

Tahmid's blog

Microcontroller and Power Electronics

[Index - View All Posts](#)



Saturday, February 16, 2013

Sine Wave Generation without ECCP - Using single CCP Module of PIC16F877A

I had previously shown how to generate sinusoidal pulse width modulation (SPWM) signals using the ECCP module in a PIC for generating a sine wave output for use in DC-AC inverter. I have had requests from people asking how to generate the same SPWM signals with other microcontrollers that don't have the ECCP module, such as the super popular PIC16F877A.

So, here I talk about how to generate the same SPWM signals using just one CCP module as can be commonly found on so many microcontrollers. This allows much greater flexibility in microcontroller selection.

You should go through the other articles related to generating SPWM with the ECCP module (if you haven't already gone through them, that is) to get an idea of what I'm talking about regarding sine wave generation with the ECCP module and about sine wave generation in general, really:

- [Generation and Implementation of Sine Wave Table](#)
- [Smart Sine - Software to generate sine table](#)
- [Generation of sine wave using SPWM in PIC16F684](#)
- [600W 50Hz sine wave inverter test circuit](#)
- [Feedback in sine wave inverter \(PIC16F series based\)](#)
- [Demystifying The Use of Table Pointer in SPWM - Application in Sine Wave Inverter](#)

The code I had previously used (utilizing the ECCP module) is:

```
//-----  
//Programmer: Syed Tahmid Mahbub  
//Target Microcontroller: PIC16F684  
//Compiler: mikroC PRO for PIC (Can easily port to any other compiler)  
//-----  
  
unsigned char sin_table[32]={0,25,49,73,96,118,137,  
159,177,193,208,220,231,239,245,249,250,249,245,  
239,231,220,208,193,177,159,137,118,96,73,49,25};  
  
unsigned int TBL_POINTER_NEW, TBL_POINTER_OLD, TBL_POINTER_SHIFT, SET_FREQ;  
unsigned int TBL_temp;  
unsigned char DUTY_CYCLE;  
  
void interrupt(){
```

```

    if (TMR2IF_bit == 1){
        TBL_POINTER_NEW = TBL_POINTER_OLD + SET_FREQ;
        if (TBL_POINTER_NEW < TBL_POINTER_OLD){
            CCP1CON.P1M1 = ~CCP1CON.P1M1; //Reverse direction of full-bridge
        }
        TBL_POINTER_SHIFT = TBL_POINTER_NEW >> 11;
        DUTY_CYCLE = TBL_POINTER_SHIFT;
        CCP1L = sin_table[DUTY_CYCLE];
        TBL_POINTER_OLD = TBL_POINTER_NEW;
        TMR2IF_bit = 0;
    }
}

```

```

void main() {
    SET_FREQ = 410;
    TBL_POINTER_SHIFT = 0;
    TBL_POINTER_NEW = 0;
    TBL_POINTER_OLD = 0;
    DUTY_CYCLE = 0;
    ANSEL = 0; //Disable ADC
    CMCON0 = 7; //Disable Comparator
    PR2 = 249;
    TRISC = 0x3F;
    CCP1CON = 0x4C;
    TMR2IF_bit = 0;
    T2CON = 4; //TMR2 on, prescaler and postscaler 1:1
    while (TMR2IF_bit == 0);
    TMR2IF_bit = 0;
    TRISC = 0;
    TMR2IE_bit = 1;
    GIE_bit = 1;
    PEIE_bit = 1;

    while(1);
}
//-----

```

That's the previous code. Now let's look at the code based on a single CCP module and not the ECCP module. For this, I chose the super popular PIC16F877A microcontroller. The chosen frequency, like before, is 16kHz. The code is:

```

//-----
//Programmer: Syed Tahmid Mahbub
//Target Microcontroller: PIC16F877A
//Compiler: mikroC PRO for PIC (Can easily port to any other compiler)
//-----

unsigned char sin_table[32]={0, 25, 50, 75, 99, 121, 143, 163, 181,
198, 212, 224, 234, 242, 247, 250, 250, 247, 242, 234, 224, 212, 198,
181, 163, 143, 121, 99, 75, 50, 25,0};

unsigned int TBL_POINTER_NEW, TBL_POINTER_OLD, TBL_POINTER_SHIFT, SET_FREQ;
unsigned int TBL_temp;
unsigned char DUTY_CYCLE;

```

```

sbit MOSA at RD0_bit;
sbit MOSB at RD1_bit;
sbit MOSC at RD2_bit;
sbit MOSD at RD3_bit;

unsigned char FlagReg;
sbit Direction at FlagReg.B0;
//0 -> MOS A + D
//1 -> MOS B + C

void interrupt(){
    if (TMR2IF_bit == 1){
        TBL_POINTER_NEW = TBL_POINTER_OLD + SET_FREQ;
        if (TBL_POINTER_NEW < TBL_POINTER_OLD){
            //CCP1CON.P1M1 = ~CCP1CON.P1M1; //Reverse direction of full-bridge
            if (Direction == 0){
                MOSA = 0;
                MOSD = 0;
                MOSB = 1;
                MOSC = 1;
                Direction = 1;
            }
            else{
                MOSB = 0;
                MOSC = 0;
                MOSA = 1;
                MOSD = 1;
                Direction = 0;
            }
        }
        TBL_POINTER_SHIFT = TBL_POINTER_NEW >> 11;
        DUTY_CYCLE = TBL_POINTER_SHIFT;
        CCP1L = sin_table[DUTY_CYCLE];
        TBL_POINTER_OLD = TBL_POINTER_NEW;
        TMR2IF_bit = 0;
    }
}

void main() {
    SET_FREQ = 410;
    PORTD = 0;
    TRISD = 0;
    PR2 = 249; // 16kHz
    CCP1L = 0;
    CCP1CON = 12; //PWM mode
    TRISC = 0xFF;
    TMR2IF_bit = 0;
    T2CON = 0x04; //TMR2 on
    while (TMR2IF_bit == 0);
    TMR2IF_bit = 0; //Clear TMR2IF
    PORTC = 0;
    TRISC = 0;
    TMR2IE_bit = 1;
    GIE_bit = 1;
    PEIE_bit = 1;

    while (1);
}

```

Now let's talk about the changes I've made in order to be able to use a single CCP module instead of the ECCP module.

When the ECCP module is used, it generates the SPWM signals and sends the modulation signals to the required "MOSFETs" (of course there's a drive circuit in between) depending on the "direction" as dictated by CCP1CON.P1M1 (bit 7 of CCP1CON register). Since this bit does not exist in the CCP module (obviously, since it's "uni-directional"), this functionality must be achieved in software. Since we don't have the ECCP module and have chosen to use a single CCP module only, the 4 drive signals come from other pins not associated to the PWM module. I've chosen PORTD bits 0 to 3. Of course, you can select any other 4 pins.

This is the circuit diagram of the SPWM signal generation portion:

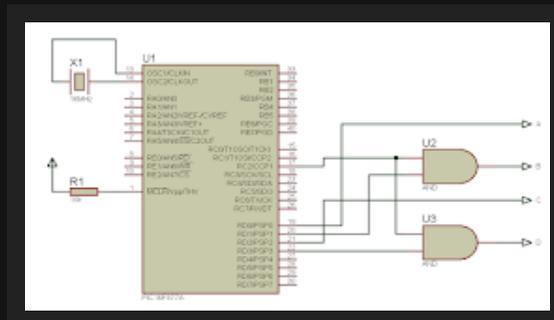


Fig. 1 - Circuit diagram of SPWM generation section - microcontroller + AND gates (Click image to enlarge)

Below (Fig. 2) is the circuit diagram for the configuration of the MOSFETs and the drivers - and the synchronization with the signals generated from Fig. 1 above.

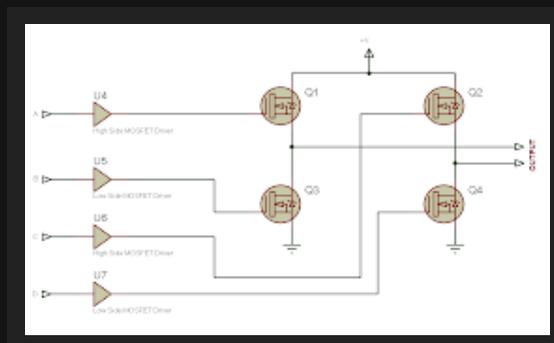


Fig. 2 - MOSFET Configuration Section (Click image to enlarge)

The SPWM generation is done by the single CCP module and which MOSFETs to send the signals to is set by the "Direction" bit and the hardware trick employing the AND gate. When "Direction" is equal to 0, the high side MOSFET A is kept on for 10ms during which time the SPWM signals on CCP1 output (RC2) are sent to low side MOSFET D by sending a "1" to RD3, which, with the help of the AND gate "diverts" the CCP1 signal to the low side MOSFET D (see Fig. 1 above). The same thing is achieved when "Direction" is equal to 1, just with high side MOSFET C and low side MOSFET B. When MOSFETs A and D are operated, MOSFETs B and C are kept off and vice versa. The MOSFETs are first turned off before the other two are turned on, as can be seen in the code block:

```

if (Direction == 0){
    MOSA = 0;
    MOSD = 0;
    MOSB = 1;
    MOSC = 1;
    Direction = 1;
}
else{
    MOSB = 0;

```

```

MOSC = 0;
MOSA = 1;
MOSD = 1;
Direction = 0;
}

```

To understand how the timing and the table pointer operation work, go through this:

[Demystifying The Use of Table Pointer in SPWM - Application in Sine Wave Inverter](#)

I've modified the sine table to increase the deadtime. Notice how there's a 0 at both the start and the end. This achieves the additional deadtime. See Fig. 4 below. I did this by using my software "Smart Sine" to generate a sine table with 31 values and then adding a 0 at the end.

