Chapter 10

Circuit Manufacture

Introduction

Printed Circuits Boards consist of an insulating material forming the PCB substrate onto which conductive tracks are placed. Components forming the circuit are connected to those tracks.

There are two basic ways of producing printed circuit boards:

Conductive Layer Removal

A conductive layer, like copper, covers one or both side of the PCB substrate and the conductive layer is removed where there are to be no tracks. The metal can be removed using Chemical Etching, Milling or Laser Ablation.

For conventional Printed Circuit Board manufacture, chemical etching of copper clad printed circuit boards is the most common PCB manufacturing technique.

Depositing Tracks and Vias

In this method, the conductive tracks are deposited onto the substrate. For the production of plated through holes, this process is always required. The areas of the PCB where tracks are to be placed are processed through a series of chemical baths to chemically deposit a thin copper coating of approx 1.5 μm. Electroplating is then used to build up the thickness of the tracks or vias.

For Thick and LTCC circuits, the tracks are screen printed on a bare substrate material and then fired to make them permanent.

Printed Circuit Board Materials

Most Printed Circuit Board substrates are made by having thin copper sheets glued to an insulating material. The copper thickness depends on the applications. Typical thicknesses are 2, 1, ½ and 1/4 oz copper per square foot, which corresponds to a copper thickness of 70 μm, 35 μm, 18 μm or 9 μm respectively. The copper thickness together with the track width must satisfy the current carrying capability and attenuation requirement of the transmission lines. For printed circuit boards the copper that is not wanted is removed, by etching or milling.

Conventional PCB substrates

FR3

This is a substrate consisting of a mixture of paper pulp and epoxy. It is mainly used in low cost consumer products, and is not normally used for multilayer boards. FR3 has a Dielectric Constant of approximately 2.3 but this is not controlled and the loss tangent is not specified. FR3 is less tolerant to scorching heat than FR4.

An advantage of FR3 and similar paper based laminated are that holes and slots can be punched. This laminate is not recommended for RF designs. At JCU we have been using FR3 for PCB's since the fibreglass used in FR4 is carcinogenic. Very special precautions need to be made to ensure that the particles produced by the PCB milling process are not inhaled. FR3 is safer in that respect. FR3 is normally light brown with a smooth texture.

Figure 1 shows a typical low cost AM/FM radio manufactures using FR3 circuit board. The advantage of being able to punch out a particular shape can be seen..

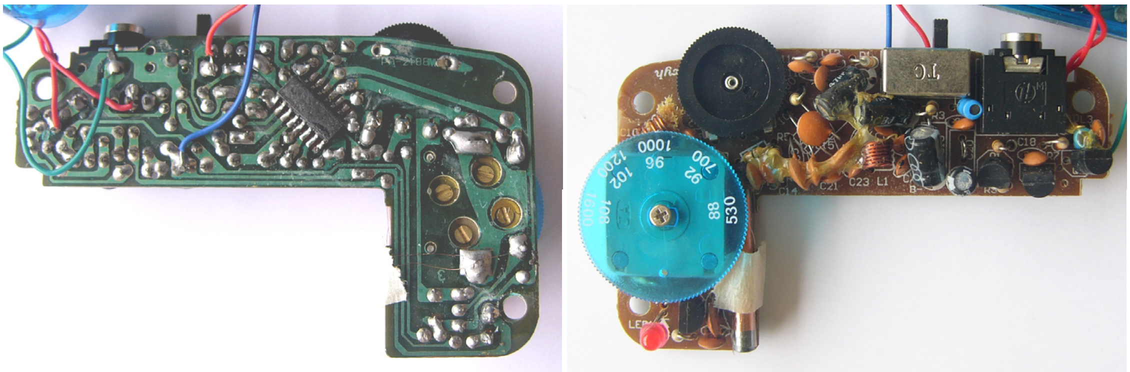

Figure 1 FR3 Printed Circuit Board for a low cost AM / FM radio.

