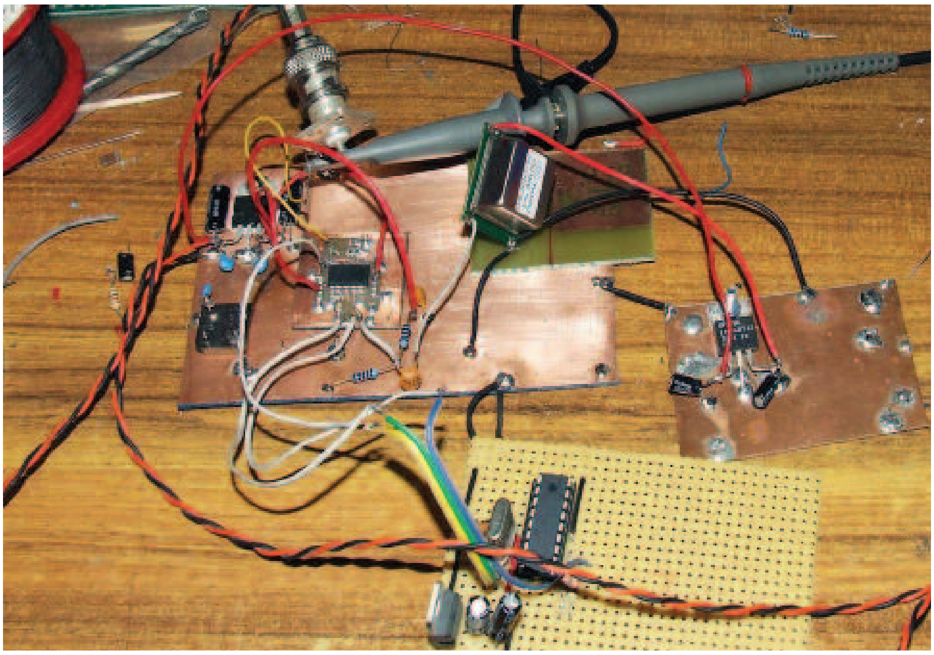
Homebrew

Working with surface mount ICs.



What you don't usually see! This month's second project, a DDS board (middle) under test with a 100MHz reference oscillator (silver box, right) and PIC driver circuit (veroboard, foreground).

SCARILY SMALL. A fine-pitch IC with several dozen pins and a pin-to-pin spacing of much less than 1mm can be a terrifying sight for the amateur constructor. Commercial manufacturing, reworking or repair of circuits using such devices calls for specialist knowledge and expensive equipment. However, with a little bit of practice, it is possible for the amateur to work with SM ICs using simple and readily available tools.

In many respects, SM devices are easier to use than their through-hole counterparts. Soldering SM ICs is often a single pass operation where all pins are soldered at once. De-soldering and removing SM ICs can be very much easier than removing through-hole DIP devices. SMT soldering is not a black art that is only practised by a few highly-skilled wizards. Regular readers will know that I am not very skilled in the fine art of soldering: I know the theory, but I often come unstuck when it comes to the practice. Some of my homemade boards feature excessive solder, solder bridges, bad connections, dry joints… just about everything you would expect to find in a list of common soldering defects. In short, if I can successfully solder fine-pitch SM ICs, you can do it too.

MAKING THE BOARDS. SMT PCBs are designed and made using the same methods that we have previously used for through-hole PCBs. A little more care is required because of the small track widths and pad spacings used in SMT layouts. However, the basic process is exactly the same. Simple single sided layouts are very convenient because of the lack of holes and 'vias' (drilled inter-layer connections). Complex double sided layouts with a large number of vias are more difficult, especially if you are using hand soldered wire vias. Careful consideration at the design stage will minimise the difficulties encountered in the later stages of making the PCB.

