smart surface coating

Resource-saving rework of printed circuit boards

from
Dirk Kaschel
APL Oberflächentechnik GmbH, Germany

Summary
APL Oberflächentechnik GmbH is a service provider for functional surfaces in the electronics and pcb industry. It specializes in treating the surface with immersion tin (iSn - immersion Sn) under the product name smarttin®. In addition, they offers service in refreshing of iSn-circuit boards. This method allows to put those 'not more- or bad solderable pcb' back into a solderable state without much effort. In 2014 another post-processing step was made, it is the so-called Final Clean process was introduced. This enables us to restore the solderability of ENIG and ENEPIG surfaces without the use of stripping chemicals. Both two Post-treatment methods enable the resource-saving in terms of electronic boards reuse, because with these method a successful rate of >95% can be achieved without much effort.

A short Biography
Mr. Dirk Kaschel was graduated in 1994 after training as a surface coaters (formerly depositor), his engineer training at the professional school for PCB and electroplating in Schwabisch Gmund equipped him as a state-certified technician who specialized in printed circuit board technology. Then he worked 4 years at a company called PPE in Schopfheim (the predecessor of today's Würth Elektronik). Mr. Kaschel worked in the fields of metallization, pattern plating and precious metal coating, later he led the analytical lab and the wastewater treatment area. Since 1998 he has worked as Technical Director of the Production and Quality at APL in Lörrach.

1 Introduction

The last step of manufacturing pcbs is to coat the surface finish. The surface finish prevents oxidation of the copper of the pcbs, which ensures following process actions of the pcbs (e.g. soldering, bonding, contacting or press-fit). The purity and cleanliness of the end surface plays an enormous role, it is the cleanliness, not given by ionic impurities, residues or other contaminants, which creates often problems and leads to operation failures. Furthermore, the circuit board often can not be processed if the predetermined maximum storage time has been exceeded. If the pcb once is manufactured and already coated, there are often high chances that its end surface got strip off with big cost and eventually has to coat new or to scrap the circuit board. Both processes are very expensive and sometimes very harmful to the environment. Adding this point that means the circuit board is exposed to huge stress, which can ultimately lead to the complete failure of the circuit board.

Currently a couple of reworks so-called 'refresh processes' are available in APL. Hazardous chemicals are not introduced into this process at all. Through this simple process, the circuit boards can be rescued. However, these methods can only be applied to a certain end surfaces. The refreshing Surface Technology is offered at APL for more than five years, it is also qualified by international companies (EMS). The following document is intended to explain relevant resource-friendly processing of printed circuit boards.

2 Which established end surfaces are currently being offered by the market?

End Surfaces / Application	iSn* Sn: 1.0 µm	ENIG* Ni: 4-7 µm Au: <0.1 µm	ENEPIG* Ni: 5 µm Pd: 0.1 µm Au: <0.1 µm	OSP* 0.3 µm	iAg* 0.15 – 0.45 µm	HAL* 1-20 µm
Leadfree Soldering	■	■	■	■	■	■
Al-Wire Bonding	◆	■	■	◆	▼	◆
Au-Wire Bonding	◆	▼	■	◆	▼	◆
Cu-Wire Bonding	◆	◆	▲	◆	▼	◆
Cu/Pd-Wire Bonding	◆	◆	■	◆	▼	◆
Pressing	■	▼**	▼**	▼**	▼**	▼**
High-frequency Technology	■	●	●	■	■	■
Fine line	■	▼	▼	■	■	◆
Planarity	■	■	■	■	■	◆
Storage Period	12 Months	12 Months	12 Months	6 Months	6 Months	12 Months

■= very good ▲= good ●= qualified ▼= bad ◆=very bad

*** iSn: immersion tin**
*** ENIG: electroless nickel/ immersion gold (electroless nickel/ gold)**
(Risk of cracking of the copper plating in the holes, due to the high hardness of nickel)**
*** ENEPIG: electroless nickel/ electroless palladium/ immersion gold (electroless nickel/ palladium/ gold)**
(Risk of cracking of the copper plating in the holes, due to the high hardness of nickel)**
*** OSP: organic surface protection (organic surface passivation)**
(Risk of contact corrosion in the bore after first reflow (layer is no longer 100% proof). It can form a galvanic element between copper sleeve and pin)**
*** HAL: hot air leveling**
(Risk of chip formation during the pressing because of inhomogeneous layer thicknesses, further there is strong changing forces due to different layer thicknesses)**
*** iAg: immersion silver**
(Risk of contact corrosion between the silver layer and pin and/ or sulfide corrosion of the exposed silver surfaces in the bore)**

