

This leads to a cost increase as in the table above:

2 layer > 6 layers	=2.0X
HASL > ENIG	=13%
Standard Tg > Medium Tg	=3%
1.60 mm > 2.00 mm thickness	=6%

The increase in the number of layers will increase the base price from SEK 10 to SEK 20. Other supplements total 22% (13 + 3 + 6), which leads to total price in this example of about SEK 24.40. However, it is important to remember that more layers can provide an improvement for other PCB parameters, as well as significantly facilitating functional characteristics and requirements, especially to fulfil EMC requirements.

Chap. "4.1.1 Design for Compliance"

4.4 Development tools

Today's CAD (computer aided design), and CAM (computer aided manufacturing) tools are very powerful and make work much easier for the developer and manufacturer. CAD is used by the developer to drawing schematics and PCB layout during design work. CAM is used by PCB manufacturers to make PCBs. The interface between CAD and CAM tools is often Gerber files or ODB++ (which contain more information).

There is a wide range of CAD programs, ranging from freeware to highly advanced proprietary packages. Which software to choose depends on the technical level of the electronic design and budget.

Freeware software is a good option for students, hobbyists or simple designs, especially in the initial stages. "You get what you pay for" is highly relevant to CAD tools. As freeware is available in abundance, test what is available to see which type best suits requirements.

Considerations when choosing CAD tools:

- How complex is the design? (Number of layers, high speeds etc).
- Can it specify all the requirements?
- Should other people be able to work with the same documentation?
- Are import and export formats 100% compatible with other software? Does it support the most modern file formats?
- What support functions are available? For example, simulation, error correction and other warnings.
- How complete does it make the production documentation?
- How accessible will the tool be over time? Will it still be available if documentation needs to be referred to some years later to make changes and updates?
- How are program updates organised? For example, configuration to Windows updates.
- Is help available? User forums are often available, but is highly qualified support available when needed?
- Does the design of various functions feel logical and easy to use?
- Are component libraries with footprints for schematics and layouts available?

Unfortunately, problems often occur late in the project that were caused by choosing too simple a CAD tool.

An alternative to a freeware program is to use a test version of a professional tool. These are free of charge but have limited functionality. They can, however, be excellent for smaller or simpler designs. If the project expands later on, simply purchase a license, to access the enhanced functionality without having to change the tool.

A further alternative to purchasing a CAD tool is to hire a consultant developer with expert competence in PCB design, and who has access to a professional tool and component library.

What is it that costs in a CAD tool?

- The ability to handle several layers
- The ability to handle large or complex PCBs
- The ability to purchase component libraries
- Reference designs are often made in the major tools
- Built-in simulation and other analysis functions (although there are separate simulation programs)
- Compatibility with other tools such as PDM systems and revision management systems



5. Reliability

The knowledge of ensuring the reliability of soldered printed board assemblies has become an increasingly difficult, but underestimated, area. With electronics in almost every application, many of them with high safety requirements, this skill needs greater attention. The Smarter Electronic systems program has funded research projects in which Per-Erik Tegehall, a reliability expert and researcher at RISE, has done extensive research work in collaboration with several different companies. The results of these and other projects are presented in the handbook: Reliability Assurance of Soldered Printed Board Assemblies. The introduction follows below.

At the same time as there are demands/expectations from customers that the reliability of electronic products should be constantly improved, many ongoing changes entail an increased risk of reduced reliability.

Examples of the most important changes are:

- New technologies are implemented more and more rapidly in new products. It may be to improve functionality, lower manufacturing costs or due to legal requirements.
- New technologies are developed primarily for consumer products with a short life expectancy and a mild operating environment.
- Product cycles are getting shorter and shorter, which places demands on an increasingly shorter ”time-to-market”, but it may also cause components to become obsolete after a short time on the market.
- More and more products are portable and/or used in harsher operating environments.
- More parties are involved in the product development chain due to extensive outsourcing.
- Pressure to minimise costs is strong at all stages of the product development chain.

If electronics are manufactured with high reliability requirements and a long-life expectancy, or are used in harsh environments, users need to be aware how these changes affect reliability in order not to decrease reliability or delay product launch.

The traditional way of working to ensure the quality of electronic hardware is to design, manufacture and test according to ”how-to” standards and guidelines, but the rapid development of new technologies has made it impossible to update documentation at the same pace. Another problem is that many standards are based on ”best manufacturing practice”. That is, they are based on the experience of mature technologies. Therefore, requirements in standards and guidelines not be sufficient, or even relevant, when implementing new technologies.

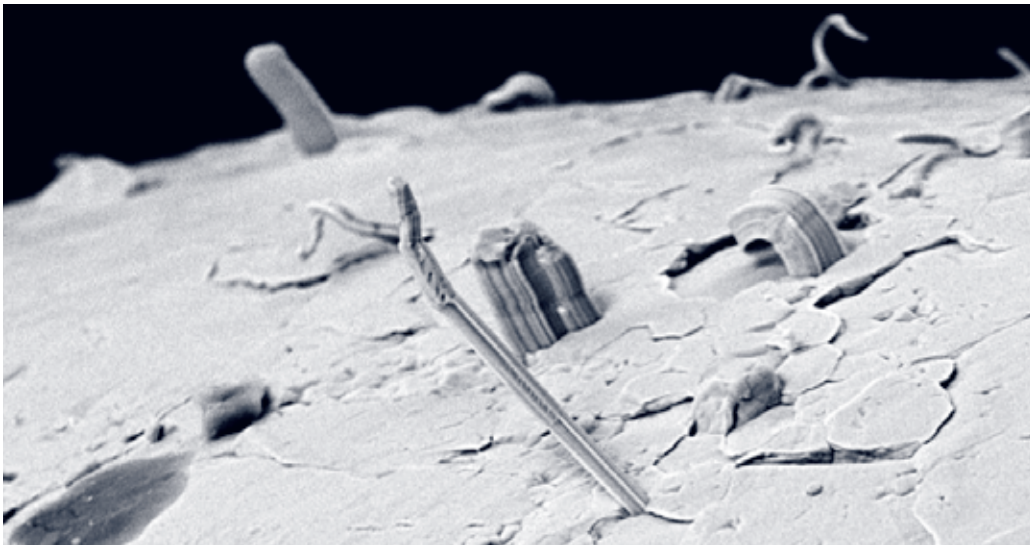
Consequently, it is important to set relevant reliability requirements when procuring PCBs and printed board assemblies PCBAs to ensure that the products are reliable during the expected lifetime. This applies especially to products with high reliability requirements, a long-expected life and/or a harsh operating environment.

Hence, due to the rapid implementation of new technologies, ensuring reliability requires a different way of working than the traditional standard-based approach. It requires knowledge of the failure mechanisms that may affect reliability and how reliability may be assessed and verified. Therefore, it is usually referred to as ”physics-of-failure based”, ”performance based” or ”knowledge based”.

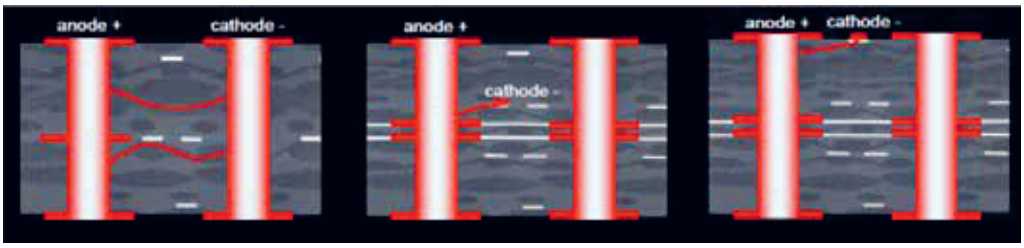
Standards for reliability programmes based on this approach have been developed by IEEE, SAE and GEIA. Although meeting requirements in traditional standards may not be sufficient to ensure that a product is reliable, it is a good starting point for ensuring reliability. Failure to meet these requirements carries a high risk of poor reliability.