Unless there is some compelling reason to use a double sided layout, you should try to design your circuit using a single layer. If your circuit is so complicated that a second layer is required, use large areas of the bottom layer as a ground plane or for power supply tracks. Try to avoid putting large numbers of signal tracks on the second layer because this will require a large number of hand soldered vias. Don't put vias underneath ICs. This is common practice on commercial PCBs with electro-plated via holes, but it is almost impossible on a homemade PCB. Try to avoid using very small vias because they require ridiculously small drill sizes. Even if the PCB will not be mounted using screws, it is a good idea to put holes in the corners so that they can be used as a guide for aligning the top and bottom layers.

Through-hole boards are usually made using wave soldering equipment. All components are soldered in a single pass by running a wave of molten solder across the surface of the PCB. A standard soldering iron can easily be used for rework or repair. Wave soldering is an ideal method of soldering through-hole boards, but it is not particularly well suited to SMT.

The basic principles of SMT soldering are the same as any other type of soldering. Flux and solder are applied to the joints and the temperature is increased until the solder melts. Most commercial SMT boards are made using 'reflow' soldering.

In a typical reflow system, the solder and flux are applied to the PCB pads in paste form. The paste can be applied through a metal mask using a process similar to silkscreen printing or it can be applied one pad at a time using a computer controlled paste dispensing nozzle. The components are then placed on the board using a pneumatic pick-and-place system. SM components are usually supplied in reels of plastic or paper tape with perforations along the edge of the tape. The perforations are engaged by the feed system of the pick-and-place machine. Once the components are placed, the entire board is heated so that all of the components are soldered at once.

Figure 1 shows a typical thermal profile for reflow soldering. There are four stages or 'zones' in the process, beginning with the initial ramp (or preheat) stage where the board is heated at a rate of a few degrees per second up to about 150°C. The board is held at this elevated temperature during the thermal soak stage, which is typically more than one minute. During the soak zone, the flux in the solder paste is activated and all components will reach an equal temperature.

For the reflow stage, the temperature is increased to above the melting point of the solder. At this point, the liquid solder can flow around component leads and pads to

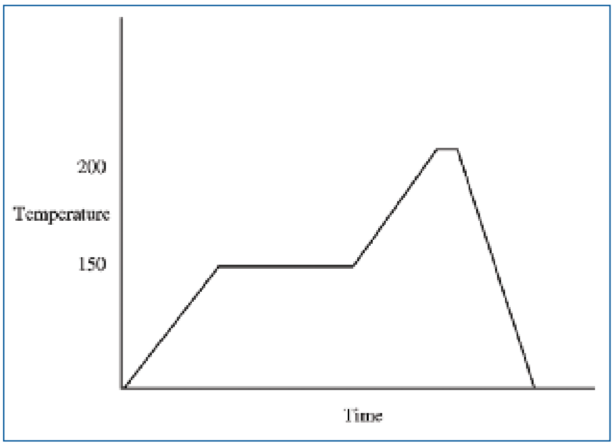


FIGURE 1: Typical thermal profile for reflow soldering.

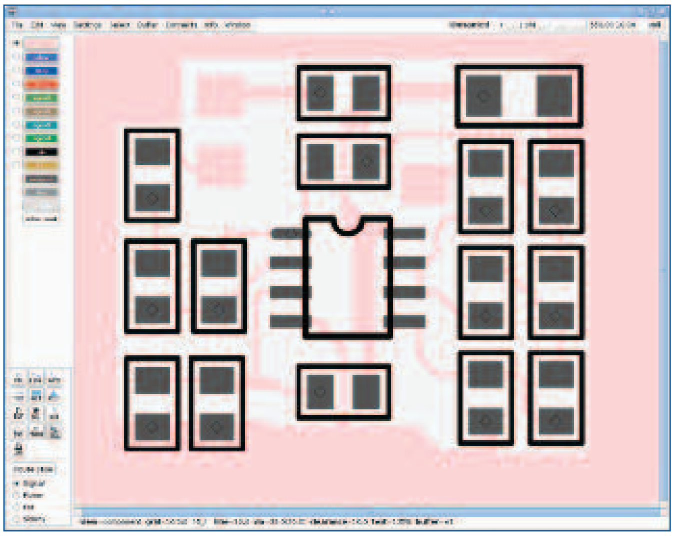


FIGURE 2: Drawing the PCB layout.

