**1. Pre-production**

**Engineering**

This is a detailed and critical process. Its purpose is simple though - to take electronic data, or Gerber data from the customer, and generate the necessary tools and working files so the factory can build the PCB correctly. We also review any specific requirements and ensure that they're followed during production. Failure here, can be carried into production and we can potentially build a perfectly wrong PCB.

We begin with checking the customer's Gerber files to ensure they are complete. This involves confirming we have all the necessary files, and that the hole files align with the drawings. Mistakes occur quite frequently; in fact, over 30% of the data we receive has some sort of concern that we need to address.

We also define the Manufacturing Instructions, they include both the process steps and the tooling needed to build the PCB in the correct way.

Then we take the customer's original Gerber files and generate the working Gerber files. These files are used in the production process to define the circuitry of all copper layers, the solder mask layer with clearances for pads, and programs that outline the correct drilling locations and drill sizes.

We have our own PCB specification to guide us on features that might be overlooked in a customer's specification. We also have agreed routines for how the factory should handle customer data. For instance, the factory cannot make any changes from the original files without written approval. Such requests, or engineering questions – EQs, are always submitted in writing using a standard NCAB EQ template and supported by screenshots through engineering questions, including a suggested solution to ensure the PCB is built according to requirements.

**Material issue**

At this stage, the factory identifies which type and amount of material that is needed for a specific construction, and then cuts the material into the right size and release it for production

The factory sources the materials from approved laminate suppliers. One type of material is the inner layer core, which comes with copper foil already bonded to the outside - they can be purchased with different core and copper foil thicknesses. Since these cores are already fully cured, they can be stored under normal storage conditions.

Another type of material is prepreg, it´s similar to the cores but without copper foil. Since prepreg isn't fully cured, it must be stored in controlled temperature and humidity conditions and has a maximum shelf life of 6 months.

It's crucial to follow a strict first-in, first-out approach when using both materials.

All material comes with a batch number for traceability and a Certificate of Compliance. Additionally, the factory inspects each batch before storing it in the warehouse or using it for production.

Cores and pre-pregs have various versions with different performance features like mechanical, thermal, or electrical characteristics. The selection is based on specific customer requirements.

We only approve specific brands of materials - brands we know are reliable. We also routinely check material stock to ensure nothing is past its use-by date and that the materials are stored in the correct way and grouped by their different characteristics.

**2. Inner Layers**

**Image transfer**

This process allows us to transfer the circuit design from customer-supplied Gerber data to the copper layers of an inner layer core. The challenge here is to ensure an exact representation of the circuitry, so the copper design on the core matches the Gerber data precisely.

After receiving the correctly sized cores, we add a laser engraved Data matrix code for identification during production. Then we chemically clean and slightly roughen both sides of the copper surface to ensure good adhesion of the film. The photosensitive film is then applied to the panel.

This process occurs in controlled clean rooms with fewer than 1000 particles per cubic foot. The clean rooms are protected from UV light, so all windows and lights are covered with a yellow UV-blocking film.

There are two common methods for applying the circuitry pattern, to the film.

The first uses artworks which are placed on top of the film. Then the panel is exposed to UV light. Where there are clear areas, the UV light is allowed to penetrate through the artwork, defining the circuitry pattern.

The second method is laser direct imaging which is increasingly popular. It offers improved registration of the pattern and better definition of smaller features, especially useful for more complex designs. The principle is somewhat similar to the previous method: when the laser contacts the film, it polymerizes the film, creating a pattern that matches the desired circuitry.

In both methods, the polymerized film stays on the panel for the next process.

**Develop Etch Strip**

After applying the circuitry pattern to the inner layer core, we must remove the unwanted copper, leaving only the desired circuitry design.

In previous steps, specific parts of the UV-sensitive film were polymerized by UV or laser, matching the circuitry pattern. In this step, the unexposed or unpolymerized film area will be chemically removed, revealing the undesired base copper on the inner layer that we want to eliminate. The exposed film areas will stay. The unwanted copper is then etched away using either an ammonia or cupric chloride-based solution.

Now, we have the desired copper, resembling the circuitry from the customer-supplied data. However, it's still covered by the exposed film that wasn't removed. To reveal only the wanted copper, we strip away this remaining film through another chemical process, followed by a final cleaning step. This completes the inner layer core.