In conclusion, ensuring reliability of soldered assemblies is a huge challenge, especially if new technologies are used to produce the assemblies and/or if they are to be used under harsh conditions. Ensuring reliability according to the ”new” reliability standards will require deep knowledge of the failure mechanisms that may affect the reliability of electronic hardware and how reliability can be assessed and verified.

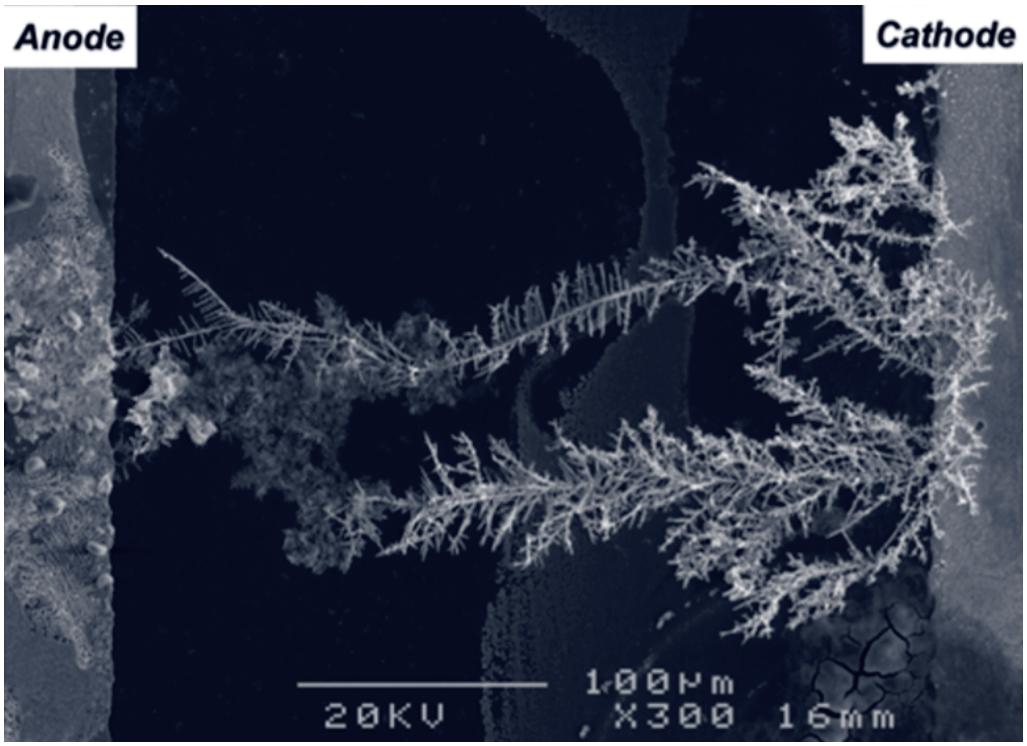
➡ **Read more: ”Reliability assurance of soldered printed board assemblies” (Handbook published 2024)**



EXAMPLES OF WHISKERS SHOWN IN HIGH REINFORCEMENT THROUGH A SWEEP ELECTRONIC MICROSCOPE.



PRINCIPLE SKETCH SHOWING WHERE CAF CAN OCCUR.



EXAMPLE OF CAF SEEN IN STRONG MAGNIFICATION THROUGH A SWEEP ELECTRON MICROSCOPE.

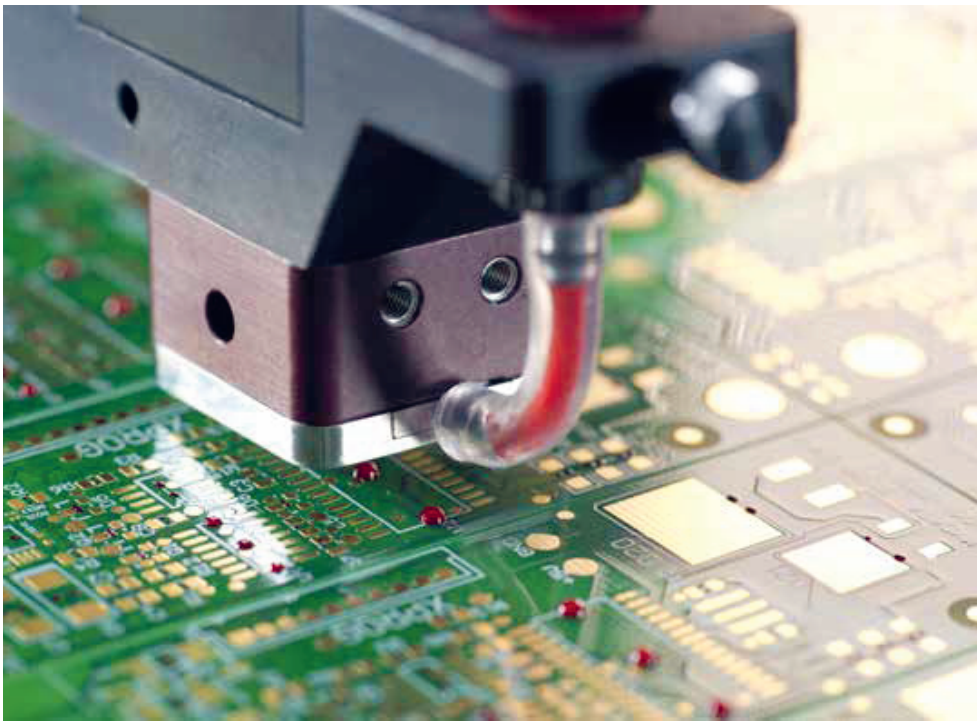
6. Before manufacturing

This manufacturing chapter is the core of the handbook and starts with a brief description of the way a PCBA is produced. More detailed information is available, but this falls outside the framework of the handbook. The main part of this chapter consists of descriptions of required production documentation, examples of which can be found among the templates.

Anyone considering ordering PCBA production who has never visited a PCBA factory, should do so. It is informative to go through the product and documentation with the manufacturer on site.

The period when the product or prototype is manufactured can sometimes be regarded by the product owner and developer as mere waiting time, but it is advisable to contact the manufacturer immediately and then check whether any items are unclear.

The manufacturer should have access to contact persons if questions or problems occur, especially longer holidays, like summer.
The result will be best if there is an ongoing dialogue, in particular if it is the first prototype to be manufactured.

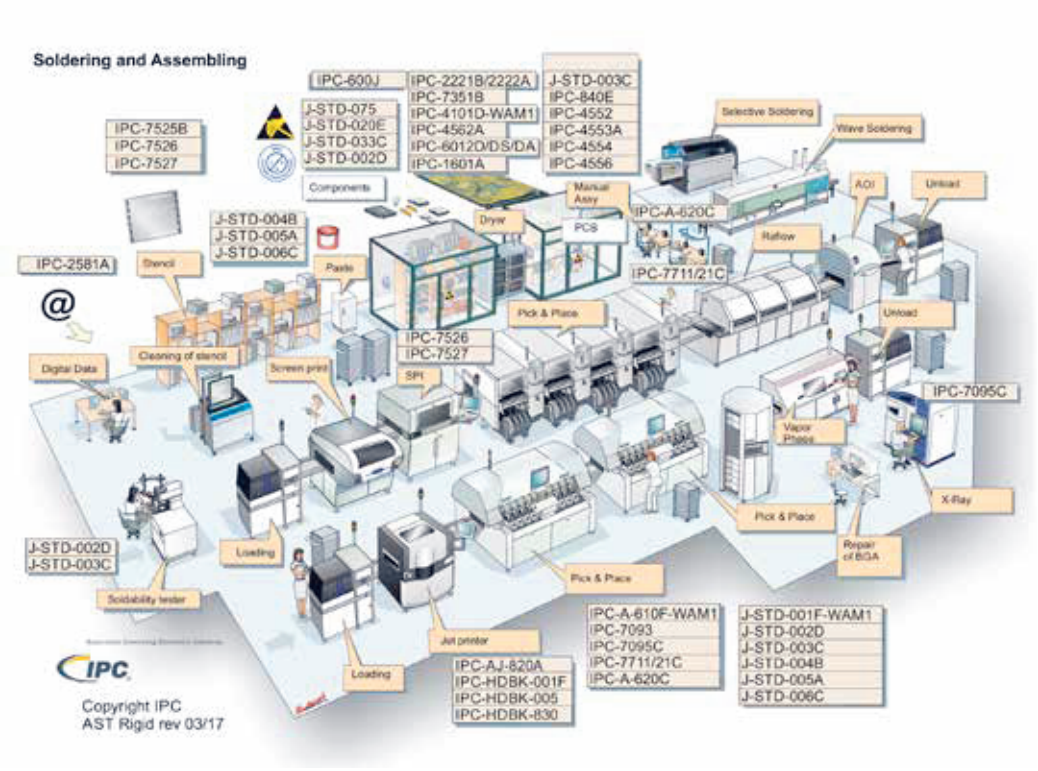


6.1 Electronic manufacturing described

Electronic manufacturing today is a modern process industry, in a pleasant working environment where the process is quiet and requires a controlled climate. It would be an excessively comprehensive task to describe in detail all the tasks and activities included in electronic manufacturing. A visit to the manufacturer selected for the project project is recommended to gain the best possible understanding of how manufacturing is done.

