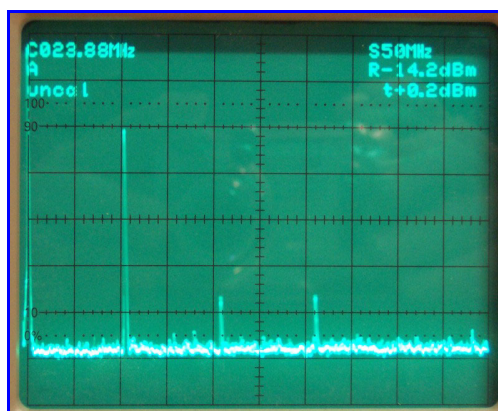
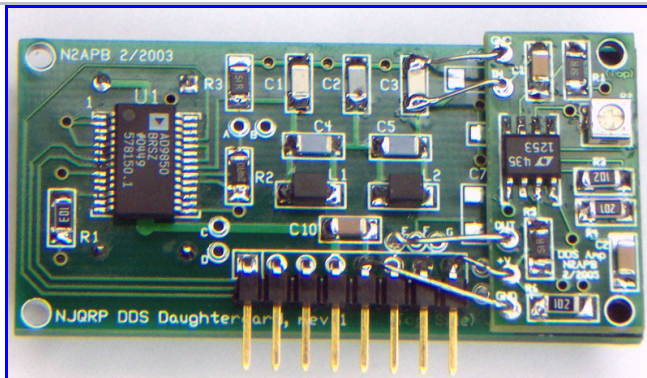


DDS Daughtercard Kit

... a modular way to add stable signal generation to your next transceiver or VFO project

How many ways can you use a self-contained, high-precision dc-30 MHz signal source contained on a 1" x 2" plug-in circuit board? How about as a stand-alone VFO, a signal generator for your bench, a replacement LO for your Sierra or NC40 transceiver, or perhaps as the heart of an antenna analyzer! Control it with your favorite microcontroller, or even hang it off the parallel port of your PC. Any way you do it, you'll be generating quality signals for under \$35



Any homebrewer worth his salt can tell you that Direct Digital Synthesis (DDS) captures the experimenter's imagination and excites him like no other technology these days. Exceptional signal precision, accuracy, stability, programmability and signal quality are all easily and inexpensively achieved. But two quite formidable problems still remain.

The first problem is that these surface mount DDS chips are so tiny, with lead pitches as fine as 0.65 mm, that it is nearly impossible to homebrew with them using conventional techniques. It is quite difficult to tack-solder a fine wire onto one of these small SOIC package leads, let alone putting 16 wires to the chip!

The second problem is that these DDS chips must be interfaced to a microcontroller that provides that frequency programmability. A 40-bit control word must be loaded into the DDS chip to command it to generate a specific frequency. There are many projects around that control a DDS chip with a PIC, an Atmel controller, a BASIC Stamp, an SX chip, and so on. I don't know about you, but the VFO I will ultimately need is likely to use some controller that I don't technically "know" and cannot program. This makes it tough to use that controller for anything but commanding the DDS, thus raising the cost of the entire project, increasing the amount of board real estate needed, and raising the power needs for the entire project.

The Solution

To get past these two problems, we designed a self-contained functional module that generates a top-notch RF signal by using a small pc board to contain just the bare DDS essentials – an Analog Devices AD9850 DDS chip, a 100 MHz clock oscillator, a 5th-order elliptic filter and an RF amplifier add-on module called the DDS Amp. Additionally, an onboard 5V regulator is provided so you only need provide a single 12V battery or power supply ranging anywhere from 8-16V DC. The three digital control lines, the power supply, and the output signal are all available on a pin header at the board edge. The schematic is shown on page 3.

The 8-position pin header at the board edge serves to allow DDS Daughtercard to be plugged into and used in any project you might have on your bench, regardless of which microcontroller is employed. Just provide a single strip socket (e.g., a 16-pin IC socket split lengthwise) on the project board and plug in the DDS Daughtercard. Heck, you don't even need a dedicated microcontroller – use a cable connected to the parallel printer port of your PC and use public domain PC software to control the DDS board! See the Controller section here for a number of custom solutions for you to easily control your DDS Daughtercard.

Once your controller-of-choice serially loads the 40-bit control word into the DDS, the raw waveform is presented to an elliptic filter that removes unwanted high-end frequency components, resulting in a signal of sufficient quality to serve as a local oscillator for a transceiver. Refer to the AD9850 data sheets for signal purity specifications.