Controlling the process poses significant challenges. As with all of the chemical processes we must have tight control of the many elements relating to the chemistry (pH, temperature, concentrations, copper content for etching) and we must also manage other elements such as conveyor speeds for different copper weights, rinsing and spray pressures to maintain precise etching rates. The goal is to remove unwanted copper without affecting the desired circuitry by etching either too much or too little. Higher technology factories often use vacuum etching lines for better control and minimal variation from top side to the bottom side.

**AOI**

Automated Optical Inspection, AOI, is a way to scan the copper circuitry on inner layer cores and compare it to the design in the Gerber data.

Before scanning with AOI, we may observe factories cleaning the inner layer cores to remove any debris or surface contaminants. The first stage of AOI involves the machine scanning the cores, comparing the scanned image to the customer's Gerber data. AOI identifies errors or faults, preventing the use of flawed inner layers in the production of the circuit board moving forward.

The second stage is the verification stage, where operators check if concerns found during scanning are acceptable or not. They inspect the circuitry for issues like shorts, opens, or damage, ensuring that the conductor width and spacings meet customer requirements and IPC tolerances.

Short circuits or excess copper can be repaired but according to our PCB specification, we don't allow open circuits to be repaired or welded. This enhances reliability by eliminating the risk of repair failure and does not compromise signal integrity.

We also ensure that only the most suitable equipment, offering good definition and imaging qualities, is used for our products.

**3. Lamination**

**Oxide layer**

In this process step the inner layer cores are prepared for bonding. Later, during lamination, they will be bonded together using prepreg sheets as glue and insulation between the conductive layers on the inner layer cores. The oxide layer is necessary to prevent copper from corroding and helps the copper stick well to the resin in the prepreg.

We clean and slightly roughen the inner layer cores through a simple process of cleaning and micro-etching. Then, we apply an oxide layer.  While it may seem straightforward, it's crucial to carefully manage the chemistry and handle the treated cores with care. The thin oxide layer is very sensitive to damage, and any damage could risk a failed bond during lamination, potentially leading to delamination of the bonded multilayer.

**Layup**

To construct a multilayer PCB with the inner layer cores we've made, we need to stack them carefully, making sure they're perfectly aligned, and then securely bond them together.

First, we prepare the inner layer cores and the correct prepreg sheets as specified in engineering. We make sure to stack all cores in the right order and orientation during lay-up. If not, we won't achieve the right connections in later processes, hindering circuit connections across all layers.

We alternate between laying cores and prepreg to build up the layers, aligning using the pins. After aligning, we may weld the stack at various points on the outer frame for the needed alignment and layer-to-layer registration. This process repeats with an additional set of cores and prepreg.

Afterward, we just need to add the final outer layers of pre-preg to complete the package before moving to the last step, where copper foil is added, and the stack undergoes pressing.

It should be noted that we start with a plate on the bottom of the stack and finish with one on the top of the stack so that the stack, especially the thin copper foils, is not damaged during the bonding process.

NCAB only accepts materials - laminates and pre-pregs - from well-known manufacturers. The lay-up process is crucial for a quality multilayer, and it's a key focus during our audits. To get approval, the lay-up process must meet specific requirements, including cleanliness, a controlled environment, excellent handling, and well-maintained equipment.

**Bonding**

The purpose with this process step is to bond the individual materials together into a solid panel. If done right, with the correct materials, it can handle heat, cold, and mechanical stresses in tough environments.

This is achieved through a process where the product goes into the lamination press where it is hot-pressed with specific pressure and temperature based on the desired build and types of materials used. The resin within the prepregs melts and will glue the prepreg to both the inner layer cores and the copper foils.

Once hot pressing is done, the product is then transferred to a cold pressing to cool down noting that a slow and controlled cooling is important to avoid bow and twist of the panel. Once cooled, the resin is cured, bonding the layers together, and the multilayer we created is now very stable.

After the cycle is done, the tooling and the bonded multilayer stack leaves the lamination process. We carefully unpack the multilayer panel and move it to the next step. The tooling goes for cleaning to get ready for the next cycle.

