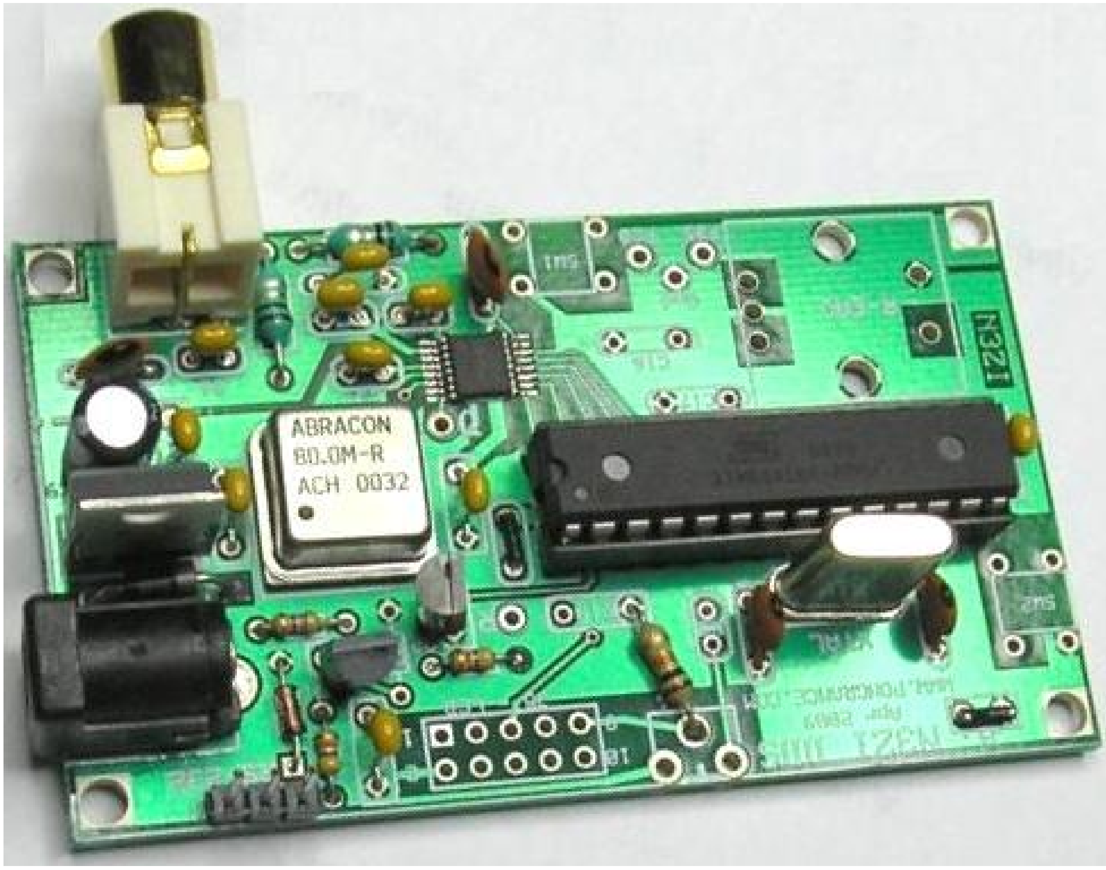
Preliminary Manual for the N3ZI DDS
WWW.PONGRANCE.COM