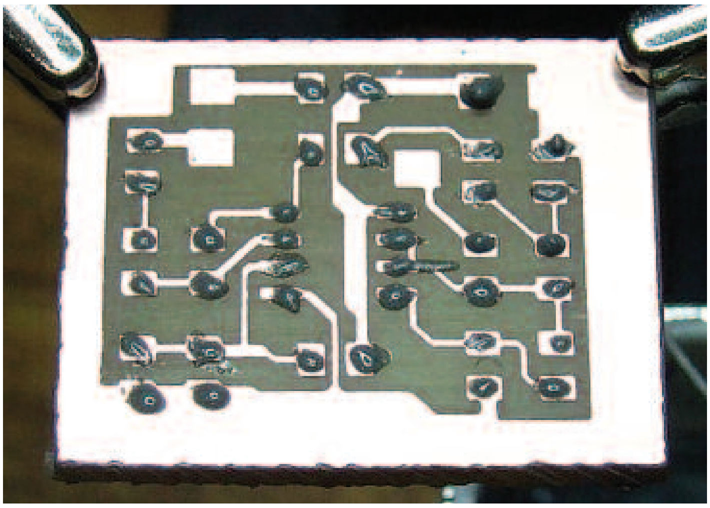


PHOTO 1: Blobs of solder paste have been applied to the PCB pads using a cocktail stick.

form a good solder joint. Modern lead-free solder has a melting point well above 200°C.

The final stage is the cooling zone where the PCB is cooled back down to ambient temperature. The exact temperature, ramp rate and time duration of each zone depends on the specifications of the solder/flux paste, the components and the PCB laminate. The reflow zone is the most critical period. The high temperatures required for lead-free soldering are quite close to the absolute maximum limits for many semiconductors and PCB laminates. Some components, like plastic capacitors and PCB mounted plastic connectors, are particularly prone to heat damage. A suitable profile must be hot enough to give good solder flow, but the duration of the reflow zone must be kept short so that components are not damaged by excessive heat. From start to finish a flow soldering process typically takes several minutes.

SMT REFLOW SOLDERING TOOLS. The amateur must find cheap and readily available replacements for the specialised and very expensive SMT equipment used in industry. The amateur pick-and-place machine is a simple pair of tweezers. Ordinary metal tweezers are suitable for most SMT work. Plastic or bamboo tweezers can be used for handling more delicate components. Solder paste is available in syringe style dispensers. A plunger is used to force the paste through a small metal or plastic nozzle. This a very convenient way of applying solder paste. Unfortunately, solder supplied in this type of dispenser has a relatively short shelf life. At room temperature, the shelf life is typically about one month. To achieve a longer shelf life, the paste must be stored in a refrigerator. This is very inconvenient for the occasional constructor. Solder paste supplied in small plastic pots can usually be stored for long periods of time at room temperature. This type of solder/flux paste can be applied to the PCB pads using a small pointed tool like a wooden cocktail stick or toothpick. I use Lodestar L309 paste and a cocktail stick as a dispenser. The two most common types of reflow heating used by home constructors are hot gas and infrared. In the world of kitchen table technology, hot gas generally means hot air. A cheap hot air gun can be used for reflow soldering. The air temperature at the nozzle of a standard DIY type of hot air gun is really too high for reflow soldering. For best results, use a gun which has a high/low switch to control the temperature and airflow rate. A skilled operator can control the PCB temperature by adjusting the spacing between the gun and PCB. This technique can be very effective for reflowing an entire PCB. This type of gun is not suitable for rework or repairs where it is necessary to remove and replace individual components. Special hot air rework tools are now available at reasonable cost. Hot air SMT rework stations can be bought new for less than £100. The amateur equivalent of the infrared reflow oven is a toaster oven of the type that is used for making toasted sandwiches (at least, that was used for that before being put to a far better use!). As with the hot-air gun, using an unmodified toaster oven requires great skill on the part of the operator. Some amateurs have built microcontroller based temperature control systems that are used to control the current to the oven heating elements. The best designs use a thermocouple as a temperature sensor. This type of oven can give excellent results. Obviously, modifying mains powered appliances is potentially very dangerous. A modified oven with a smart control system can produce excellent results. Some amateurs have also used an electric hot plate or even an inverted clothes iron for SMT reflow.