FR4

This is a substrate consisting of a mixture of fibreglass and epoxy. It is suitable for professional quality PCB's and has the following properties:

Dielectric Constant	4.7 at DC decreasing to 4.4 at 10 GHz.
Dissipation factor (Tan(δ))	0.014
Q at 50 MHz	95
Max Temperature	130 °C

Table 1. FR4 Properties.

FR4 PCB's can be made with many layers. Typical PCB's have 4 layers, with one layer being earth, one layer being the supply voltage and the outer layers containing the tracks that are connected to the IC's and passive components. FR4 is normally pale green and semi-translucent, often this can be seen at the left edge of the board as in figure 2. The large component at the left edge of the board is a ceramic diplexer.

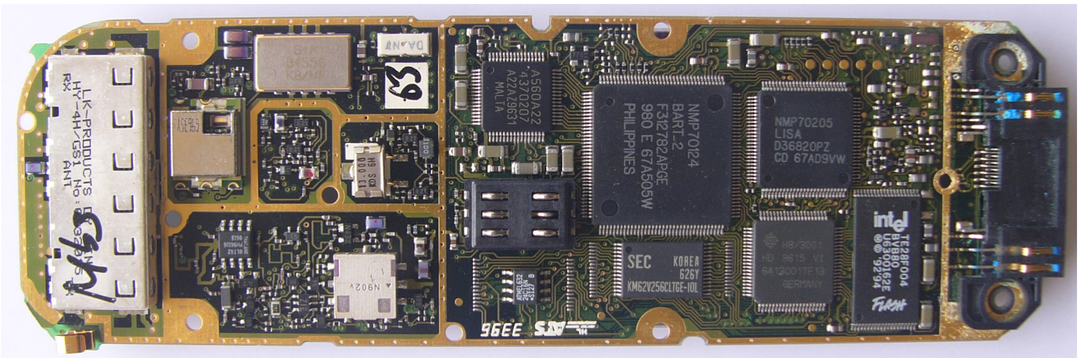

Figure 2. FR4 Multilayer Printed Circuit Board. (Nokia Mobile Phone).

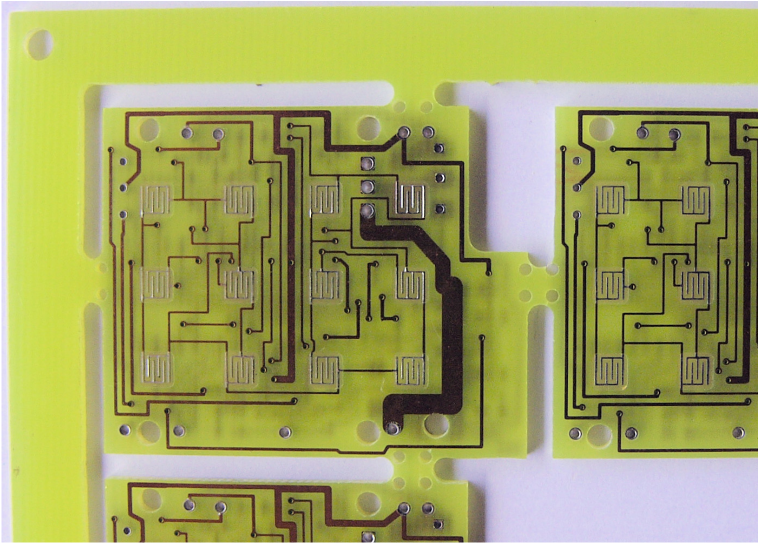

Figure 3. FR4 Printed Circuit Board, showing cut-outs. (JCU Layouts).

Normally many printed circuits are etched together on one sheet as shown in figure 3. Cut-outs are provided to enable the individual circuits to be removed easily.

Microwave and RF printed Circuit Board Materials.