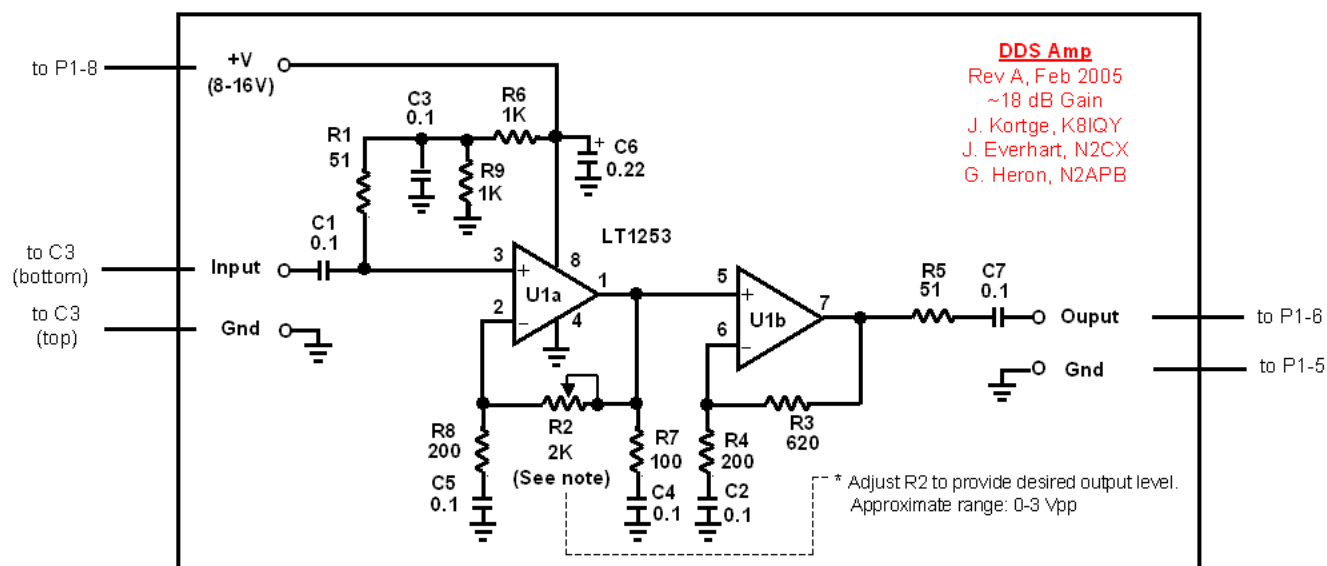
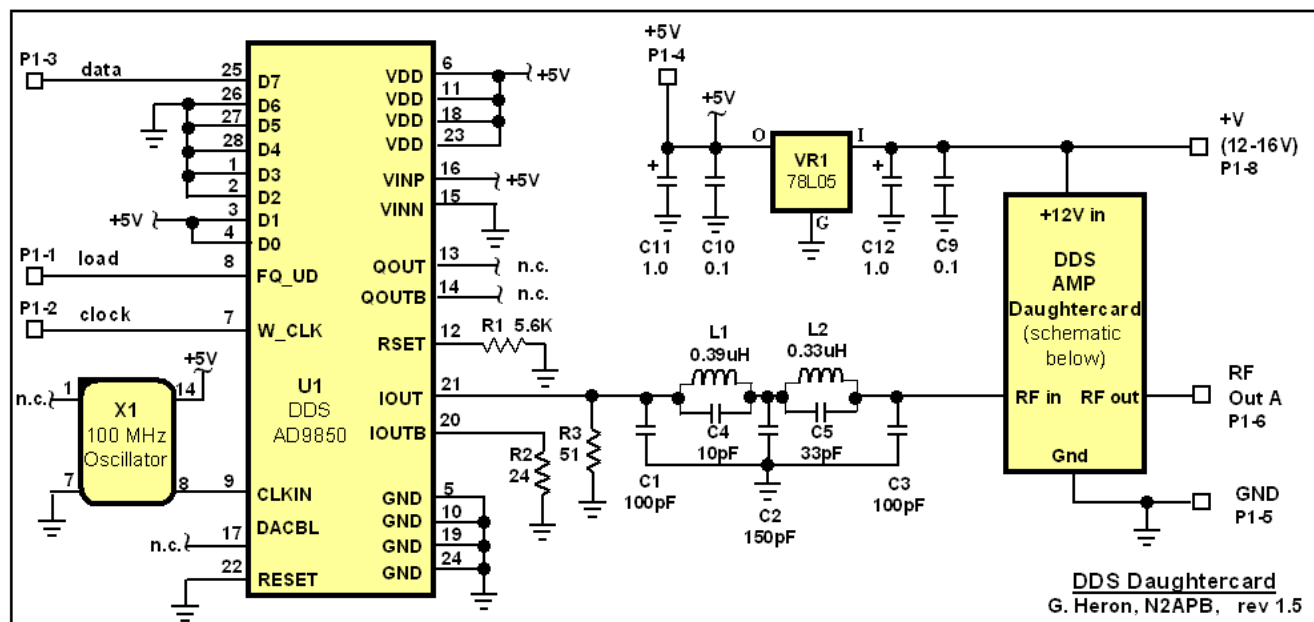
The signal generated by the DDS is quite small so we use an amplifier called the DDS Amp to provide 18 dB of gain to boost the signal to about 2V p-p, which is quite usable in a variety of applications. The DDS Amp is an LT1253 dual-stage video amplifier IC that fits on a small 1" x 1/2" pc board. The board sits on the end of the DDS Daughtercard when the existing output amplifier components are removed and connects to the main DDS circuit board with short wire jumpers. This amplifier module replaces the single-chip MMIC amps used in earlier versions of the DDS Daughtercard. Our new DDS Amp is much improved compared to the previous design by offering unconditionally stability ($k > 1$) and yielding spectrally-clean signals with harmonics down more than 40 dB. It is an ideal signal source for making impedance measurements in the Micro908 Antenna Analyst and other demanding designs. A trimpot allows precise setting of desired output levels. The design provides a good signal using supply voltages from 16V all the way down to 8V, thus conveniently allowing for battery operation.

The amplified signal is then presented to P1 pin 6 on the pin header where it can be used as a 50-ohm source input signal. If not used as an input to any other component or module, the output should be terminated with a 50-ohm resistor in order for the stated specifications to be realized.

Specifications

- Power requirements: 8-16V DC at 180 ma (max).
- RF Output – fully adjustable to +10 dBm, or about 3V pp.
- Output signal unaffected by varying +V supply voltage – great for battery operation.
- Near-constant output level from 1-30 MHz – only 1.8 dB drop occurring from 28-30 MHz.
- Good signal purity – harmonics down approximately 40 dB from the fundamental.

SCHEMATIC



Assembling the DDS Daughtercard “main board”

Although many of the components are surface mount devices, assembly of the pc board is still simple and straightforward. The board layout is fairly open and the components are all very accessible. You will need a fine-tipped soldering iron and some thin solder to best attach the components. You'll also likely need a magnifying lamp in order to see the small leads of the surface mount components.

A trick to soldering surface mount devices to pc boards is to (a) pre-solder one of the pads on the board where the component will ultimately go; (b) hold the component in place on the tinned pad with needle nose pliers or tweezers; (c) re-heat the tinned pad and component to reflow the solder onto the component lead, thus holding the component in place; and lastly (d) solder the other end of the component to its pad.

1) Attach DDS (U1) – Following the technique described above, pre-solder the corner pad for U1 pin 1, then carefully position the leads of U1 over its set of pads on the pc board. Re-heat the solder of pad 1 to reflow the solder onto U1 pin 1. This should leave the IC attached by pin 1. Again making sure the IC pins are aligned over all pads, carefully solder the opposite corner lead (pin 15) to its pad. This should leave all other pins of the IC aligned over their respective pads, making it easier to solder them to the pads. Go ahead and solder each of the other pins to their respective pads, being careful not to bridge across any adjacent pads. If this does happen, grab some solder wick or a solder sucker and use it to draw off the excess solder, which should be fairly easy and clean because of the solder mask on the circuit board. (See Note 3 describing a resource that can be used if you need help attaching this IC to the pc board.)

2) Attach oscillator can (X1) – Insert the metal oscillator can on the backside of the circuit board, carefully noting location of pin 1, denote by the one sharp-corner on the can. Solder the leads in place and snip off the excess lead length.