3 Which surface finishes can be reworked?

Table 1: The following surfaces can be reworked:

iSn	qualified process since 2010 – smarttin® refresh process
ENIG	qualified process since 2014 – FinalClean process
ENEPIG	qualified process since 2014 – FinalClean process
OSP	establishment and validation of the process ongoing - OSP refresh process
iAg	possible with some restrictions (on request)
HAL	not abailable at APL Oberflächentechnik GmbH

4 How can iSn PCB to be reworked?

Due to the formation of intermetallic phases between iSn layer and the copper, the circuit boards by superposition or with contaminated iSn surfaces often can no longer be soldered. iSn post-processing method, the so-called refresh process, pcbs are treated with solder problems again back into a solderable state. The post-processing is a "tin on tin process", a previous stripping is not necessary. APL has achieved in the last five years a success rate of >95%. In APL, the process runs under the name "smarttin® refresh".

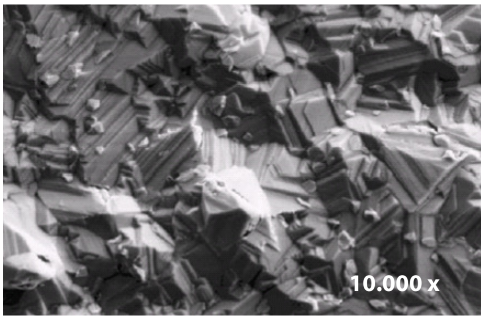

Fig. 1: Copper + smarttin®

5 What are intermetallic phases?

Intermetallic phases are formed by diffusion of at least 2 output metals (here copper and tin). The copper diffuses into the tin layer, two phases are formed Cu_6Sn_5 (tin rich) and Cu_3Sn (poor in tin). The intermetallic phases are formed already during the deposition of iSn. The growth of the phase depends on time and temperature, which is a natural physical process. After the maximum storage time, copper can diffuse to the tin surface through the growth of the intermetallic. Copper oxice can occur, thereby the pcbs will show a bad or even not solderable condition.

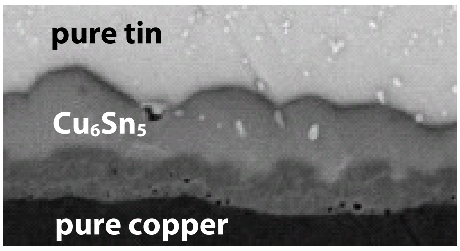

Fig. 2: Layer structure Cu/ Sn

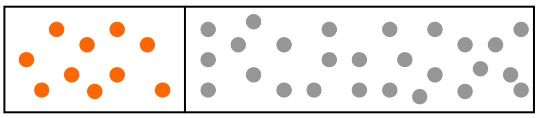

Fig. 3: Schematic description of the diffusion (left Cu, right Sn)[KAD15]

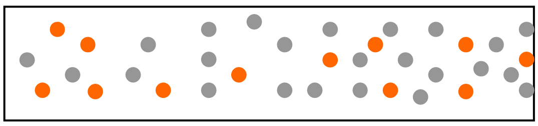

Fig. 4: Schematic description after the diffusion process[KAD15]

5.1 How does the process work?

During the smarttin® refresh process copper contaminations, on/ or in the original iSn layer, intermetallic phases and undefined tin oxides are removed and/ or dissolved. In parallel a fresh, pure tin layer (>0.1 µm - max. 0.3 µm) is deposited especially on such areas where copper is leached out. We speak in such a case from a "self-healing effect".