A SIMPLE SMT PROJECT. As usual, we will start with a relatively simple project. This month's first project is a simple microphone amplifier and audio LPF. Through-hole versions of this circuit have been used in several previous projects **[1]**. The new PCB layout was drawn using PCB software on a Linux PC and is shown in Figure 2. This is a very simple single sided layout. The board was produced using photographic methods as previously described [1]. Instead of using the laser printer, the artwork was printed on Stephens brand OHP transparency film using a HP D5160 photo quality inkjet printer. The resulting positive artwork is at least as good as the best laser printed artwork that I have used for previous projects. Two sheets were printed and stacked to make sure that the black track pattern was absolutely opaque. After the board was exposed, developed, etched and cleaned. The solder paste was applied to the PCB pads. **Photo 1** shows how

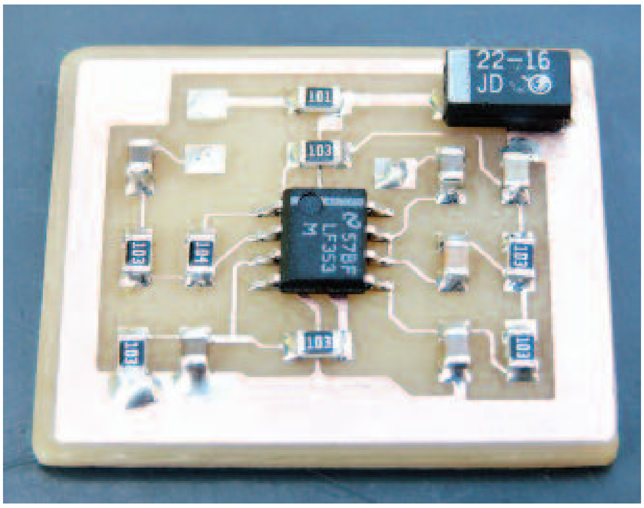


PHOTO 2: The finished amplifier PCB, built entirely of surface mount devices.

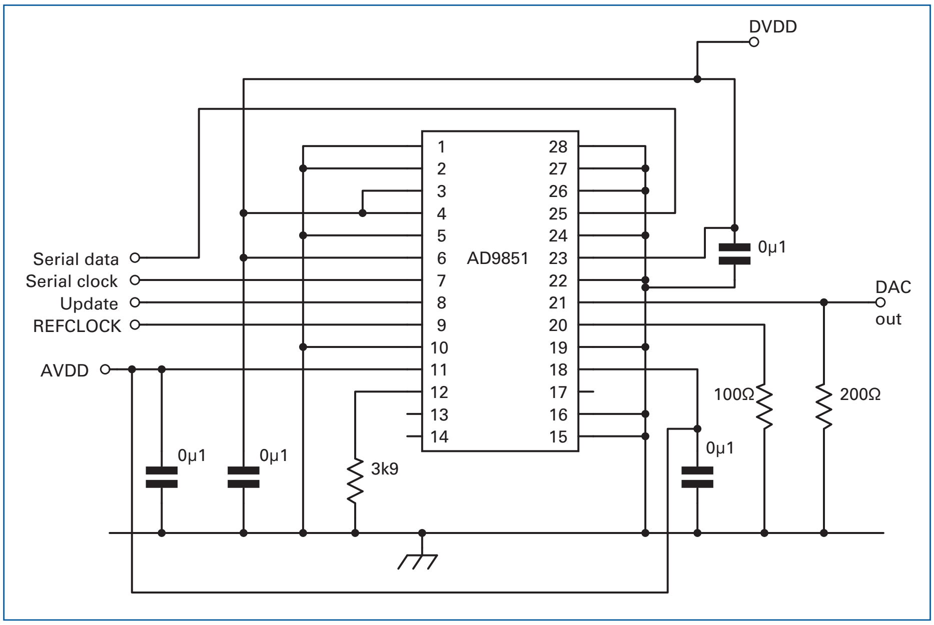


FIGURE 3: Circuit diagram of the AD9851 direct digital synthesiser project.

