

A credit card sized Ethernet Arduino compatible controller board

by [drj113](#) on July 4, 2010

Table of Contents

License: Attribution Non-commercial Share Alike (by-nc-sa)	2
Intro: A credit card sized Ethernet Arduino compatible controller board	2
step 1: Here is the Schematic Diagram	2
File Downloads	3
step 2: The PCB Layout	3
File Downloads	3
step 3: Soldering the Components	4
step 4: Programming the Firmware	5
File Downloads	5
step 5: But what does it do????	6
step 6: Parts List	6
step 7: KiCad Files	7
File Downloads	7
Related Instructables	7
Advertisements	8
Comments	8

License: [Attribution Non-commercial Share Alike \(by-nc-sa\)](#)   

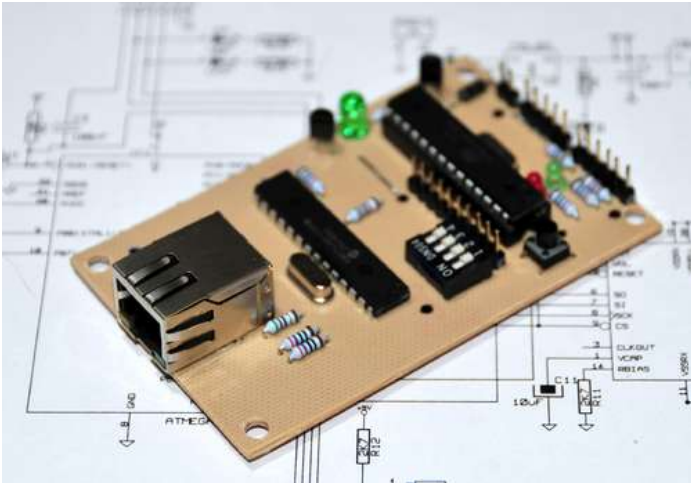
Intro: A credit card sized Ethernet Arduino compatible controller board

I love the Arduino as a simple and accessible controller platform for many varied projects. A few months ago, I purchased an Ethernet shield for my Arduino controller to work on some projects with a mate of mine - it was a massive hit - for the first time, I could control my projects remotely using simple software.

That got me thinking - The Arduino costs about \$30AUD, and the Ethernet board cost about \$30AUD as well. That is a lot of money - Could I make a simple, dedicated remote controller for much cheaper? Why Yes I could. Could I make it the size of a credit card? Why Yes - I could!!

This project is my simple Arduino compatible controller that has embedded Ethernet, and the capacity to drive some extra I/O lines for projects, such as a Remote thermometer, a Remotely accessible Fridge controller, and a Remote Humidity sensor. I have to say from the start that I didn't write all of the software, my mate Mikal did that - but this instructable is about making your own controller board!

Lets start!



step 1: Here is the Schematic Diagram

For the curious, this is the schematic diagram of my simple Ethernet board.

As you can see, there are a number of exposed header connectors that can be used to connect peripheral devices to.

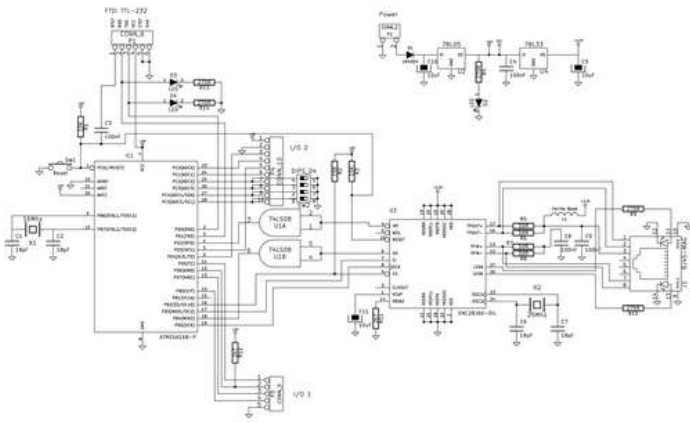
The board is powered with a supply of between 7 and 12v. It contains voltage regulators to provide +5v and +3.3v for the Ethernet controller.

There is also a 4 position DIP switch that can be used to allow programmed functions to be modified. A failing of the standard Arduino Ethernet library is that the IP address for the board has to be set in code. Using the DIP switch, a block of addresses can be selected from as required. You can make 16 boards, and have each board automatically select a different address based on the switch setting. This is *really* handy when you have deployed 10 sensors around the house. All you need to do is set a switch and then they are configured.

The pinouts of the I/O connectors are;

I/O1 - 1 - PD5 (Arduino Pin 5)
I/O1 - 2 - PD6 (Arduino Pin 6 +pullup to +5v) - Used to connect a DS1820 Temperature sensor.
I/O1 - 3 - PD7 (Arduino Pin 7)
I/O1 - 4 - PD8 (Arduino Pin 8)
I/O1 - 5 - GND

I/O2 - 1 - +5v
I/O2 - 2 - GND
I/O2 - 3 - PD4 (Arduino Pin 4)
I/O2 - 4 - PC0 (Arduino Analog 0)
I/O2 - 5 - PD3 (Arduino Pin 5)
I/O2 - 6 - PC1 (Arduino Analog 1)
I/O2 - 7 - PC2 (Arduino Analog 2)
I/O2 - 8 - PC3 (Arduino Analog 3)
I/O2 - 9 - PC4 (Arduino Analog 4)
I/O2 - 10 - PC5 (Arduino Analog 5)



File Downloads



Arduino-Ethernet-schematic.pdf ((595x842) 59 KB)

[NOTE: When saving, if you see .tmp as the file ext, rename it to 'Arduino-Ethernet-schematic.pdf']

step 2: The PCB Layout

Here is the PCB layout.