Lamination is a crucial step in multilayer production. To pass the NCAB audit, each factory must show they have press parameters for various materials and constructions. Additionally, they need to prove through reliability tests that the bonded panel can withstand specific forces or pressures without coming apart.

**4. Drilling**

**X-Ray drilling (tooling holes**)

After lamination, all internal reference markings that shows where we want to image the outer layers and where we shall be drilling in relation to the position of the inner layer features, are now hidden in the multilayer panel.

Failure to align upon these internal reference points correctly could result in our drill location not making precise contact with the copper feature on the inner layers risking potential weak connections.

In this process, we use an X-Ray to detect the internal references and provide new tooling holes that align with inner layers.

In simple terms, the X-Ray is used to find the internal reference markings on different cores. The equipment then figures out the best alignment for drilling, ensuring we align with the internal pattern. Once this is determined, new tooling holes are drilled in the panel to help align the multilayer in the next steps of the process.

We measure and record data to see how much the inner layer cores have changed in size during bonding. This information is then used to adjust for these changes in later processes, like drilling, to make sure the pattern accuracy requirements for all layers are met.

At this stage we also we cut away the excess material all around the edge of the panel as this is considered waste and to be recycled.

In NCAB´s own PCB Specification, we have limited the allowed differences in size between the Gerber data and the actual PCB pattern. This ensures that the customer's printing stencil fits the PCB pattern correctly.

**Mechanical Drilling**

Drilled holes are essential as they, once plated, serve two purposes.

* The first is to create electrical connections between all the layers, usually going through the entire PCB – this is called a via hole.
* The other is for solder through-hole components - that is the component hole.
* There are also non-plated holes used for housing alignment or securing the PCB into its final product.

First, we determine where to drill, using drill files from customer data, including positions and drill size details.

Before drilling, we place a baseplate on the machine bed to prevent drilling into the machine and avoid burrs on the panel sheet's bottom side. On top of the baseplate, we stack the panels, aligned using the tooling holes we added in the previous step, usually three panels at a time, but this can vary depending on thickness of the panel and copper thickness. Aluminum entry sheets are added to prevent drill deflection, ensuring hole accuracy, and also avoiding burrs and scratches.

We control the drill head's feeds and speeds to prevent issues like broken drills, excessive resin smear and poor hole wall quality.

Additionally, we control the number of 'hits' that each drill makes, before the machine automatically replaces it for a new drill and the old one is removed for re-sharpening.

It should be noted that before the drill machine starts drilling with any new tool, the drill will be checked automatically to ensure that it is the correct size and correct length of the flute to ensure that the drill is not broken.

**5. Plating**

**Desmear and Electroless plated through hole**

The reason we need this step is that we need to plate copper inside the hole. The hole wall is a non-metallic surface – and to plate on a non-metallic surface is hard to do. Electroless is a special plating process that can plate on a non-metallic surface – so this is the first stage before we can plate the desired thickness of finished copper.

Before we start plating, we make sure to remove any resin smears left from the drilling process, using a chemical solution for this.

Then, we activate the panel's surface, including edges and drilled holes, to enable us to plate on to the non-metallic areas.

In the next step, a copper deposit is plated onto the non-metallic and metallic areas of the whole panel, including through the drill holes, onto the hole wall. The copper deposit is very thin, around ≤ 1um thick. This process can be done vertically for multilayer products or horizontally for HDI products.

This process, like other chemical processes, depends on precise controls due to its specialty - plating metal on non-metallic and metallic surfaces. Having this step under control is crucial because without this thin deposit, we won't have connections through the holes. We use the backlight test, which is a method to check the deposit. Relying only on microsections is challenging as the thin deposit is hard to measure, and using a probe to measure it risks damaging the deposit.

**Galvanic Panel plating**

The previous process only plates a very thin layer of copper, and this thin deposit isn't sufficient for the next steps, nor is it sufficient for the final working circuit. That's why we need to add a thicker layer of copper to the surface.

The panels are processed using the vertical electrolytic plating line to receive the first stage of additional copper.

Before plating, the panels are cleaned with an acid cleaner and rinsed before we begin the plating process, to ensure cleanliness.