The fact that Sweden has considerable knowledge of electronic manufacturing is underlined, not least by the fact that the country has one of the world's leading manufacturers of production lines for PCBA manufacturing.

The core business of the manufacturers is manufacturing PCBAs. But this is often supplemented by services such as test development, testing, assembling circuit boards into systems, enclosure, repairs, claims management, purchasing, storage and distribution (logistics) and many more services that a product owner can use, depending on their needs. Since electronics is found in more and more equipment and devices, the manufacturers have widened their service offering and often make completely finished products for their customers. This means that in addition to electronics, they handle plastic and metal parts, cable assemblies, meters, buttons, displays, marking, packaging etc.



THE ILLUSTRATION SHOWS A SCHEMATIC MANUFACTURING INSTALLATION, GIVING A GOOD ILLUSTRATION OF ALL THE STEPS THAT ARE REQUIRED BEFORE A CIRCUIT BOARD IS FINISHED. THE ILLUSTRATION ALSO SHOWS THE RELEVANT IPC STANDARDS THAT AFFECT EVERY STAGE.

6.2 Description of different tests

The product owner should have a comprehensively developed and agreed test strategy for the company's PCBAs. In other words, the developer/product owner must make a decision early on about how each fully developed circuit board will be tested in future serial production.

Test strategy. It is difficult to develop a test for a PCBA that is not fully developed, but the test strategy is an important aid in the endeavour to develop a production test in good time. It is not unusual to take more or less the same time to get a reliable test in place as it takes to make the first prototype of a PCBA, so please keep this in mind. The test strategy should describe (simple) questions such as:

- How are simple boards tested?
- How are boards with digital/analogue functions tested?
- How are boards with a processor/microcontroller tested?
- How are boards being power-supply tested?
- How are boards tested where there are high voltages/high currents?
- How should test data be stored?
- How should access to test data be handled?
- What standards are applicable to the test system (dangerous voltages, pinch risk etc)?

Well formulated and realistic requirements for tests are decisive for cost efficiency. Ensure that the requirements can be implemented in practice by collaborating with the contract manufacturer in drafting and inspecting the test requirements. It is important to determine a balance between planned batch size and testing cost at an early stage. It is advisable to ask the manufacturer's test personnel to offer views on the ease of testing a design early in the design stage.

The manufacturer can then collaborate and influence the quality and producibility of the project by selecting the correct subcomponents and correct test method to ensure that the components are mounted in a safe and efficient manner.

In addition, a production test should not just be assessed and qualified against a specification. The test system should be run on one or several production batches, to set the correct test limits for tested boards.

After some time in production, analyse where (in that test) the faults in a specific PCBA are discovered. Not all tests may be needed. However, it may be necessary to increase testing to further fine-tune production and optimise production results.

Frequently used tests:

AOI (Automated Optical Inspection)

- **What?** A camera with vision software analyses finished PCBAs and assesses whether assembly has been done correctly in accordance with the PCBA documentation. AOI sees whether components are mounted or not, whether they are correctly mounted and correctly aligned when polarity is important. Solder joints can also be assessed if visible. For some types of components, it is not possible for AOI to determine whether the correct component is mounted or if it is incorrectly aligned, e.g. if the component lacks text or other identification.
- **When?** Normally done when PCBA manufacturing has stabilised, it is an excellent aid in finding faults early in the production line. Can also be set up and used on early prototypes. AOI is not normally something that is demanded by the developer/product owner, but is a useful aid for a manufacturer in controlling production processes, and helps make a cheaper product in collaboration with the product owner.

💡 If components are too close to each other on the PCBA, AOI can find it difficult to discover faults. The camera requires a certain angle to do visual inspection correctly. A conservative angle, which most cameras can manage, is 30°.

💡 If it does not affect the design, select a component that is easier to inspect visually. For example, it is easier to visually inspect an operation amplifier in a large SO package than in a QFN (Quad Flatpack No Leads) or in the CSP (Chip Scale Package).

Boundary scan

- **What?** A mainly digital test method sending signals and stimuli to circuits under test, in a similar manner to ICT (described later in this chapter). The PCBA typically has one or several circuits equipped with a so-called JTAG port, which can communicate via a special protocol.
- **When?** Boundary scan is an excellent aid in testing advanced digital circuits that are otherwise difficult to test, such as BGA circuits. Test vectors can be transmitted to the circuits on the PCBA via the JTAG port, and circuit functions and their connections to the PCBA can be tested via a clever test pattern. Even if the circuits cannot run at full speed under test, the test frequencies are much higher than are possible with ICT for example. Some analogue testing can also be done if the PCBA or test equipment has a D/A converter that can be controlled via circuits with a JTAG port.

💡 Connecting the JTAG port to all circuits on the PCBA that are equipped with this port makes testing much easier.

Burn-in test

- **What?** A test that is done on a finished product under load. This includes tests at raised temperature, accelerated short duration tests, or tests during a longer period. The goal is to ensure that each of the products will function in its application.
- **When?** In stable serial production, for products that will be heavily loaded in operation and where faults can cause severe problems.

E-test (Electrical test)

- **What?** PCB E-test examines the electrical conductivity of the PCB.
- **When?** An "electrical test" is normally done on the PCB in the PCB factory. This test is an excellent way for the PCB manufacturer to discover whether the PCB, as produced, complies with the PCB documentation. There are typically two levels for this test. Either one PCB is found to be "similar" to all the others and the board passes the test, or the board is tested more reliably in accordance with its correct netlist. The latter requires, of course, that the machine is programmed with the netlist of the board.

FCT (Functional Circuit Test)

- **What?** Function testing is a good aid for testing a final product, or several PCBAs that are individually ICT tested and then assembled. It is often advisable to test the product as soon as value-adding measures have been taken. FCT can have multiple meanings.
  An FCT normally consists of a function test similar to what the product will be subjected to in its application. FCT equipment does a number of tests that the developer/product owner stipulates in a specification.
  Please refer to:

  Test Data Specification

- **When?** Mainly in serial production, but also during the start-up phase of new products. FCT is normally a requirement from the developer/product owner, but is also a useful aid for a manufacturer to ensure that the products will function in its application. Parties should enter into dialogue early on related to FCT, then follow up test coverage and fault discovery, and consider how they can be maximised.

One reason for specifying a function test is when the PCBA handles high current in its application. It is possible to test the continuity of conductors all the way to connectors on a board in an ICT, but there is no good way to test high current through solder joints and connectors.

Another occasion when a function test is needed is when standards require it, for example regarding electrical safety. Another occasion is products that handle dangerous voltages. These must normally be isolation tested and earth connections must be tested for continuity (ECT, Earth Continuity Test) to allow them to be marketed.

Modules that are also used as spare parts require the same tests as when the product is tested together with other components of the entire system. For this reason, spare parts must also be function tested. For example: A final test of a machine is done in production for isolation and GTC. In this case, the modules that are supplied individually as spare parts must be tested in the same manner.