As with all of my projects, I make the circuit boards using press-n-peel blue as a toner transfer, and cuperic chloride as the etchant.

This layout can be printed onto a laser printer directly and used as artwork.

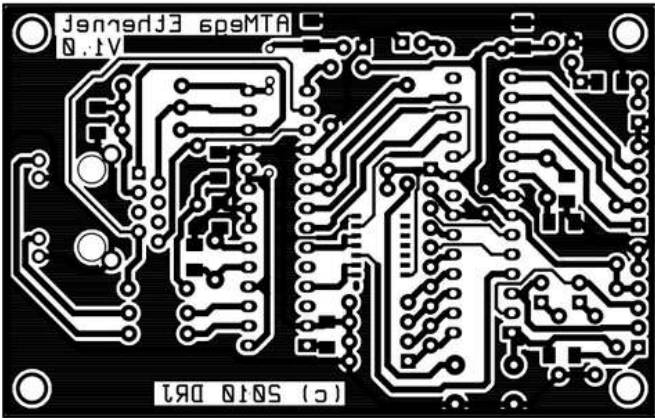
I wrote detailed instructions in my Arduino Wordclock writeup that you can follow to etch your own board. Check out <http://www.instructables.com/id/The-Word-Clock-Arduino-version/step2/The-hardware-Make-the-controller-board/> for details.

One thing with this layout - I had to use a surface mount IC (a 74HC08) - Please do not be too scared by the surface mount technology - it is extremely simple to solder - as we will see in the next step.

Anyway - Go on - make a PCB, and check out the next step for how to assemble the board.

Remember - download the PDF file as the master - not the PNG picture - the PNG is just there so that you can see what it will look like - it is almost certainly not to scale! If you print the PDF full size (without scaling) then it is the exact correct size for Toner Transfer.

Udate - I just made another controller and realised that the pin spacing on the Ethernet jack was slightly off.... It has been fixed in the attached PDF layout.



File Downloads



Arduino-Ethernet.pdf ((595x842) 47 KB)

[NOTE: When saving, if you see .tmp as the file ext, rename it to 'Arduino-Ethernet.pdf']

step 3: Soldering the Components

Now that we have a PCB, it is time to solder the components onto the board.

Download the attached top and bottom pictures of the board to let you know where the various parts are placed.

Also download the file to let you know where the 3 jumpers have to be installed on the top side of the PCB.

Lets start by getting the surface mount 74HC08 onto the board. I promise that it is not as hard as it looks.

Firstly - clean your work surface - there is nothing more frustrating that working in a mound of junk, when you have to do something carefully.

Continue by tinning the pads where the IC will be mounted - just apply a little bit of solder, not a great mound. Once the pads of the PCB have been tinned, get the part, place it onto the tinned pads, double check it is oriented correctly - The PCB has a dot where pin 1 should be - make sure that the part is facing that direction.

Then using a fine tipped soldering iron, touch one pad on a corner of the chip to heat it. Let the solder melt, and then let it cool. Look closely at the part to verify that it is still oriented correctly, and aligned with all the pads. Then solder the opposite corner.

Now, under a decent light, using a magnifying lamp, spend a minute double checking that the device is sitting correctly on the pads - if it isn't, just re-heat a pin, and fix it up.

Once you are happy that the alignment is correct, heat the rest of the pads to melt the solder and connect the device. If necessary, add just a tiny bit of solder. Again - double check that your work is neat and clean - If you end up shorting pins together, don't panic - just use a little bit of Solder Wick to remove the excess solder.

Once you have the 74HC08 soldered, pat yourself on the back, and go and show your fine work to a significant person in your life!! - You did it! You can now solder Surface Mount parts! A whole new world awaits you!

Next solder down the surface mount capacitors on the back of the board - they are *simple*, just use a similar process to what you have already used - and do not be afraid to add a little bit of solder to keep the joints neat.

Next, turn the board over and install the jumpers on the top side.

Continue assembly by mounting all of the resistors, capacitors, the ferrite bead, IC sockets, LEDs and connectors.

Finish off by plugging the Micro controller into the socket, and the Ethernet controller into its socket.

Finally - spend a couple of minutes under a strong light double checking your work - If I had a dollar for every stupid time I forgot to solder a pin, and spent a night debugging something silly - I would be able to spend the rest of my life writing Instructables projects.... Sadly..... Anyway, where was I? Ahhh.

There - You have done it - you now own your first Ethernet micro board!

Now we just have to load up some software.