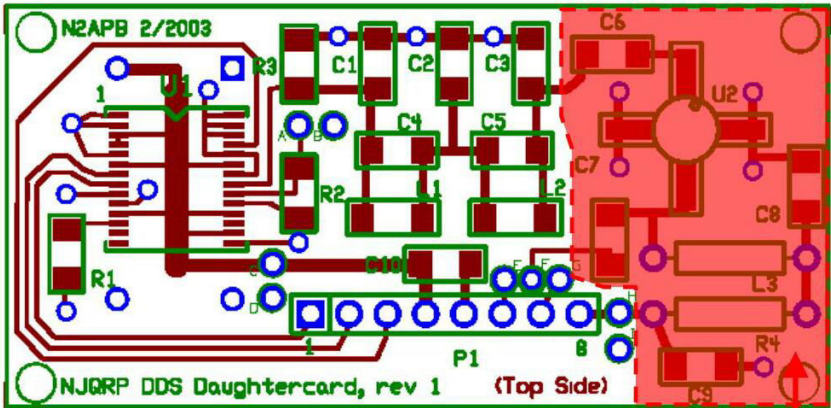
3) Attach the SMT R's, C's & L's – Attach surface mount capacitors (C1-C5, C10-C13), resistors (R1-R3) and inductors (L1 and L2) – Use the technique described at the start and you shouldn't have any problems. Note: Capacitors C6-C9 are not used. C11, C12 and C13 are to be mounted on the bottom of the board. C11 and C12 are miniature electrolytic capacitor “cans” with the negative polarity denoted with the black mark, as shown in the photo.

4) Attach the voltage regulator (VR1) – VR1 is soldered to the pads on the back side of the board. Orient the flat side of the package against the board and bend the leads down such that they are in full contact with the three pads. Snip off the leads to the proper (short) length before soldering.

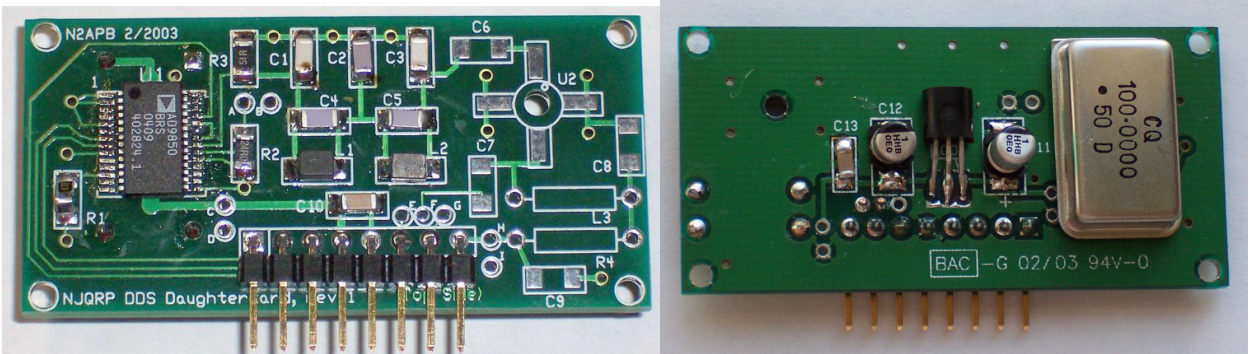
5) Attach pinheader (P1) – Insert the pin header on the top of the board and solder all eight pins to the pads, ensuring that the longer pins are parallel to the plane of the board.

Main Daughtercard Parts List:

QTY	Designator	Description	NOTES
1	X1	Oscillator, 100 MHz, 14p DIP can	Large metal can. Pin 1 at square corner.
1	U1	DDS integrated circuit, 28 pin SSOP	Not supplied in kit. Contact Analog Devices for free sample. (See text.)
1	R3	Resistor, 51, 1206 SMD	on SMT card (black side has value markings)
1	R1	Resistor, 5.6K, 1206 SMD	on SMT card (black side has value markings)
1	R2	Resistor, 24, 1206 SMD	on SMT card (black side has value markings)
2	C10, C13	Capacitor, 1206 SMD, 0.1uF 25V	on SMT card
1	C5	Capacitor, 1206 SMD, 33pF 25V	on SMT card
1	C2	Capacitor, 1206 SMD, 150pF 25V	on SMT card
1	C4	Capacitor, 1206 SMD, 10pF 25V	on SMT card
2	C1,3	Capacitor, 1206 SMD, 100pF 25V	on SMT card
1	L1	Inductor, 0.39uH, 1206 SMD	on SMT card (small black "cube")
1	L2	Inductor, 0.33uH, 1206 SMD	on SMT card (small black "cube")
2	C11,12	Capacitor, electrolytic, SMD, 1.0uF, 50V	Small silver "cans". Black stripe denotes negative lead.
1	VR1	Voltage Regulator, 78L05, 5V	Install with flat side toward board. Bend leads down to fully contact pads.
1	P1	Pin Header, 0.1", 8 pin	Install with longer pins parallel to the plane of the board.
1	PCB	PC Board	Top has DDS, LPF components and P1 connector. Osc can on bottom.



Shaded component positions are not populated.
These parts are not required.



Top side of main DDS Daughtercard and the bottom side.

Assembling the DDS Amp “grand-daughtercard”

Refer to the diagrams, photos and parts list on the next page.

1) Trimpot (R2) – The three leads for this tiny trimpot are located (mostly) underneath two edges of the component. Tin one pad on the board as indicated in diagram on the next page, and carefully hold the trimpot in place with fingers of one hand while reflowing the soldered pad with a soldering iron held in the other hand. Ensure that the trimpot is oriented over the pads such that each of the three leads can be soldered. (The pads are closely spaced, so be sure the first solder connection is done with the lead at the very edge of the pad. You’ll see.) After the first pad is soldered, and ensuring proper alignment on the others, solder the remaining two pads. Using U1 pins 1 and 5 as test points, measure the resistance of the trimpot to ensure proper connection. You should see about 1K-ohm. If you read more than 2K-ohms, you did not properly attach the trimpot. If you read zero ohms, you have a short in the trimpot connection to the board. Do not proceed until you get the correct reading. (Note: These 2K-ohm trimpots come from the factory set approximately to mid-position.)

2) Video Amp IC (U1) – Using the techniques described for attaching the DDS chip on the main Daughtercard, attach the 8-pin LT1253 SOIC chip for U1. The lead spacing is greater here and the task is much easier.

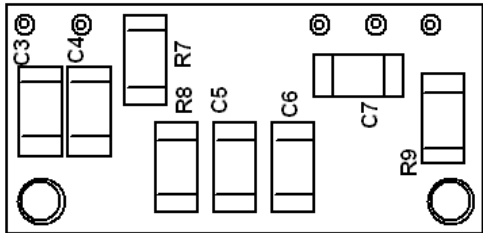
3) Attach SMT R’s & C’s – Detaching only one value component at a time, solder the SMT resistors and capacitors to the top and bottom side of the grand-daughtercard. For example, install R1 and R5, then R6 and R9, and so on. NOTE: When it comes to attaching C6, the positive end of the capacitor is marked with a black bar and it should be mounted toward the center of the board, as shown below.