PHOTO 3: You'd never know it, but this 100 pin IC has been removed & replaced in an amateur workshop. Excess flux was cleaned off after the operation.

the paste was placed on the copper pads using a wooden cocktail stick. Once the paste was applied, the components were placed on the board using a small tweezers. A clean cocktail stick was used to nudge the components into exact alignment with the pads.

SOLDERING. Although I have got 'proper' hot air rework tools, I decided I would do the soldering the Homebrew way using a £10 hot air gun from a local DIY store. In the low heat position, this gun pushes out 280 litres per minute of air at a temperature of 300°C. The air temperature is a bit too high, even for lead free soldering and the high air velocity will tend to blow the components off the board if the nozzle comes too close. You can use a multimeter with a K thermocouple to monitor the surface temperature of the board while you are practising this technique. Please be extremely careful when trying these sorts of techniques because it's very easy to end up with a nasty burn – or even to set fire to things.

The following procedure gives good results every time:

1. The board is held in a 'helping hands' type of clamp so that there is easy access to both sides of the board. The PCB should be level or the components will tend to slide off the pads. Use a rough wooden or metal bench. A more delicate surface will be damaged by the heat.
2. The PCB is preheated from underneath to a temperature of 100 - 150°C. This is basically too hot to touch. This is the preheat, or thermal ramp zone. Even with the heat gun at a distance of 30 – 40cm this takes less than one minute.
3. The gun is then quickly moved to the top (component side) of the board for the reflow zone stage. The gun should be moved closer to the board so that the temperature will increase above the melting point of the solder. Don't allow the gun nozzle to get closer than 10 – 20cm or the components will probably blow off the pads. Hold the gun so that it is blowing directly down on the components rather than across them. A few seconds after all the joints have flowed, the heat gun should be withdrawn vertically and the board allowed to cool.

Two identical boards were made using this method. The finished boards turned out quite well. No solder bridges were found. Both ICs were perfectly centred on the pads, with good solder connections at the pins. One of the boards had a couple of 'pasty' looking joints, which indicates that I didn't heat the entire board to a uniform temperature or the reflow zone temperature was a little too low. It only took a few seconds to rework these joints using a soldering iron. Photo 2 shows one of the finished boards.

SMT REPAIRS AND REWORK. It is quite easy to remove and replace SM chip capacitors and resistors. There are special tools and soldering iron attachments available, but even with a standard iron it is not difficult to melt the solder at both sides of the device and slide it off its pads. Removing SM ICs can be a bigger challenge. Special tools are available for this purpose, but they usually require a separate head for each package type and size. DIY hot air guns are not ideal for this job because they tend to heat the entire board. Special hot air rework pencils are usually the best tool. Special air nozzles are available for each type of IC package. These are handy for removing large QFP (quad flat package) devices. The standard set of round nozzles supplied with these tools are quite adequate for most other types of IC. A typical SMT rework station will have a hot air or 'air bath' type of preheater underneath the board. Preheating the board makes soldering and desoldering easier and reduces the stresses caused by localised heating of a small area by the hot air pencil. While a preheater is desirable, it is not absolutely essential in an amateur setup.

Once the components have been removed from the board, old solder can be removed using desoldering braid (solder wick) and the board can be cleaned using PCB cleaner or a suitable alcohol based solvent. The new components can then be soldered in place using hot air reflow or a soldering iron.