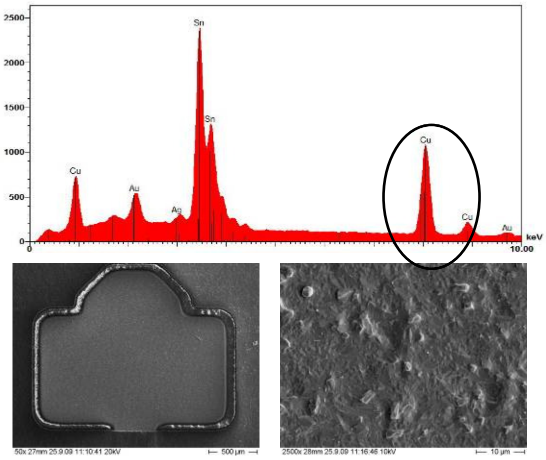

Fig. 5: Before smarttin® refresh[FRH09]

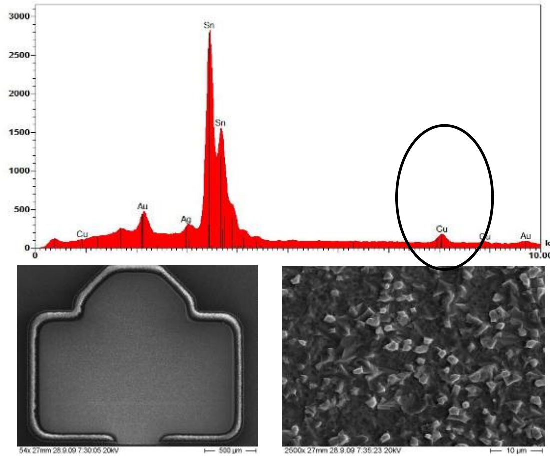

Fig. 6: After smarttin® refresh[FRH09]

5.2 Why and when is the smarttin® refresh process used?

The solder and press fit surface iSn can generally be refreshed under industrial conditions. smarttin® refresh process is economical, very effective and prevents the scrapping of the pcbs. It is from the European industry extensively tested and an approved process. It is carried out daily at APL´s plant for several European electronic companies. By this, pcbs with iSn are processed a second time in the iSn-line at higher speed, so that Sn-oxides and Cu_xSn_y intermetallic phases could be removed. [FRH09, FRH14, ATO11, BOS10]

5.2.1 The refresh is carried out if:

- The solder ability is not sufficient for multiple soldering processes
- The original iSn layer is overlaid (>6 resp. >12 months)
- The iSn thickness is below <0.80 µm/ <1.00 µm
- There was a copper redeposition on iSn at the first tinning process
- The ionic contamination is >1.55 µg/cm² NaCl-Equivalent

5.3 Performance tests Fraunhofer ISIT:

In the years 2009 and 2014, studies were conducted to refresh pcbs by the Fraunhofer ISIT. It was about the solderability of chemically tinned (smarttin®) printed circuit boards according to different load simulations and storage time (real-time). The first examination was carried out in 2009. In order to obtain a verification of the results, the study was repeated in 2014. The test circuit boards were coated at the APL with 0.8 micron Sn and stored without foil package and air conditioning in the metrology lab for 6 to 12 months. Subsequently, the pcbs were refresh, and then artificially aged. The artificial aging after the refresh process is defined by:

1 x Reflow Profile depart; Peak ~245 °C / without solder paste
2 x Reflow Profile depart; Peak ~245 °C / without solder paste
3 x Reflow Profile depart; Peak ~245 °C / without solder paste

After that, the boards were soldered and rated (3x reflow or reflow 2x/ 1x wave soldering; SAC305 solder; Peak ~245 °C). The hole filling through the sleeves and the wetting was evaluated. The hole filling must be at least 75% according to IPC-A610D. The hole filling in all tested pcbs was 100%. The SMD pads also point to a very good wetting.:

„With the smarttin® refresh process, overaged pcbs can be verifiably transformed into a solderable condition"[FRH09]