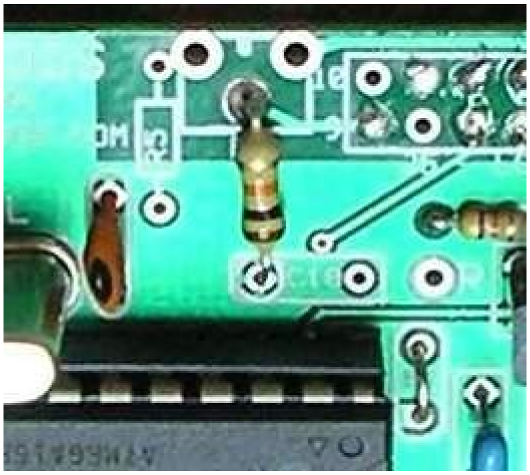
Version 1.3 Nov 2009



Assembly

Verify that you have all the correct parts for your kit. Be very careful unpacking the DDS chip. It's a tad fragile, I usually pack it in with the PCB. It's in a small piece of antistatic rail tube. It's best not to take it out of that until you are ready to solder it. Start by soldering in the power supply components. D1, U4, C8, C9, C10, you can use a power connector in J2, or just solder your power wires directly to the PCB. Double check the polarity on D1, C9, and the orientation of U2. Apply 12VDC power and check to see if 5VDC is present at U2 pin 7. If this is all ok, next soldering down the DDS chip. It's the tiny 16 pin chip AD5932. It will be easier to solder it down if he rest of the board is nearly empty. The best method I have found for this chip is the use a low wattage soldering iron, 25 watts or less. Use a sharp fine tip. (I just sharpened my old tip with my dremmel tool) Carefully align the chip in the right location. Make note of the direction. Pin 1 goes in the lower right. Pin1 on the chip has a dot indent in the plastic package. Take a look at the close up photo to the left. Align the leads on the pads and make sure it's centered left to right. I usually hold the chip down with a piece of tape leaving one side of the chip exposed. Solder one of the corner pins. Re-align the chip, and solder the rest of the pins on that side. Remove the tape and solder all the remaining pins, don't worry too much about solder bridging. After you are done use solder wick to remove excess solder and any solder bridges. If the solder wick becomes frozen (soldered) to the board, don't rip it off, heat it up with your iron and pull it up carefully. Inspect it carefully by eye under a magnifying glass. Clean it with flux remover, and make sure there is no debris under the chip or elsewhere. Use a DVM or continuity tester to make sure there are no pin to pin shorts, no sorts to power or ground, and all the connections are made but testing right as the lead exits the package, then on the PCB at the appropriate point.

Next install the 4 capacitors next to the DDS chip, C1, C2, C3, C4 be careful with these to avoid any solder shots, and trim the leads neatly. Solder in the rest of the components. Initially I'll be shipping kits with a 28 pin socket for the micro, please use it, if there is a software bug, it will be easier to reprogram your micro that way. The XTAL can be mounted standing up, or laying down, though it hangs over the edge a little bit when paying down. Solder in the remaining components.



R6 installation: R5/R6 set the LCD voltage. When using the Standard LCD display R5 is not used, and a fixed 10K resistor is used instead of the trimmer for R6. When installing the 10K resistor solder one end in the middle hole for the R6 trimpot, and the other end (ground) in the left hole for C18 (C18 is not used) See photo at left. If you choose to use a different LCD, use a 1K resistor for R5, and a 1K trim pot for R6. There are a few LCD's that don't need an adjustment, just ground the center pin of R6. And a few that require a negative voltage, you're on your own with that. See my latest LCD info here: http://www.pongrance.com/dds-lcd.html

Even if you are going to use a different LCD I recommend that you initially build the kit with the Serial LCD module supplied, then change it out later once every thing is working ok. A number of components can be installed on the front or the back of the PCB. Which you use will depend on how you plan on mounting the DDS. The Switches, the rotary encoder, the LCD connection, the RCA jack, all can be mounted on the back if you like. All function the same regardless of the side they are mounted on except possibly the LCD connection (see below.)

Power connector: The PCB is laid out to accept a 2.1mm power connector, but it isn't really needed. A connector that fits is available from All Electronics, they have 3 pins. Just break off the side pin. This is a 2.1 mm coaxial power connector, +12v inside, ground outside. If you don't have a power connector, just solder your power wires directly into the PCB holes.