The panels are connected to a cathode, and the plating solution in the plating bath is connected to an anode, and with the applied current the copper ions move from the plating solution to the panels, depositing copper all over the surface of the panel, including into the holes.

We deposit about 5 to 8 um of copper on the outer layer surface. We don't add more because we will remove some copper in later stages.

To ensure quality, factories must use reliable and well-known chemical brands for the plating chemistry and have good process control, as well as good management of all other plating parameters including time in baths and control over plating currents. We also expect at this stage deposit rates to be checked as well as surface / through hole thickness of deposits.

**6. Outer layer**

**Image transfer**

Following the addition of copper as a result of panel plating, we may now apply an image on the outer layer copper foils, matching the Gerber data. This allows us to form the outer layer pattern that will both define the functionality of the printed circuit board as designed by the customer and allows the customer to add components to the circuit board delivering the functionality of the electronics device.

This is a similar process to the one for inner layer imaging, where we apply a circuit pattern onto photosensitive dry film using UV light or laser direct imaging. The main difference here is that for outer layer imaging, we want to remove the dry film where we want to keep the copper, where we want to define the circuit pattern. It might sound odd, but this is because, during the later stages, we wish to add some extra copper only where the circuit pattern is, in order to achieve the final thickness for the outer layer copper (both for surface and through hole).

As before, there are two ways to apply the outer layer image to the panel - the first is by using Laser direct imaging, and the second is by using artworks or films and exposing them to UV light to create the image.

It's important to have good control over the imaging process, starting with a well-applied layer of dry film being applied to a perfectly clean panel. And, that no dust or debris is within the department as dust or debris stuck between the panel and the artwork can cause problems as we try and image the circuitry, therefore it is crucial this process must take place in a very tightly controlled clean environment using the right equipment.

**Developing**

Similar as for the inner layer, we want to remove photoresist from the panel's surface, but this time, it's in the areas where we want to have tracks.

We chemically remove the photosensitive film that the laser or the UV light didn't polymerize. This reveals the copper we want to keep. Without dry film in these areas, we can add more copper to achieve the final desired thickness.

The key challenges with the process are controlling the chemistry and the processing of the panel through the developer so we only remove what we wish to remove. This is one thing we check during our audits. Also, we check the developers to make sure we rinse right after developing to remove the unwanted resist and prevent it from sticking back to the panel.

**Pattern plating**

To put it simply, in this process, we add extra copper to the panel only where we want it, as we continue to define the outer layer circuitry.

This is another **electrolytic plating process**, adding copper only to the areas exposed during outer layer imaging and developing, including through the holes, ensuring the final plating thickness meets either (i) NCAB's requirement of an average 25 um through the hole, or (ii) aligns with the customer's surface thickness preferences should they have a specification beyond the NCAB requirement.

Once the additional copper has been plated then a deposit of **tin** is applied on top of the plated copper, in the same pattern, to protect the plated copper underneath during the subsequent processes.

We check and audit the control of all plating parameters including the process control for all of the chemistry and the setup of plating currents, which determine deposit rates.

Ensuring a uniform deposit can be a challenge due to design, with isolated external areas potentially plating higher. The right set up on plating flight bars is important as are thickness/deposit checks to ensure good levels of plating uniformity are achieved across all panels.

Final plating thickness is verified through microsections and non-destructive methods.

**Strip Etch Strip**

In this process we remove all the remaining dry film, then etch away the unwanted copper from the panel's surface, leaving only the copper we want. This creates a functioning circuit.

First, we chemically remove the remaining dry film, exposing the unwanted copper underneath. Then, we use etchants to remove the unwanted copper under the removed dry film.

Once the unwanted copper is gone, we need to remove the tin deposit, which protected the desired copper during the etching process. This is done using an acid solution, leaving behind the final outer layer copper circuitry.

Like in previous chemical processes, we finish with rinsing and drying. It's worth noting that at this point, the PCB is fully electrically usable.

During etching, tracks can sometimes be etched in areas they shouldn't be. However, based on our own demands, we don't permit the repair or welding of open circuits, as it could impact reliability. To ensure quality in this step, we check with the factory to understand how they inspect the physical PCB track widths as they come off the line. Additionally, we conduct various chemical & SPC checks. [Statistical process control]

Once again, we have a very critical process reliant upon chemistry, so it is very important that the factory has excellent SPC in place to control the chemistry and thus provide a reliable process with a reliable, predictable output.