Besides that, the other functionality are the same - the PWM initialization and setting, the table and table pointer are used the same way as before. So make sure you go through this tutorial if you aren't completely clear regarding it:

[Demystifying The Use of Table Pointer in SPWM - Application in Sine Wave Inverter](#)

For the MOSFET drivers, you require high/low side MOSFET drivers. One of the most popular such driver is the IR2110. For a thorough tutorial on using the IR2110, go through this tutorial:

<http://tahmidmc.blogspot.com/2013/01/using-high-low-side-driver-ir2110-with.html>

Here are the simulation results:

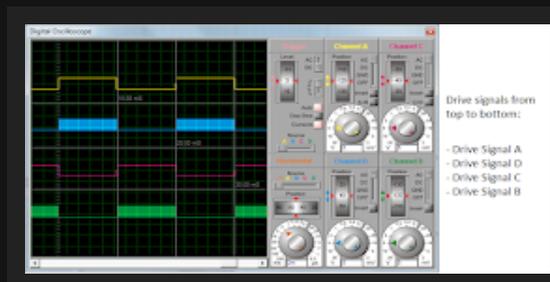


Fig. 3 - Generated SPWM Drive Signals (Click image to enlarge)

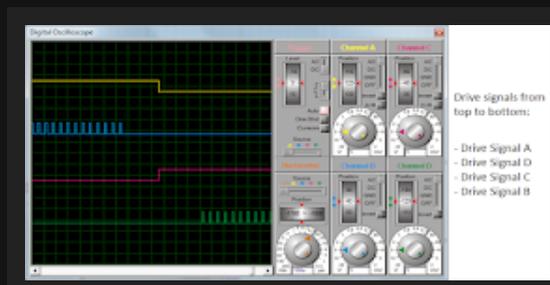


Fig. 4 - Clear demonstration of the "deadtime" (Click image to enlarge)

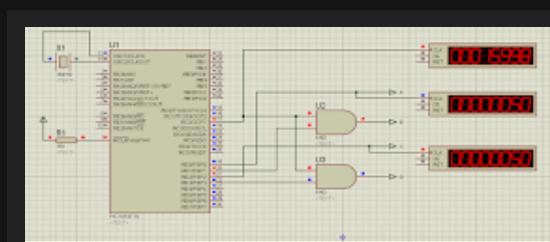


Fig. 5 - Simulation results showing signal frequencies (Click image to enlarge)

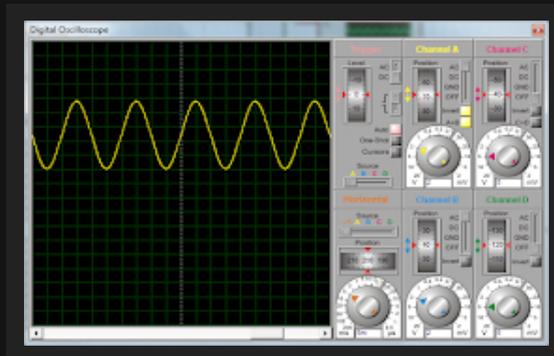


Fig. 6 - Generated Sine Wave Signal (Click image to enlarge)

The operation is quite simple to understand. The trick lies in a simple software modification and the use of the external AND gates. It's quite simple really! All we've needed are 5 IO pins from the PIC16F877A leaving all the other IO pins unused - for use for so many other tasks you can carry out. Observe how the main function in the code is not doing anything and all is done in the interrupt. Notice the empty endless while(1) loop where you can carry out any other required task.

I hope you've understood how to generate SPWM signals using just the single CCP module of a microcontroller and can now use it for all your applications! Keep in mind that this isn't restricted to only PICs but can be used for any microcontroller containing one PWM module. Let me know your feedback and comments.

Tahmid at 8:45 AM



196 comments:

Anonymous February 16, 2013 at 5:11 PM

can you share a hex file because when i compile this code on mikroC appear many errors

Reply

▼ Replies



Tahmid February 17, 2013 at 3:35 AM

Sharing the hex file will defeat the purpose of this tutorial as many people will choose to just download and use the hex file.

If you are facing problems, elaborate on the problems you are facing and then, I can help you solve those problems. However, there is no error in the code posted since I have successfully compiled and simulated it. So, elaborate on the issues you are facing and I can try and help fix them.

Regards,
Tahmid.



Nizomuddean February 15, 2014 at 10:48 AM

Hello Tahmid,
may i know how to change this program to 3 phase.. what i need to do..

Anonymous October 4, 2014 at 11:47 AM

Nice job Tahmid, but I saw some errors here, if you turn on the high side MOSFET A for 10 ms and at the same time turn on the low side MOSFET D with SPWM signal, you will get a shoot through-condition.

Shoot-through is defined as the condition when both MOSFETs are either fully or partially turned on, providing a path for current to "shoot through" from +V to GND. this condition will destroy the MOSFETs.

So I think, this circuit maybe works on PROTEUS simulation, but not on real work.

To correct the error, when "direction" bit is equal "0", the high side MOSFET A is kept on for 10ms during which time the SPWM signals on CCP1 output (RC2) are sent to low side MOSFET B (not D) by sending a "1" to RD1 (not RD3), both MOSFETs D and C are kept on inactive state (OFF) and when "direction" bit is equal "1", the high side MOSFET C is kept on for 10ms during which time the SPWM signals on CCP1 output (RC2) are sent to low side MOSFET D (not B) by sending a "1" to RD3 (not RD1), both MOSFETs A and B are kept on inactive state (OFF)

Juan Miguel Perez
Microtronic Technology



Tahmid  October 12, 2014 at 5:07 AM

Good catch! There was a typing mistake in Fig. 2. Thanks for pointing it out! I'll make the correction. It should be ABCD instead of how it's shown now.



Unknown June 11, 2017 at 6:15 PM

hello tahmid in your code never you use sbit Direction at FlagReg.B0; Nowhere do I see that this is being used Then the change of direction appears by magic



Customer care helpline number 9162902829 6291663434 June 13, 2020 at 4:26 AM

Customer care helpline number
6291663434 9162902829



Customer care helpline number 9162902829 6291663434 June 13, 2020 at 4:27 AM

Customer care helpline number
6291663434 9162902829



Customer care helpline number 9162902829 6291663434 June 13, 2020 at 4:28 AM

Customer care helpline number
6291663434 9162902829



Customer care helpline number 9162902829 6291663434 June 13, 2020 at 4:29 AM

Customer care helpline number
6291663434 9162902829

Customer care helpline number

6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829



Customer care helpline number
6291663434 9162902829

Customer care helpline number

6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829



Customer care helpline number 9162902829 6291663434 June 13, 2020 at 4:30 AM

Customer care helpline number
6291663434 9162902829

Customer care helpline number

6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Customer care helpline number
6291663434 9162902829

Reply

Anonymous February 17, 2013 at 9:45 AM

it is ok , the code run Successfully
thank you

Reply

▼ Replies



Tahmid  February 17, 2013 at 10:05 AM

That's great! I wish you success on your project!

Regards,
Tahmid.

Reply

Anonymous February 17, 2013 at 11:55 AM

i want to use IGBT , can you help me to find the driver Circuit? ?

Reply

▼ Replies



Tahmid  February 17, 2013 at 1:16 PM

You can use IR2110. The drive circuit for IGBT is similar to that for a MOSFET.

Go through this for a tutorial on IR2110:

<http://tahmidmc.blogspot.com/2013/01/using-high-low-side-driver-ir2110-with.html>

Regards,
Tahmid.

Reply

Anonymous February 21, 2013 at 8:16 AM

hi tahmid, thanks for the info, been great understanding the concept of making the pure sine wave. I've simulated this over proteus, pin RD0-RD3 would show the clock showing signal frequency at '50'. But when I place the counter terminal tool on the CCP1 it does not show '16000', it shows '0'. can you share if you have come across this problem before? thanks!

Reply

▼ Replies



Tahmid February 21, 2013 at 10:02 AM

As you can see in Fig. 5, the frequency counter clearly shows a frequency approximately 16kHz. If it shows zero, you're doing something wrong. Ensure that the program compiled properly and you set up the simulation "hardware" properly.

One thing to remember: the frequency counter in Proteus works by checking the clock over one second intervals. So, until one second (in Proteus simulation, shown at the bottom of the screen) occurs, the frequency counter will show zero. So, after simulation starts, wait for at least a minute, perhaps more than that and then check to see if it still shows zero or not.



Unknown May 13, 2014 at 5:21 PM

i have the same problem ccp1/RC2 pin shows nothing it stops :(it doesn't have 16khz freq.



Unknown May 14, 2014 at 7:24 AM

its still zero...i have comiled the code successfully but freq counter shows 3 on 50Hz..and 0 on 16kHz..i have waited for atleast 6 min but shows nothing..the ccp1/RC2 pin not running...plz help

Reply



abm February 22, 2013 at 5:24 PM

This comment has been removed by the author.