Bill of Materials

Designator	Part	Quantity Supplied	Description / Appearance
U1	Micro ATMEGA168	1	28 Pin Dip
U1	Socket	1	28Pin DIP socket
U2	DDS Chip AD5932	1	Tiny 16 pin TSSOP
PCB	Circuit Board	1	N3ZI DDS
XTAL	13.44 MHz XTAL	1	Marked ECXR3392
U3	Oscillator	1	Metal can 50MHz
J5 LCD	LCD	1*	KTM-S1201
C1,C2,C3,C6	Bypass Capacitor	4	See packing list
C11, C12, C15	Bypass Capacitor	3	See Packing list
C8, C10	.05uF Capacitor	2	Brown .05
C16, C17, C18	Not Used	0	
C13, C14	27pF Capacitor	2	Brown
C9	Electrolytic Cap	1	Black Radial 20-100uF
C4, C7*	68pF Capacitor	2*	Orange 68
C5	150pF Capacitor	1	Orange 151
L1,L2	1.5uH Inductor	2*	Brown, Green, Gold
U4	Volt Reg	1	7805
D1	Diode	1	Black epoxy
D2	Signal Diode	1	Glass
Q1,Q2	2N4124	2	To-92 2N4124
R2,R4	47K Resistor	2	Yellow, Purple, Orange
R3	4.7K Resistor	1	Yellow, Purple, Red
R7	Not used	0	
R1	0 (Shorting Jumper)	0	Jumper
R5	See Text	0	For Character LCD's
R6/opt	See Text	0	For Character LCD's
R6	10K Resistor	1*	Brown, Black, Orange
R-ENC	Encoder	1*	Varies
SW1,SW2	Push button Switch	2*	
J4	Shorting Jumper	0	Jumper
J1	RCA Jack	1	
		* Not supplied with basic (RS232 only) kit	
	U3, L1, L2, C5 have different values with the overclocked version.		

Note the appearance color and value of some parts change from time to time with differences from various suppliers. You should receiver a packing list with your kit, which should have to current description for the parts in your kit, thus superseding this list.

Shown below is the wiring connections to the LCD. If you are using the supplied serial LCD module, you can see from the chart that connections are straight through if you mount the LCD module and DDS PCB back to back.

The pin numbers are clearly marked on the PCB, and carefully wire the power per the diagram. If you choose to use a connector to remote your LCD be especially careful. You can use a 10 pin header a connector for the LCD, and if you mount this to the front, and then you plug in a 10 conductor flat cable with a standard IDC connector, then the even and odd pins will be reversed.

(If you put the header on the back of the PCB, as in the photo, then pin numbering will be fine.) This is not a problem unless you forget to account for this when you connect the other end of your cable to the LCD. Be careful, and double check all connections before applying power. Reversing Power and ground will likely damage your LCD module. If you are using the standard LCD and want connectors and a cable. Solder a 10 pin header on the front of the DDS-PCB, and one on the back of the LCD module. Then use a standard 10 pin flat cable with standard connectors to interconnect them.

PCB Pin Number	Standard LCD Connection		HD44780 LCD Connection	
1	Vcc	1	Vcc	2
2	Gnd	2	Gnd	1,5,7,8, 9,10
3	SCK	3	E	6
4	SI	4	DB4	11
5	C/D	5	RS	4
6	Reset	6	DB5	12
7	/BUSY	7	N/C	
8	/CS	8	DB6	13
9	Vlcd	9	Vlcd	3
10	GND	10	DB7	14

There are two special I/O pins on the lower center of the PCB, labeled PB5 and PC0. PB5 is used for VFO A/B select. An open will select VFO B, a short to ground will select VFO A. You can connect this to a switch, or relay, but any voltage over 5v or negative, will likely damage the microprocessor. Use an opto isolator to connect to anything that may have a high voltage. Only 200uA will flow when PB5 is grounded.

The other special pin is PC0, this can be used for blinking an LED whenever the frequency is changed. It basically acts as an open collector output with a weak pull up. Connect the anode of the LED to +5v, connect the cathode of the LED to a 1K resistor, and the other side of the resistor to PC0.