4) Attach input/output leads – Locate five (5) scrap pieces of wire on the bench, each approximately 1” long. For example, these could be clipped-off leads from axial resistors of some previous project. Place a short 1/8-inch 90-degree bend at one end of each lead and insert the bent end of each wire into the edge holes marked as GND, IN, OUT, +V and GND. Solder the wires in place and align them to be parallel to the plane of the board, and perpendicular to its edge, as shown in the photo on the next page.

DDS Amp “Grand-Daughtercard” Parts List:

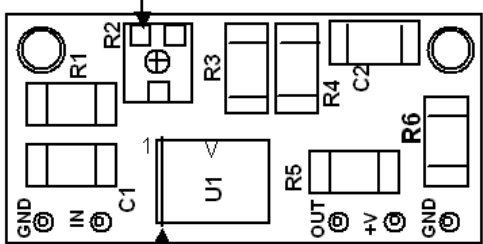
Qty	Designator	Description	Notes
2	R1, R5	Resistor, 51 ohms, SMD 1206	Supplied on SMT parts card
2	R6, R9	Resistor, 1K ohms, SMD 1206	Supplied on SMT parts card
1	R3	Resistor, 620 ohms, SMD 1206	Supplied on SMT parts card
1	R2	Trimpot, 2K ohms, SMD	Supplied on SMT parts card
2	R4, R8	Resistor, 200 ohms, SMD 1206	Supplied on SMT parts card
1	R7	Resistor, 100 ohms, SMD 1206	Supplied on SMT parts card
6	C1, C2, C3, C4, C5, C7	Capacitor, 0.1 uF, SMD 1206	Supplied on SMT parts card
1	C6	Capacitor, 0.22 uF, tantalum, SMD 1206	Supplied on SMT parts card. Positive end has black band.
1	U1	Dual op amp, LT1253, SOIC	Supplied on SMT parts card. Side with pin 1 has beveled edge.
1	PCB	PC board, approx 0.5" x 1"	

Solder parts to bottom side after top side components are attached.



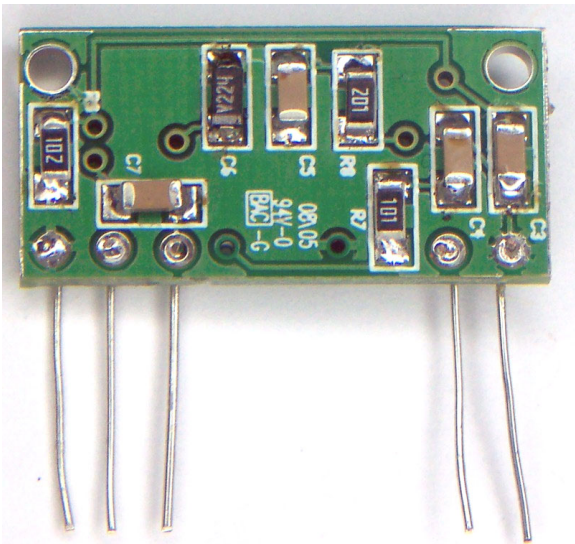
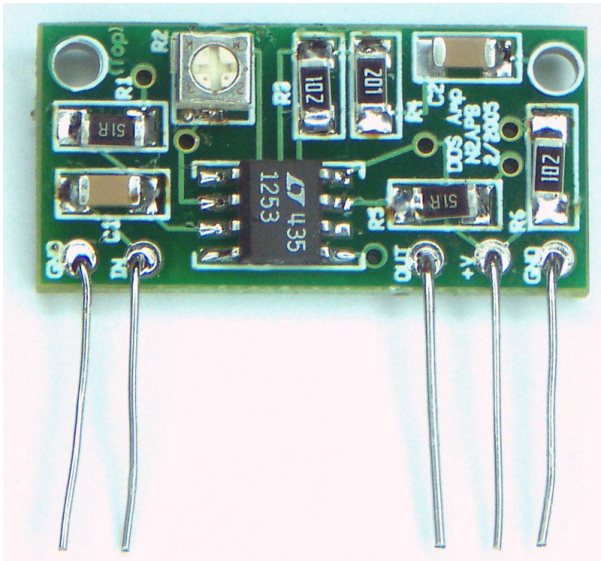
Bottom side

Add R2 trimpot first. Tin one pad first, then carefully hold the trim pot in place with tweezers, and re-heat the pad. Solder the other two pads. Measure about 1K ohm on traces to ensure successful connection.



Top side

Small beveled edge on package denotes side for pin 1.



Top side of the DDS Amp “Grand-Daughtercard” and the bottom side.
(NOTE: In the left photo, the value for R3 should actually read “621”.)

Attaching the Amp Grand-daughtercard to the main DDS Daughtercard

Once the two assemblies are individually complete, it is time to attach the Amp “grand-daughtercard” to the main DDS Daughtercard.

1) To be safe, place a small piece of electrical tape onto the main daughtercard to prevent shorts from occurring to the Amp pcb that will be placed above it.

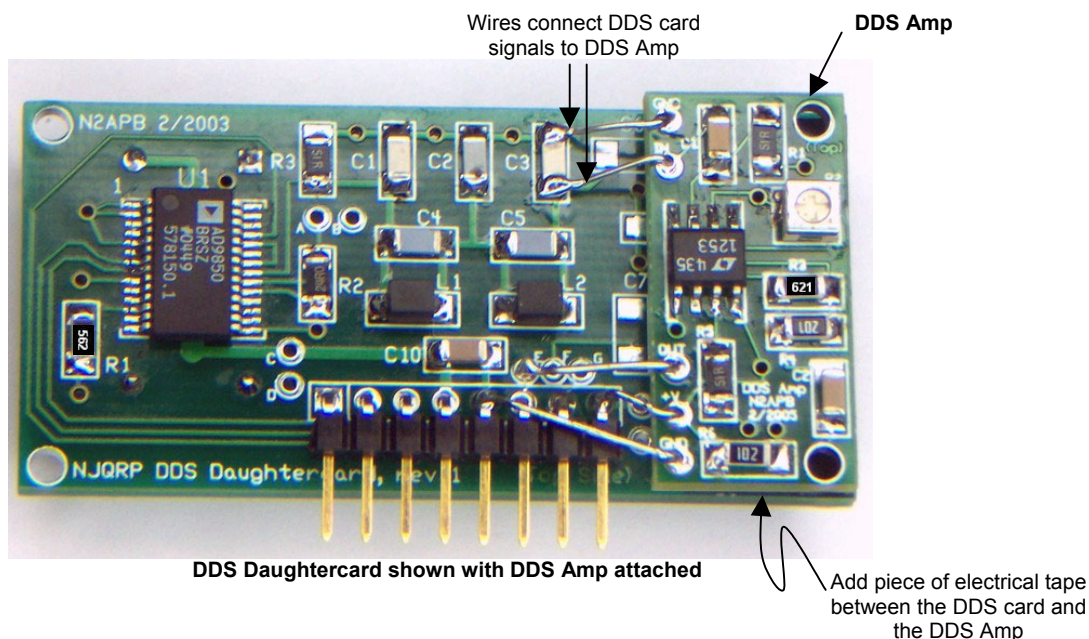
2) Position the small Amp grand-daughtercard over the right-hand side of the main board as shown in the photo below and first arrange the top two wires (GND and IN) to straddle the C3 capacitor. Cut the leads and solder in place.