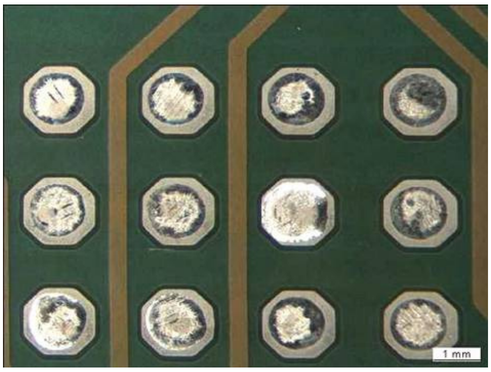

Fig. 7: 6 months storage + refresh + 3x reflow and wetting by wave soldering

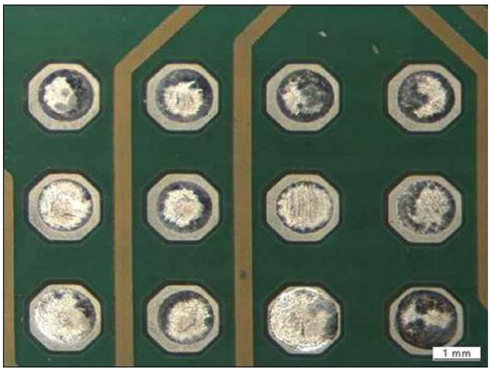

Fig. 8: 12 months storage + refresh + 3x reflow and wetting by wave soldering[FRH09]

6 How can ENIG and ENEPIG PCB to be reworked?

FinalClean is a new process to clean and active pcbs which show bad solderability. It is also used for overlaid pcbs. The FinalClean process is used for ENIG- (**e**lectroless **n**ickel/ **i**mmersion **g**old) and ENEPIG- (**e**lectroless **n**ickel/ **e**lectroless **p**alladium/ **i**mmersion **g**old) surfaces.

6.1 How does the process work?

The simplified process permits the treatment to be accomplished in horizontal mode of equipment. The process starts with a cleaning module with ultrasonic support. The cleaning module contains sulfuric acid and a special solution which is called *Aurotech FinalClean* (special solution). These two components will ensure a perfect cleaning of the pcb surface. After the ultrasonic cleaning module the pcbs will be rinsed with deionized water (DI water). The DI water shows a very low conductivity. The drying will be proceeded in a special drying module. After drying the pcbs will not show any residual moisture.

6.2 What happens on the pcb surface?

ENIG/ ENEPIG is a direct electroless deposition of nickel on copper. In addition to the nickel layer a gold deposition on top is necessary. The immersion gold process is a replacing mechanism of nickel and gold. During the immersion process one gold atom will replace two nickel atoms which lead in some cases to an incorrect positioning of the gold atoms.

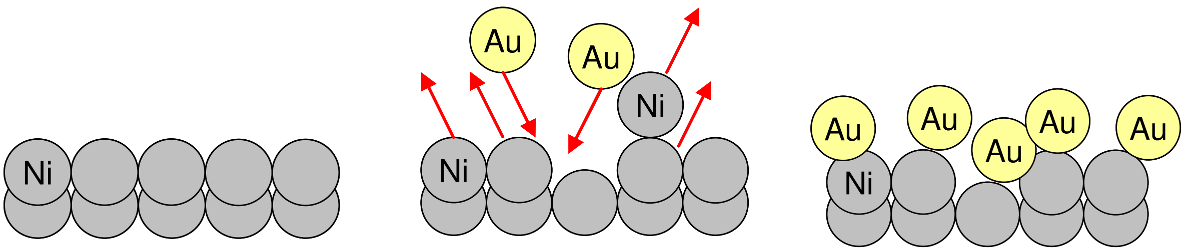

Fig . 9: Schematic description of the electroless Ni/ immersion Au process[ATM11]

After the Ni/ Au deposition, the pcbs will be assembled. During the assembly and other process steps the pcbs will be exposed to environmental influences like temperature and humidity. These influences lead to an artificial aging of the pcbs. Through the investigation to environmental influences nickeloxyd can occur on the pcb surface.