Reply



abm February 23, 2013 at 4:14 PM

Yeah! i got the code working?

here it is

```

*****
'* Name : UNTITLED.BAS *
'* Author : *
'* Notice : Copyright (c) 2013 *
'* : All Rights Reserved *
'* Date : 2/22/2013 *
'* Version : 1.0 *
'* Notes : *
'* : *
*****

DEVICE 16F877A

XTAL 16

ON_HARDWARE_INTERRUPT GOTO isr

DIM tblpointernew AS WORD
DIM tblpointerold AS WORD
DIM tblpointershift AS WORD
DIM setfreq AS WORD
DIM tbltemp AS WORD
DIM dutycycle AS BYTE
DIM flag AS WORD
DIM SINVAL AS BYTE
DIM direction AS BIT

SYMBOL mosa = PORTD.0
SYMBOL mosb = PORTD.1
SYMBOL mosc = PORTD.2
SYMBOL mosd = PORTD.3
'SYMBOL direction = flag.0 '0 = mosa + mosd and 1 = mosb+mosc

SYMBOL PEIE = INTCON.6 ' Peripheral Interrupt Enable
SYMBOL GIE = INTCON.7 ' Global Interrupt Enable

'SYMBOL TMR1IF = PIR1.0 ' TMR1 Overflow Interrupt Flag
SYMBOL TMR2IF = PIR1.1 ' TMR2 to PR2 Match Interrupt Flag
SYMBOL TMR2IE = PIE1.1 ' TMR2 to PR2 Match Interrupt Enable

GOTO main

DISABLE
isr:

CONTEXT SAVE

IF TMR2IF == 1 THEN

update:
tblpointernew = tblpointerold + setfreq
IF tblpointernew < tblpointerold THEN
IF direction == 0 THEN
mosa = 0
mosd = 0
mosb = 1
mosc = 1
direction = 1

ELSE
mosb = 0
mosc = 0

```

```
mosa = 1
mosd = 1
direction = 0
ENDIF
ENDIF

tblpointershift = tblpointernew >> 11
dutyicycle = tblpointershift
SINVAL = LOOKUPL dutyicycle, [0, 25, 50, 75, 99, 121, 143, 163, 181,198, 212, 224, 234, 242, 247, 250, 250, 247, 242, 234,
224, 212, 198,181, 163, 143, 121, 99, 75, 50, 25, 0]
CCPR1L = SINVAL
tblpointerold = tblpointernew
```

```
TMR2IF = 0
ENDIF
```

```
CONTEXT RESTORE
```

```
main:
setfreq = 410
tblpointernew = 0
tblpointerold = 0
tblpointershift = 0
dutyicycle = 0
TRISD = 0
PORTD = 0
TRISC = 0
PORTC = 0
PR2 = 249
CCPR1L = 0
CCP1CON = 12
```

```
TMR2IF = 0
T2CON = $04
```

```
TMR2IF = 0
TMR2IE = 1
GIE = 1
PEIE = 1
direction = 0
```

```
WHILE 1 = 1
```

```
WEND
```

Reply

▼ Replies

Anonymous September 28, 2013 at 4:29 PM

Plz add Feedback code on this code.....

Reply

Anonymous March 1, 2013 at 9:11 AM

hi tahmd,,,

i was simulated your circuits „but at h bridge i get shape like thick sine wave,,not smooth like yours whats should i do please help me for this;;;

regards
abdul

Reply

▼ Replies



Tahmid  March 1, 2013 at 11:54 AM

That's an indication of a problem with filtering. Adjust the values of the filter components you are using.

Regards,
Tahmid.

Reply

Anonymous March 1, 2013 at 12:09 PM

Thank you so much Tahmid , Is there anyway to get the sine signal at the output of the h bridge without using filtering ? Or do we have to use it certainly ? also do you have any other communication way , like skype , facebook etc..

Best ,
Abdul

Reply

▼ Replies



Tahmid  March 1, 2013 at 12:14 PM

Filtering is a MUST. That's the basis on which sine wave generation with SPWM works - you approximate the sine wave with square wave digital signals, that upon filtering give a nice sine wave output.

Regards,
Tahmid.

Anonymous March 1, 2013 at 12:36 PM

thanks again for your replay
,but why did you add and then invert in the output h bridge signals through oscilloscope



Tahmid  March 2, 2013 at 2:23 AM

The oscilloscope in Proteus is ground referenced. I inverted one channel and added the two channels because I needed to show the signal "with reference to each other".

Regards,
Tahmid.

Reply

Anonymous March 3, 2013 at 7:36 AM

hi tahmid ,,
thanks so much for sharing your knowledge,, i just want to us you about your filter using to get that pure sine wave i have

tried many types but i couldn't get result like yours,,if possible can you specify components of your filter hope you will help me in this,,

regards
abdul

Reply

▼ Replies



Tahmid  March 3, 2013 at 9:16 AM

My one shown here is a simulation result.

In practice, it is very time-consuming to pick the correct components for proper filtration. You must continuously tweak the value of the components if you are using trial and error method.

Regards,
Tahmid.

Reply

Anonymous March 3, 2013 at 10:14 AM

thanks again tahmid for your replay ,,
some time when i changes the value of the filter components i got an error during simulation in a Proteus its display like this `,,, program or EEPROM data has invalid address[2000] for this device `,,,
i wonder what is a reasons for that,,,,,

Reply

▼ Replies



Tahmid  March 3, 2013 at 10:55 AM

That is indicative of an error in the program and not the hardware. Ensure that you have compiled the program properly and are using the correct device in the simulation.

Regarding the LC filter, you should adjust them in practice - actual hardware - instead of in Proteus simulation.

Reagards,
Tahmid.

Reply

Anonymous March 3, 2013 at 12:32 PM

hi tahmid,,
i finally got pure sine wave like yours after add transformer(1 to 1 ratio) working as load and then 10uF capacitor at the output of h bridge but problem is i got too small voltage in terms of mV and desired output volts is 12V,,

also i used ir2101 driver and i connected exactly as in your tutorial ,,

Reply

▼ Replies

Tahmid  March 6, 2013 at 7:48 AM



I'll say what I've said before:

Regarding the LC filter, you should adjust them in practice - actual hardware - instead of in Proteus simulation.

Regards,
Tahmid.



Unknown June 1, 2014 at 2:53 PM

sir can you send me ur proteous simulation.it would be helpful fr me to detect my problem....

Reply



jtrao March 4, 2013 at 2:21 AM

Hi Tamid,
Nice tutorial.

Would you please explain the PWM generation and feedback correction for Battery charging also. I mean with the same transformer and Mosfets working in reverse to charge the battery with mains.

Thank you
J.T.Rao

Reply

▼ Replies



Tahmid  March 4, 2013 at 11:27 AM

I'll try to write one tutorial regarding that topic soon.

Thanks for the suggestion!

Regards,
Tahmid.

Reply

Anonymous March 5, 2013 at 2:23 PM

hi Tahmid,

thanks you very much for sharing your knowledge to us,,,am new in this kind of inverter circuit ,,can you help me by sending to my email you completely design so as i can learn after seen a working simulation circuit,,, so that letter i can modify or making my own after learning from yours

i real appreciate if you can help me.....

my email gandolf_12@yahoo.com;

regards,
ahmad

Reply

▼ Replies



Tahmid  March 6, 2013 at 7:33 AM

Unfortunately, I don't have a complete design with working simulation. You can learn by yourself by searching for information online. There's also quite a lot of material on this blog itself.

Regards,
Tahmid.

Reply

Anonymous March 9, 2013 at 5:48 AM

hi
great job. pls can you post the code in .ASM ..and can i do without h bridge drivers using the pic..thanks.

Reply

▼ Replies



Tahmid  March 9, 2013 at 10:09 AM

Hi,
The code is only in C. You can download mikroC for free from their website.

You can't drive a H-bridge without some sort of H-bridge driver. I prefer to use dedicated driver chips such as IR2110, L6385E, etc. You can do it using discrete components as well.

Regards,
Tahmid.

Reply

Anonymous March 13, 2013 at 6:06 AM

```
void interrupt(){
if (TMR2IF_bit == 1){
TBL_POINTER_NEW = TBL_POINTER_OLD + SET_FREQ;
if (TBL_POINTER_NEW < TBL_POINTER_OLD){
//CCP1CON.P1M1 = ~CCP1CON.P1M1; //Reverse direction of full-bridge
if (Direction == 0){
MOSA = 0;
MOSD = 0;
MOSB = 1;
MOSC = 1;
Direction = 1;
}
else{
MOSB = 0;
MOSC = 0;
MOSA = 1;
MOSD = 1;
Direction = 0;
}
}
TBL_POINTER_SHIFT = TBL_POINTER_NEW >> 11;
DUTY_CYCLE = TBL_POINTER_SHIFT;
CCPR1L = sin_table[DUTY_CYCLE];
TBL_POINTER_OLD = TBL_POINTER_NEW;
TMR2IF_bit = 0;
}
```

icould not understand the above (interrupt)
can you explain it in a simple way

Reply

▼ Replies

Anonymous March 16, 2013 at 11:30 AM

This comment has been removed by the author.

Anonymous March 16, 2013 at 11:34 AM

thanks tahmid for demystifying the technique of the SPWM. I already understand it perfectly well. Concerning Generating SPWM without ECCP such as in this case. There is just a simple question i want to ask u.

```
unsigned char FlagReg;
```

```
sbit Direction at FlagReg.B0; These lines in ur code is actually making me not understanding this stuff.
```

The FlagReg which is a register was created. The FlagReg.B0 is what? I dont understand the second line? Because FlagReg was not set as an array how is then that FlagReg.B0 came about..... Why cant i use "unsigned char Direction;" in place of the two lines... Thats my question. I am using mikroC version 6.2 and i Guess u are using mikroC pro version 5.6 so the code written above is not compatible with mine and hence cannot run on version 6.2 mikroC compiler. I Tweaked your code a little... bur i got stuck at the point of.....unsigned char FlagReg;..... i see dis line as a created register;

```
sbit Direction at FlagReg.B0;..... i see dis line as a SFR (special function register). whats the difference between FlagReg and FlagReg.B0? why cant i use Direction just like that instead of making FlagReg and then direction?.....
```

Dats why i wanted to confirm these two statements. These Questions would make me understand this code fully.