3) Arrange the next wire (OUT) to be over the thru-hole marked ‘E’ on the main DDS card. (Placing a small 90-degree bend at the appropriate point will help the wire go down into the hole.) Solder the wire in place and cut off any excess length protruding from the bottom.

4) Arrange the next wire (+V) to be over P1 pin 8. Similarly, place a small 90-degree bend in it and cut excess length such that the wire sits right next to the “vertical” part of P1 pin 8. Solder in place.

5) Arrange the last wire (GND) to go over to P1 pin 5 and again place a small 90-degree bend in it to allow good contact with the vertical part of pin 5. Cut it to the proper length and solder in place.

6) For a neat appearance, make sure the Amp grand-daughtercard is parallel to the main DDS card and is well-aligned to its edges. The soldered input/output leads will hold the Amp board securely in place.



Initial Tests for the DDS Daughtercard

Apply 8-16 volts to P1 pin 8, and ground to P1 pin 5. The DDS Daughtercard will not generate any signal on its output until an external controller delivers a control word to the DDS, as described in the next section. However at this point you can ensure that proper voltages are present and that excessive current is not being drawn.

Check to see that you have 5 V on the output of VR1, which is also available at P1 pin 4.

The board should be drawing about 150 ma. The voltage regulator, DDS, and oscillator can will all be warm to the touch, but not unbearably so. Note that since the voltage regulator is located on the bottom side of the board beneath the Low Pass Filter components, it may seem that the LPF resistors and capacitors are getting warm, but this is not the case.

If you have an oscilloscope, you should see an approximate 4Vpp 100 MHz signal being delivered from the oscillator can (X1) on pin 8. Note that your 'scope would need to have a bandwidth of at least 200 MHz in order to see the full-level signal as described here.

Controlling the DDS

It won't be too long before you consider ways to use the DDS Daughtercard. Well, you are in for a real treat as there are a whole bunch of convenient and powerful techniques already developed that can be employed to provide a user interface and controller for this RF signal generator module.

In general, the DDS chip needs to be sent a control word by an external controller before the Daughtercard is capable of generating a signal. No matter what controller you use, it must serially send a 40-bit control word to the DDS Daughtercard on the DATA line (P1 pin 3). The controller clocks each bit into the DDS by the toggling the CLK control line (P1 pin 2). To complete the sequence, the accumulated control word is loaded into another internal DDS register by a toggling of the LOAD line (P1 pin 1), thus instructing the DDS chip to generate the frequency just loaded. Then *bingo*, the new frequency appears at the RF Output of the DDS board (P1 pin 6).

The AD9850 data sheet, and an Interactive Design Tool applet on the Analog Devices website:

(http://www.analog.com/Analog_Root/static/techSupport/designTools/interactiveTools/dds/ad9850.html)

provide all the details on how to construct the five byte, 40-bit control word with your microcontroller-of-choice. Just enter 100 MHz in the reference clock field and the desired operating frequency, and the applet will display the five control bytes that you need to send serially to the DDS Daughtercard in order for it to generate that frequency.

There are many software programs available that already accomplish sending the control words to the DDS card, thus enabling it to serve as a general purpose VFO. Any of the techniques outlined in the next section may be adopted if you plan on using the specific processors in your project. They are also great examples on which you can model a software program using a different microcontroller.

Many Uses!

Here are some great ways for you to build up a project using the DDS Daughtercard to generate AF and RF signals.

QuickieLab

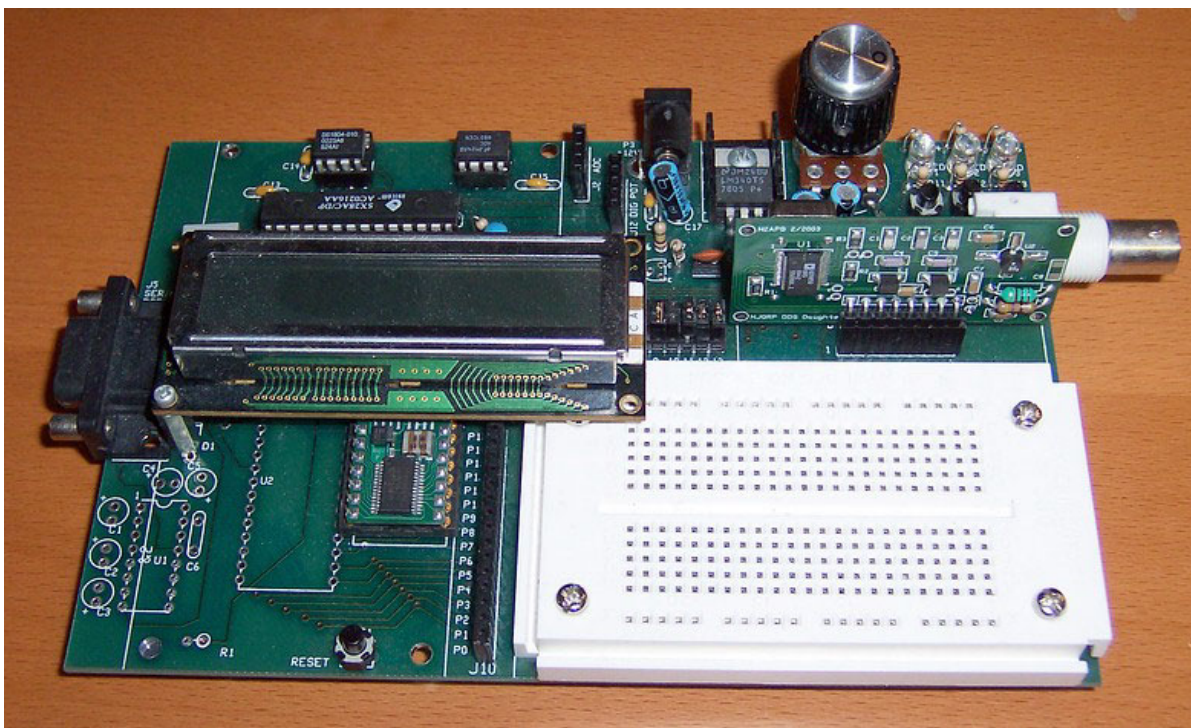
The DDS Daughtercard is ready-made for plugging into the NJQRP “QuickieLab”, which is an experimenter’s platform provided several years ago in kit for hobbyists to easily build up circuits on a plugboard and connect them into various standard circuits provided on the board for audio amplification, A/D conversion and LCD display. So all you need to do is plug the DDS Daughtercard into the appropriate socket on the QuickieLab and use some of the canned software that’s provided to have the hardware serve as a basic VFO.