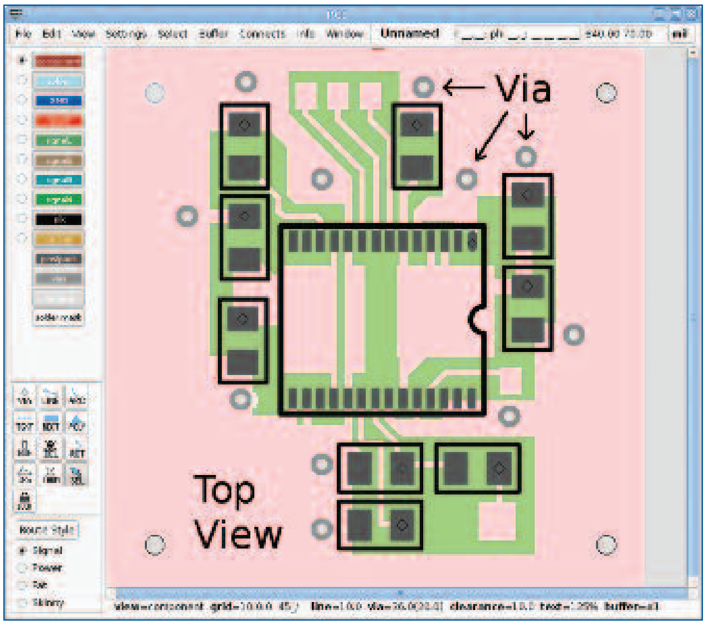


FIGURE 4: Overlay of the AD9851 DDS board.

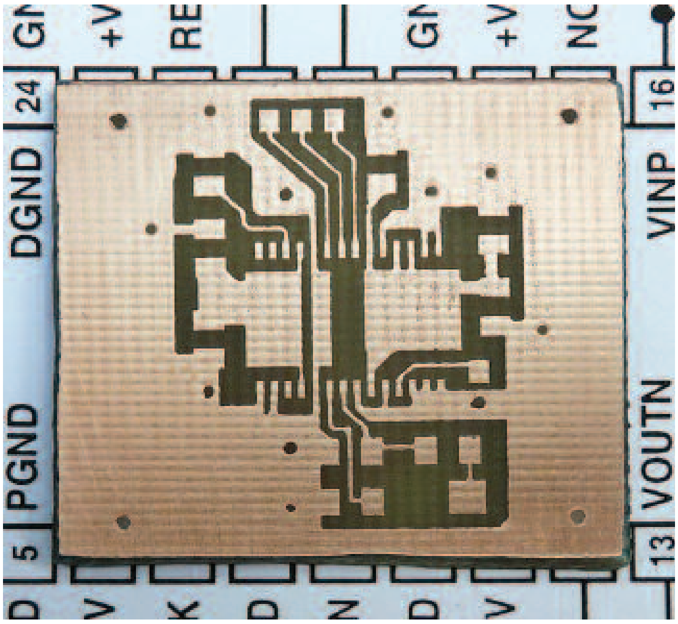


PHOTO 4: The etched DDS PCB is a respectable copy of Figure 4.

HAND SOLDERING. It is possible to solder SM ICs using an ordinary soldering iron, for which there are several techniques in common use. You can solder the pins one-by-one using a fine tip and very fine cored solder. This is reasonably easy to do for SOIC packages, but it is extremely difficult for smaller devices with very close pin spacing. One approach that is popular with radio amateurs is to tack a few corner pins to keep the IC in place, then solder all of the remaining pins without worrying too much about solder bridges. Excess solder and any solder bridges are removed using desoldering braid.