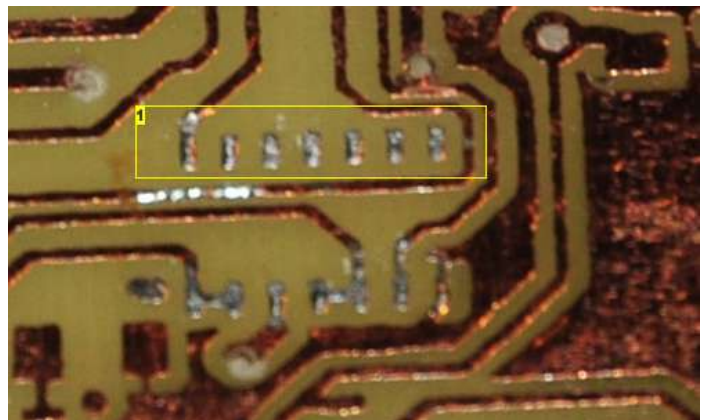
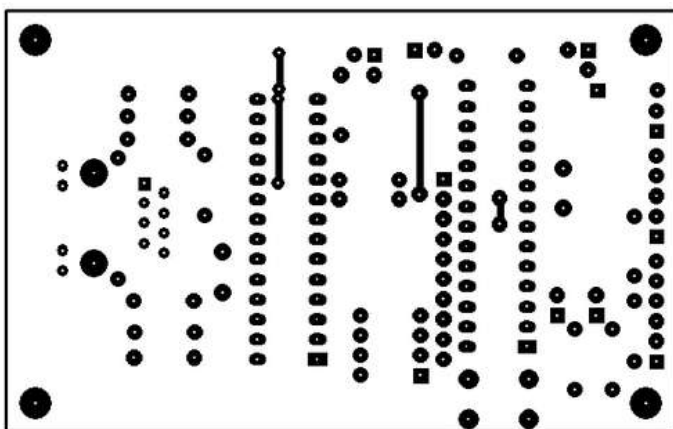
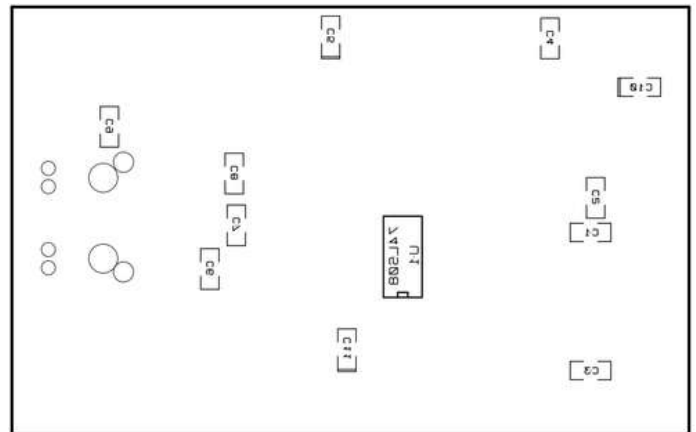
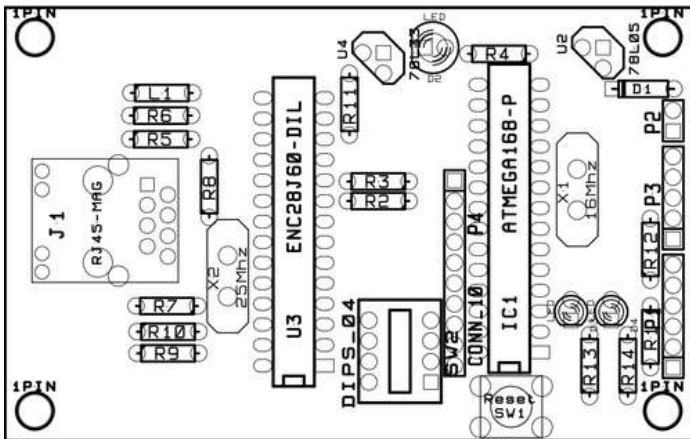


Image Notes

1. Tinned Traces

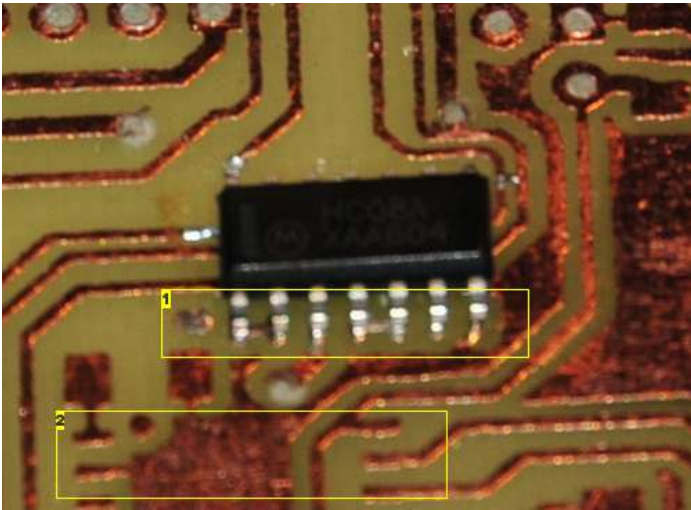


Image Notes

1. Pins gently soldered in place
2. Don't worry about these extra pads in this photo - this board is actually a full surface mount version of the controller that I am working on....