FPT (Flying Probe Test)

- **What?** Test probes connect to solder pads or test points on the PCB or on the PCBA. This test can determine whether the PCB or PCBA is correct according to the PCB documentation or the PCBA documentation, compared with ICT. Partly assembled separate circuit boards can also be tested if there are accessible areas for the probes.
- **When?** It is a cost-effective option for prototypes and low-to-medium volume production, even when testing fully assembled PCBAs. It takes longer to test with FPT compared with ICT, but FPT does not require a bed-of-nails.



Flying probe test equipment is available with test probes that can test areas as small as 50 µm x 50 µm.

ICT/ICA (In-Circuit Test/In-Circuit Analysis)

- **What?** The fully assembled PCBA or a complete panel is attached to a fixture, a so-called 'bed-of-nails' in test equipment. This test equipment gives multiple opportunities for testing single components and the function of all or parts of the circuit on the PCBA. The test equipment connects various drivers and sensors via nails (pins) on the bed-of-nails, so that the results can be measured by the test software in the test equipment. Good opportunities can arise to do various types of advanced tests, depending on how advanced the test equipment is. Please remember, however, that advanced tests take time, which raises production costs. To achieve highest possible testability, adapt the design to suit the ICT test by ensuring there is space to site the relevant test points.

  The test fixture, the bed-of-nails, is specific to every PCBA. If the PCB has to be modified at a later date, this may mean that the fixture also has to be modified. The larger the change, the more it costs. Ensure there are guide holes in the PCBA that fit the fixture to make testing faster and more reliable.

  The location of the test points requires planning. Do not place the test points too close to each other, because there is a minimum dimension that manufacturers' ICT equipment can handle. Moreover, it would be more difficult to supplement the fixture if new board revisions require it. However, the test points should be as close to the circuit under test as possible to prevent conductor resistance or capacitance from influencing the measurements. For this reason, several measurement points may be required for the same conductor.

  Test point design can be decisive. A square test point may be preferable to a round one, as the entire area is larger and the test point is easier for the pin to find. Please consult the manufacturer about the size the test points need to be. A conservative dimension that works for most ICT equipment is 1mm in diameter or square. Also remember to spread the test points as evenly as possible over the PCBA to prevent the pins from applying uneven pressure to the PCBA, which can cause cracking of the PCB or the components. The test points must also be free from solder mask.
- **When?** As this test method requires that tests to be developed for each specific PCBA and prepare a fixture (which is relatively expensive), Developers/product owners often prefer to delay introducing ICT until the board is stable and fully developed. The use of ICT is specified by the developer/product owner, but it is also a useful aid for a manufacturer in controlling their production processes once in place.

  By collaborating with the product owner, it is easier to obtain a product with fewer faults and at lower cost. Ensure communication among those involved is good in relation to ICT is good, to follow up test coverage and fault discovery, and consider how to optimise.

Each manufacturer has their own specific ICT equipment. It is seldom possible to move production without making a new (expensive) fixture that suits the new manufacturers' ICT equipment.

If the PCB has to be modified after the bed-of-nails has been made, try to make sure that old test points are the fixture or make a new one.

If the ICT has full access to all the pins on a processor or other programmable logic, it can control the circuit and verify its function. If necessary, the ICT can shut down the clocks and oscillators on a PCBA and generate its own clock during ongoing testing. To facilitate this and increase test coverage, the control signals on the circuits must be provided with pull-up or pull-down resistors to the supply voltage or to earth, respectively.

Manual inspection (sometimes called HOI, Human Optical Inspection, compared with AOI)

- **What?** An operator checks and visually inspects the components and solder joints manually.
- **When?** Primarily, small batches, when programmable AOI is not financially justified. There is a risk of missing faults, however, as the operators' task easily becomes monotonous and the risk of human mistakes increases.

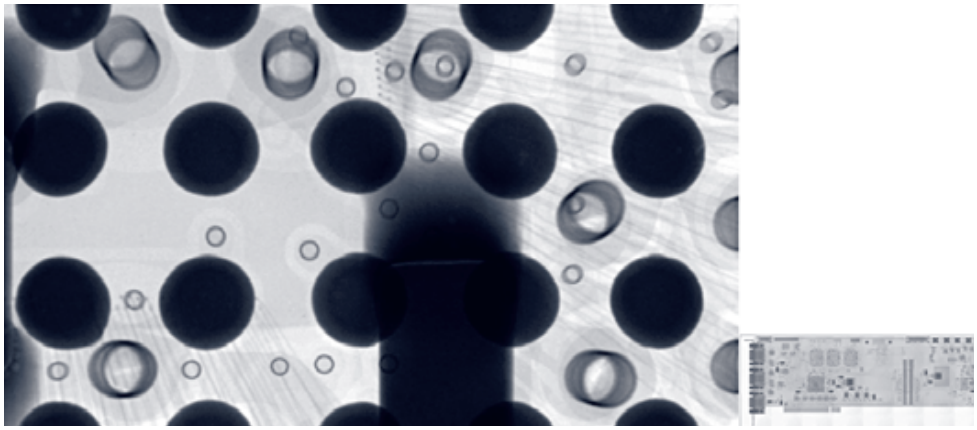
ROSE/SEC (Resistivity of Solvent Extract/ Solvent Extract Conductivity)

- **What?** An analysis method for analysing the quantity of ionic contamination on a PCB and PCBA. Several IPC standards such as IPC-6012 for rigid PCBs and J-STD-001 for soldered PCBAs contain requirements for cleanliness, measured by this method.
- **When?** Even if IPC standards have requirements for cleanliness, this is not a relevant method for assessing whether boards are sufficiently clean to be reliable. This is a useful method, however, for verifying that cleanliness does not change over time in ongoing production.

Automated X-ray Inspection /AXI

- **What?** The fully assembled PCBA is analysed by x-ray equipment. X-ray inspection helps uncover hidden faults in solder joints underneath a BGA package etc. AXI is a more advanced version of x-ray equipment.
- **When?** In serial production, or for fault finding on individual PCBAs. Soldering of a BGA packaging can only be inspected by X-ray. The layers on the PCB and bond wires on IC circuits can also be visually inspected if the resolution of the equipment permits it. Fake components (correct package but wrong or no silicon chip) can be discovered. AXI is not normally something that is demanded by the developer/ product owner, but is a useful aid for a manufacturer in controlling production processes and assists in making a cheaper product in collaboration with the product owner.

Each manufacturer has specific X-ray/ AXI equipment. The size of details that can be discovered depends on the resolution of the equipment.



RÖNTGENAVSYNING/AXI (AUTOMATED X-RAY INSPECTION)

SPI (Solder Paste Inspection)

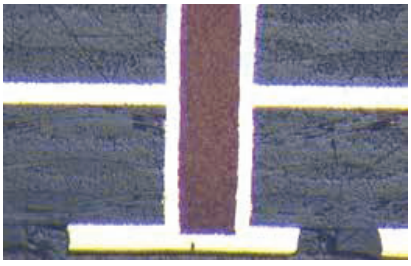
- **What?** A camera with vision software analyses the applied/jetted (dispensed) solder paste on unmounted PCBAs and assesses whether the applied solder paste coincides with the PCBA documentation.
- **When?** This is usually done when production of a PCBA has been stabilised. It is an excellent aid for finding faults early in the production line. This can also be set up with early prototypes. SPI is not normally demanded by the developer/product owner but is a useful aid for a manufacturer in controlling production processes and helps make the product cheaper in collaboration with the product owner.

Cross-section analysis

- **What?** This is a destructive test. A PCB manufacturer often has equipment for analysing a cross-section of a PCBA when requested by a developer. A selected portion of a PCBA is potted in plastic and the desired cross-section is then cut, polished and analysed in a measurement microscope. The geometries, vias and conductors on the board can then be analysed in this manner.
- **When?** Some applications (such as medical) require that this test should be done on each batch of PCBs or that samples are saved for later analysis as required.