**AOI**

Automated Optical Inspection [AOI=Automated Optical Inspection ] is a way to scan the copper circuitry on outer layers and compare it to the design in the Gerber data to verify that the pattern defined is correct.

Before scanning with AOI, we clean the outer layers to remove any debris or surface contaminants.

Then, the machine does the scanning, comparing the scanned image to the customer's Gerber data.  The AOI scanning identifies any concerns, such as errors or faults with the pattern, thus preventing the use of faulty outer layers in the production of the circuit board moving forward.

The second stage is the verification stage, where operators check if the concerns identified during scanning are acceptable or not. They inspect the circuitry for issues like shorts, opens, missing copper or other damage to the copper pattern, ensuring that the circuitry, including conductor width and spacing, meet both customer requirements and IPC tolerances.

Short circuits or excess copper can be repaired but according to our PCB specification, however we don't allow open circuits to be repaired or welded. This enhances reliability by eliminating the reliability risk of a repair failure and does not compromise signal integrity. We also ensure that only the most suitable equipment, offering good imaging qualities, is used for our products.

**7. Solder mask**

**Via Hole Treatment**

We fill via holes to make the PCB more reliable and aid in the assembly process. This enhances the PCB´s reliability by reducing the likelihood of any chemistry becoming trapped within a partially filled via and prevents liquid leakage onto the surface.

To achieve this and fill the via holes, in accordance with the IPC 4761 guideline, specifically type VI filling, we usually use solder mask ink or special via plugging inks along with a prepared aluminum stencil. The stencil has holes that match the via holes we want to fill. We use a screen-printing process to push as much ink into these holes as we can.

In the assembly process one benefit is when conformal coating is required, via plugging prevents the liquid or resins from leaking through to the bottom side.

According to IPC, there are 7 types of vias, but types 1 to 4 are not recommended. We base the NCAB demands upon the Type 6 recommendation for plugging the holes. This involves filling the holes from one side, and then followed by an application of solder mask to cover the surface of the circuit board, including the plugged holes. We choose Type 6 because it ensures a good solder mask fill and with our own demands we exceed the guideline calling for at least 70% fill, minimizing the risk of any issues with via holes during assembly.

**Printing & pre-cure**

In this process, we cover the surface with a photo-imageable ink called solder mask. The ink provides protection where needed and can be removed where we don't want coverage, such as in areas where components will be soldered. The solder mask also protects the copper, preventing oxidation, and provides insulation for the circuitry.

First, we need to prepare the copper surface by roughening it for better adhesion of the liquid solder mask. This can be done through mechanical brushing with pumice and/or a chemical etching process. Etching is good for fine tracks that could be damaged by mechanical roughening alone. After roughening, the surface is rinsed and dried to prevent copper corrosion.

Next, the liquid solder mask is applied using either a silkscreen or spray coating process, depending on the factory's setup, technology requirements, and solder mask type. In this case, we observe screen printing, where the liquid solder mask is deposited on one side of the PCB. The panel is then flipped, and more solder mask is printed on the second side.

Since we can't handle a panel covered in wet ink, we pre-cure the deposit of liquid solder mask slightly. This process hardens the outer layer of the solder mask, making it handleable.

We check and approve the solder mask inks used by the factory, ensuring they comply with class T of the IPC specification SM-840. In line with our own demands, we specify the thickness range of the solder mask needed on the PCB - not just on the surface, but also on the knee of the track. This ensures a reliable coverage of ink. All these aspects are thoroughly checked and reported in the NCAB CoC.

**Soldermask exposure**

Now that the board is coated in solder mask, we need to image the areas where we want to remove the solder mask we don’t need, like pads for surface-mounted components. Challenges include accurately removing small amounts of solder mask to define small pads and ensuring precise alignment with the copper circuitry.

To expose the solder mask, we use different methods.

The most common one in series or volume production, is using UV-light and artworks for both the top and bottom sides to expose a pattern onto both sides of the panel.

The process is quite straight forward, with different parameters for various solder-mask types. The machine aligns solder-mask films to the panel on both sides using cameras and fiducials on the production panel.