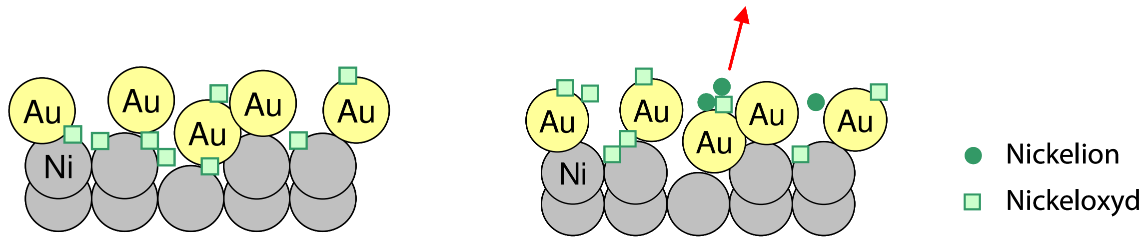

Fig. 10: Schematic description of the nickeloxyd formation[ATM11]

Nickeloxid will be transformed to ionic nickel through the investigation of corrosive solutions. Ionic nickel will diffuse to the surface of the pcbs. On top of the gold layer nickeloxyd will occur again. *Aurotech FinalClean* contains special chemicals. Those special chemicals will wet the surface all-over. The composite of *Aurotech FinalClean* and sulfuric acid will dissolve the nickeloxyd on the gold layer.

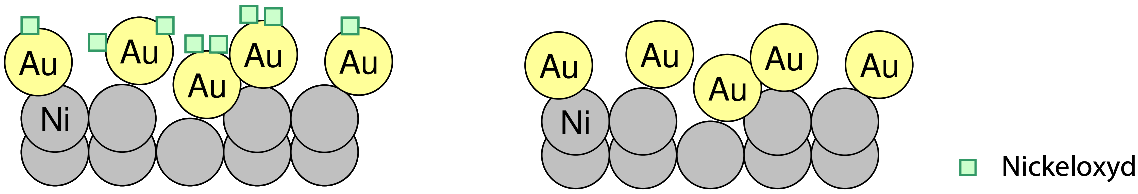

Fig. 11: Schematic description of the nickeloxyd resolution[ATM11]

Afterwards the pcbs will be rinsed in a 4 step DI water cascade which lead to an excellent rinsing result. After rinsing, the pcbs will be dried in a horizontal way.

The cleaning process activates the surface finish of the pcbs. After activating the surface it is possible to solder the pcbs again. The further assembly and solder process should be done directly after the FinalClean process. The storage time should be kept to a minimum in order to prevent oxidation on the pcb surface again.

6.3 How does it look in practice?

Pcbs (DC4299) were soldered before and after the FinalClean process. The following pictures show the improvement of soldering on 15 year old pcbs. The pcbs were stored in a cabinet without any packaging and special climatic conditions. Pictures were taken at APL with a stereomicroscope at the same magnification.

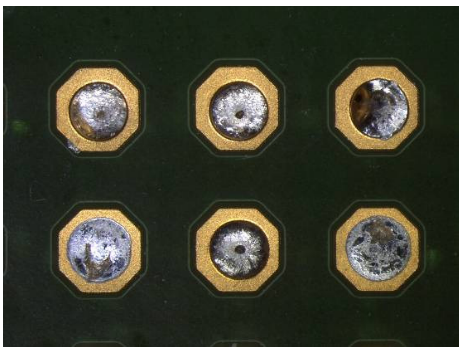

Fig. 12: Before FinalClean

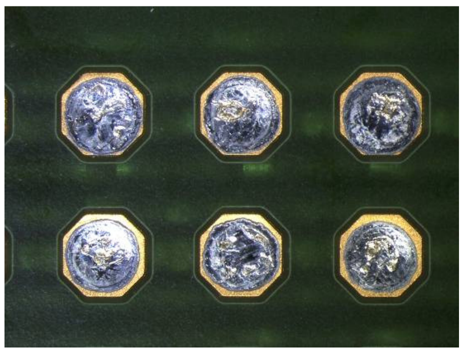

Fig. 13: After FinalClean

All the pcbs show a better soldering result. The hold filling of pcb drilling is much better than from the pcbs which are not cleaned before the solder process.