Here is the one I tweaked.

```
#define MOSA PORTD.F0  
#define MOSB PORTD.F1  
#define MOSC PORTD.F2  
#define MOSD PORTD.F3
```

```
unsigned char FlagReg;  
#define Direction FlagReg
```

```
unsigned char sin_table[32]={0, 25, 50, 75, 99, 121, 143, 163, 181,  
198, 212, 224, 234, 242, 247, 250, 250, 247, 242, 234, 224, 212, 198,  
181, 163, 143, 121, 99, 75, 50, 25,0};
```

```
unsigned int TBL_POINTER_NEW, TBL_POINTER_OLD, TBL_POINTER_SHIFT, SET_FREQ;  
unsigned int TBL_temp;  
unsigned char DUTY_CYCLE;
```

```
//sbit MOSA at RD0_bit;  
//sbit MOSB at RD1_bit;  
//sbit MOSC at RD2_bit;  
//sbit MOSD at RD3_bit;
```

```
//0 -> MOS A + D  
//1 -> MOS B + C
```

```
void interrupt(){  
if (PIR1.F1 == 1){  
TBL_POINTER_NEW = TBL_POINTER_OLD + SET_FREQ;  
if (TBL_POINTER_NEW < TBL_POINTER_OLD){
```

```
//CCP1CON.P1M1 = ~CCP1CON.P1M1; //Reverse direction of full-bridge
if (Direction == 0){
MOSA = 0;
MOSD = 0;
MOSB = 1;
MOSC = 1;
Direction = 1;
}
else{
MOSB = 0;
MOSC = 0;
MOSA = 1;
MOSD = 1;
Direction = 0;
}
}
TBL_POINTER_SHIFT = TBL_POINTER_NEW >> 11;
DUTY_CYCLE = TBL_POINTER_SHIFT;
CCPR1L = sin_table[DUTY_CYCLE];
TBL_POINTER_OLD = TBL_POINTER_NEW;
PIR1.F1 = 0;
}
}
```

```
void main() {
SET_FREQ = 410;
PORTD = 0;
TRISD = 0;
PR2 = 249; // 16kHz
CCPR1L = 0;
CCP1CON = 12; //PWM mode
TRISC = 0xFF;
PIR1.F1 = 0;
T2CON = 0x04; //TMR2 on
while (PIR1.F1 == 0);
PIR1.F1 = 0; //Clear TMR2IF
PORTC = 0;
TRISC = 0;
PIE1.F1 = 1;
INTCON.F7 = 1;
INTCON.F6 = 1;

while (1);

}
```

yeah thanks tahmid for your anticipated support.....



Unknown July 13, 2015 at 8:15 AM

Hello Tahmid. I use Hitech C Compiler. I have "translated" the whole program. But I am not able to understand "the same two" lines. Can you plz explain what those lines mean so that we can try to "translate" them.

The lines:

```
unsigned char FlagReg;
sbit Direction at FlagReg.B0;
```

Reply



c s roy April 15, 2013 at 5:49 AM

My Dear Tamid Mahabub (A great popular figure of Power electronics)

I am extremely happy to learn that you got inspired to enter into the fascinating world of power electronics through my very elementary book POWER ELECTRONICS DEMYSTIFIED. Thank you for pointing out some inadvertent printing mistakes found in this book I consider your statement about my book in your blog as one of the greatest rewards received by me in my professional life as a design engineer, a man of science and an author of technical books.

I am amazed by the huge popularity of your blog and your keen interest in sharing the knowledge among the thousands interested in power electronics.

I am also highly impressed by your uncanny ability to master the following two seemingly opposing topics;

1. Microcontrollers and the software
2. Hardcore Power electronics hardware domain

Apart from this, what pleased me most was nothing other than the photo you posted on all the circuit boards with dead MOSFETS, IGBTs etc and candid declaration of many of your failures before your remarkable success you have achieved today.

I have some humble request to you

Please contribute some good articles to our website www.microkut.com on power electronics & microcontroller technology.

Be an active part of this website microkut.com. In this connection please visit Calcutta and hold an official meeting with our team.

Waiting for your reply

Chandra Sekhar Roy

Director

Digital Power Corporation

+91 9830053365

ronixin@gmail.com

www.microkut.com

Reply

▼ Replies



Tahmid April 22, 2013 at 5:26 AM

Dear Sir,

Sorry for the late reply. Due to my school examinations, I could not manage time to reply.

Your book is a great book has been a stepping stone to my future endeavors and efforts in power electronics. In fact, when someone asks me for book recommendations for power electronics, this is one of the books I recommend.

Thank you for the compliments. I am always trying to make my blog better. I try to share as much as I can so that I can reach out to help as many people as possible.

I am always looking to use microcontroller in power electronics for most, if not all, control in the designs. I believe that a strong ability and grasp of microcontrollers gives one complete control of the design/circuit, while at the same time resulting in the simplification of the design, improved efficiency of the design and, in most cases, a reduction in cost as well.

I will certainly contribute to www.microkut.com . I will contact you through email regarding this.

Visiting Calcutta may be difficult on my part now, since I have my GCE A Level examinations in May-June and will be leaving for the USA for university towards the beginning of August. If I do get time in between, I will certainly try to visit Calcutta and shall contact you then.

I am truly honoured that a man such as yourself has visited my blog, commented on my post and has become a member. Any further comments or suggestions for improvement you may have regarding the blog will be highly appreciated.

With thanks and regards,
Tahmid.

Reply

Anonymous May 8, 2013 at 7:56 AM

`TBL_POINTER_NEW = TBL_POINTER_OLD + SET_FREQ;`

what does this means?

`SET_FREQ=410` (How?please explain.)

what is the frequency of clock?

why TMR (prescaler) not used?

Reply

▼ Replies



Tahmid  May 17, 2013 at 9:18 AM

Go through this:

<http://tahmidmc.blogspot.com/2013/02/demystifying-use-of-table-pointer-in.html>

Regards,
Tahmid.

Reply

Anonymous May 8, 2013 at 3:24 PM

how can i made a dead time between the square wave (Rd0 , 1 , 2 , 3) ????

Reply

▼ Replies



Tahmid  May 17, 2013 at 9:19 AM

Use a sine table with a maximum value that is quite a bit lower than $(PR2 + 1)$. That can act as deadtime.

Regards,
Tahmid.

Reply

Anonymous May 13, 2013 at 11:10 PM

Long life to you sir Tahmid

Reply



Unknown May 14, 2013 at 7:49 AM

HI TAHMID

This is Ganesan from tamilnadu(india).I am the new one for power electronics field.I visited your blog . that was very helpful for me. i am trying to make single phase sine wave inverter. i designed your circuit "Sine Wave Generation without ECCP - Using single CCP Module of PIC16F877A" it is running successfully. but i am facing a problem in the full bridge inverter side in proteus simulation. i have attached the file with this. and also in which side i have to connect the CRO probe in the proteus. please give me a suggestion.

Reply



Unknown May 14, 2013 at 8:04 AM

HI TAHMID

This is Ganesan from tamilnadu(india).I am the new one for power electronics field.I visited your blog . that was very helpful for me. i am trying to make single phase sine wave inverter. i designed your circuit "Sine Wave Generation without ECCP - Using single CCP Module of PIC16F877A" it is running successfully. but i am facing a problem in the full bridge inverter side in proteus simulation. i have attached the file with this. and also in which side i have to connect the CRO probe in the proteus. please give me a suggestion.

Reply



Unknown May 14, 2013 at 8:20 AM

hi

i have attached the two files. first one is working . second one is not working.
http://search.4shared.com/postDownload/e5jv9qVz/sine_wave_inverter.html

Reply



Adeel May 19, 2013 at 3:17 PM

Hi Tahmid!

visiting your blog is always helpful. you guy is doing a great job. God bless you.

I copy your code in Mikro C compiler and build the hex file. Simulate the circuit in Proteus as shown in tutorial. PORTD is working correctly. but no signal at CCP1 pin. What can be wrong. please guide me.

thanks!

Reply

▼ Replies



Tahmid  May 20, 2013 at 5:20 AM

Check to make sure that the code is exactly as shown. It should be working fine, as I've shown in the simulation images.

Regards,

Tahmid.



Adeel May 21, 2013 at 9:56 AM

Thanks Tahmid for your concern.

I have checked the code three times word by word. its exactly same as shown by you. I am also amazed. code should be worked correctly.

I am designing a single phase inverter as a part of my Final Project. I am stuck with it. Try again and again to edit the code but no way.Could you please send me your Hex file?

I shall be very thankful to you for this regard.



Tahmid  May 22, 2013 at 9:34 AM

There could be a problem with the simulation. You should try it on hardware. Just check that the output signals are coming from the PIC. If you get the outputs, then you can conclude that the error is in simulation.

Regards,
Tahmid.

Reply

Anonymous May 20, 2013 at 4:56 PM

Hello Tahmid, should this code work with any pic with a CCP module? I tried the code on a 18f2525 and the CCP1 pin only stays HI, I think the rest of the pins worked as they should, any advice?

Reply

▼ Replies



Tahmid  May 22, 2013 at 9:32 AM

It should work with any PIC with a CCP module, provided you initialize the CCP module and use it properly. You should attach the code and DSN file. I could take a look if I get time. Upload to an online storage site such as rapidshare.

Regards,
Tahmid.

Reply



Unknown May 21, 2013 at 10:41 AM

hello tahmid. i worked your code. it is working fine.i got sine wave output from your code.but i have faced one problem. when i run the 1/2 hp motor using this inverter IGBT IS heating.motor is not rotating.but drilling machine(500w) is working using this inverter. and here i have used igbt for H bridge driver.what will be the mistake? please guide me?

Reply

▼ Replies



Tahmid  May 22, 2013 at 9:29 AM

Do the IGBTs get too hot? Have you used sufficient heatsinking? Have you used a fan? Did you measure the output voltage? What's the output voltage with the 0.5HP motor?



Unknown May 25, 2013 at 1:20 PM

i have used h25r1202 as IGBT. IGBT IS very hot in the running condition of the 0.5 hp motor. i gave dc supply to the IGBT s collector terminal from the mains (Ac supply).i.e i converted ac to dc via bridge (10A).from this i got 350 v dc. will it be problem for running the motor?i have not used fan.

Reply

Powell May 23, 2013 at 10:10 AM



Hello Tahmid i was the one that asked you about the pic 18f2525.
The links below have the DSN file and code i used.
I really hope you get the time. Thanks in advance.
<http://www.mediafire.com/?anq4kjbo4204ara>
<http://www.mediafire.com/?n30rz14ra83c0sz>

Reply

▼ Replies



Tahmid  May 24, 2013 at 1:58 PM

I'll take a look and let you know.