The programming sequence is easily accomplished. In one of his renowned “Joe’s Quickies” columns in QRP Quarterly magazine, N2CX uses the following three lines to instruct the BASIC Stamp controller to produce a 7.040 MHz signal with the DDS Daughtercard plugged into the QuickieLab:

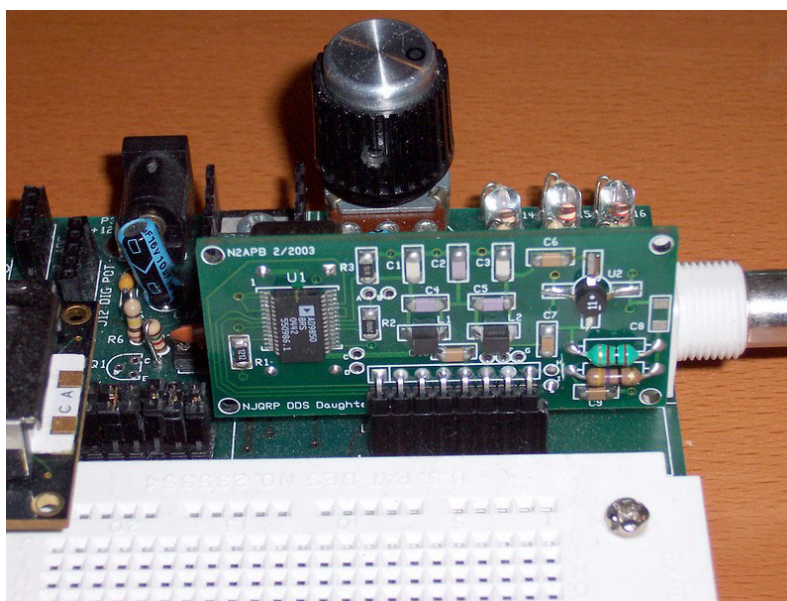
```
shiftout 7,8,0,[$02,$BC,$05,$12,$00]
out9 = 1
out9 = 0
```

The first instruction line shifts out the 40-bit (5-byte) value on port P7, using P8 as the clock. The second and third instruction lines toggle the LOAD pin going to the DDS, at which point the RF signal is generated at the output of the DDS Daughtercard.

(If you do use the DDS Daughtercard with the NJQRP QuickieLab, remember to cut the +5V trace on the bottom of the QuickieLab pc board that leads to P1 pin 4 on the DDS Daughtercard socket. This is because the DDS Daughtercard produces its own internal 5-V supply and doesn’t require that the QuickieLab supply it.)



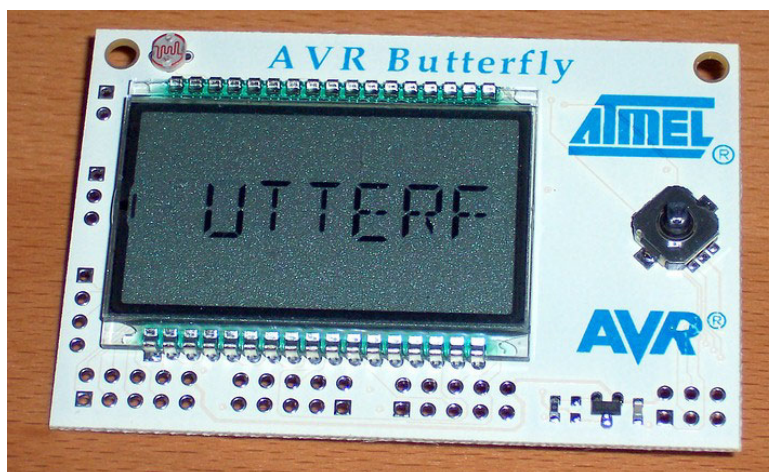
Here's the QuickieLab set up with the DDS Daughtercard in the socket located just above the white plugboard. The BASIC Stamp processor, located on the main board beneath the LCD, commands the DDS chip on the daughtercard in order to generate RF signals that are pumped out the BNC connector on the right side of the board.



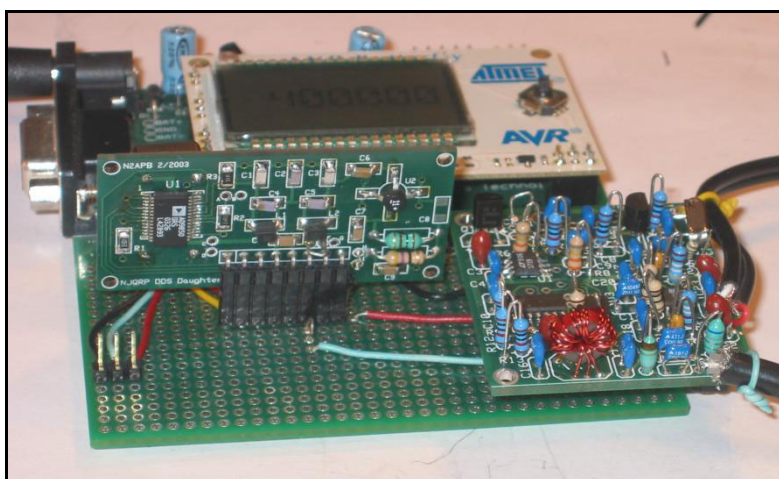
Closeup of the DDS Daughtercard plugged into the QuickieLab. Note that DDS card used here is one of the original ones that used the MMIC amplifier on the right side of the card. (The MMIC amplifier has been replaced by the addition of the DDS Amp "grand-daughtercard" in the current design.

Butterfly Controller Board

Software currently exists for the AVR Butterfly card to present a simple user interface for frequency entry and display and to command the DDS Daughtercard to generate those frequencies. See http://www.atmel.com/dyn/products/tools_card.asp?tool_id=3146) for more information on the Butterfly controller card. Steve Weber, KD1JV developed the DDS card driver software and has it on his website, located at <http://www.qsl.net/kd1jv/bfydds.HTM>.