Another occasion for using a cross-section cut is to investigate faults that have occurred in a PCBA that are not visible on its surface. This analysis is mainly done to ensure that the geometry of the PCB coincides with the PCB documentation. You can then get answers to the following questions:

- Is the impedance specified for conductors correctly implemented?
- Are the layers connected as intended?
- Do the vias have sufficiently thick walls?



Test coverage table

Fault type	Fault cause, based on IPC	X-ray	B-scan	ICT	Flying Probe	AOI	ATE
Component fault	Defective Component (electrical fault)		X	X	X		X
	Wrong value component			X	X		X
	Component leads under tension						
	Component not worked on						X
	Component damaged	X	X	X	X	X	X
	Component not programmed		X				X
	Component lead bent		X	X	X	X	X
	Component lead, wrong bending radius						
	Component leads not in line (coplanarity)					X	
	Component leads, incorrect lead shape						
	Component leads/cables, incorrect length						
	Component leads bent under body	X		X	X		X
	Component lead is not tinned						
	Solder, poor solderability					X	
	Board, foil fault			X	X		X
	Board, base material damaged						
	Board, blisters, reasoning, wrinkling						
	Board bowed or skewed						
	Connector damaged	X	X	X	X	X	X
	Gold not removed	X					
	Incorrect strain relief						
	Pin, flange faulty						
	Incorrect isolation distance						
	Isolation or conductor damaged						
	Marking incorrect						
	Cable incorrectly manufactured						X
	Wire damaged						
	Wire not tinned(when required)						
	Wire unravelled						
	Software fault, unprogrammed component		X				X
Monting Fault	Solder, solder balls, solder splash	X					
	Lacquer missing						
	Lacquer wrinkled						
	Lacquer where they should not be lacquer			X	X		X
	Glue where there should not be glue			X	X		X
	Glue, not enough glue						
	Glue, too much glue						
	Potting compound unhardened						
	Potting compound missing						
	Potting compound where there should not be any potting compound						X
	Potting compound wrinkled						
	Board not clean						
Placing fault	Component missing	X	X	X	X	X	X
	Component incorrectly rotated	X	X	X	X	X	X
	Component Billboard	X				X	
	Component Tombstone	X		X	X	X	X
	Component incorrect height					X	
	Component incorrectly placed, side, end, toe overhang etc					X	
	Component incorrectly assembled (incorrect value)		X	X	X	X	X
	Component/components/hardware loose/missing/faulty						
	Component upside down					X	
	Crimping fault						
	Incorrect mount						
	Minimum isolation distance encroached						
	Assembly height exceeded						
	Cable incorrectly connected						X
	Wire incorrectly connected						X
Connection fault	Component, lacquer meniscus in hole 7.3.5.8						
	Component, incorrect leg protrusion						
	Solder pores, outgassing, voids	X					
	Solder, solder bridge	X	X	X	X	X	X
	Solder, unacceptable wetting					X	
	Solder, un-wetted connection	X	X	X	X	X	X
	Solder, no clear wetting					X	
	Solder, solder tips 5.2.10	X					
	Solder, solder disturbed						
	Solder, not enough solder					X	
	Solder, solder joint cracked 5.2.9	X					
	Solder, cold soldering			X	X	X	X

6.3 Documentation for electronic production

The documents described in this chapter are those required to achieve the most problem-free process, leading up to, and during, production. Some of the documents are described in greater detail.

Important documents for ordering:

- **Purchase orders** for electronics production are often generated in the client's business system and are handled by the purchasing organisation. Information such as part number, revision number, quantity, price, reference number, requested delivery dates etc are normally included in the document. An order for electronic production should also contain general requirements for manufacturers such as IPC class, RoHS declaration and, where relevant, specific process requirements. If these specific manufacturing requirements are not found in the business system, ensure that they are included in a separate document. If the order refers to prototypes, the order may be simpler and less detailed.
- It is important to include **a document overview and revision control** from the beginning. Using outdated and/or incorrect documentation may have serious consequences.

  Chap. "6.3.1. Document overview and revision management"

- **PCB documentation**, including documentation for paste masks in the form of Gerber or ODB++ files together with the BOM, are the most important foundation for the final PCBA. If there is a fault here, it is seldom possible to redress the situation. Documents need to include correct requirements for the intended manufacturing process, with reference points for machinery and fixtures included.

  Chap. "6.3.4 Ordering of Printed Circuit Boards/PCB"

- **PCB drawing/PCB specification**. In addition to the Gerber or ODB++ files, a specification of the PCB, including layer structure and material requirements, is required. These may either be extracted from the CAD program in the form of a drawing or supplemented by a specification. The PCB can also be described in a text document.
- **The BOM** should clearly specify the exact position of each component. If it is permissible to choose freely within the framework of given data, it is beneficial to specify general requirements at the beginning of the BOM (or in the product specification). These include voltages, power, and dielectric for component groups such as resistors and capacitors.

  Chap. "6.3.3 Bill of Material/BOM"

- **The text** on a component placement drawing must be readable. It should clearly indicate which component the text in the drawing refers to, and how the component should be polarised.
- **Coordinate data** (Pick and Place files) for components for the pick and place machines are needed for preparing the board. The manufacturer can normally handle most formats and data. Origin points often consist of a sight mark that is also related to the centre of the component body for each component.

Documents if required:

- **The product description** is a general description of what the product should do:

  Chap. "6.3.2 Product description"

- **For custom components** (e.g. inductive components, EMC shields, mechanical components), the specifications/drawings for these should be included.
- **Panel drawings** show how the circuit board should be located on the panel. These are normally prepared by the manufacturer.

  Chap. "6.3.5 Panel Drawing"

  Requirements that refer to the panel, such as areas where break points must not be located, should be included in the manufacturing documentation.

- **The test specification** describes the tests to be done. A good test plan cannot be created in production. It is developed during product development. The manufacturer cannot guarantee function without testing.

  Chap. "4.1.5 Design for Test (DfT)"
  Chap. "6.2 Description of different tests"

- **The test instruction** is generally related to test equipment and describes how the testing procedure should be carried out.
- **Schematics** are design documents that are used mainly by the manufacturer if testing is included. Schematics can also facilitate communication with the client or help identify problems when faults occur.

  Chap. "4.2 Schematics"

- **The assembly instruction** is the client's means of transmitting instructions to the manufacturer. This can be used to describe the assembly sequence, mechanical installation, assembly torques for screws, etc.
- **Conformal coating** instruction describes the areas to be coated with protective film, cleaning requirements, the components that need to be masked, etc.
- **The potting instruction** describes how the PCBA should be potted, including degree of filling, requirements for masking, etc.

6.3.1 Documentation overview and revision management

Lack of order in documents and their revision level have caused enormous costs and delays to many products over the years.

An overview of all the documents related to the product and their latest revision status is essential to reduce the risk that an incorrect document might be used as input material. A wide range of systems are available for handling the revision status of documents. The most important aspect is not which is selected or whether it is home-grown, but that a system is in place for revision management – and that it is used.

Communicate all changes and document updates in writing. When changes and updates are made, use email as it automatically provides a time stamp from when the message was sent. Referring to "the latest document" is not a reliable method.

Change management during ongoing serial production is found in greater detail in a separate chapter.

Chap. "7.4 Change management"

A simple form of revision management is that the file name contains a date, for example BoardXXXBOM_YYMMDD. Remember that the revision level must also be visible inside the file.

6.3.2 Product description

The product description should include the PCBA to be manufactured in a context in which the entire the value chain can contribute with their knowledge and experience, and can offer suggestions for alternative solutions and improvements. Even if the requirement specification is the document that formally determines the properties of the product, the product owner should supplement this with a product description that guides the developer and manufacturer in the right direction.

A good product description will also help the developer to find imperfections in the requirement specification that could otherwise be expensive. If the product owner in the example below had forgotten to include US radio certification in the requirement specification for example, the developer would be able to discover this as the product description states that the product will be sold in North America.

Example Product Description

The product is a navigation computer for marine use with a touch screen, built-in GPS receiver and Wi-Fi.

Environment: The computer can either be located in the cockpit or under deck. It can be used in strong sunlight, but also for night-time navigation in tropical, temperate and Arctic climates.