FR3 and FR4 can be used for PCB's to 1 GHz, however above these frequencies the losses become significant. To avoid reflections, at microwave frequencies many of the connections are made using transmission lines of a specified impedance. For FR3 and FR4, the dielectric constant and thickness varies from one batch to the next, so that the characteristic impedance of transmission lines on the PCB also varies. This is not suitable for many RF designs. Rogers Corporation (http://www.rogerscorporation.com), are a large RF PCB substrate material manufacturer and make the following laminates:

RT/duroid 58X0

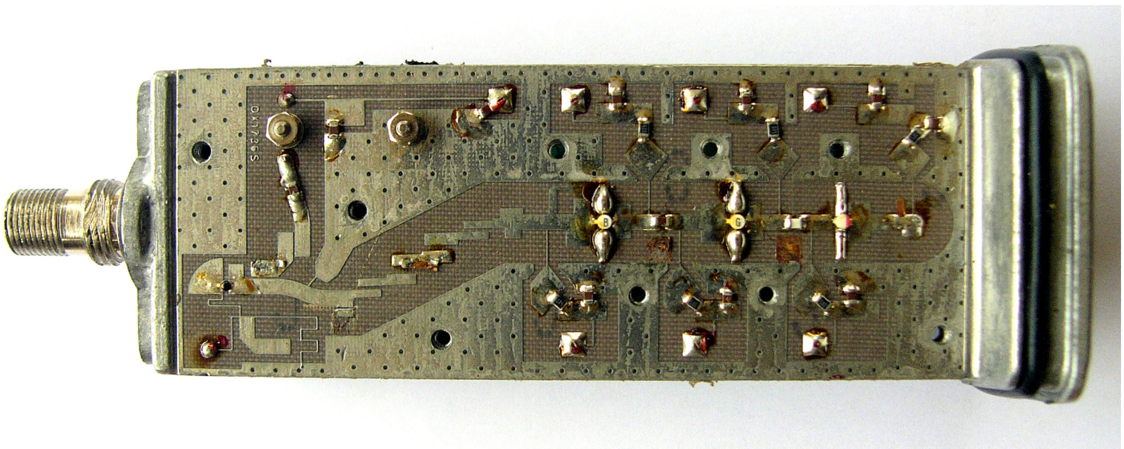

Figure 4. Ku band (12 GHz) Low Noise Converter (LNC) using RF/duroid (Norsat).

RT/Duroid 58X0 is a Glass microfiber reinforced PTFE (Teflon) composite. The two products in this family are RT/duroid 5870 (ε_r =2.33) and 5880 (ε_r =2.2) materials.

These are the oldest Rogers products and were developed in the 1960's. The low dielectric constant and the low dielectric loss of typically 0.0005 make these materials best suited for high frequency/ broad band applications where dispersion and losses need to be minimized. A high dielectric constant reduces the size of the circuit. For frequencies above 20 GHz circuits with a high dielectric constant become too small. Because of their low water absorption characteristics, RT/duroid 58X0 is good for applications in high moisture environments. The LNC of figure 4 is located at the focal point of antenna, which is sometimes very hot, sometimes cold and sometimes very wet. RT/duroid is thus a good choice for the PCB material. RT/duroid is flexible and for thin laminates requires care in mounting them securely. RT/duroid is normally brown and the woven glass mat can clearly be seen.

RT/duroid 6000 (PTFE/Ceramic Laminates)

There are three laminates in this family RT/duroid 6002 (ε_r =2.94), RT/duroid 6006 (ε_r =6.15) and 6010LM (ε_r =10.2). The thermal coefficient of the laminate matches that of copper. The laminates are more rigid and thus easier to mount. This product is ideal for applications in thermal changing environments (space). Multilayer boards are possible with this laminate. RT/duroid 6000 is a military specified substrate and thus very expensive.

RO3000 and RO3200 (PTFE/Ceramic Laminates)

The RO3000® materials family consists of four grades, RO3003™ (ε_r =3.0), RO3035 (ε_r = 3.50), RO3006 (ε_r =6.15) and RO3010 (ε_r =10.2). These laminates are aimed at a commercial market and are a lot cheaper than the corresponding RT/duroid 6000 laminates. The temperature coefficient of expansion matches that of copper and FR4 in the X-Y plane. Multylayer RO3000/FR4 boards can easily be made.