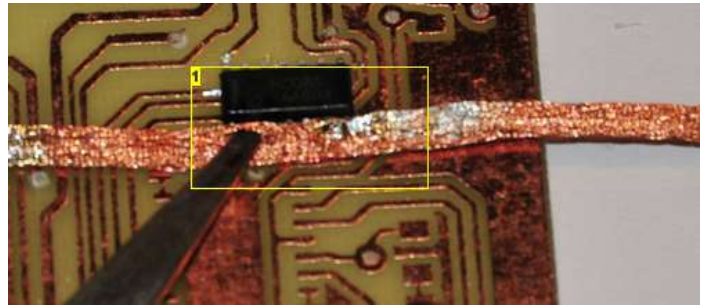


Image Notes

1. If you short some pins don't panic - just use a little bit of solderwick, and it will be better!

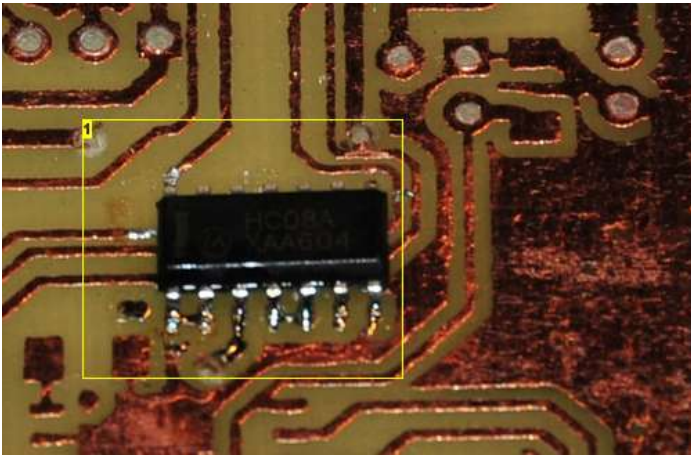


Image Notes

1. Completely soldered - Easy!

step 4: Programming the Firmware

Now - we can load up the firmware.

Attached is a sample project - in this case, it is the trivial web server example from the Arduino library - it uses a DS1820 chip connected to Pin 2 of I/O 1.

I have to say that I did not write the web temperature project - it is the sample one from the Ethernet library - It works beautifully on this board!

Use a FTDI-TTL cable to download the code - you can get one of the cables from <http://evilmadscience.com/partsmenu/130-usbttl> They only cost \$20, and I can guarantee that every project you make in the future will have that magic 6 pin interface....

File Downloads



etherShield_web_temperature.pde (9 KB)

[NOTE: When saving, if you see .tmp as the file ext, rename it to 'etherShield_web_temperature.pde']

step 5: But what does it do????

Ahhh - I was wondering when you would ask this question.

I have these boards strewn all around Mikals house - they are doing things ranging from measuring temperature, to measuring humidity, to controlling the temperature of the beer fridge.

Here is a photo of one of them in situ - in this case it is measuring the humidity near Mikal's books. This is an old version of the board - but it has been doing sterling service.

If you want a copy of Mikals code from any of his systems, check out Mikals blog at <http://www.stillhq.com>

The old version of the Beer controller is at: <http://www.stillhq.com/arduino/000003.html>

and the Hygrometer is at: <http://www.stillhq.com/arduino/000007.html>



Image Notes

1. Arduino Ethernet Board
2. Hygrometer
3. An old Microsoft DVD - Good as an extra 12mm of shelf padding!

step 6: Parts List

Here is the parts list:

Semiconductors:

Micro - ATmega168 or ATmega328 - DIP (the 328 provides extra code space)
Ethernet - Microchip ENC 28J60 - DIP
74HC08 - Surface Mount
3mm LED X2
5mm LED
1N4004 Diode
78L05 Regulator
78L33 Regulator
16 Mhz Crystal
25 Mhz Crystal

Resistors

50R x 4
270R x 5
2k7 x 2
10K x 3

Caps:

18pF x4 - Surface Mount
100nF x 4 - Surface Mount
10uF x 3 - Surface Mount

Hardware:

28 pin socket x 2
4 pin dip switch
RJ45 MAGJACK - Has to be a MagJack, as that has the ethernet isolation transformer inside it http://www.sparkfun.com/commerce/product_info.php?products_id=8534
10 pin header
6 pin header
5 pin header
2 pin header
Small Pushbutton switch

<http://www.instructables.com/id/A-credit-card-sized-Ethernet-Arduino-compatible-co/>

step 7: KiCad Files

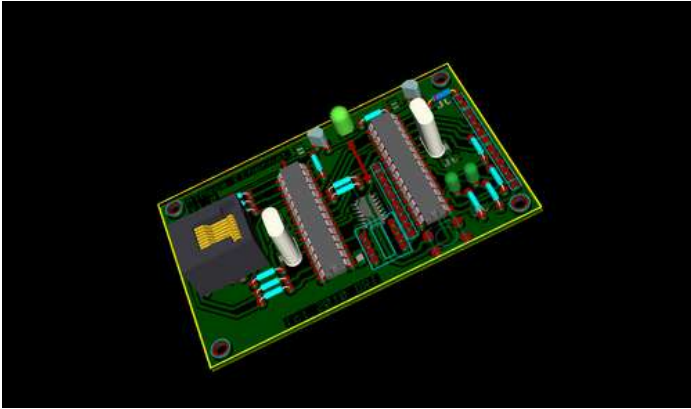
Many people have been asking for the source files for this project.

Here they are. They are in KiCad format. KiCad is Open Source, Free, Accessible, and does not apply arbitrary license restrictions. As a community, I feel that we should be supporting open source software.

I appreciate that some people would like to see Eagle versions, but I simply can not condone Eagle's licensing model. I am a private user, but the size limitation of the boards in Eagle means that I can not use that product to make trivial 150mm x 150mm PCBs for my clocks. So I simply do not support their product.

Anyway - here is an archive of the project tree from my system - I hope you find it useful! Please keep in mind the CC attribution license for my projects.

update I had received some feedback that the custom libraries that I made for the MagJack and the enc28J60 were missing - I have added them to the download - it *should* work - but these things are always hit and miss if you haven't tried them on another machine... (You do build up a collection of useful libraries that you don't even realise you use...) I also removed the LCD module library reference, as there is no LCD module installed.... It was just the default, as many of the projects that I make have LCD screens on them.