Output filer: This DDS uses a D/A converter to generate the sine wave output. This D/A converter is clocked at 50Mhz. The maximum frequency that can be output is 25Mhz, this is refereed to the nyquist rate. All D/A converters have an image output above the nyquist frequency. (If you go to wikipedia, and search for nyquist you will find a nice article describing the theory) So if you generate a 24Mhz signal, which is 1 Mhz below the nyquist frequency, there will be a mirror image exactly 1 MHz above the nyquist rate, at 26Mhz. If you generate a signal at 15Mhz the mirror will be at 35Mhz. Normally an output filter is used to filter out this signal. My DDS PCB has component holes for up to a 5th order output filter. However, depending on your application you may or not be concerned about the mirror signals. For example, if you have a radio with a 9MHz IF, and are generating a VFO in the 5.0MHz range for operation on 80 meters. You will have a mirror signal from the DDS at 45MHz range. It's not likely that this signal will cause any problems.

With the standard output filter, (1.5 uH inductors, 68pF input and output caps, 150pF center capacitor) will allow operation up to 20MHz, with about 20db of attenuation of the 30MHz mirror. If you need something better than that you will have to design your own filter, keep in mind that the output impedance is 200ohms. In the overclocked version, the master clock is 80MHz and the output filter is changed (1.0uH inductors, 100pF center cap) to allow operation up to 30MHz.

Jumpers: You need to install a shorting jumper in the R1 space if you have not done so already. And if you do not have L1 or L2 in your kit, you need to jumper those. If you are using the standard serial LCD you need to jumper pin 10 to ground. You can either use J4 on the PCB, or put a jumper from pin 10 to ground on your LCD module.

Operation

Memories: There are a total of 12 memories, each holds a Frequency and an IF. In most cases all the IF's will be set the same. Memories 0 though 9 and standard memories but are tunable. That is if you turn the dial while at that memory the frequency will change. When you switch to the next memory, the last used frequency will be saved for the prior memory location.

The other two memories, are VFO A and B. These are different in that a separate control line selects between the two. You can program a different IF in for VFO A and B, but I doubt you will want to. You can set any or all IF's to 0.00.

Normally SW1 and SW2 control memory up/down, and the RENC push button controls tuning step. But you can reassign these using the rs232 advanced commands. You can also remote any of these and mound different buttons elsewhere, it's easiest to simply wire the remote button in parallel.

With the standard serial LCD a single character on the left side of the display indicates the memory number (0 to 9) or A, b for the VFO's With the character based 44780 LCD's MEM#0 to MEM9, and VFO#A, VFO#B will be displayed.

Dial speedup. The default frequency increment is 50Hz per tick, but there is a fine tune mode with 10Hz per tick, and fast tune modes for 1KHz, 10KHz, , 100KHz, and 1MHz steps. Fast tune modes will quantize lesser digits to 0. When using the serial LCD the fine mode will be indicated with an extra decimal point at the far right. The fast tune modes will be indicated by blinking the corresponding digit. You change speeds by pressing the rotary encoder. Many parameters associated with tuning can be changed using the rs232 port, including default tune step, fast tune quantiziing, cursor type, etc. Refer to rs232 manual for details.

Setting IF's To enter the IF set mode, remove power, hold either SW1 or SW2 and apply power. It will then come up in a mode, which will allow you to set the IF's. "IF" will be indicated on the LCD display. The DDS will output the IF frequency in this mode, which can be used to zero beat with your BFO if applicable. Then when complete, toggle through all memories, to make sure they are all ok, then remove power. It is a little bit easier to set the IF's through the RS232 port, and you can do so without powering in the special IF mode.

Additive IF: This VFO supports 3 different IF modes. The simplest is the additive, and example of this is the one used on the BITX20. The IF is 11MHz and the VFO tunes 4.0 to 4.35, resulting in an operating frequency of 14.0 to 14.35. For this type of operation enter the IF as a positive number, and enter the operating frequency as a positive number. It will compute the proper VFO frequency to generate. Note that this type of radio can never operate below the IF.

Subtractive IF: Now with subtractive IF schemes, there are two options for the VFO, high side, and low side VFO injection. For example, many 40m QRP radios have a 11MHz IF, with a VFO ranging from 4.0 to 3.7MHz. Tuning is backwards, so the higher the VFO the lower the operating frequency. To program you DDS VFO to work this way, enter the IF frequency as a negative number, i.e. –11.000.0 in this case, and also enter your operating frequency as a negative number, to indicate low side injection. i.e –7.000.0, the DDS microcontroller will computer the proper VFO frequency to use, 4.000.0 in this case. Note that if you are using the serial port, you can turn on the debug mode, and it will display the IF, VFO, and operating frequency every time. Note that most radios will also work with high side injection without modification. For example you can charge the operating frequency to +7.000.0 and you will be on the same frequency, but with high side VFO at 18MHz. Usually when you do this, the opposite sideband is selected.