After alignment, the machine secures the panel with a vacuum.

UV-light exposure begins with defined energy and for a specified time. The UV light interacts with solder mask polymers within the ink, making them resilient for the next steps.

This process can be automated with a line that autoloads, aligns artworks, and exposes. Alternatively, it can be done semi-automatically with machines that use camera alignment, where operators load panels and expose each side separately.

Other methods involve using laser or laser-LED light to create the pattern.

These methods work well for designs with tiny features, like those in HDI designs, as they offer precise resolution, and the exposure image can be adjusted based on fiducial alignment.

However, these machines tend to work at a slower pace compared to the UV exposure method.

We check exposure settings in all machines and evaluate the cleanliness of the room to limit foreign particles. This process is a key focus during all NCAB factory audits.

**Solder mask development**

After solder mask exposure, we can remove areas with unwanted solder mask through a development process.

The production panels are placed in a developing line, where chemicals are sprayed onto the solder mask layer from both sides. The unexposed solder mask areas are not resistant to the developing chemicals and are effectively washed off. The areas with exposed, polymerized solder mask are resistant against development chemicals and remain on the PCB surface.

Whilst the process ends with rinsing to remove any chemical residues, followed by a final drying of the panel, there is typically an in-process quality control step directly after – where the deposit of soldermask is checked considering thickness, adhesion, registration and other elements that may influence quality or reliability. This step is closely examined in all NCAB factory audits, with a focus on checking chemical controls.

Once again…. Another process reliant upon chemistry, and this means that we pay attention once more to the SPC that the factory runs at this stage.

**Soldermask final cure**

We must give the solder mask a final cure to fully set the ink. This ensures the solder mask can withstand further processes and is securely attached to the panel's surface.

Now that we have the solder mask in the right places, we need to give the panels a final cure. The panels are loaded onto a conveyorized oven, moving through different temperature zones for a set time. Within these temperature zones, the polymerization is completed, solvents evaporate from within the ink, and the solder mask layer is fully cured.

This step is a key focus during NCAB factory audits - We check the profiles to prevent solvents from erupting within the ink, which could cause bubbles in the solder mask deposit.

**8. Legend**

We mark the PCBs to include component identification, factory markings, part numbers, and date code or batch references for traceability. All marking except the component identification can also be added using negative solder mask.

In some of the factories, we use inkjet technology for direct legend printing. Like other processes, precise registration is crucial to avoid printing on exposed copper features. The machine reads fiducials on the panel, aligns it, and prints the legend ink directly onto the PCB surface, following the customer's Gerber data. The ink is then cured with UV light during the process.

The second method is quite similar but involves a traditional screen-printing technique to apply the ink. A screen with open areas, matching the customer's Gerber data, is used. After printing, the panels undergo thermal curing, ensuring the ink becomes resilient and adheres to the panel's surface.

**9. Surface finishes**

**ENIG**

ENIG stands for Electroless Nickel Immersion Gold. It's the most common surface finish for high-reliability PCBs. Surface finishes are necessary to prevent copper oxidation over time and allow customers to solder components to the PCB.

The ENIG process is very complex and requires careful control.

It involves a series of chemical baths, starting with an activator that cleans and prepares the copper for nickel deposition.

Step 2 is the Nickel bath, where Nickel is plated onto exposed copper areas.

Then comes the Gold plating step, adding a thin and consistent layer of gold less than 1um to protect the nickel from oxidation.

Afterward, we rinse and dry the panels.

This process ensures a reliable surface finish for soldering components to the PCB.

NCAB has dedicated Quality Staff in the factory to oversee the ENIG line, ensuring it functions correctly and conducting tests for superior quality, particularly in solderability. We closely monitor Nickel (Ni) and Gold (Au) thickness [We closely monitor Nickel (Ni) and Gold (Au) thickness] records and check all chemistry parameters. Given the complexity of the process, maintaining good SPC is crucial.

**Immersion Tin**

Immersion Tin is a more affordable surface finish compared to ENIG and is often chosen for applications involving Press Fit pins. It is better for the environment, but some chemicals used in the process require strict control to prevent contact with humans.

On the contrary to the ENIG process, this is a horizontal process.