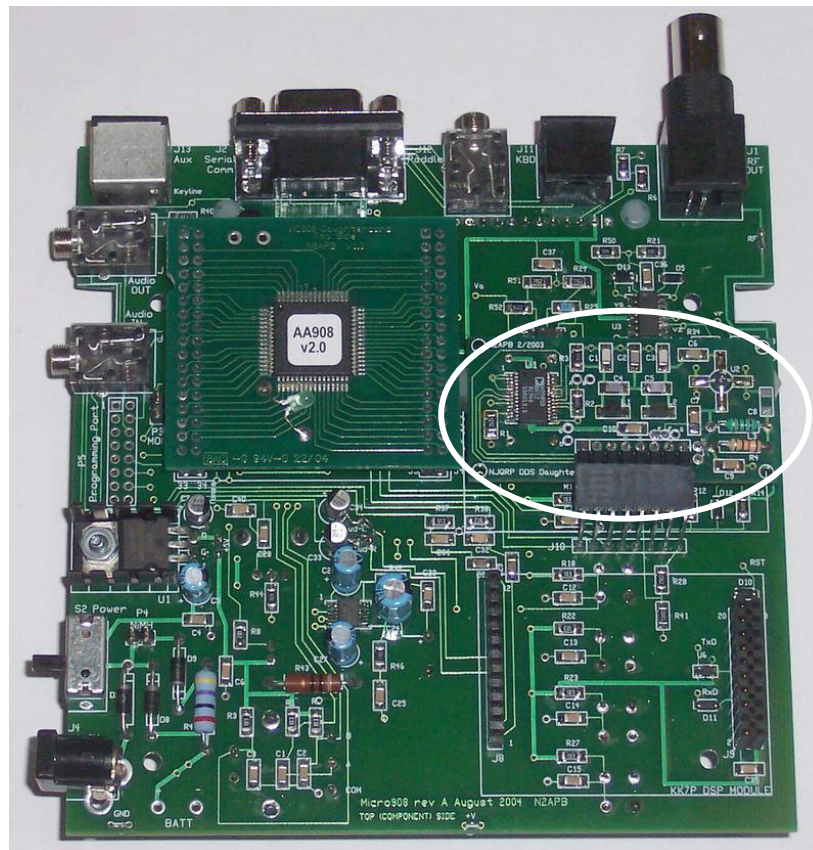
The "Butterfly" controller card from Atmel is a convenient and inexpensive way to get a fairly powerful processor on a card with some universal I/O of an LCD and "joystick/pushbutton" being used to navigate.



Here is pictured the DDS Daughtercard plugged into an ERCOS Butterfly Carrier board, which allows the Atmel processor to control circuits interfaced to the Butterfly board. The additional circuit card on the right is an earlier version of the SoftRock40 SDR receiver project described elsewhere in this issue of Homebrewer. In this application, the Butterfly controller board controls the DDS Daughtercard to generate RF being used as the local oscillator for the little receiver card. (The ERCOS Carrier Kit can be found at <http://www.ecrostech.com/Products/Butterfly/Intro.htm>)

Micro908

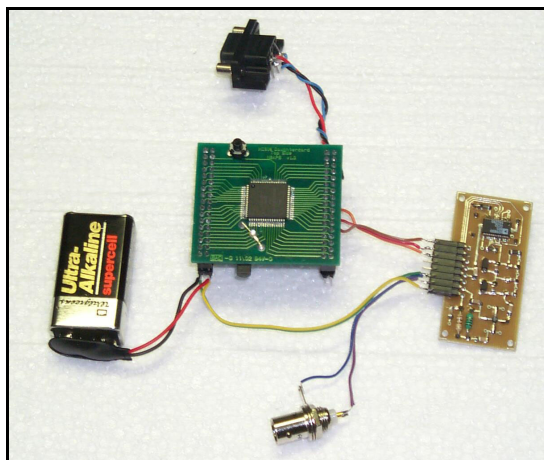
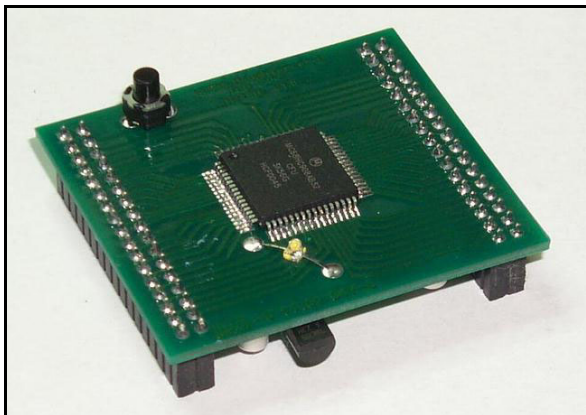
The DDS Daughtercard is already incorporated into the Micro908 kit from the AmQRP Club and is integrally used to make the instrument function as an antenna analyzer. So if you are able to write a specific VFO program for the Micro908, or perhaps just use the VFO function provided with the AA-908 software load on the Micro908, you can use the instrument to generate signals “right out of the box”.



The DDS Daughtercard (identified by the white circle in the photo above) is controlled by the HC908 Daughtercard on the Micro908 motherboard. It plugs into an 8-pin right-angle connector that holds the DDS card up off the motherboard, above the reflectometer circuits of the Micro908.

HC908 Daughtercard

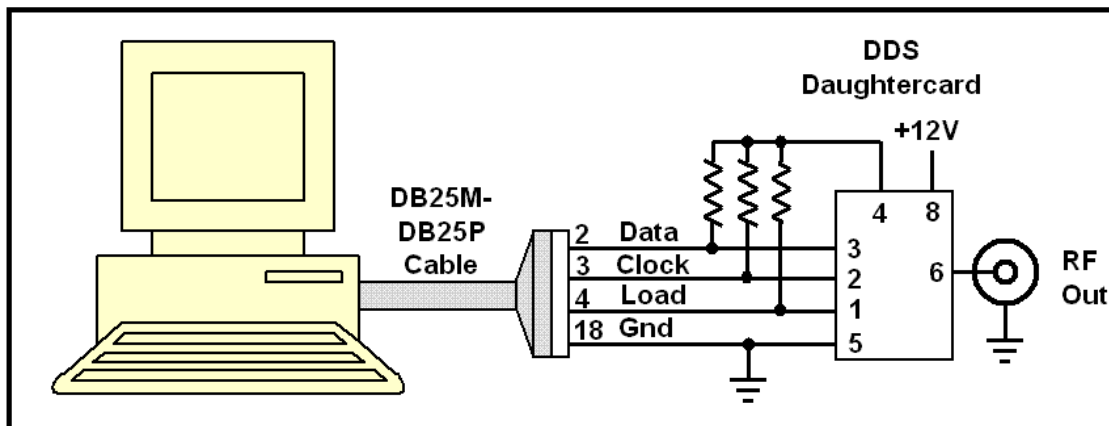
You may also plug the daughtercard into the HC908 Excerciser board described in the Digital QRP Breadboard project and use the software provided with that project to have it serve as a programmable VFO.