Dial Direction: The dial direction may be reversed because of the use of negative IF's and/or high vs. low VFO injection. The dial direction can be reversed by changing the sign of parameter 9, (tick_div) via the serial port. 0.02 or –0.02 are values to try for this parameters.

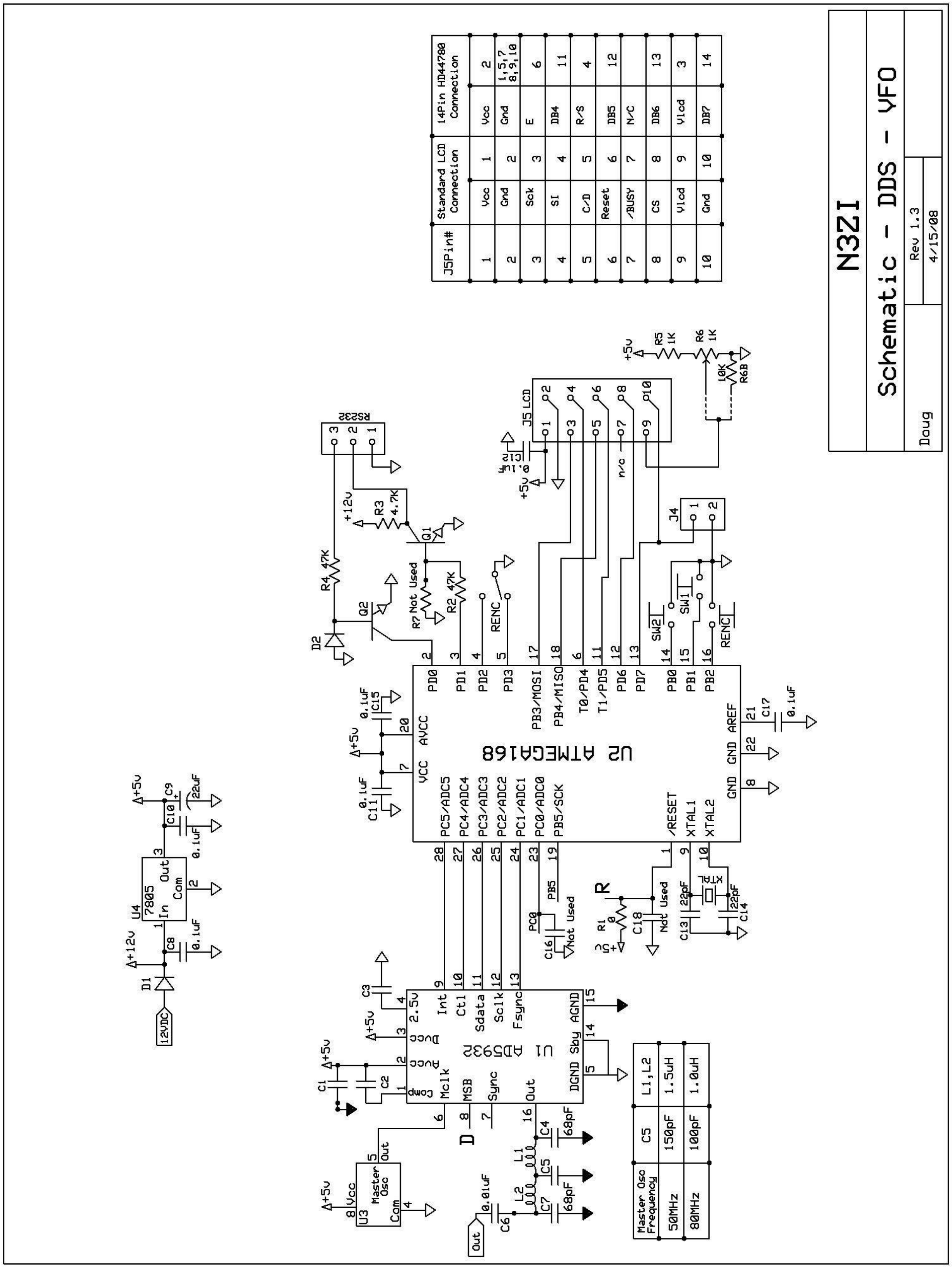
Expanding your radio's coverage: In the case mentioned above, the BITX20 with an 11MHz additive IF scheme, you can get this radio to receive 10MHz WWV, with a 1MHz or 21Mhz VFO. To get your DDS to do this, make the IF negative, i.e -11.000.0 and the operating frequency to +10 or –10 MHz. If you use low side, which results in a 1MHz VFO you will also receive the image on 12MHz. You could also cover the 30M amateur band. It's interesting to do this with the receiver, but you should never try to transmit unless you have redesigned the filters for the new band. In most cases receive sensitivity will be substantially lesser, but it will frequently work for WWV or other strong signals.