File Downloads



ArduinoEthernetController.zip (678 KB)

[NOTE: When saving, if you see .tmp as the file ext, rename it to 'ArduinoEthernetController.zip']

Related Instructables



A Remotely Programmable Relay Controller (Christmas Lights or Home Automation Controller) by drj113



A watering controller that can be home networked by drj113



Arduino HVAC Servo Thermostat/Contr by tikka308



The Arduino AA Undershield by Artificial Intelligence



Displaying Twitter feed without a PC! by fabrizio.granelli





Awesome Moving Gears Controlled by Awesome Buttons (to be continued) by waynefan


Comments


50 comments [Add Comment](#)

[view all 115 comments](#)

 **wcbzero** says: Sep 15, 2010. 2:02 PM [REPLY](#)
Any chance you have a part number for the mag jack? I picked some up and found they had a different pinout.

 **drj113** says: Sep 15, 2010. 2:19 PM [REPLY](#)
It is this one: http://www.sparkfun.com/commerce/product_info.php?products_id=8534
I have amended the parts list to include that link. I don't see why any other MagJack couldnt be used, although the board may need to be changed.

 **wcbzero** says: Sep 15, 2010. 3:38 PM [REPLY](#)
Mine wont work because the pins are strait across in a single row, but thanks!

 **madeintaiwan** says: Sep 11, 2010. 10:53 PM [REPLY](#)
Hey,
Just in the midst if building this and wanted to double check a couple items:


1. Is "L1" (upper right above R6) "R1" or am I missing something...
2. What is "R1P1" (bottom right directly below R12) Is that "R1" ?
3. Could you make a note in your materials list what is surface mount and what is through hole, as I have caps but it wasn't until I got to this page and read that they were SMD caps did I realise. It's my fault really for not reading the whole thing first. Maybe I can solder the normal caps (making it more than credit card size) for round one.


The only other question I had was , in your opinion, would it be possible to adjust the magjack for this one : <http://www.allelectronics.com/make-a-store/item/MT-86/SHIELDED-RJ-45-JACK-W/YEL/GRN-LEDS/1.html> (i'm in LA and can buy them locally) the things is the pins for the LEDs are in back not in front like the one from Sparkfun....


Otherwise really brilliant Instructable, I have about seven million uses for this and can't wait to get it up and running.

Thanks

Nathaniel

 **drj113** says: Sep 11, 2010. 11:30 PM [REPLY](#)
Hi Nathaniel,
L1 is a piece of wire with a small ferrite bead on it.
R1P1 is the blend of the label for the resistor (R1), mixed gently with the label for the pin connector P1. Opps!
Sorry about the materials list - all of the Caps are SM, as is the 74HC08 chip. I will fix that.
The jack that you mentioned does not have the ethernet transformers there, so it should not work. I say should not, as one enterprising person simply replaced the MagJack with a RJ45 socket, fiddled with the pinout connections (believing that I had incorrectly specified the wrong connections) and it worked. I would not recommend it.

 **madeintaiwan** says: Sep 12, 2010. 1:29 PM [REPLY](#)
Hey,
While I was a bit of a dummy and didn't read the guide in total first, I did read your conversation about the RJ45 and Mag Jack difference. As I'm trying to build this sort of hurriedly I wanted to source the Mag Jack locally. I saw the one from All Electronics and then spent a while on the phone with Bell Stewart (which is the original manufacturer of the part) the other day. The guy I spoke to said while part # SI-50014 has been replaced with a RoHS part that the circuit diagram for both is the same and that the RoHS version does have magnetics. Here's a link: <http://www.datasheetarchive.com/pdf-datasheets/Datasheets-6/DSA-119029.html> . Are those the transformers (magnetics)? Hopefully they are, if not I'll order some from SF.
Again, thanks a bunch for a great Instructable. I'm trying to get it into Fritzing at the same time as building the actual part.
Nathaniel

 **drj113** says: Sep 12, 2010. 2:35 PM [REPLY](#)
That jack looks great - the pinout is pretty close - it looks as though the LEDs are on the back of the connector, not up the front, so there will need to be some board layout changes.
How are you making the board?
Doug



madeintaiwan says:
Hey Doug,

Sep 12, 2010. 5:07 PM [REPLY](#)

That Jack is pretty good, cheaper than the SF, although it may be more emotionally taxing going in the Valley to get them... Let me know if you can swap the pins around to fit that jack otherwise I'll figure a way to modify that drawing

I was going to try and make the board in Fritzing too (have you used Fritzing?), although no one has yet to make a couple parts: ethernet jack, ENC28J and the SMD components.

Currently I've got the resistors, the DIP, the crystals, the jack (i've had to trim off the LED legs on the jack and then solder connections back over to the front connections), voltage regulators and the diode connected. I'm going to order SMD caps as I can't rightly make this incorrectly.



drj113 says:

Sep 12, 2010. 7:06 PM [REPLY](#)

Not a problem - I will adjust the layout for you and submit a new PDF for toner transfer.

I have not used Fritzing. I use KiCad, and it works beautifully for me.

If you haven't heard anything within a week, just chuck me a small reminder - Things here are hectic.



pocketscience says:

Sep 1, 2010. 6:44 PM [REPLY](#)

Hi, How are you getting away with those small TO-92 regulators? I thought they could only provide around 100-150mA? Max requirement on the ENC is 180mA, and once you start adding external components you'll need more than 150mA for the ATmega as well right? Cheers,



drj113 says:

Sep 1, 2010. 8:41 PM [REPLY](#)

I agree that it is tight, but my measurement of current consumption is about 135mA for the lot. The kinds of sensing components that I add only add a couple of mA to the load.