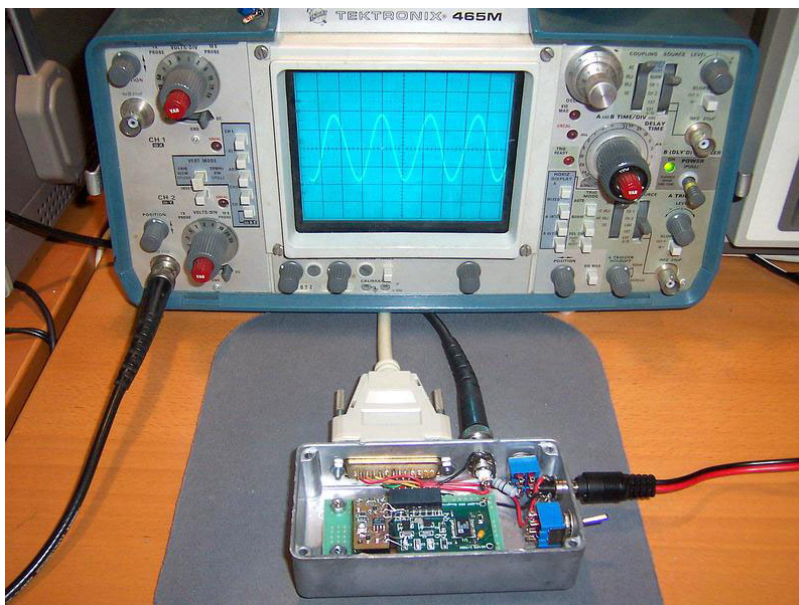
The HC908 Daughtercard controller is a popular and recurring component with the DDS Daughtercard. Together they make a flexible combination of modules that can be used in many different applications, using a common set of software libraries (subroutines). The configuration of the two modules shown in the right photo is the minimal hardware arrangement needed for a “dumb terminal” (like HyperTerm on a Windows PC or ProComm on a handheld computer) to command the HC908 to generate frequencies ... in effect, the HC908 and DDS daughtercards become a remotely-controlled signal generator!

Parallel port control from your PC

You can even control the DDS Daughtercard directly from your PC. All that's needed is to wire the daughtercard to the printer port cable and run one of several public domain custom software programs to have the PC serve as VFO controller. The programs from KA2UPW, or WA6UFW and WA6FXT (located on the project website) each have an attractive display screen that offers specific frequency entry, phase settings, and even scanning controls!



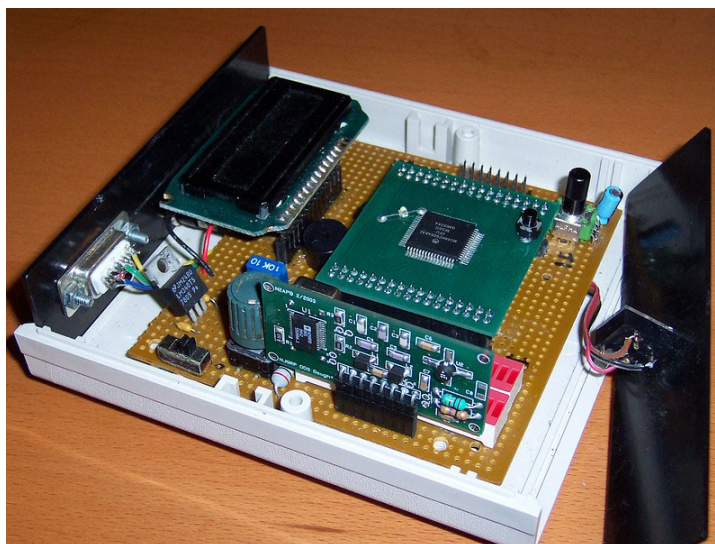
If you wire the DDS Daughtercard to the PC's parallel port connector and a power supply, as shown in this diagram, any signal from 0.1 Hz to 30 MHz may be generated by the software application running on the PC.



Here is shown the “PC VFO” assembly being controlled by a PC over its parallel printer port. The DDS Daughtercard is housed in a rugged die-cast aluminum enclosure with connectors for the parallel port control lines, RF output on a BNC, and power input on a coaxial power connector. Switches provide power control and the ability to switch a 50-ohm load resistor in/out of the circuit. The oscilloscope display in the background shows the nice and clean 10 MHz signal currently being generated by the DDS card, as commanded from the PC screen.

Custom Controller

Of course you can also design the DDS Daughtercard into your own custom project, using your favorite microcontroller chip as the “brains”, be it a PIC, Atmel, Ubicom, or 8051-derivative processor. The possibilities are limitless!



This photo shows the DDS Daughtercard included in a circuit I call the Exerciser ... which is a test fixture that all HC908 Daughtercards get plugged into in order to get tested and qualified before being programmed with the appropriate software before shipping. (The Exerciser schematic and software program are located in the HC908 Daughtercard manual.)

Conclusion

This simple DDS Daughtercard solves both of the problems described at the beginning of this article. It enables the homebrewer to easily take advantage of the positive attributes of the DDS chip to produce a high quality homebrew variable-frequency signal source. Let us know how you end up using your DDS 'card!

Notes

1) The DDS Daughtercard Kit may be purchased from the AmQRP Club. Refer to the DDS Daughtercard project web page for all current technical and ordering details:
<http://www.njgrp.org/dds>.

2) The AD9850BRS DDS chip is not provided because homebrewers can obtain a free sample from the Analog Devices website at:
<http://products.analog.com/products/info.asp?product=AD9850> . Just go to this Internet location, register with Analog Devices (i.e., give them your mailing address), and within a week or so you will receive a free sample of the DDS chip by mail.

3) The NJQRP has lined up a great resource to assist in soldering the DDS chip onto the printed circuit board. Once you've acquired your free AD9850BRS DDS chip from Analog Device, send the chip and your DDS Daughtercard circuit board, to Mike, WA6OUW, at "KitBuilders". For \$6 he will attach this surface mount chip to the pc board and return it promptly by mail. It's not tested because at that point it's only the DDS chip on a bare pc board, but Mike does excellent work. (The NJQRP uses KitBuilders for assembly of the HC908 Daughtercard product, so we know the quality is there!) Just place your DDS chip, pc boards and a \$10 check or M.O. payable to "KitBuilders" into a padded envelope and send to: KitBuilders, 1630 San Miguel Canyon Road, Watsonville, CA 95076. You can contact Mike by email at wa6ouw@aol.com if you have further questions.

4) Details of the NJQRP QuickieLab project can be found at www.njgrp.org/quickielab .

5) Details of the HC908 Daughtercard project can be found at www.njgrp.org/hc908 .

6) George Heron, N2APB may be contacted by email at n2apb@amgrp.org , or by mail at 2419 Feather Mae Ct, Forest Hill, MD 21050.