Powell May 24, 2013 at 8:06 PM

Ok thank you

Reply

Shafeeq May 25, 2013 at 8:33 AM

hai,tahmid....i used pic 16f877a to get pulses to switch buckboost circuit.inorder to avoid the source and ground problem i used IR2111 driver ic.Mosfet i using is IRF540.But IR2111 will not be directly fed from pic since its logic high is 5V.IR2111 needs a logic high about its Vcc=12V.For resolving this problem i used an optocoupler between pic and IR2111.So i got optocoupler output with a logic high as 12 V with optocoupler Vcc as 12V.

Anyway i got the pulses having a voltage level sufficient to turn on IRF540 (Vgs=12V).But now the problem is that,at 0.1 V input to buck boost is working.But when the input voltage is increased,it is not working.But,i noticed that the problem is that when voltage is increased pulses from pic is not there.

Can you help.....it is urgent.....

Thank u
shafeeq

Reply



Unknown June 1, 2013 at 9:08 PM

hi tahmid,,,in your fig 2, A and D are high and low side of one bridge respectively and in figure 3, A and D are simultaneously high,,, so i have a doubt, if A and D are both HIGH at a same time then it will be short circuit ,, please clear my confusion,,

Reply

▼ Replies

shafeeq June 8, 2013 at 10:26 AM

hai,,i think its just a mistake in naming B and D.think interchanging the connection B and D.

Reply

Anonymous June 25, 2013 at 1:14 PM

Hello Hahmid,

I construct circuit as you do using Sine Wave Generation without ECCP - Using single CCP Module of PIC16F877A.And I have some problems.

In Proteus simulation, the output of pic are ok,as you showed and they have 5V in each output(5 pins).But in real construction, 17 pin has over 3V(may be 3.2V) and the remaining pins(19,20,21,22) have over 2V(may be 2.3V). Why don't they have 5V each? VDD is nearly 5V.I gave the pic double supply and ground(31,32,11,12).How to set the configuration bits when i burn the hex file to pic.The power supply to MOSFETs is 12V.The two outputs have around 3V each(may be 3.4V).

It is very less.I don't know exactly what type of transformer to be used.(When i create a mikroC pro project,how much frequencies for pic16f877a is used?)The two output and ground is connected to the transformer, there is no output from the transformer.

What should i do? Please help me! urgent!!!!!!!!!!!!!!

My mail is myinthtun08@gmail.com.

Reply

Anonymous July 9, 2013 at 8:26 PM

Hi Tahmid,

As with the PIC16F684 sine wave generation i'm working with this one to.

And here i'm getting the same saw tooth wave at the rc filter aswell?

I've tried many many rc configuration and even use a program to calculate the best rc combo for 50 and 60hz but i'm still getting the saw tooth wave.

Hope you gotten my email with all the info about this one to.

Waiting patiently for your reply.

Cheers

Grandi

Reply



Unknown July 14, 2013 at 7:41 AM

hi Tahmid , i construct your circuit the i used hef4081 and gate ic ... when i used and gate ic no output form and gate

Reply

▼ Replies



Tahmid  August 8, 2013 at 2:34 AM

Do you get the proper outputs from the microcontroller?

Reply

Anonymous August 4, 2013 at 11:08 AM

Hello Tahmid,me I used the PIC16F877A but the signals I am getting are not the same at all there is a difference on their on and off time ;

mainly those signals which pass through the AND gate are not good and their on and offtime are different to the other two signals A and B

which I think should cause a short circuit.

Please help on this,firstly I used PIC16F876A and I had the same erro even on the board and simulation but I changed and took PIC16F877A

and paste your codes as they are but I considered a difference on the on/off time meanwhile the signals are not the same as the one of yourFig. 3 - Generated SPWM Drive Signals (Click image to enlarge).

Tell mehow may I proceed.

Thans.

Reply

Anonymous August 7, 2013 at 12:33 PM

I Tahmid,I want to generate a spwm signal of 28Khz which must be used for controlling H-Bridge ,what can I do?

I generated one referring to your tutorials but the frequency I get is 50Hz so with that the MOSFET became very hot so I want to change the controlling signal.

Noe:This DC-AC Converter I am repairing before in its normal operation it used 28Khz for controlling the MOSFET (in H_Bridge).

Thanks and waiting for your Good reply.

Reply

Anonymous August 13, 2013 at 5:51 AM

Hi Tahmid

I ported your code to another controller from Freescale and it seems to be working fine for now.
Can you suggest a method to change the base frequency (up to 100Hz) based on an ADC input?

I want to congratulate you on your wonderful blog and useful libraries...cheers!!!!!!

Reply

▼ Replies



Tahmid September 6, 2013 at 10:05 PM

Please refer to this:

Demystifying The Use of Table Pointer in SPWM - Application in Sine Wave Inverter
<http://tahmidmc.blogspot.com/2013/02/demystifying-use-of-table-pointer-in.html>

I think you'll get your answer from here.

If you still don't, feel free to ask specific questions.

Hope this helps!

Regards,
Tahmid.

Reply

Anonymous August 20, 2013 at 7:24 AM

Hi Tahmid, I am trying to run the code in MPLAB using Hi-Tech C. I however get so many errors and the code doesnot compile well. Could it be that the code cannot be run using MPLAB's Hi-Tech C?

Thanks in Advance

Reply

▼ Replies

Tahmid September 6, 2013 at 9:47 PM



Of course it can. But it will take quite a lot of changes due to syntax and library differences between mikroC and Hi-Tech C.

Regards,
Tahmid.

Reply

Anonymous September 13, 2013 at 8:45 AM

hi tahmid thanks for shairing your knowladge,,
am desigining inverter by using arduino uno ;i create 32 array by using sinetable to control pwm i saw in your blog,,,
but the problem is my output frequency is not stable(i mean not costant) and my pwm change all time not stablized ;
this is my code;

```
// avr-libc library includes
#include
#include
int pwm1= 3;int pwm2= 11;
byte flicker[32] = { 0, 25, 49, 73, 96, 118, 139, 159, 177, 193, 208, 220, 231, 239, 245, 249, 250, 249, 245, 239, 231, 220,
208, 193, 177, 159, 139, 118, 96, 73, 49, 25};
void setup() {
pinMode(pwm1, OUTPUT);
pinMode(pwm2, OUTPUT);
// initialize Timer1
OCR2A=0;
OCR2B=0;
TCCR2A = 0;
TCCR2B =0; // put your setup code here, to run once:
TCCR2A = _BV(COM2A1) | _BV(COM2B1) | _BV(WGM21) | _BV(WGM20);
TCCR2B = _BV(CS22);

}

void loop() {
for(int i=0; i<31; i++) // loop equals number
{ // of values in array
analogWrite(pwm1,OCR2A= flicker[i]);
analogWrite(pwm2,0);

}

for(int i=31; i>=0; i--) // loop equals number
{ // of values in array
analogWrite(pwm1,OCR2A= 0);
analogWrite(pwm2,OCR2B= flicker[i]);

}

}

}
```

regards Abdul

Reply

Anonymous September 17, 2013 at 4:15 AM

Hi Tahmid, thanks for the great work

I have failed to find IR2110 in Proteus ISIS professional but instead found IR2101. How can I connect it?

Also, pin 17 of the PIC is always at 0 voltage, during the simulation. What could be the problem?

Regards Henry Kato

Reply

▼ Replies



Tahmid  September 21, 2013 at 8:57 PM

Pin 17 is not supposed to be always at zero. Use the oscilloscope to check the signal at pin 17.

You can use IR2101 in place of IR2110. I think all you need to do is to account for the pin-out differences.

You might find it helpful to refer to this:

<http://tahmidmc.blogspot.com/2013/01/using-high-low-side-driver-ir2110-with.html>

If you still can't do it, then feel free to ask and I'll try to help you further.

Regards,
Tahmid.



Unknown May 14, 2014 at 10:39 AM

i have the same issue plz help...

Reply

Anonymous September 28, 2013 at 4:09 PM

How to make H-Bridge Sine wave Inverter with Pic16f72 with feedback at 50hz fixed frequency? Plz Help.....

Reply

Mr. Soe October 22, 2013 at 1:10 PM

HI TAHMID , I used your mikroC code without ECCP module. when I combile this code with MikroC PRO version 5.01, i got hex file. But i see ccp1 pin is not working when i do simulation ISIS proteus. So i can't see sine wave in ISIS. That is my problem. I want to know MikroC PRO version you combile. Depand on MikroC PRO version? PLS help me , i have MikroC PRO version 5.01 only,.. so pls share code mikroC PRO version 5.01. Thank You. Mr. Soe

Reply

▼ Replies

Mr Soe October 24, 2013 at 2:02 PM

PLS suggestion to me, Can i use which mikroC version for 16F877A without ECCP module?



Tahmid  December 29, 2013 at 10:30 AM

The mikroC version shouldn't matter. Double check everything you did.

Reply

Obrown November 1, 2013 at 8:55 PM

Hi Tahmid, i compiled the code and get hex file, the program is running fine when i simulate in proteus, but the SPWM has some sort of distortion, is not as clean as yours, and when i connect the COUNTER TIMER in proteus, instaed of displaying the frequency it counts upward continuously. Pls. Tahmid help, what am i doing wrong

Reply

▼ Replies



Tahmid  February 22, 2014 at 12:54 AM

I recommend trying the circuit in actual hardware.

You need to change the mode of the timer/counter to frequency measurement. Double click the timer/counter and you can see the setting to change to frequency measurement.

Regards,
Tahmid.

Reply



Unknown November 17, 2013 at 4:22 AM

Hi Tahmid, how to calculate dead time. Plz Help

Reply

▼ Replies



Tahmid  December 29, 2013 at 10:32 AM

You need to select something that you think won't cause a short circuit. If you use a good drive circuit, this won't be too long.

You also need to consider the MOSFET you're using and how long it takes for it to turn off completely.