RO4000

The RO4000 series is a woven glass/ ceramic loaded thermoset plastic resin Laminate and were developed as a low cost commercial laminate, that is specially suited for multilayer boards using RO4000 and FR4. The boards have a glass transition temperature (Tg) >280°C and can thus be used with conventional PCB processing techniques, such as wave soldering and surface mount techniques. Unlike PTFE based microwave materials, no special through-hole treatments or handling procedures are required. Therefore, RO4000 material circuit processing and assembly costs are comparable to epoxy/glass laminates. RO4003C has an ε_r of 3.38 and RO4350B has an ε_r of 3.48.

Multilayer Boards

There are many applications that require RO4003 material to be bonded. One example is a stripline RF circuit one other is a RO4003/FR4 multilayer board for both RF and digital signal processing. Several types of adhesives can be used depending on the laminates to be bonded. Figure 5 shows a typical construction requiring an RO4003 stripline circuit bonded to an FR4 multilayer construction.

For constructing a stripline circuit by using two RO4003 laminates, a high frequency bond-film like the RO4400 "prepreg" sheet from Rogers is required. The bond-film should have a dielectric constant similar to the laminates to be bonded and the loss tangent of the bond-film should match that of the laminates.

For bonding of an RF substrate to an FR4 substrate, a higher loss bond-film can be used. In addition the dielectric constant is not critical. A good low cost option is to use the standard bond films used for FR4 for making multilayer PCBs.

For bonding the laminates are located precisely using locating pins and the boards are clamped at a pressure of 2.8 MPa and a temperature of 225°C for more than one hour. Since air has an $\varepsilon_r = 1$, it is important to prevent air bubbles, particularly when high dielectric substrates are used.

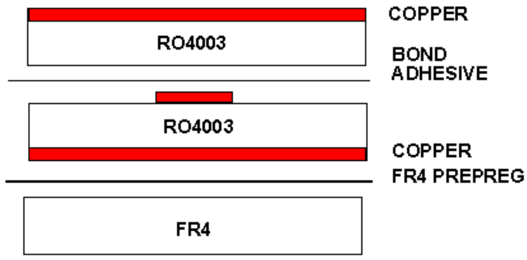

Figure 5. Bonding laminates to form a multilayer board (Courtesy Rogers Corporation).

Non-Clad Substrates

Companies like Morgan Electro Ceramics, (http://www.morganelectroceramics.com) and Coors Ceramics (http://www.coorsceramics.com/) produce dielectric materials that can be used for RF substrates. The dominant substrate material is Alumina, but other materials with higher dielectric constants are also used.

These non-clad substrates are used for precision microwave circuits using Thin film techniques and for low cost thick film circuits at both RF and low frequencies.

Companies like UltraScource Inc. (http://www.ultra-source.com/company.html) specialise in the design and production of thin film circuits, predominantly for space and military applications.

Companies like Hybrid Electronics, (http://www.hybrid-electronics.com/index.htm) specialise in the design and production of thick film circuits.

As an indication of the types of materials available and their RF applications, the following is a list of some of the RF and Microwave materials from Morgan Electro Ceramics.

D6

This is a low permittivity material that is used predominantly for supports for dielectric resonators however it can be used effectively for substrate applications and also for GPS patch antenna applications. The dielectric constant is 6.

Alumina Substrates

Alumina is the dominant substrate material for Thick and Thin film applications. The dielectric constant is close to 10, with both the dielectric constant and cost increasing with the purity of the substrate. For lower cost applications, Thick film substrate materials with 85% to 95% purity are used. In many cases these substrates are used "as fired" for low frequency applications or ground flat for low cost microwave applications. For precision, low loss microwave applications 99.5% to 99.8% purity substrates are used and these may be ground and polished in order to remove any surface irregularities and ensure a constant thickness of the entire substrate. Alumina is ideal for demanding low loss applications, such as substrates for thin film circuits.