Users: Our customer surveys indicate that the people who buy our products are boat owners with good computer skills, are quality-aware, and do not accept problems with technology.

Market: The forecast is an annual volume of 10,000 units per year for two years. Manufacturing costs at that volume must not exceed SEK 1200. The product will be sold in Europe and in North America.

Lifecycle: The product will be sold for about four years without redesign. The warranty time is 2 years. Units will be repaired and supported for at least 5 years.

6.3.3 Bill Of Material/BOM

The BOM (Bill of Material) saves problems in the future if time is allocated to it and it is updated.

The more information included in the BOM, the more precise will the quotation from the manufacturer be when a request for quotation is issued.

The team also saves time because of all the questions the manufacturer does not have to ask. The probability that a PCBA will function the way as conceived will increase if the correct components with the correct values are mounted in the correct places. Where the make of a component is critical for the function of the circuit, it is important that the make is clearly marked in the BOM so that the manufacturer knows that it may not be replaced.

Two templates can be downloaded that are created in a slightly different manner. Please use the one that suits best. Both templates should be seen as a good starting point. If more columns are required, simply add them.

BOM_A, BOM_B

The BOM is often designed so that all the components with the same specification are assembled on one line (the so-called total BOM) that contains information about position(s), component manufacturer's name and part number, manufacturer's part number, electrical performance, RoHS spec, short description and preferably second source if there is one. At least as important as second source information is clearly noting whether a component must not be changed because its properties are unique for the function.

Another way of drafting the BOM is that one component is listed per line, irrespective of the number of times this component occurs.

It is easier for the manufacturer to sort out specific attributes by using this type of BOM. For example, it is easier to sort out all the components that will be mounted on the primary side. If possible, talk to the manufacturer about the sort of BOM that they prefer.

AML and AVL (Approved manufacturer List and Approved Vendor List). Different product owners have different structures for BOM and AML. Some have BOM and AML information in the same document, whereas others prefer to separate them.

Revision management is very vital. Simply referring to "the latest document" can have catastrophic results. All changes must be made in writing. Verbal agreements must be confirmed in writing to reduce the risk of misunderstandings or that something is forgotten. Make it a habit to always update relevant documents and check revision management to ensure that it is up to date.

File format. The BOM is both a working document and an original document, so it can be sent in PDF and Excel format. This gives the manufacturer a document that can easily be processed to work with this as efficiently as possible, at the same time as the original PDF is there for reference. A comma-separated ASCII format document will work instead of Excel for most manufacturers.

Send the BOM in both PDF and Excel formats to the manufacturer.

Ensure that the information for every component states whether it is surface or hole mounted and whether it is mounted on the primary or secondary side. This information makes life much easier for the manufacturer.

6.3.4 Ordering printed circuit boards/PCB

Usually, the manufacturer will order the PCBs. The documentation that the developer hands over to the manufacturer must contain all the information that allows the manufacturer in their turn to be able to order the PCB from a PCB supplier. The documentation must be complete so that the PCB can be manufactured without the active involvement of either the product owner or the developer.

Ensure that there is no double/contradictory information in the documentation. It is not unusual to find different information in the specification, Gerber and ODB+++ files and text files, which leads to unnecessary technical questions and delays in manufacturing. Please refer to:

Chap. "4.1.3 Design for Manufacturing/ DfM"

Chap.4.3 "Designing PCBs"

The manufacturer ensures that the correct information, including panel drawings, is sent to the PCB manufacturer.

The following need to be included:

- Circuit board specification. This includes information about the IPC standard, board structure, copper thickness, plating, surface treatment etc. Please refer to:

  "PCB Specification"

- Gerber data + Drill data or ODB++/IPC-2581 describes the conductors on every layer, solder pads (footprint), screen printing, drill holes, etc.

The manufacturer needs to supplement this with the following information:

- Size/design of the panel if requested
- Location of sight marks on the panel
- Design of break points
- An optional area defined for solderability tests

The PCB supplier is expected to return the PCBs plus:

- Paste file. This contains repetitive Gerber documentation for ordering paste screens.
- PCB certificate (CoC, Certificate of Conformance) contains the PCB supplier's declaration that the defined standards have been complied with.

6.3.5 Panel drawing

PCBAs are normally assembled in larger panels containing several PCBAs to make manufacturing more efficient. Standardised sizes of PCBs (panels) reduce the number of times that the pick and place machines have to be reset, which makes manufacturing more effective. Another effect is that PCB manufacturers can optimise their processes.

Chap. "4.1.3 Design for Manufacturing/ Production, DfM/DfP"

Panel design is something the developer can leave to the manufacturer based on the requirement specification. Panels should be designed to be as stable as possible, because unstable panels cause considerable uncertainty in the manufacturing process. Common standard sizes PCB manufacturers offer are 130×200, 180×200, 200×270 and 270 x 420 mm. These are the panel sizes the PCB is fitted into. Individual PCBs can be handled directly by pick and place machines down to sizes of about 80 x 80 mm, but handling often becomes much more rational when entire panels are used.

Several aspects must be considered when panels are designed:

Machinery programming and visual inspection are simplified if the boards are symmetrically located in the panel. If no production engineering aspects prevent it, mirrored, rotated and displaced panel rows should be avoided.

The boards must sit stably in the panel because:

- Screen printing becomes a problem if the boards do not lie closely against the screen.
- The components flex against the board and are easily moved out of position if the board flexes during mounting.
- Unstable boards warp further in the oven, causing problems in continued handling and verification, such as for component mounting on the secondary side of the board.
- Automatic visual inspection becomes problematic or, in some cases, impossible to do on bowed panels.
- The mechanical stress caused by bending and skewing means that the board will not reach its expected service life.

Edge design on panels must enable them to be handled without problems in the transport system. If the edges are too sharp, there is a risk that the panels will catch in the machinery and the force of the transport system is sufficient to damage the PCBs. If the boards are supposed to have straight edges and sharp corners, add a rounded break-off tab on the front edge. Please refer to the illustration below. The manufacturer can advise about the recommended radius.



ROUNDED BREAK OFF TAB IN DIRECTION OF TRAVEL.

The minimum distances to solder surfaces, components and reference points should also be considered when the panel is handled in machinery. The selected manufacturer can provide information about the minimum distance required for the machinery used.

Break-off points. PCBs that are planned to be in panels must have break-off points located so that the boards are securely located in the panel and are prevented from moving during handling. The number of break-off points is determined by the size of the PCB, thickness, weight, material, the size of holes drilled in break-off points and the design of the board. The manufacturer can provide information about a suitable design and position for break-off points, so that they suit the relevant pick and place machinery. The location of break-off points can also affect the layout.

Reference points or fiducials, are used by the assembly and visual inspection equipment to achieve sufficient accuracy. If there are no reference points, accuracy is impaired, which affects the result, and at worst, makes component mounting and automatic visual inspection impossible. There are three types of reference points:

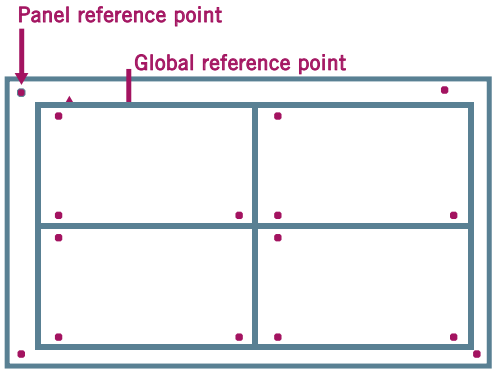
- **Panel** – used to find a panel and verify that it is correctly rotated in the machine.
- **Global** – used for greater accuracy on each PCBA in the panel. Two, preferably three, global reference points should be placed in every panel to facilitate mounting and visual inspection. More than three reference points is of no use.
- **Local** – used for components with fine lead spacing, which requires particular accuracy in positioning and visual inspection.

Positioning of reference points is important as they can be used to simplify the manufacturing process and prevent issues such as incorrect mounting, etc. The figure below shows how Panel and Global reference points should be located to achieve good results. Because the distance between reference points should be as large as possible to achieve good results, it is best to position the points close to the corners of the boards. Reference points should be located asymmetrically to eliminate the risk of turning a board the wrong way round in the pick and place machine.