A similar technique called 'drag soldering' can be used to solder high pin density ICs. The board is cleaned and then fluxed using SMT flux from a syringe or a flux pen. I use a CircuitWorks CW8100 no-clean flux dispensing pen (Maplin N63AA). A few corner pins of the IC are tacked with solder to keep the IC in place. Make sure that all of the IC pins are sitting properly on the PCB pads before you start the final soldering. Using a relatively large and flat iron tip and copious amounts of flux, solder the IC pins, several at a time while gradually dragging the tip along the IC pins using cored solder. This is not quite as difficult as it sounds because surface tension will tend to draw the solder to the pads and IC pins. The hot iron will easily flow the solder from pin-to-pin so that excess solder tends to disappear and solder bridges are avoided. Once soldering is complete, do a visual check and remove any bridges using desoldering braid. The drag soldering stage of the operation is easier if the board is mounted vertically during soldering. Start soldering at the top of each row of pins so that gravity will assist the flow of solder as you move along the row of pins. This is known as 'vertical drag soldering' **[2]**. **Photo 3** shows a 100 pin IC which was removed from an old Ethernet card and then replaced using drag soldering. I used an Antex 25W iron with the standard flat angled tip for this soldering job. Fine pointed tips are not suitable for drag soldering.

AD9851 DDS. Our second project is a DDS synthesiser that will be used as a signal generator. The AD9851 **[3]** comes in a 28-pin SSOP package that has a pin spacing of 0.65mm. The PCB uses a double sided layout, but the bottom layer is only used as a continuous ground plane so that no etching of this layer is required. The schematic is shown in **Figure 3**, the layout is shown in **Figure 4**. The board was made using the toner transfer method **[4]**. Although I felt that I was pushing the boundaries of what was possible using a ten year old laser printer and cheap glossy photo paper, the PCB turned out better than expected. **Photo 4** shows the etched PCB. The ground plane layer was protected by a couple of strips of vinyl tape while the board was in the etchant.

A total of 11 vias were used to connect critical points of the top layer directly to the PCB ground plane. As recommended in the AD data sheet [3], separate 5V power supplies were used for the analogue (AVDD) and digital (DVDD) supplies. The digital and analogue ground connections are tied to the main ground plane at strategic points in the circuit. **Photo 5** shows the finished DDS unit. The hand soldered wire vias are a bit ugly, but they are perfectly functional. A PIC16F628 microcontroller was used to generate the data, clock and update signals to program the 40 bit frequency/control word into the DDS. The module was tested with a 100MHz OCXO as a reference clock and also with a 29.5MHz TTL oscillator and the internal 6x Refclock multiplier to give a Refclock frequency of 177MHz. RF performance seems good. Tests were performed at various frequencies from 500kHz to 50MHz. Spur levels were generally in line with the AD9851 data sheet figures.

PCB layout files are available by e-mail at the address above.

REFERENCES:
[1] Homebrew August 2006; also see May 2009
[2] http://tinyurl.com/3chfhp
[3] AD9851 data sheet Rev. D. Analog Devices
[4] Homebrew July 2006

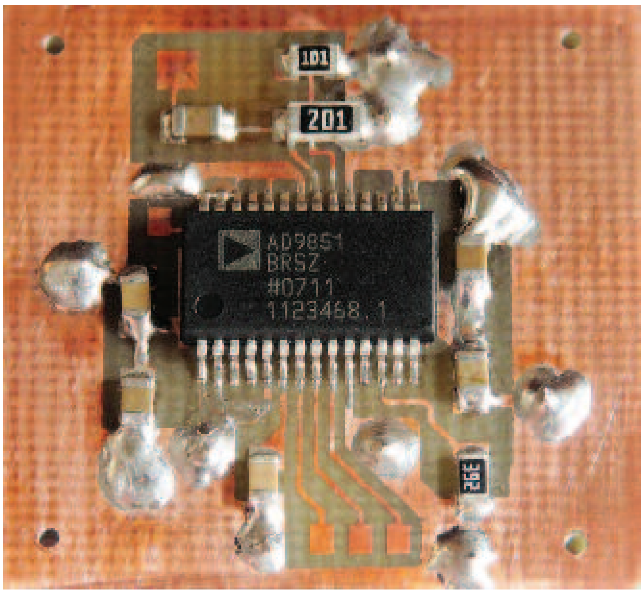


PHOTO 5: Surface mount makes for a very neat finished board. Note that this is the opposite way up from Figure 4 and Photo 4.