Regards,
Tahmid.

Reply



AkMaL November 26, 2013 at 12:05 PM

This comment has been removed by the author.

Reply

Anonymous December 8, 2013 at 3:45 AM

Hi Tahmid, i am really surprise to see your knowledge about power electronic specially microcontroller. This is amazing for me from BD. I need information about inverter, which u explain here but is it pure sinewave inverter in practically ? i use TC4420 mosfet driver but i can't get sin wave ,it has some distance to show half cycle and then other. what can i do pls tell me? i am waiting for ur reply.

Reply

▼ Replies



Tahmid  December 29, 2013 at 10:29 AM

The output should be a sine wave when passed through a low pass LC filter. You should be able to observe the waveform using an RC filter. But for practical use, you need to use an LC filter.

You can't use a TC4420 MOSFET driver to drive a MOSFET H-bridge. The two upper MOSFETs require high-side drive. TC4420 is just a low-side driver.

Regards,
Tahmid.

Reply



Unknown December 23, 2013 at 2:08 AM

Dear tahmid
how low pass filter is designed.
must reply

Reply

▼ Replies



Tahmid  December 29, 2013 at 10:26 AM

Search on Google for "LC low pass filter".

Regards,
Tahmid.

Reply



Unknown January 3, 2014 at 10:00 AM

Dear Tahmid

Firstly I would like to wish you a Blessed New Year for 2014, I have read through your blog and have attempted to design a pure sine wave inverter. I am using a H bridge topology with its high side mosfets driven by the mosfet driver IR2110. The mosfets are driven by a microcontroller, PIC16F877, which is programmed to drive the mosfets using SPWM. I have used your SPWM code and changed its base frequency to 60Hz using the same switching 16kHz switching frequency and 32 elements in the sine wave array. I am using Proteus 8 Professional and have upload the simulations that I have done so far.

<http://www.4shared.com/rar/xT5DGSgice/Files.html?>

Problems:

1. I have read your comments where you included a 0 at the end of the array to increase deadtime as well as when using the smart sine wave the peak value lower than $(PR2 + 1)$. I have done this and have realised that the dead time is not evenly spaced during certain intervals. Can you help me on this??

2. I interfaced the PIC with the H Bridge and I am getting an error stating:

{Spice} TRAN: Timestep too small; timestep = 1.25e-019: trouble with node #00052.

This node joins resistor 4, R4, and the right low side mosfet, Q4, together. I do not know how to remedy this problem. From your blog I have seen you posted your resultant sine wave after using a low pass filter on your H bridge which means you have gotten your circuit to work. I hope you can help me in getting me through this problem so I can get mine to work.

Reply

▼ Replies



lord aldebaran September 29, 2015 at 8:49 AM

Hello my friend!!! i've downloaded your schematic!!!! See the diagram for IR2110!!!
<http://www.avislab.com/blog/ir2101/> and see datasheet: <http://www.irf.com/product-info/datasheets/data/ir2101.pdf>!!! See the diagram!!!

Reply

Anonymous January 9, 2014 at 10:17 AM

Sir,

I want arduino uno code for 3 phase "Sinusoidal pwm"

It must be such that we must get totally 6 outputs- from pins 11,10,9 and 3,5,6 of arduino.

Please help me , it is an urgent....

My mail id is rhlk92@gmail.com

Reply

▼ Replies



lord aldebaran September 29, 2015 at 8:52 AM

for arduino uno you can generate output in square and convert to sine wave using filters then the output transformer!!!!

Reply



fmtech January 13, 2014 at 6:18 AM

Hi Tahmid, On ur pic16f72 sinewave, pls how can u help me out, 1, to change the push reset switch to normal off / on switch. 2, to increase the 3.2khz to 5khz thanks. fmtech83@yahoo.com

Reply



fmtech January 13, 2014 at 6:40 AM

pls can i use this pic16f72 circuit to drive 180volt inverter with IGBT?

Reply

▼ Replies



Tahmid February 22, 2014 at 12:42 AM

Yes you can. The PIC 16F72 has a CCP module that you can use for pulse width modulation for whatever you need. You just need a driver stage between the PIC and the IGBT.

Regards,
Tahmid.

Reply

Anonymous January 26, 2014 at 1:39 PM

HI TAHMID , i going to use the 16f877a microcontroller for producing sine wave.it have 2 ccp module.i want to produce the sine wave for 3 phase .so which pic i have to use.also tell about driver circuit.

Reply

▼ Replies



Tahmid February 22, 2014 at 12:51 AM

Take a look at the PIC16F777.

Additionally, you can look at the motor control DSPICs. Take a look at the dsPIC33MC series of microcontrollers.

You need to use 3 high-low side drivers for your circuit.

Regards,
Tahmid.



Unknown March 24, 2014 at 2:27 AM

thank you for your reply.i selected the pic 16f777 for producing 3 phase sine wave.by using this i want to produce space vector modulating wave using sampled reference frame algorithm .can you send the coding for this.

Reply

Anonymous January 27, 2014 at 12:43 AM

hi tahmid ,can you explain the program the sine wave generation using pic 16f877a

Reply

▼ Replies



Tahmid February 22, 2014 at 12:41 AM

Which part confuses you?

Reply

Anonymous January 29, 2014 at 9:13 AM

I didnt get the deadtime in my simulated output.

Pls give suggestion for modifying the sine table

I'm using 512 samples in the sine table. My oscillator freq = 20Mhz and carrier freq is 20Khz

Reply

▼ Replies



Tahmid June 10, 2014 at 11:44 AM

Add zeros at the ends of the sine table to include a sort of deadtime.

Reply



psgtech January 30, 2014 at 12:08 AM

hi i am using nuc 140ve3cn arm boardmay i know how to produce SPWM?

Reply

▼ Replies



Tahmid February 22, 2014 at 12:41 AM

I haven't worked with that microcontroller. However, the general idea should be the same. Go here:

<http://tahmidmc.blogspot.com/p/blog-page.html>

Look under SPWM and sine wave generation and inverter. Read through those to get a general idea of how SPWM works. Then use the onboard PWM module (I assume there's at least one) and create your own implementation of SPWM. If you get stuck, I'll be more than glad to help you try and resolve the issues - however I can only provide a high level support as I don't know the internals of that controller. Good luck!

Regards,
Tahmid.

Reply



Unknown February 19, 2014 at 1:27 PM

hi Tahmid,

Great tutorial. I am facing problem with output filtering circuit. Please give me idea about filtering circuit so that I will get pure sine wave.

Reply

▼ Replies



Tahmid February 22, 2014 at 12:38 AM

You will need to use a low-pass LC filter. Set the "cutoff frequency" to about 500Hz and keep on experimenting from there (trial and error method) to determine the best values of inductance and capacitance.

Regards,
Tahmid.

Reply



Unknown February 19, 2014 at 1:37 PM

Can I use TLP250 to drive the MOSFET using this tutorial?

Reply

▼ Replies



Tahmid February 22, 2014 at 12:36 AM

You will need to either use an external power supply or a bootstrap drive method to drive the high side MOSFET.

Regards,
Tahmid.

Reply



Unknown February 19, 2014 at 1:41 PM

can I use ccp1 and ccp2 both at a time for different pwm generation?

Reply

▼ Replies



Tahmid  February 22, 2014 at 12:37 AM

Yes you can. They are independent modules. Be careful though: they have the same timebase. Keep that in mind.

Regards,
Tahmid.

Reply



shivansh February 20, 2014 at 12:56 AM

sbit Direction at FlagReg.B0;
i cannot get this point..what is it used for ?

Reply

▼ Replies



Tahmid  February 22, 2014 at 12:35 AM

I created a variable called FlagReg. I defined the bit 0 of that variable and called it "Direction". I use this bit to understand/tell "which direction to go". When Direction = 0, current flows through the H-bridge in one direction (due to how the program is written; the program uses the Direction bit to tell which MOSFETs to drive to control direction of current through the H-bridge). When Direction = 1, current flows through the H-bridge in the other direction.

Regards,
Tahmid.

Reply



Unknown February 22, 2014 at 9:48 AM

thanks a lot tahmid for replying and giving me solution. If due to inductive load the nature of sine wave changes so can we control pwm using closed loop to maintain the pure nature of sine wave?

Reply



Unknown February 22, 2014 at 9:51 AM

Right now I am working on closed loop digital controller for dc dc converter using PV module (solar). Can you have any tutorial which related with this? Please send me link if it is available.

Reply



Unknown February 22, 2014 at 9:53 AM

Right now I am working on closed loop digital controller for dc dc converter using PV module (solar). Can you have any tutorial which related with this? Please send me link if it is available.

Reply



fmtech March 2, 2014 at 11:59 PM

Hi Tahmid, On ur pic16f72 sinewave, pls how can u help me out, 1, to change the micro switch to toggle off / on switch. 2, to increase the 3.2khz to 6.8khz on inverter and 20khz on charging thanks. fmtech01@yahoo.com

Reply

▼ Replies



Tahmid April 2, 2014 at 4:28 PM

Without idea of the inverter itself, I can't provide help.

Reply

Anonymous March 4, 2014 at 12:20 PM

Sir ,I have copied the same program and when I run the circuit in proteus, the frequency of pin ccp, D0, D2 keeps on increasing and never ends...

why is it so? I use the same pic, and compiler is mikroc.

please reply

Reply

▼ Replies



Tahmid April 2, 2014 at 4:28 PM

You're probably measuring time and not frequency. Double click the frequency counter and change measurement option from time to frequency.

Reply



Unknown March 22, 2014 at 11:39 PM

Dear Mr.Tamid, You posted Sine wave PWM Generator 16F877 working well. I would like to control speed by fixing pot 10k or 100k ohm.

could you help by sending Mikroc code.

Reply

▼ Replies



Tahmid April 2, 2014 at 4:27 PM

You can adapt the idea from here to vary duty cycle:

<http://tahmidmc.blogspot.com/2012/11/feedback-in-sine-wave-inverter-pic16f.html>

Reply



Abrar Jahin Antor April 2, 2014 at 4:46 AM

Hi, Tahmid...