D30

Morgan Electro Ceramics D30 dielectric composition has a dielectric constant of 30 and a Q_o typically 50,000 at 2 GHz and 15,000 at 10 GHz. The material is specifically designed for telecommmunications filter applications where a high Q is an absolute requirement. This ceramic is predominantly used for resonators.

D43

Morgan Electro Ceramics D43 material has a high $\varepsilon_r = 43$, with a Q_0 typically 30,000 at 0.9 GHz and 22,000 at 2.0 GHz. This material is very good for resonators and filters for mobile radio applications.

D88

Morgan Electro Ceramics D88 material offers a high dielectric constant of $\varepsilon_r = 88$. The material offers maximum miniaturisation whist still offering a temperature coefficient that is tuneable through zero.

Manufacturing

RF printed circuit boards can be designed in a similar way to conventional PCBs, so that programs like Protel, (Altium) can be used to produce the circuit board layout. For circuits like stripline filters, and other RF circuits, where the characteristics and lengths of transmission lines are critical, then Software like Microwave Office or ADS is required. All of these programs can produce Gerber Plot files, which are used by either commercial PCB manufacturers, or by the milling machines used to produce prototype RF boards. The Gerber Plot files can also be imported into Protel to produce a combined RF and conventional PCB. Such technology is required for the Multilayer boards shown in figure 5.

Normal PCBs are protected using a soldermask. Unless the thickness and the dielectric constant of the soldermask is taken into account in designing the RF circuit, it should not be used. To prevent corrosion, Gold plating can be used on the RF circuits.

Manufacturing using PCB milling machine

At JCU we use a milling machine to produce PCB's. The ceramic filler used in the RO4003 substrates are much more abrasive than the epoxy, paper and fibreglass used in the FR3 and FR4 type substrates. As a result the cost of milling bits is a major cost in the production of those boards. For milling RT/Duroid, the substrate needs to be thick enough to be able to be held flat as part of the milling process, so that the more rigid RO4003 substrates are the best to use in PCB milling machines.

Layout Hints

Many PCB manufacturers can etch to an accuracy, such that the thinnest tracks can be 0.2 mm wide and have a minimum spacing of 0.2 mm between tracks. Many of the modern ICs have 0.5 mm pitch of the leads on the IC. The 0.5 mm track pitch is close to the maximum resolution of present day PCB manufacture.

The minimum drill size that should be used is 0.5 mm diameter. The minimum pad size for vias or leaded components is the drill size plus 0.5 mm, however small pads will lift off easily when components are soldered to them. For through-hole components use a drill size that is approximately 0.1 mm larger than the lead size and use a pad that is approximately 1 mm larger than the drill size. The recommended hole size for a standard leaded resistor or capacitor is 0.9 mm with a pad size of 2 mm. To ensure that that the circuit can be changed if needed, for prototyping boards a bigger pad size is advisable

For milling of PCBs, the tolerances are slightly different. A minimum recommended track size is 0.5 mm, with a 0.2 mm minimum spacing. (0.2 mm wide milling bits are the smallest milling commonly used). Having a 0.25 mm spacing ensures that 2 milling cuts are made, thus reducing the chance of short circuits. For short track lengths, track widths of 0.2 mm are possible, however such thin tracks should be avoided for long tracks since the milling machine positioning accuracy is 0.1 mm. When milling circuit boards, as much groundplane as possible should be left on the board, to minimise the amount of milling that has to be done and minimise the milling time and cost.

Thin Film

Thin film circuits are normally produced by firstly applying a very thin (1 to 2 micron) seed-layer of gold to an alumina substrate. A photographic mask is then applied to the places where no track is required and the tracks are then electroplated to the correct thickness of several tens of microns. The photo-resist is then removed and a quick etch removes the seed-layer, where no tracks should be, but does not completely remove the tracks. Solder-pads are then added where required, together with dielectric dams to prevent the lead from solder leaching into the gold. The polished Alumina used for the Thin film substrates has a very low dielectric loss. Thin film circuits can be made to better accuracies and smaller sizes than etched PCB's

Thin film techniques are expensive compared with other techniques and are thus used for circuits where the other techniques cannot be used. Examples are the 20 Watt Class C 800-900 MHz amplifier hybrid, shown in figure 6. The whole circuit including heat-sink is 56 mm long. The bond-wires for the large output transistor on the right can clearly be seen.