6.4 What is the result of analytical investigation?

In addition to the solder process, three pcbs of the same batch (DC4299) were analyzed on the surface. First of all the coating thickness was measured at APL with the x-ray XULM (Fischer GmbH). Afterwards the pcbs were sent to an independent laboratory to analyze the surface on a defined pad by XPS (x-ray Photoelectron Spectroscopy). After recovering the pcbs they were treated with the FinalClean process, packed in shrink-wrap foil and sent back to the laboratory for another analysis on the same pad again.

Table 2: Coating thicknesses of the analyzed pcbs

	NiP [µm]	Au [µm]
Test panel – 1	3.657	0.127
Test panel – 2	3.572	0.125
Test panel – 3	3.637	0.133

Table 3: Analytical results of the XPS-measurements in atomic percent[ATU15]

before the FinalClean process:

	Na	Ni	O	N	C	Cl	Br	S	Si	Au
Test panel – 1	-/-	4.19	19.96	1.89	59.43	0.08	-/-	0.98	1.19	11.14
Test panel – 2	-/-	5.49	22.65	1.89	56.97	-/-	-/-	1.81	-/-	9.34
Test panel – 3	0.36	3.43	19.68	1.73	62.84	0.23	0.08	1.00	0.98	8.43

after the FinalClean process:

	Na	Ni	O	N	C	Cl	Br	S	Si	Au
Test panel – 1	-/-	0.85	14.07	1.32	53.14	-/-	-/-	2.84	1.20	19.48
Test panel – 2	0.28	1.11	17.45	0.83	50.19	-/-	-/-	2.67	1.52	18.79
Test panel – 3	-/-	0.29	11.67	1.28	61.19	-/-	-/-	2.17	0.65	17.04

The x-ray photoelectron spectroscopy shows a significant reduction of Ni and O which approves the FinalClean operation mode. Nickeloxyd will be removed by the FinalClean process, which explains the improved solderability. This leads to a higher amount of gold on the surface after the FinalClean, elements like N, S and Si leached out of the shrink-wrapped foil.

7 Why must OSP coated printed circuit board to be reworked?

The OSP coating (organic surface preservation) is an organic coating from substituted imidiazolen based organic solution. The chemical reacts with the copper and forms a closely fitting protective layer. To coat with OSP horizontal systems are used. The usual layer thicknesses of the OSP protective layer can be between 0.2 microns and 0.6 microns. The maximum storage time is specified in the rule with 6 months. If the maximum storage period of 6 months exceeded, the layer may lose its protective properties and it is possible that copper is exposed selectively to the surface and got oxidized. Should such initial coating with OSP parameters not exactly adhered to, it may also lead to soldering problems with the film. The drying temperatures during the first coat must be strictly adhered to. The layer can be very "spongy" at a low temperatures, and too high temperatures, it will be "glassy". This can decrease the performance of the OSP layer. The working window of end surfaces as iSn or ENIG is smaller at OSP. An OSP refresh is always displayed in superposition and poor soldering performance.

7.1 How does the process work?

In the OSP initial coating the pcbs are usually cleaned and etched with exposed copper micro. The copper removal amounts to ≤1 - 2 microns. The OSP layer is then generated in a 2-stage operation. The process chemistry reacts directly with the copper and forms an approximately 0.2 to 0.6 microns thick organic layer of protection. After a very good rinse with clean water, and good drying process is completed. The processing is carried out in a horizontal plant technology.

7.2 What happens to pcb surface?

In OSP refresh, the existing OSP layer must first be removed and the copper are again etched weak micro. The copper removal amounts to ≤1.0 microns. The dissolution/ destruction of the OSP layer and recoating are carried out in a horizontal installations. The advantage of the OSP refresh is lower copper removal ≤1.0 microns, little stress on the base material and solder mask, as well as fast availability and cost savings.