I would *love* to identify a better switching reg, but locating one that is (a) cheap, and (b) easily available is a bit tricky.



meurig says:

Aug 3, 2010. 4:15 AM [REPLY](#)

I bought the wrong capacitors (non-surface mount) so I'm trying to go in to kicad to change the board layout. Sadly I'm getting errors like the following: "The following libraries could not be found: rj45-mag dips-s relay_spdt microchip-enc28j60" Are these libraries/modules you've created yourself? I'm new to KiCad so not really sure where to look or even what I'm looking for.



bigjeff5 says:

Sep 1, 2010. 1:07 PM [REPLY](#)

I'm in the same boat, I think these may be custom libraries because I can't find anything but the dipswitch library. If you try to edit the pcb an LCD library pops up as well, which doesn't even appear to be used anywhere, it's just in the library list.



drj113 says:

Sep 1, 2010. 2:57 PM [REPLY](#)

There you go - I just re-loaded the zip file, with what I hope are the missing libraries - if you have any more problems, let me know.



drj113 says:

Sep 1, 2010. 2:26 PM [REPLY](#)

Yes, they are libraries that I made for components that I couldn't find - I will try to extract them from my machine and attach them to the Instructable.



Oldfrt46 says:

Aug 31, 2010. 6:33 AM [REPLY](#)

Do you have an estimate of total cost to build the device? Also sources for obtaining the parts and hardware would be helpful. Really great instructable.



petergams says:

Aug 27, 2010. 6:00 AM [REPLY](#)

Please excuse my newby question. Is it possible to have about 5 dry Com, NO ,NC contact switching? I have an idea but all I need is about 5 clean no voltage contacts to switch from open to closed contacts. Is it possible? Poiter



drj113 says:

Aug 27, 2010. 3:18 PM [REPLY](#)

HiPoiter

It certainly is. You have just reminded me of what I was going to do later. I got sidetracked on another project....

You would drive each relay using an NPN switching transistor (BC548). It isn't very complex. The Arduino would drive the base (Via a 470R resistor), the Emitter would be connected to ground, and the relay coil would be connected between a 9-12V supply and the transistors collector.

You would also put a protection diode (1n4004) between the collector and the + supply, with the anode connected to the supply - to protect the transistor from back EMF when the relay switched off.

Watch this space.... I plan on having a PCB ready very soon!



petergams says:

Aug 28, 2010. 12:58 AM [REPLY](#)

Hi drj113, Thanks heaps for your reply. I'm starting to get ideas. How would one go if the Ethernet connection is lost? Would it be possible for the board to run under a default pre-programmed mode? So live controlled thru the adaptor & auto mode with set instructions if connection is lost. Hope you understand what I mean.



drj113 says:

Aug 28, 2010. 4:28 AM [REPLY](#)

Thanks for your question -

What I am doing at the moment, is finishing the layout for a board that supports 6 dry contact relays. Each relay is capable of driving 240v @5A.

What I am planning is to have a default program that operates the relays in a controlled mode if there is no Ethernet driving it, and have the programing capability happen via Ethernet.

Alternately, it can just be a 'slave' board, so that a computer can drive 6 high voltage outputs as it likes. Either is possible. There is a ton of spare code space in the AtMega328 chip left over.

You can email me at doug@doughq.com, and I will send you a picture of the board as soon as it is finished -

I was planning on completing the hardware this weekend, but some other jobs have gotten in the way, pushing it to next weekend. The completed board is 3inches x 3.5 inches, and runs of 12v.



pocketscience says:

Aug 25, 2010. 9:21 PM [REPLY](#)

I bought a ENC28J60 based shield and it was rubbish, but I'm going to give this board a try. Have already built my own stand-alone Arduino-compat boards before, so this should be easy. The SMD soldering will be new though. Just ordered some (cough cough) "samples"... :-)



mfmjos says:

Aug 15, 2010. 7:32 PM [REPLY](#)

This may be a super newbie question. But are you sure you meant 2k7 resistor? Should it be 27k or 2.7k perhaps?



drj113 says:

Aug 15, 2010. 8:35 PM [REPLY](#)

2K7 means 2.7k The way of writing it means that even when the document is photocopied, the decimal point never goes away. 2.7k, after a couple of copies, can easilly turn into 27k, as the decimal fades.....



liodel says:

Aug 11, 2010. 8:49 AM [REPLY](#)

Hi,

I just finish to realise an arduino shield using your schematics and had a trouble with the ethernet plug.

The pins you use are not using the ethernet standard and I had to reroute the signals around the ethernet plug in order to fix the trouble.

The standard is below

- 1 TX+
- 2 TX-
- 3 RX+
- 4 n/c
- 5 n/c
- 6 RX-
- 7 n/c
- 8 n/c

Moreover, there is a link on the schematics between one side of the EMI suppressor and the pin 3 which normally don't exist on the ENC28J60 datasheet (page 7) or on the nuelectronics ethershield based on the same component (pdf is here)

Anyway, thank's for this instructable, I build my shield (picture attached) from your inspiration !

The image contains a schematic diagram of an Ethernet shield circuit and a photograph of the physical board. The schematic shows an ENC28J60 Ethernet controller connected to an Arduino Uno. It includes an EMI suppressor (X2) connected to pin 3, a 2k7 resistor, and various capacitors (100nF, 100pF, 10uF). The photograph shows the shield mounted on an Arduino Uno with an Ethernet cable plugged in.