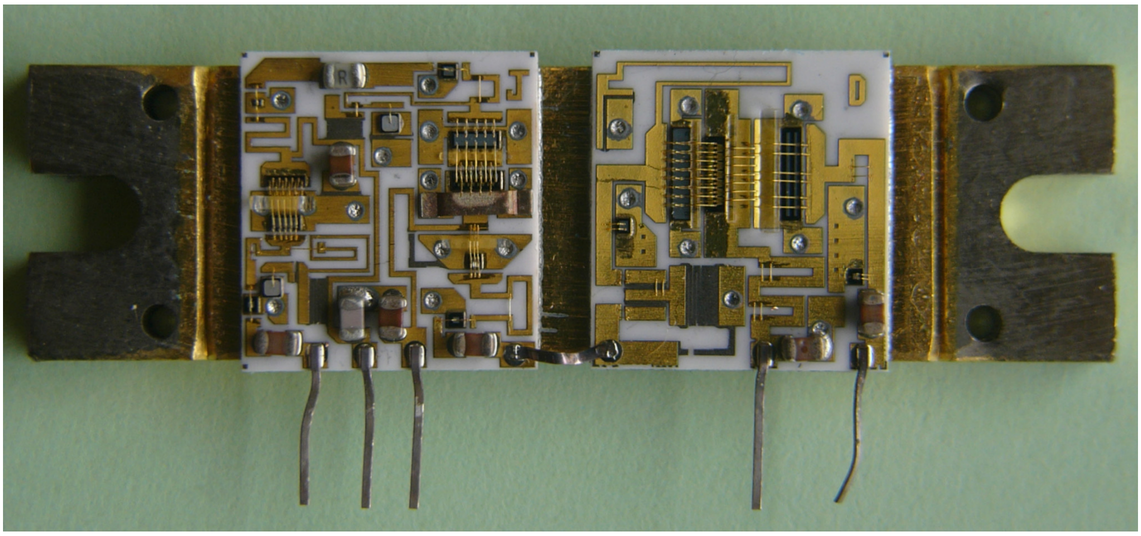

Figure 6. 20 Watt Class C Amplifier Hybrid MHW820 (Motorola).

Thick Film

Thick film techniques have a slightly higher circuit set up cost as several screen masks need to be made, but they have a lower increment cost as no etching or milling of the PCB's are required. In thick film techniques, a substrate, which normally is an alumina substrate, has conductive, resistive or insulating inks applied to them using precision screen printing technology. The substrate is then fired at typically 850 °C to harden the tracks and thus provide the correct circuit connections. Dielectric layers can be made, so that tracks can cross each other. For a multilayer track circuit, several separate firings may need to be made. Similarly capacitors can be made by firstly using a conductive ink to form the one electrode and firing that. Then a dielectric layer is applied and fired, followed by another conductive layer for the other electrode. Resistors can be made using resistive inks and inductors can be made by printing conductive spirals. This results in a high-density circuit board of low cost. The low thermal coefficient of expansion (TCE) of ceramic materials guarantees mechanical stability and is closely matched to that of ICs for bare chip assembly.

The screen has a photographically produced mask applied to it, with the screen being open where the tracks need to go and being blocked everywhere else. The screen is placed in contact with the substrate and the ink pastes are applied with a squeegee to the screen. The inks can also be applied using other techniques, such as spraying, dipping using the appropriate screening techniques to ensure that the inks are only deposited where they are supposed to be.