This is the awesome tutorial on sine wave inverter. But how I add feedback option to this code (without ECCP)???

Thanks.

Reply

▼ Replies



Tahmid  April 2, 2014 at 4:26 PM

The idea is the same for feedback, with and without ECCP. So, you can adapt the idea from:

<http://tahmidmc.blogspot.com/2012/11/feedback-in-sine-wave-inverter-pic16f.html>

Reply



Unknown April 5, 2014 at 1:36 PM

Assalamualaikum Tahmid,

I'm Arman from Malaysia. currently i'm doing a project on SPWM multilevel inverter where i'm using 2 set of CHB to produce stepped output voltage. do i need to use 2 set of PIC to drive the CHB since the 2nd CHB need to be shifted 120 degree? thanks

Reply



Unknown May 14, 2014 at 7:42 AM

Hi Tahmid!

visiting your blog is always helpful. you guy is doing a great job. God bless you.

I copy your code in Mikro C compiler and build the hex file. Simulate the circuit in Proteus as shown in tutorial. PORTD is working correctly. but no signal at CCP1 pin. What can be wrong. please guide me.

thanks!....plz help..

Reply

▼ Replies



Tahmid  June 10, 2014 at 10:40 AM

Try the circuit in hardware and see if you get an output of CCP1 pin. Might be some sort of issue on simulation.

Reply

Unknown May 19, 2014 at 4:01 PM



thats a great job tahmid , im just wondering if it is possible to control this sine wave frequency from PC using UART , In other words , i need to know if I can vary the frequency from this Code ??? and what is the range of frequencies that can be applied to this project??? I need help in tha ASAP please, got 3 days for my presentation and I need to find an alternative for my DDS please replay to my email : hasanabdr@gmail.com

Reply



Unknown May 27, 2014 at 2:27 PM

i want to make inverter about 1200w can u give me schematic for this?

Reply

▼ Replies



Tahmid  June 10, 2014 at 10:42 AM

I don't have any readymade schematics that I can give. However you may be able to find some using Google.



Unknown June 27, 2014 at 7:06 AM

sir i am using same code for pic18f452.but 16khz frequency is not coming at ccp pin.and also tell me about function of flag register in this code.i am using ir2112 driver ic in simulation but high sided mosfet is not drive by driver ic.

Reply



Unknown July 16, 2014 at 3:36 PM

Kindly share the necessary code adjustments for running this code on PIC 16F877A please.

Reply

▼ Replies



Unknown July 17, 2014 at 3:54 AM

no adjustment required for this code.....

Reply

Anonymous September 4, 2014 at 10:54 PM

Hi Tamhid:

I have read carefully the explanation of the operation of the full bridge and have a question that is:

If A transistor turned on for 10 ms and D transistor receives the PWM signal, that branch of the full bridge is making some short circuits. Also if the transistors B and C are off which receives the load voltage.

In my view ought to function like this:
half cycle: A on for 10ms

D off
B receives the PWM signal
C receives the PWM signal complement

another half-cycle:

C on for 10ms
B off
D receives the PWM signal
A receives the PWM signal complement

such that two transistors of the same branch working in complement.

Please clarify if there is any confusion in the drawing.

greetings

Jose

Reply

▼ Replies



Tahmid  October 12, 2014 at 5:11 AM

You are correct. There was a mistake in Fig. 2. I've corrected it now.

Reply



Abrar Jahin Antor September 7, 2014 at 1:38 PM

HOW TO GET SINE WAVE LIKE FIG. 6 IN PROTEUS SIMULATION??

Reply

Anonymous October 28, 2014 at 7:47 AM

Hi bro.....I hope u r in good health and in good mood....

I go through ur post for sine wave generation using PIC 16F877a.....

its a nice ... i am also trying to make a single phase inverter....but I am using MPLAB with XC8 compiler.....

i could not understand this portion of Code because its in MikroC....

.....

```
sbit MOSA at RD0_bit;
```

```
sbit MOSB at RD1_bit;
```

```
sbit MOSC at RD2_bit;
```

```
sbit MOSD at RD3_bit;
```

```
unsigned char FlagReg;
```

```
sbit Direction at FlagReg.B0;
```

```
//0 -> MOS A + D
```

```
//1 -> MOS B + C
```

```
.....
```

please explain me for what purpose these instructions are used.....

Thanks in advance..... :)

Reply



Unknown November 7, 2014 at 8:13 PM

Hello Mr Tahmid could u help me y showing how to connect feed back transformer net work to pic for voltage regulation
> thank u

Reply

Ashab Uddin January 29, 2015 at 2:09 AM

Hi Tahmid:

Can you tell me about the filter you used for producing pure AC wave from SPWM.

Reply



Evgenii Litvinenko March 1, 2015 at 8:03 AM

Hey Tahmid, the Smart Sine software links that you gave is not available any more, if I ask you can you share it again. Thanks a lot.

Reply

Anonymous March 4, 2015 at 1:38 AM

how to generate 12 gating signals.

Reply

Anonymous March 4, 2015 at 2:15 AM

how to generate 12 gating signals

Reply



arunkumar March 4, 2015 at 2:23 AM

how to generate 12 gating signals

Reply



Imad March 15, 2015 at 2:14 PM

Thanks Mr Tahmid ,but the question is how does the same pure sine wave circuit using H bridge and same transformer can charge the battery knowing that the AC voltage is very small with respect to the needed for charging the battery

Reply

▼ Replies



Tahmid  May 9, 2015 at 1:07 AM

You will PWM the low-side MOSFETs in the bridge to create a boost circuit along with the transformer. The upper two MOSFETs are just used for their body diodes.

Reply



Unknown April 15, 2015 at 11:02 PM

hello I'm Tahmid current research on this topic. So you can give me gmail or email address so I can not learn from you. hope you can help me and nt in my Gmail address: nvtuan.dvt.91@Gmail.com I would like to address your mail

Reply



Govindaraj April 26, 2015 at 4:40 AM

Mr Tahmid SPWM is very usefull for me. I faced a problem with it whrn i compile your code using MPLAB IDE , HITECH C COMPILER , some times it shows too many errors nd some time it shows one error in _main. i have changes to acces single bit sbit MOSA at RD0_bit; to #define MOSA RD0; and sbit Direction at FlagReg.B0; to #define Direction FlagReg.B0; is it correct. please help me to overcome this.

Reply



Unknown May 3, 2015 at 4:47 AM

Hi Tahmid thanks for your valuable blog.. I tried the code it worked well in proteus. But while writing the HEX file generated by microc in my PIC16F877A using PICkit 2 there is absolutely no output. Even an LED blinking program does not work. But all works fine if i write an HEX file generated with MPLABX and XC8. Why is this so ? Does hex file genarated by MicroC does not work on real hardware ?

Reply



Unknown May 25, 2015 at 4:58 PM

hi mr tahmid i got some questions i hope (inshallah) u answer it all for me.

1st what (tbl pointer new,old,shift,mosa and tmp)refer to?

2nd i dont get the equation of 410 set_fre to get 50 hz frequency?

3rd why u wrote the(if (Direction == 0){

```
MOSA = 0;
```

```
MOSD = 0;
```

```
MOSB = 1;
```

```
MOSC = 1;
```

```
Direction = 1;
```

```
}
```

```
else{
```

```
MOSB = 0;
```

```
MOSC = 0;
```

```
MOSA = 1;
```

```
MOSD = 1;
```

```
Direction = 0;
```

```
}) under the interrupt void not under the main void?
```

4th (TBL_POINTER_SHIFT = TBL_POINTER_NEW >> 11;)what no.11 refer to here?

5th (CCP1CON = 12; //PWM mode) i changed no.12 to (0x12) once and to(0b 00010010)another time and the ccp1 pin didnt work,why?

6th (Pulse Width = (CCPR1L,DC1B1,DC1B0) * Tosc * TMR2 Prescale Value) i need to understand this equation with example if its possible?

*i know my questions are too much but im accounting on your kindness to help me to unerstand those questions

thanks in advance

Reply



errachidia solaire May 28, 2015 at 7:37 AM

slt comment brancher lcd 016 avec ce montage merci

Reply



Unknown May 29, 2015 at 11:51 AM

mr tahmid i wonder why you are not responding to us hope you are ok (inshallah) ?

Reply



Unknown June 18, 2015 at 7:18 AM

i fail DUTY_CYCLE = TBL_POINTER_SHIFT;CCPR1L = sin_table[DUTY_CYCLE]; i don't know why . Can you help me ? i use program MPLAB .And i want to use 16f887 with mosfet output to sine wave ,single phase full bridge inverter .

Reply



Unknown September 19, 2015 at 3:25 AM

hi Mr Tahmid I am following your blog for powers supply it is really helpful and its really one of nice article which u explain in simple words. I am using pic16f877a with MPLAB X IDE 3.10 and xc8 compiler. I compiled your code successfully. I changed it for 60HZ ac out. Now i am getting the 60HZ sine wave but with some garbage in it also if i increase DC line voltage like i increase from 30v DC to 45v DC after 45V DC the output sine wave distorted. all component are same as u suggested in diagrams here just Mosfet are different coz later my DC line voltage will be 250V DC..

please help me i am stuck here.

Reply



Unknown September 20, 2015 at 3:55 AM

Hi Mr Tahmid I done successfully is project for 50Hz and 60 hz. I make changes up to 200HZ but i just wondering for 400 HZ AC out. I am a hobbyist trying to make a Variable frequency drive (VFD) for Ac motor control. will u help me to get it up to 400 HZ. need help please

Reply



kikiloaw November 13, 2015 at 7:40 AM

wow its so nice to have a project like this. but is there any way not to use this IR2110 driver ? i have STP55NF06L mosfet here available. how do i use this? thanks in advance.