Pin	Label
1	RD+
2	RD-
3	TD+
4	NC
5	NC
6	TD-
7	NC
8	NC



drj113 says:

Aug 11, 2010. 4:28 PM [REPLY](#)

Hi, Thanks for your feedback. It seems as though you and I are using different ethernet MagJack connectors. I am using the ones from sparkfun, where the external pinout (on the bottom of the connector) does not correspond to the ethernet connector pinout - There are transformers included in the socket itself. The pinout that I have specified is correct for the connectors that have been used. It is important for others to note that you can not simply connect the EXC28J60 to an ethernet socket - there must be magnetic components used between the ethernet socket and the ethernet controller. Regarding the connection for pin 3 - In the sparkfun connectors, pin 3 is the centretap of the transmit transformer - It "must" be tied to 3.3v for the transmit circuit to operate as shown on page 7 of the enc28J60 datasheet. It must not be tied to pin 3 of the ethernet connector without magnetics - that simply won't work. The datasheet for the sparkfun connector that I used is here <http://www.sparkfun.com/datasheets/Prototyping/MagJack.pdf> Where did you get your ethernet magnetic connector?



liodel says:

Aug 12, 2010. 6:15 AM [REPLY](#)

Hi,

I understand better my confusion, because the plug I use is without insulation transformers, and it works, but I clearly understand it's not safe.

I'll integrate these transformers in my next design, thank's for this information, I'll try to find a french distributor for these plug and keep you in touch about the model.



drj113 says:

Aug 12, 2010. 2:24 PM [REPLY](#)

Hi,

Thanks for that - I was confused as well... WOW - I am stunned that it worked, because the transformers are there to remove common mode noise from the Receive lines. You will probably find that your project doesn't work well at the 100m limit for Ethernet cables. :-)

Anyway - I would love to see your future design, and good luck.

Doud



BOSCO says:

Jul 31, 2010. 10:05 PM [REPLY](#)

Great idea. I just have a question on the MagJack. You stated that you used the Sparkfun unit. "<http://www.sparkfun.com/datasheets/Prototyping/MagJack.pdf>" This MagJack has 9+10 and 11 +12 pins for the internal led's. Your schematic shows 11+12 and 13 +14 as the internal led's. There are different types of Magjacks and Sparkfun could have changed the ones they sell. I just bought some from them with no markings on them that are 9+10 and 11+12 for the led's. Are you sure that you bought them from Sparkfun? The reason I asked is, I have built one of your boards and my yellow led on the jack just blinks a few times after a reset the solid yellow. Green led doesn't even come on. Also, maybe related, how does the program changing of the ip address and mac address relate to the switch settings? If I wanted an ip of 192.168.1.15 for example, what would I set the dip switches at? By the way, I programmed the atmega 328p with no problems. TIA for your help



BOSCO says:

Aug 4, 2010. 7:19 AM [REPLY](#)

drj113, could you please address my post. I need help. thanks BOSCO



drj113 says:

Aug 4, 2010. 2:35 PM [REPLY](#)

Hi Bosco, Sorry about not replying sooner, I was a little frazzled from returning from a trip, and I needed to think. Regarding the pinout - the pin numbering is different because when I was assigning pin numbers for the library PCB part, I missed by 2 when I started the LED numbering. I used the PDF that Sparkfun provided for the component, but just got the pin numbering wrong - I am very confident that the part has not changed. The yellow blinking is the library turning the LED on and off during the initialization code - that tells you that the part is operating correctly. How have you set the IP address for the board? In the code provided, the IP address is set using the following 3 lines; `static uint8_t mymac[6] = {0x54,0x55,0x58,0x10,0x00,0x24}; static uint8_t myip[4] = {192,168,1,15}; static char baseurl[]="http://192.168.1.15/";` the code sets the address to 192.168.1.15 I have not provided code to set the IP using dip switches in this instructable. The code is still not ready for release, but essentially I define the variables as non-static, allowing them to be changed. I examine the current settings for the switches, and modify the variables accordingly. It is not elegant. I get the impression from your comment that you can't get the board to work. There are a couple of things you could check based on the original layout that had the spacing errors on the ethernet socket. There is a surface mount cap beside the magjack - it needs to be grounded. It is normally grounded through the case of the magjack. You should probably check that. There are also a couple of jumpers - did you get them all installed? Without them, the project won't work.



BOSCO says:

Aug 5, 2010. 5:19 PM [REPLY](#)

Hi,drj113, and thank you for responding. I found my problem to be the reset switch turned the wrong way on the board. Now I can call up the web page for the temperature, but unable to get a reading. It just stays at 0c. I have the board set with a reserve ip address on the router. I can ping it and get a round trip time. I see both led's on. Yellow solid, green flashes when I press the"Get Temperature" button. So, I know the two are talking to each other. I am using a DS18S20+ temp sensor with pin 1 to gnd, pin 2 to pin 2 on io 1, pin2 also goes thru a 4.7k resistor to board +5v, pin 3 goes to +5v on board. I tried different configurations for the sensor with no luck, even changed out the sensor. For some reason, the board is not outputting data. If it does, the green led on the magjack should blink,correct? TIA for your help BOSCO



drj113 says:

Aug 5, 2010. 6:28 PM [REPLY](#)