For double-sided boards, the points should be placed so that they cannot be found if the board is placed upside down.

The time each panel takes in component mounting can be reduced if the reference point locations are specified by coordinate data (pick and place file) and on the place drawing.

Reference points should be located at least 5 mm from the edge of the board. If not, the points will be concealed by the card -fixing supports in the machinery.



PANEL WITH FOUR SINGLE BOARDS

The design of reference points is just as important as their positioning. Incorrectly designed reference points may mean that the equipment fails to localise them. Reference points shaped as a point or a plus sign are recommended. Please refer to the illustration below. They should not be less than 1.0 mm in diameter and not greater than 2.0 mm. The recommended size is 1.5 mm. It is important that the points are designed so that there is a clear contrast against the colour of the PCB. If not, problems may arise oherwise occur in distinguishing the reference points from the background.

The surface of the reference points must be even. Reference points with convex surfaces are often obtained on tin-/lead-coated (HASL) PCBs, resulting in poor accuracy in reading. Demand accuracy from the PCB supplier or use chemical tin or gold, for example, as a coating that gives completely flat points. The same oxide protection can normally be used both on reference points and the rest of the board. The surface must still be flat, however.

Points smaller than 1.0 mm in diameter, oblongs and circles with holes can be handled, but should be avoided. Please refer to the illustration below. Reference points designed as double plus signs or double squares, also so-called diamonds, are not supported by all equipment and should thus be avoided. In "multi-board" operation, all reference points on all PCBs should be of the same type and have the same area.

Thus, there should be reference points on all PCBs, irrespective of whether they are located in panels or are single boards. This ensures exact fixation of the PCB.

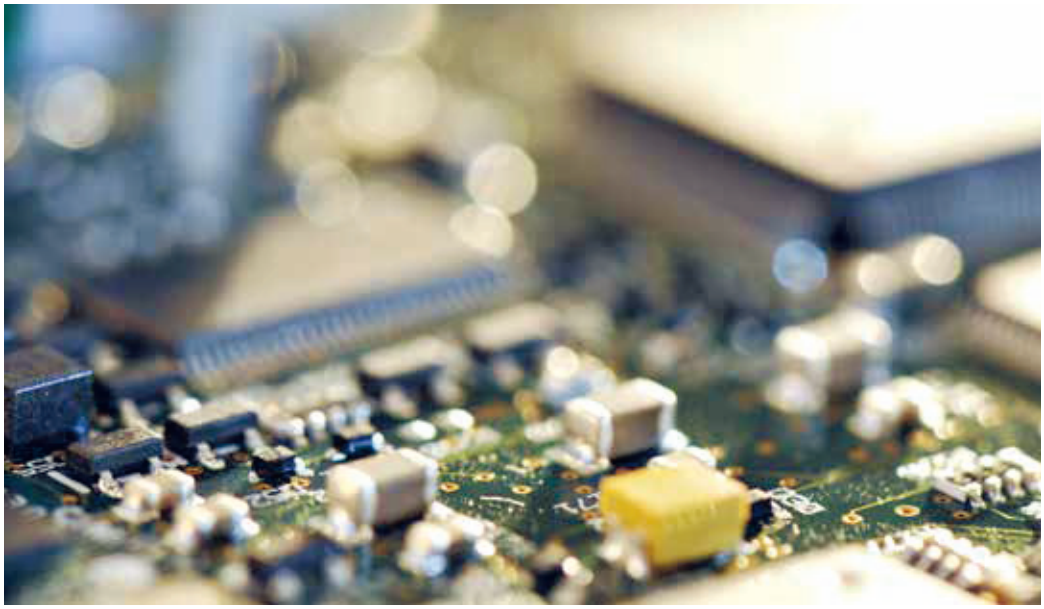


RECOMMENDED DESIGN OF REFERENCE POINTS.



DESIGNS OF REFERENCE POINTS THAT SHOULD BE AVOIDED.

7. Manufacturing follow-up



Manufacturing is complete and the product has been delivered. In serial production, this is a routine event. But if a prototype has been delivered, it might be the first time the product owner and developer get to see their vision in real life, which can sometimes feel like Christmas Day. Although the goal of the handbook is not to describe how to evaluate a prototype, this chapter provides tips to use in the period following manufacturing.

7.1 Arrival inspection and verification

Manufacturers can normally control their processes, but errors might have occurred, so always check one or a few samples of delivered goods.

If a prototype has not been given a serial number by the manufacturer, put a label with an ID number on each unit as soon as it is taken out of the package to create at least a minimum of traceability.

If there is no advanced serial number system it might suffice to write "P1 - 1", "P1 - 2" ... It can also be useful to have an "individual log" to track the prototypes and any special characteristics they have, or changes that have been made. The scope of verification work can be enormous, so here are a few tips:

Document the tests that have been performed on that revision of hardware and software.

Document the tests that have been performed on that revision of hardware and software.

If there are differences in components mounted or function between different units, document the fact that units have been tested and how.

7.2 Producibility

Much information, comments and tips can be obtained from the manufacturer after the first production batch and later during serial production. They are well qualified to offer views and tips about producibility, documentation, design procedures, component selection, etc.

- Does the PCBA contain design features or material choices that make the final product excessively expensive?
- Does the PCBA contain components with long lead times?
- Does the PCBA contain components that are difficult to obtain?
- Are there requirements for a large MOQ (Minimum Order Quantity) from the component suppliers, which means holding large stocks of components?
- How large is the fault frequency in production? Another way to express this is 'yield', which means the number of functioning circuit board assemblies expressed as a percentage of the total number manufactured.
- Are certain tasks time-consuming?
- Can all components be mounted?

Manufacturers have considerable knowledge and experience that they are happy to share. Do not neglect to ask for feedback and take the points to the next prototype release, or when serial production is due to be updated later on.

7.3 Feedback

A foundation of continuous improvements is to use the views that the product owner/developer/manufacturer expresses about the work done by the others. The client for a PCBA should actively ask for feedback about the quality of the manufacturing documentation and the producibility of the design etc. In the same way, it is important that the manufacturer is given comments about faults in produced units, including faults in soldering quality, components etc.

A continuous dialogue is an important tool for improved competitiveness. Likewise, joint efforts should be used to improve product quality, shorten lead times, and reduce production costs.



Also give feedback when everything works well. Everyone feels better hearing words of appreciation.

7.4 Change management

No matter how careful a product has been designed, it will likely have to be modified, because a component has reached its end of life, design faults occur, new functionality has to be introduced, or something else.

Many changes are often made to a PCBA between two different prototype generations, In such cases, simply send new documentation to the manufacturer who may have to restart production planning almost from the beginning. If a product is in serial production, for example, or if only small changes are to be made, it may be much more efficient to describe the change than to send all the documentation again.

ECN, ECR and ECO (Engineering Change Notice, Engineering Change Request and Engineering Change Order) are the usual three stages for product changes or in a PCBA in large organisations. In smaller organisations, an ECO (Engineering Change Order) is normally used directly.

Many different variants of document management systems are available. Large companies with a large number of their own products use special systems that also support change management and revision management.

However, in a company that does not have this kind of system, there should be a change plan in the documentation. Document the changes and communicate them to the manufacturer. Over time, this will be beneficial.



THE TECHNICIAN HAS CHOSEN TO MANUALLY FEED A PANEL INTO THIS PICK AND PLACE MACHINE FOR SUBSEQUENT MOUNTING OF COMPONENTS. THE PANELS NORMALLY ARRIVE ALONG A CONVEYOR BELT IN THE MACHINE.

Change orders are often sent to the manufacturer by the product owner or (on assignment from the product owner) by the developer. The manufacturer may suggest changes of various kinds, but these should not be introduced until clear agreement about them has been reached, for example after an ECO has been written and approved by both parties.

An ECO template is provided as an appendix to the handbook. Please refer to a brief description of the template below. Changes can also be described in a text document and sent to the manufacturer. Here, too, the tips on the next page should be of use.