Figure 7 shows some typical thick film circuits. The circuit on the left shows that thick film circuits can have a high component density. After the circuit is tested, the whole circuit is protected with a protective coating as shown on the right of figure 7. This makes thick film circuits very suitable for hazardous or wet environmental conditions.

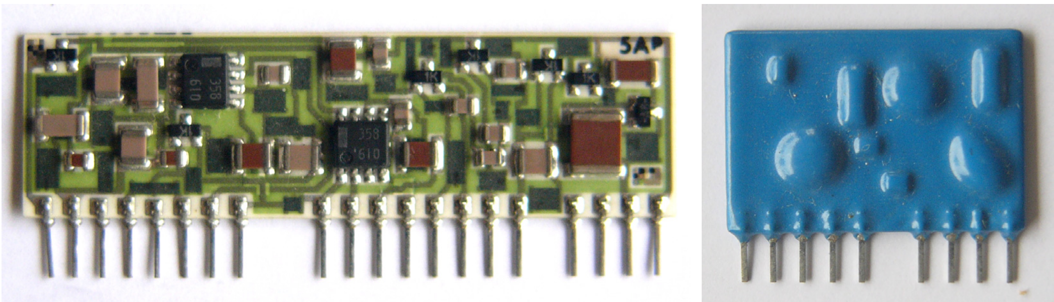

Figure 7. Thick film circuits before and after a protective coating has been applied.

Screens

The ink is normally applied using a squeegee. When the ink is applied, the thickness of the ink is close to the thickness of the screen. By varying the thickness of the screen the thickness of the tracks can be varied.

The screens used for the printing can be made from stainless steel, with a typical fibre diameter between 30 and 100 microns, a 50 to 224 micron mesh aperture and 65 to 220 micron mesh thickness with an aperture of 39 to 47%. Stainless steel screens are used for large production runs.

Screens can also be made from polyester. Polyester screens have a shorter life but are cheaper, and are thus used for smaller production runs. The screens have a typical fibre diameter between 35 and 110 microns, a 43 to 185 micron mesh aperture and 60 to 160 micron mesh thickness with an aperture of 30 to 40%.

Pastes (Inks)

Conductivity: The conductivity of the pastes is given in terms of milliohm/square. The resistance of a printed conductor of 10 mm x 10 mm, is the same as the resistance of a conductor of 100 mm x 100 mm, as long as the thickness is the same.

Thick film inks contain precious metals, glass, and/or ceramic powders dispersed in an organic medium. Specialized compositions create conductors, resistors, and dielectrics for a variety of applications and operating environments. Table 2 shows the conductivity of various conductive inks. Copper inks are difficult to use since they require an oxygen free atmosphere in the ovens, to prevent oxidisation. The palladium inks are used for solder pads.

Conductor Material	Sheet resistance (mΩ/square)
Copper	1.5
Silver platinum	2.0
Gold	3.0
Silver palladium	30
Gold palladium	30

Table 2. Comparison of sheet resistance of conductive pastes.

RF applications of Thick Film Circuits

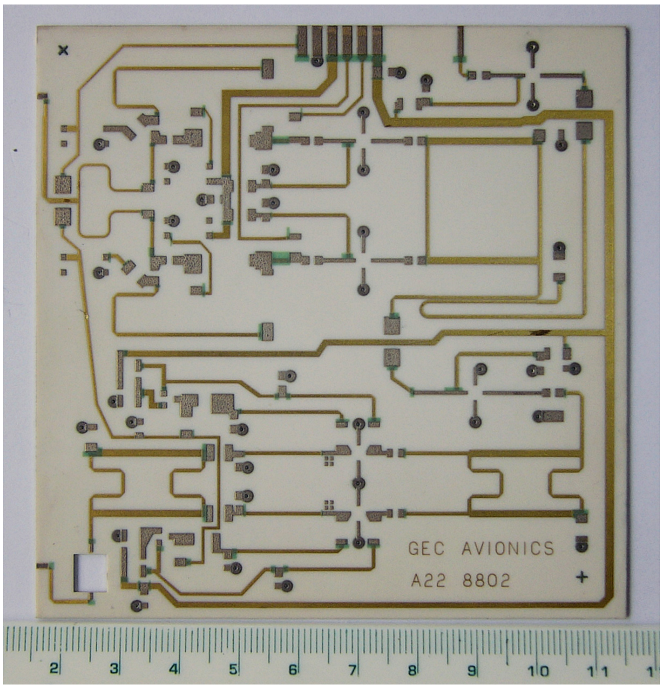

Figure 8. Thick film RF circuit on alumina. (GEC Avionics, designer Kikkert).