First, panels undergo a UV bump to fully cure the solder mask, as uncured ink may react with the tin chemistry and lead to soldering problems.

Next, a pre-cleaner with Acid removes contaminants, followed by rinsing.

A stronger acid micro etch roughens the copper surface.

Tin is then applied, forming a thin, flat layer onto the surface, forming a bond with the copper layer.

After that, the PCB needs cleaning in warm water, followed by thorough drying to prevent staining.

This process ensures a shiny surface when dried.

NCAB's Quality Staff in the factory ensures the Immersion Tin line functions correctly, they monitor controls and conduct tests for superior quality and solderability. We ensure controlled thickness of deposit and a controlled shelf life, given the right storage conditions. Maintaining the right thickness is crucial, as exposure to higher temperatures accelerates growth of the intermetallic layer, affecting soldering if it reaches the surface. Thickness records and solderability performance are closely monitored.

**HASL**

Hot Air Solder Leveling (HASL), for both leaded and lead free processes, is a common PCB surface finish process, that provides a surface finish which is similar to solder used to solder the components.  It's important to note that the surface is not as flat as ENIG or Immersion Tin, making it unsuitable for fine pitch SMD components.

First, we clean the PCB to make sure the copper is ready for solder.

Then, we apply flux to remove any oxides from the copper surface.

Next, the PCB is lowered into a bath of molten solder (for a certain time and with the solder batch at a specific temperature), which sticks to the copper areas.

As the PCB is withdrawn from the solder bath (at a certain speed), air knives running the width of the panel that blow hot air onto the panel remove excess solder, ensuring we keep the right amount on the areas we need to be solderable.

After that, we clean the PCBs to remove any leftover flux that could cause corrosion or electrical issues.

It's important to control the copper content in the solder bath and remove impurities to maintain a good solder surface.

It should be noted that this principle is the same whether we are running a lead free or a leaded HASL – of course some of the temperatures and other parameters may change, but the principle remains.

NCAB has quality staff in the factory to make sure the process works as it should, they monitor controls, and conduct tests for superior quality and solderability. We ensure controlled thickness of deposit and a controlled shelf life, given the right storage conditions.

**10. Profiling**

**V-Score**

By creating controlled depth score lines on both sides of the circuit board, our customers can easily separate individual boards from the panel after assembly.

We use special machines with rotating blades on the top and bottom to cut score lines into the panel on both sides, following the customer's design and as pre-defined in the engineering process. Before we begin cutting, we ensure the production panel is accurately aligned and set in the machine.

This process is repeated for each V-score line until they are all applied to the panel.

We make sure the factory controls the score line depth and checks the remaining thickness. We have defined our own V-score definition, which is applied if the customer doesn't have their own. We offer controlled V-score with production records in the NCAB Certificate of conformity for each delivery. Detailed V-Score and tolerance information is available in our design guidelines.

**Routing**

Routing helps cut out the customer panels or individual circuits from the production panel, creating the final outline. It offers an alternative to V-score, also allowing customers to separate the individual circuits after assembly.

First, we secure the production panel on the routing machine table using tooling holes for proper alignment with the circuitry. Usually, we stack two or more production panels on top of each other and fix them with pins on the machine table for alignment.

Once secured, we load the outline program specific to the PCB design into the machine. This program defines the rout contour and selects the necessary tools. The routing machine then cuts the PCB from the production panel, allowing them to be separated.

The final step is to wash the product to remove any particles from the surface.

Our PCB specification show the minimum tolerances that shall be applied, but if a customer requires tighter tolerances, we follow those. The results are defined in our Certificate of Conformity (CoC).

We only use approved routing machines for NCAB products.

Additionally, we support customer´s PCB designers with detailed design guidelines, specifying tolerances for PCB profile, holes, and other crucial elements.

**11. Quality**

**Electrical test**

To make sure all the circuits are correct and have the right connections, we test every PCB electrically. We look for any unexpected open or short circuits ensuring everything works as it should.

There are two common ways to test for basic open and short circuits.

One is the bed of nails or fixture machine, suitable for checking large batches in high volume.

The other is the flying probe test. [Flying probe] The flying probe method is typically used for small volumes, prototypes, and dense or fine-pitch products. It tests each electrical net individually, so it takes longer to test.