Mention as early as possible that a change is to be made. This allows the manufacturer to suggest other changes that might increase quality, reduce costs, or both, at the same time.

Briefly describe why the changes are being made.

Specify the contact person at the product owner/developer and manufacturer.

A change almost always means extra costs, process changes, surplus material etc. Reach agreement with the manufacturer about costs and how they should be allocated.

Be clear about when the change should be introduced (as soon as possible/when replaced material has finally been used up/other time) and what should happen to any residual material (e.g. scrapping). Surplus material can be eliminated if things are correctly handled, in dialogue with the manufacturer.

If previously produced and undelivered units are affected, should they be modified/updated?

A good rule of thumb is one ECO per product. If an ECO becomes complicated, it might be because it tries to cover too many unrelated changes at once. These should not all be included in the same ECO. If changes are made to several products at the same time that affect each other, specify in the ECO the order in which they should be introduced, or if all of them must be introduced at the same time.

Description of the ECO template:

The template is created in Excel, but when it is sent to the manufacturer, it should be sent as a PDF file.

If unsure what to write in each box, discuss it with the manufacturer. For example, it could be difficult for the developer to know whether a change affects production testing or manufacturing methods.

There are three types of changes in the template: Product Revision Change, Document Changes and Material Changes:

- **Product revision change:** The changes that are made are so extensive or important that the product revision number is increased. Increasing the product revision number makes changes easier to trace, which is important if it affects compatibility with other parts of the system, etc.
- **Document changes:** Revision changes in the document. These may be changes that affect the product revision or not. They do not necessarily have to do so, as they may simple be clarifications of installation instructions, etc.
- **Material changes:** This is where material changes are described in detail. For example, an R213 is changed from 10 Ω to 15Ω, or circuit board ABC123 has been replaced by circuit board CDE456.

8. Templates, Reading Tips and Other Goodies

Templates from this handbook that can be downloaded free of charge at www.svenskelektronik.se:

Some templates are in Swedish only (File name)

1. Quotation for prototypes (Offert_Prototype_Mall)
2. Quotation for serial production (Offert_Serieproduktion Mall)
3. Ordering PCBA manufacture (Order Mall)
4. Document overview (Documentoversikt Mall)
5. Document overview, example (Documentoversikt Mall Exampel)
6. Product description (Produtbeskrivning Mall)
7. Product description, example (Produktbeskrivning Mall Exampel)
8. BOM A, simple version (BOM_A_Mall)
9. BOM A, simple version, example (BOM_A_template Exempel)
10. BOM B, larger version (BOM_B template)
11. BOM B, larger version, example (BOM_B template Example)
12. BOM AML approval (BOM_AML_Approval Templet)
13. PCB Specification (PCB_specification template)
14. Test Data Specification (TDS) (Production-Test-data-specification template)
15. Change order/Engineering Change Order (ECO template)
16. Revision change (Revisionsandring mall)
17. Revision change, example (Revisionsandring Mall Exampel)

Check lists

18. RFQ_Check list (RFQ Checklista)
19. IPC CHECKLIST for Producing Rigid Printed Board Assemblies is available at www.ipc.org / www.electronics.org, where the entire range of IPC standards are available for purchase.

Appendices etc which can be downloaded free of charge from www.svenskelektronik.se.

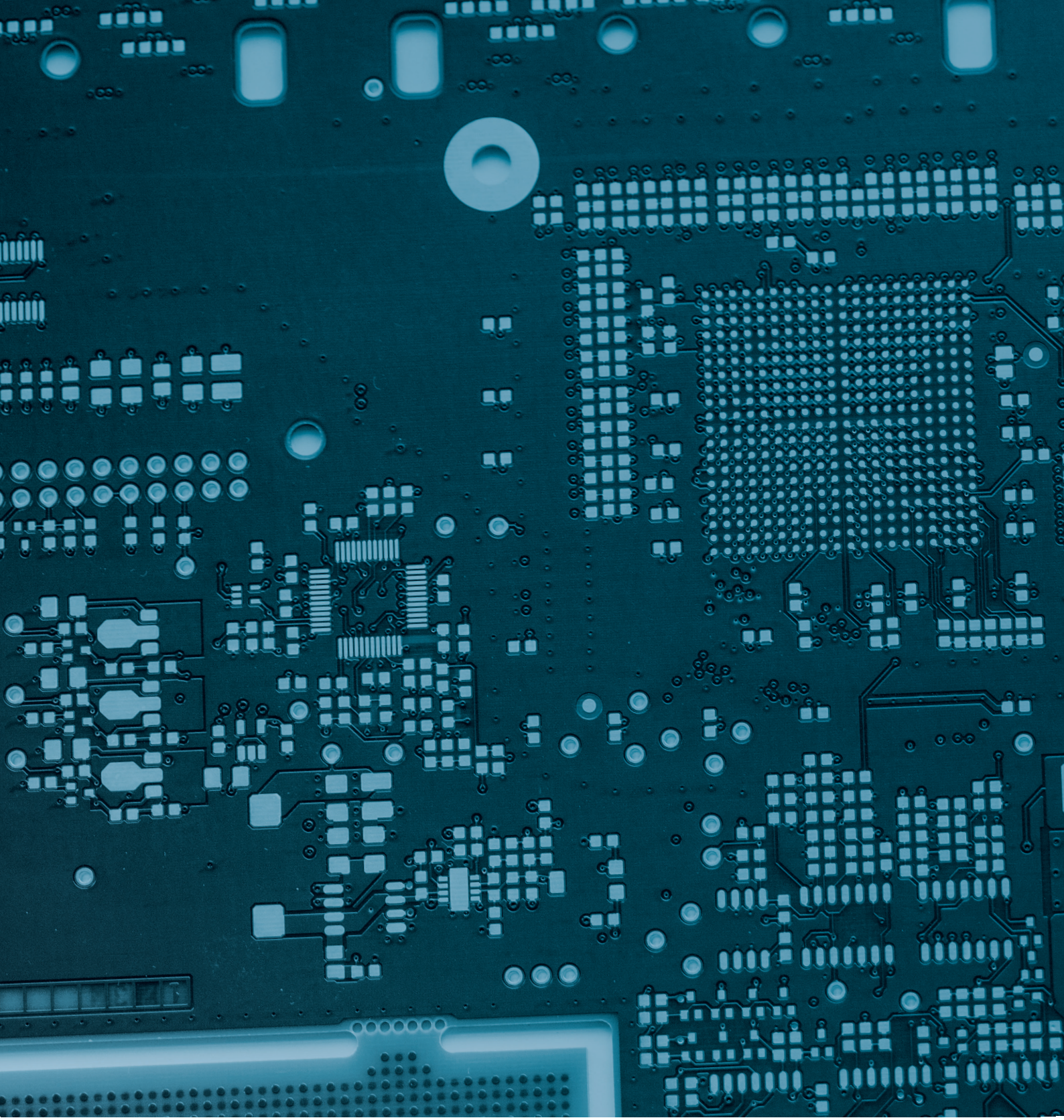
20. A vocabulary, which contains most of the expressions and abbreviations required to manage communication between product owners, developers and manufacturers (in Swedish) (Bilaga Parlör till Handboken)
21. EU directives and regulations v2 2026 (Bilaga_EU_directives_and_regulations v2)
22. Literature references (Bilaga Litteraturhanvisningar)
23. Reliability Assurance of Soldered Printed Board Assemblies. Handbook 2024

Examples of Standard Contracts/General conditions and publishers:

Avtal

a.	SEAL	Svensk Elektronik
b.	NL	Teknikföretagen
c.	SEALK	Svensk Elektronik
d.	Orgalim	Orgalim
e.	Incoterms	Incoterms

Other sources of information are the websites belonging to the developers (consulting firms) and manufacturers (EMS/contract manufacturers)



Smartare Elektroniksystem

ELECTRONIC COMPONENTS & SYSTEMS

SVENSK ELEKTRONIK

svenskelektronk.se

ISBN: 978-91-989591-7-8

PRICE: SEK 395 PLUS VAT