Ahhh - well done I discovered a small error in the code // get current temperature #define TEMP_PIN 3 void getCurrentTemp(int *sign, int *whole, int *fract); pin 2 of the I/Oq header (5 pin) is connected to Digital pin 6, so the pin definition should say: #define TEMP_PIN 6 When you connect the temp sensor, I tie the two outside pins to ground (pin 5), and the middle data pin to pin 2 of the header



BOSCO says:

Aug 6, 2010. 12:55 PM [REPLY](#)

drj113, I get +10.62C with your configuration and temp doesn't change when heat is applied. I get +.0331C when I remove pin3 from gnd to +5v from board. Temp will change as heat is applied. So it seems to work but temp should read around room temp of 25C. Even if the decimal place was in the wrong place ,and it was suppose to be +33.1C that would be way hotter for room temp. I have tried both DS18S20+ and DS18B20. no difference. Is there something else in the code that may have been overlooked? thanks, BOSCO



drj113 says:

Aug 6, 2010. 3:47 PM [REPLY](#)

Ahhh - I see the problem The DS18B20 is a programmable resolution thermometer that is compatible with the DS1822 it is not compatible with the protocol that the DS1820 uses. I am unaware of the DS18S20+ - Is it compatible with the DS1820? Things to check: Is the ATmega driven by the 16Mhz crystal? If not, then the timings could be out - also, verify that the +5V is really 5V.



mattadamsnet says:

Aug 3, 2010. 6:29 AM [REPLY](#)

I was up at 2 am putting together the parts list to order, Mistake number 1, never take the credit card out past midnight. Any who For the life of me, in my delirium, i couldn't find "Micro - ATmega368 - DIP" Anywhere. Only to wakeup this morning and realize it should be 328 or 168 -- Matt



drj113 says:

Aug 3, 2010. 2:24 PM [REPLY](#)

Arrggh! I know the feeling - and yep, I stuffed up as well - where did you see the reference to ATMegs368? I will fix that, I normally use the 328 chips, as they have extra program space, and only cost an extra dollar!



mattadamsnet says:

Aug 3, 2010. 5:54 PM [REPLY](#)

Step 6: Parts List



drj113 says:

Aug 3, 2010. 6:22 PM [REPLY](#)

Done! - thanks for that



diamantmatch says:

Jul 31, 2010. 11:57 AM [REPLY](#)

hello

i have designed my own version of this using your schematic but i can not seem to print it in the right dimensions.. i an/was using diptrace now but i have downloaded kicad aswell now but i dont fully understand how it works yet since it prints totally blank pages every time (the pages to straight thru without the jets moving over it not a printer issue) but my question was if you could upload the kicad schematic files too ? so that i can load in the scematic with one click :) all i have to do then is redesing my entire pcb in the pcb maker and then add the schematic somehow.. thank you :)



drj113 says:

Jul 31, 2010. 3:55 PM [REPLY](#)

I am more than happy to upload the archive of the KiCad files. It will be on the last step of the Instructable shortly.



sydkahn says:

Jul 31, 2010. 5:36 AM [REPLY](#)

Boy I wish I could get a Eagle cad of this board- would send it off to a board house and get a few made.



wa7jos says:

Jul 11, 2010. 9:08 AM [REPLY](#)

Installing SMD components can be intimidating at first. One product I have found to be a great help is a Flux Pen. It's similar in size to a dry erase marker containing liquid flux and a felt tip applicator. I flux the pads and then place the part on the pads. The extra flux makes it unnecessary to tin the pads first, AND it's sticky, so it helps to keep the parts from moving around until you get a pad soldered. The other important item is to get the smallest diameter solder you can locate. This will prevent the capillary action from forming huge solder balls and a bridging mess. And finally, have some SMALL solder-wick handy. WHEN (not if) you bridge some pins, the solder wick will let you remove the excess. You mentioned good lighting. If you can get some magnification that works wonders too. I bought a stereo microscope for SMD work, but a head mounted visor is a big improvement over the naked eye. SMD is not that tough to master, after which, you'll never use through hole again!



drj113 says:

Jul 12, 2010. 5:31 PM [REPLY](#)

Thanks for that tip - I will pick up a flux pen when I next get to Jaycar!



liodel says:

Jul 29, 2010. 2:03 AM [REPLY](#)

Hello,

thanks for this board, I'm planning to build my own version too, and your work is very helpfull.

Concerning SMD soldering, may I suggest you to have a look at [this video](#) particulary clear and full of useful stuff without expensive equipment.

And, as wa7jos says, using flux is the (magic) key...



diamantmatch says:

Jul 25, 2010. 2:35 AM

(removed by author or community request)



diamantmatch says:

i can not seem to find the beads resistance?? what bead do i need? thank you :D

Jul 25, 2010. 5:00 AM [REPLY](#)



drj113 says:

Any ferite bead will work fine - I use FX4 beads.

Jul 26, 2010. 10:56 PM [REPLY](#)



diamantmatch says:

(removed by author or community request)

Jul 25, 2010. 6:50 AM



diamantmatch says:

i am just replicating your circuit and drawing my own but still can not find the type of bead. just as im buisy with the dip switches i see that you have them connected straight to a header and to the switches witch either are ground or float since i see no pull up resistor? wouldnt it be verry usefull to put a pulup between them ? to prevent unwanted ip's? cheers diamantmatch

Jul 26, 2010. 8:21 AM [REPLY](#)



drj113 says:

The resistors are not needed, as I use the weak pullup function in the AtMega chip

Jul 26, 2010. 11:01 PM [REPLY](#)

[view all 115 comments](#)