Output: The output signal is not amplified, and the level is approx. 250mV peak to peak which works well with SA612 or similar mixers. Output impedance is 200 ohms. With vintage radios, or other applications, you may need a separate buffer amplifier. Space is provided on the PCB for up to 5^{th} order low pass filter. Depending on your application, a filter may or may not be needed, or may be redundant with other filters in your radio. If you are not using the filter at all, jumper both L1 and L2, omit C5 and C7.

It is normal in all DDS to have spurs above 25MHz, and the purpose of the filer is to remove these. The 5^{th} order filter supplied is sufficient for operation up to ~20 MHz. You can, theoretically, go to 25MHz, but you'll have to pay attention to the spurs. If you are designing a new filter keep in mind that the I/O impedance is 200 ohms. You can convert the existing filter to a 7^{th} order elliptical by adding caps in parallel with the inductors. Elliptical filters look good on paper, but in practice, they are very sensitive to precise component values, and almost always need to be tweaked to get good performance.

Calibration: It is normally not necessary to calibrate your DDS-VFO. The 50MHz reference oscillator is specified at 100ppm accuracy, but they are normally much more accurate than this. Calibration is done in software, set via the rs232 port, the parameter is called tb_cf (timbase correction factor) it is an extremely fine adjustment, much finer than the fundamental step size. The oscillation frequency of the 50Mhz timebase will not change, but the mathematics will be adjusted. 64 bit math is used in computing operating frequency to be certain, that no error would be introduced by rounding. A change of +0.20 in this parameter is approx. a 10 Hz change of operating frequency at 20MHz. Keep in mind that it is a multiplicitive correction factor, so a 10Hz change at 20MHz, will result in a 5Hz change at 10MHz. (The nominal value here is 562,949.95 when a 50MHz master oscillator is used. In the 80 Mhz overclocked version, the nominal value for this parameter is 351,843.72)

Extras: You may have noticed holes marked "D" and "R" on the PCB. The "D" hole is a digital output of the DDS chip. it's a square wave at the same frequency as the sinewave output. He amplitude is higher, but square waves contain lots of harmonics. You should refer to Analog Devices data sheet for the AD5932 if you want to use this output. The hole marked "R" is for an external reset. If you want to use this replace the jumper at R1 with a 10K resistor, and add a 0.1uF capacitor in C18. This isn't all that useful, but it allows you to re-boot the micro without removing power.



J5Pin#	Standard LCD Connection		14Pin HD44780 Connection	
1	Vcc	1	Vcc	2
2	Gnd	2	Gnd	1,5,7 8,9,10
3	Sck	3	E	6
4	SI	4	DB4	11
5	C/D	5	R/S	4
6	Reset	6	DB5	12
7	/BUSY	7	N/C	
8	CS	8	DB6	13
9	Vlcd	9	Vlcd	3
10	Gnd	10	DB7	14

Master Osc Frequency	C5	L1,L2
50MHz	150pF	1.5uH
80MHz	100pF	1.0uH