8 What are the benefits of the smarttin® refresh-, FinalClean- and OSP refresh processes?

Technological and environmental benefits

smarttin® refresh:

- smarttin® refresh process is approved and used in Europe from the pcb-, automotive-, telecommunication-, OEM- industries as well as from many national/ international trading companies
- It can keep pcbs with poor iSn from being scrapped
- It is very efficient with a yield ≥95%
- It is quickly available. If necessary, smaller pcb quantities are back in 1-2 days at the assemblers place.
- The cleanliness of the refreshed pcbs can be reduced with our rinsing technology to 0.5 µg/cm^2 NaCl-Equivalent

FinalClean:

- Gold does not have to be stripped from the pcb surface which avoids chemistry waste
- It is a very simple process to activate the surface again. A replating process of nickel and gold is not necessary.
- The stress to the base-material and soldermask is very low
- High effective process
- Success rate ≥95%
- The cleaning process does not include toxic and environmental damaging materials like cyanide, a new plating process would include cyanide
- Less water consumption because of easier, shorter process
- Less energy consumption because of low process temperature

OSP:

- It is a very simple process to transform the pcb into a solderable condition
- The stress to the base-material and soldermask is very low

Economic benefits

smarttin® refresh:

- smarttin® refresh process has a very high cost saving potential. The refresh price is about 10% less in relation to the total pcb costs.
- In nearly >85% of the refresh jobs the refresh process is much cheaper than to produceand coat new pcbs.
- Example cost comparison: new pruchase of 6 fold multilayer (ML-6) and 4 fold multilayer (ML-4) in comparison to the smarttin® refresh process (Dimensions: 233.4 mm x 160.0 mm; amount: 1,000 pcs)
- Time factor: pcbs can be processed shortly

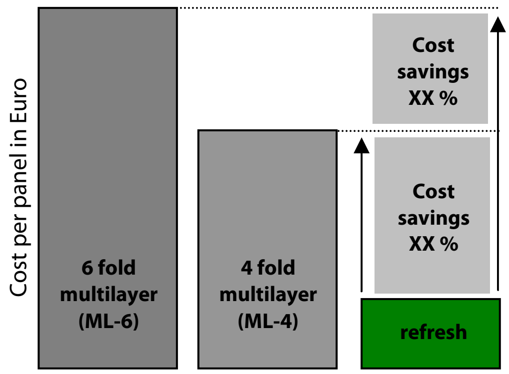

Fig. 14: Description of the cost relations

FinalClean and OSP:

- FinalClean: Cost efficient because of shorter process in compare to a new nickel and gold plating process.
- The existing pcbs can be reused, there is no need to order new pcbs
- Time factor: pcbs can be processed shortly

9 Summary

The APL technology stands in harmony with social and political guidelines regarding quality and environmental protection. APL will not only live up to these superimposed guidelines through our quality- and environmental management; we will also proactively contribute to sustainable preservation and improvement of ground, air, and water resources. APL wants to keep the balance right in terms of protection of the environment and place a great deal of value on preserving resources. In order to fulfill the own high requirements, APL has developed a comprehensive environment management system. This corresponds to the requirements of the international standard 14001:2004

Thanks to these three methods, it is now also possible to offer resource-saving surface treatments that add value to the customer to the environment. The use of hazardous chemicals will be omitted as far as possible in these processes. There is only a very small part of energy and water needed, as opposed to the production and coating of new circuit boards. A value-added three methods are provided to the customer, because a new acquisition of the circuit boards can be omitted and the circuit boards produced could be reused. This results show a large cost advantages in purchasing to the customer.

Sources Directory:

[ATM11] Atotech Deutschland GmbH; Mechanismus saurer Reiniger, 03.08.2011
[ATO11] Atotech Deutschland GmbH; PS report no. 060411; FIB Panels after 1 year storaged.pdf; 2011
[ATU14] ATU GmbH, Untersuchungsbericht 1412001, 06.02.2015
[BOS10] Bosch Automotive Electronics, Untersuchungsbericht 100121, Bosch_APL_iSn_Refresh; 2010
[FRH09] Fraunhofer ISIT Untersuchungsreihe; Bericht / Projekt 09-39456, 2009
[FRH14] Fraunhofer ISIT Untersuchungsreihe; Bericht / Projekt 11-397142, 2014
[KAD08] Kaschel Dirk, Plus 10/2008 - 2111-2114, Refreshen von chemisch Zinnschichten, 2008
[KAD15] Kaschel Dirk; Intermetallische Kupfer / iSn Phase (Diffusionsschicht); APL, 2015