The alumina substrate has a very low loss, and that is why it is used for thin film circuits. The application of the required tracks using thick film techniques is a lot cheaper than doing the same using thin film techniques. For accurate transmission line widths, fine screens need to be used and the spreading of the inks needs to be allowed for. Figure 7 shows a circuit for a low noise amplifier and I-Q gain control circuit for a receiver beam steering application at 1.6 GHz. The circuit is 100 mm x 100 mm in size.

Thick film techniques are also used to produce patch antennae for mobile radio systems. In many cases the antenna needs to be shaped to conform to the contours of the mobile phone. Thick film techniques are also used to coat the ceramic block of the dielectric filter used as a diplexer shown on the left of figure 2.

Low Temperature Cofired Ceramic (LTCC)

The Low Temperature Cofired Ceramic (LTCC) technology is a means of producing multilayer circuits at a low cost. The technique is similar to thick film techniques, except many layers of ceramic are used, each of which has conductive, dielectric and / or resistive pastes applied to them. The ceramic sheet is called a green sheet, since the ceramic still has not been fired. This normally is supplied on a tape. The printed sheets are then laminated together and fired in one process. By means of vias, connections can be made from one layer to another, as shown in figure 9.

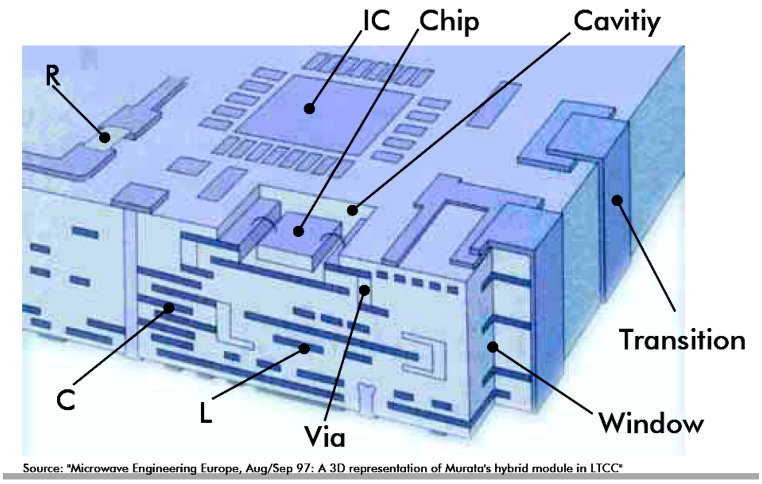

Figure 9. Typical LTCC circuit.

The low firing temperature for the ceramic green sheets allow low resistivity conductors like silver, gold, copper and alloys with palladium and platinum to be used.

The pastes are printed on the greensheet with the help of a conventional thick film techniques. Special thick-film pastes must be used since the ceramic shrinks 10 to 15 percent in x/y-axis and about 10 to 45 percent in z-axis.

It is also possible to integrate passive elements like resistors, capacitors and inductors into the substrate. Resistors are processed with the help of special pastes, which have to be printed on the tape just like conductor lines and are cofired. Capacitors and inductors can be made by forming the conductive lines into plates or coils respectively. LTCC technology permits the construction of stripline as opposed to microstrip circuits, thus reducing radiation and permitting the construction of couplers with a high amount of coupling. Combining that with the ability to include active devices on the same small circuit opens up great possibilities.

For further information see: http://www.ltcc.de/.