The bed of nails or fixture testing is very fast as it tests all electrical nets at the same time. In many cases, this machine automatically separates tested boards into pass and fail stacks. Before using the bed of nails or fixture machines, we need to build a unique test fixture for the specific PCB being tested. Passed PCBs are marked accordingly.

An additional test is the 4-wire kelvin test, [4-wire kelvin] focusing on checking net continuity and resistance. Hidden defects such as hole integrity can only be efficiently located by requiring a 4-Wire Kelvin test. This is crucial for HDI structures, assessing the reliability of through hole plating along with connections between blind via / microvia holes and their connected layers.

At NCAB, we ensure 100% electrical testing following IPC-9252 standards, and we mark PCBs for traceability after testing. We recommend our customers to connect with our local offices to discuss additional tests, like impedance or 4-wire testing, to enhance their design quality.

**Final inspection**

After going through all production process steps - each PCB must be inspected to ensure that it meets the demands of the customer, IPC and NCAB standards.

This first step is the visual checking where we are looking for any cosmetic defects or damage to the PCB that could affect meeting agreed demands or pose a risk to product functionality. Factories use both manual and automated visual inspection (AVI) to inspect 100% of the PCBs supplied to NCAB and customers. The manual inspection is straightforward, while AVI, under high magnification, compares images to highlight discrepancies or concerns automatically. AVI is crucial for detecting issues in smaller PCBs, but the inspector's skill is still essential for judgment.

The challenge is that different customers have varied demands, making it hard for factories to determine what is acceptable. Some projects may lack sufficient reference to acceptance criteria. So to address this, we have defined our own additional acceptance criteria, that ALL factories will adopt as the MINIMUM requirements for NCAB orders. It includes over 100 points, covering elements beyond IPC requirements, and thereby serving as our benchmark for quality and acceptance.

**NCAB Final Quality Check**

Every PCB design is unique because each product has a specific purpose. In many cases, the PCB must be built differently (different copper thicknesses, different sizes or with different base materials) to meet the specifications and demands of the customer. Therefore, we carefully check that the built PCB aligns with the given specifications.

The previous inspection steps mainly focus on the appearance, but visual checks can't confirm if the right build is achieved or if the correct copper thickness and ENIG deposit reliability are met. This is why many factories use additional tests like microsections. Microsections help verify builds, inspect copper thickness, and identify issues with copper plating and hole wall quality. We also check hole sizes and dimensions to ensure they match the customer's drawing and specification.

Factories can perform various tests to evaluate PCB quality and reliability, and we ensure that the factories we work with have the necessary capabilities to do so. However, these checks are often done only on the first batch.

We know that each PCB is unique, and we recognize that unchecked process or operator variation may impact products from batch to batch. Not all factories verify the same elements to the same extent, which creates risks.

To address this, we go beyond and conduct our outgoing inspection report for EACH new batch. Our report covers various aspects, including cleanliness, dimension checks as per the drawing, copper thickness measured in over 8 locations, types of materials used (laminates and solder masks), and electrical checks like impedance testing, annular ring size, or BGA pad size. We thoroughly check these aspects to ensure the product meets our standards for a reliable and durable PCB.

**12. Packaging**

After completing the PCBs, we carefully pack them for shipment. Good packaging is crucial, not only to protect the circuit boards during storage at the customer's premises but also to ensure the solderability of the PCBs.

Before packing, the boards are counted either manually or using an automatic machine, which also reads markings to check part numbers and separate different date or batch codes.

Finally, the PCBs are packed: stacked on a tabletop, a desiccant is added to remove humidity from the shrink-wrapped package. The boards then go into a shrink-wrapping machine to seal the package, creating a soft vacuum by removing most of the air.

Some factories also use a silver coloured, moisture barrier bag, that offers better protection in humid environments.

A label with identification and traceability data is added to the sealed package and packed into NCAB cardboard boxes.

Upon customer request, a Humidity Indication Card can be included inside the sealed package.

Packing materials, desiccants, Humidity Indication cards, date codes, and the quantity of PCBs or panels per sealed pack must comply with NCAB Packaging and Labeling Instructions. Both inner and outer packages should have NCAB labels.