Reply



kikiloaw November 13, 2015 at 8:13 AM

how to change this into 60hz?

Reply

▼ Replies



kikiloaw November 14, 2015 at 7:41 AM

additional ifo: i have only 20Mhz

Reply

Anonymous February 15, 2016 at 7:33 PM

Hi Tahmid,I have really gone through ur article "Sine Wave Generation without ECCP - Using single CCP Module of PIC16F877A" since I lucky ran into ur live changing blog early this year(2016). And I have also tried implementing it in Mikrobasic pro for pic as the compiler am used to, but my port pins(RD0 thru RD3) is displaying 8khz instead of 50hz on the COUNTER TIMER am using for simulation on proteus while the ccp1 pin is displaying 16khz which is normal. Even when I try playing around with SET_FREQ value it still doesn't save the situation. Pls I need ur help on this earnestly. May GOD reward u mightily.

Reply



kolleyjay February 21, 2016 at 8:31 AM

please can anyone tell me the U2 and U3 represent regards to everyone here

Reply

▼ Replies



Unknown March 12, 2016 at 3:14 AM

U2 AND U3 ARE "AND GATES" (check logic circuit basics) eg. CD4081 check datasheet....Regards

Reply



Unknown March 19, 2016 at 7:29 AM

hello friend'm trying to do this project his need to know if you could provide an example of code in C for ccs using pic 18f877a

Reply



Unknown May 20, 2016 at 11:24 PM

HELLO CAN YOU TELLE ME WHAT IS THE WORK OF PORT AND IN THIS PROJECT

Reply



Unknown July 2, 2016 at 9:17 AM

<https://www.dropbox.com/s/b90n3ts1yo3pard/Untitled.png?dl=0>
please check my Proteus result if it is correct.,

Reply



Unknown July 18, 2016 at 1:43 PM

Hi

I want to build an inverter with 4 KHZ output and 24 V DC input. the power of the inverter is limited at 24 W. do the same algorithm and principle work with this inverter

Reply



Unknown October 31, 2016 at 9:47 PM

Hi Tamid!

I have a question,please help.

With this project (using 16F684) Can I using 2 IR2101 to driving full bridge mosfet directly?(using 5VDD for uC and 12Vcc for IR2101 and H bridge?

Thank you!

Reply



Unknown November 27, 2016 at 1:31 PM

This comment has been removed by the author.

Reply



Unknown November 27, 2016 at 6:45 PM

This comment has been removed by the author.

Reply



Unknown November 29, 2016 at 2:55 PM

Resolved. I got my 50Hz.

Reply



Unknown May 5, 2017 at 3:33 PM

Hello

Dear Sir

Have a nice day

Can you help me about your code in start page why not working with PIC16F877A because no signal in pin 19,20,21,22 i am compiler the code and i program with Beeprog2 Programmer and i select the 16MHZ ?

If i would like to convert 300VDC to 220VAC what i do ?

Thank you

Reply



Unknown May 9, 2017 at 4:41 PM

How can one use this in a half bridge with center tapped transformer?

Reply



Unknown July 7, 2017 at 3:44 PM

Oh! He stopped replying!

Reply



Apex UPS August 8, 2017 at 2:39 AM

Nice Blog.

[Online UPS Dealers in Chennai](#)

[UPS Dealers in Chennai](#)

[Exide Battery Dealers in Chennai](#)

[Inverter Dealers in Chennai](#)

[Exide Dealers in Chennai](#)

[UPS Service in Chennai](#)

[Amaron Battery Dealers in Chennai](#)

[Servo Controlled Voltage Stabilizers in Chennai](#)

[UPS Suppliers in Chennai](#)

[Servo Controlled Voltage Stabilizer Dealers in Chennai](#)

Reply

nikhil September 20, 2017 at 8:54 AM

can I drive a load of 3KVA

Reply



juma June 10, 2018 at 10:35 AM

This comment has been removed by the author.

Reply



juma June 10, 2018 at 10:45 AM

/*

This code was based on Baruti SPWM code with changes made to remove errors. Use this code as you would use any other baruti's works.

if you have advice please send a message to the baruti.themigambo@gmail.com

6.10. 2018

*/

```
const int sPWMArray[] = {500,500,750,500,1250,500,2000,500,1250,500,750,500,500}; // This is the array with the SPWM values change them at will
```

```
const int sPWMArrayValues = 13; // You need this since C doesn't give you the length of an Array
```

```
// The pins
```

```
const int sPWMpin1 = 10;
```

```
const int sPWMpin2 = 9;
```

```
// The pin switches
```

```
bool sPWMpin1Status = true;
```

```
bool sPWMpin2Status = true;
```

```
void setup()
```

```
{
```

```
pinMode(sPWMpin1, OUTPUT);
```

```
pinMode(sPWMpin2, OUTPUT);
```

```
}
```

```
void loop()
```

```
{
```

```
// Loop for pin 1
```

```
for(int i(0); i != sPWMArrayValues; i++)
```

```
{
```

```
if(sPWMpin1Status)
```

```
{
```

```
digitalWrite(sPWMpin1, HIGH);
```

```
delayMicroseconds(sPWMArray[i]);
```

```
sPWMpin1Status = false;
```

```
}
```

```
else
```

```
{
```

```
digitalWrite(sPWMpin1, LOW);
```

```
delayMicroseconds(sPWMArray[i]);
```

```
sPWMpin1Status = true;
```

```
}
```

```
}
```

```
// Loop for pin 2
```

```
for(int i(0); i != sPWMArrayValues; i++)
```

```
{
```

```
if(sPWMpin2Status)
```

```
{
```

```
digitalWrite(sPWMPin2, HIGH);
delayMicroseconds(sPWMArry[i]);
sPWMPin2Status = false;
}
else
{
digitalWrite(sPWMPin2, LOW);
delayMicroseconds(sPWMArry[i]);
sPWMPin2Status = true;
}
}
}
```

Reply



asdasd June 21, 2018 at 3:37 PM

Hi Tahmid.

I think at SPWM zero points one of transformers pins is floating. Am i wrong?

Reply

Anonymous July 4, 2019 at 6:14 AM

Hi tahmid,

your code is very useful. in this code i can not control frequency. please add increment and decrement push button switch with this code.that means i want to control frequency with push button. please help me to modify this code and pls send it to my mail..milonctl@yahoo.com

thanks

Reply



tesla solaire April 3, 2020 at 8:30 AM

inverter no Feedback tested all ok tinks for yo tahmid 12v & 170v all ok 170V 320V not fonction

Reply

Chayan Mistry February 1, 2021 at 9:22 AM

Hi vaiya, thanks for this nice article.

Is it possible generate 2 SPWM without ECCP & AND Gate. Can you give any example or reference.

Reply



Antonio April 30, 2021 at 6:28 PM

Hi Tahmid, congratulations on your project,

There's like you. alter the code routine to work at 60 Hz? here in Brazil our frequency is 60Hz

Reply



Unknown August 7, 2021 at 3:37 PM

ple can u send me Hex file for pic16f877 with circuit diagram,my email is reubenmuniko727@gmail.com

Reply



christina September 15, 2021 at 5:41 AM

For over 3 year's I have been practicing telekinesis and nothing worked, until I came across a woman saying she was helped by Dr white, I never believed, but because I was desperately looking for help, I wrote to him on email and he told me he will help with with a ring and with this ring I can do all things, I followed the instructions, and now I can do all things just with my concentration and focused mind, contact him on email: wightmagicmaster@gmail.com
WhatsApp: +17168691327

Reply



Amenda November 14, 2021 at 2:49 PM

I am very grateful to Dr Dawn Acuna, for bringing back my husband who left me for another woman, that moment my husband Left me I thought I lost everything until a friend of my gave me Dr Dawn Acuna, WhatsApp contact, I messaged her and told her the pain I was going through so she told me that everything was going to be fine that if I have the faith and believe in her that the spell will surely work for me and my husband will surely come back home and she told me what to do, so those things were done and 48 hrs later my husband came back home begging for my forgiveness, am so happy and grateful to Dr Dawn Acuna, if you need her help contact her, she's accurate and sincere,

* If you want spell to conceive.

*If you want to get pregnant.

* If you want to return your lover

*If you want to cure any kind of sickness

* If you need spell to get good job. *If you want to stop having miscarriage. And E.T.C. write her on email { dawnacuna314@gmail.com }

WhatsApp: +2348032246310

Reply



Bruno February 22, 2022 at 2:18 PM

Soțul meu a cerut divorțul, dar cu ajutorul Dr. Dawn Acuna, eu și soțul meu trăim în pace, cu bucurie și fericire, Dr, Dawn acuna a ajutat-o cu o vrajă de dragoste puternică care l-a făcut pe soțul meu să realizeze cât de specială sunt și acum mă tratează ca pe o regină și mă îmbrățișează toată noaptea, asta este ceea ce mi-am dorit vreodată de la soțul meu, datorită Dr. Dawn acuna, contact pe Whatsapp:{ +2348032246310 }

*Dacă vrei să te împaci cu iubitul tău.

*Dacă doriți ca plante medicinale să conceapă.

*Dacă vrei un inel magic.

*Dacă vrei să vindeci orice fel de boală.

*Dacă vrei să devii celebru în afacerea ta.

Si altii,

Îți promit 🙏 la sută rezultat pozitiv, e sinceră, se ține mereu de cuvintele ei.

Reply



Ivy Tyler March 3, 2022 at 2:06 PM

Çdo problem ka një zgjidhje kur takoni personin e duhur! Mos kini frikë se ekziston një magjistari i shkëlqyer dhe zgjidhës shpirtëror i problemeve që është në gjendje t'i japë fund problemit tuaj. Dua te vleresoj Dr Fady qe me riktheu martesen time me magjine e saj, Pas 1 viti ndarje me burrin tim, me ndihmen e magjistares se mrekullueshme Dr Fady burri im erdhi ne shtepi dhe tani jemi te lumtur bashke per mire, edhe nje here faleminderit per Dr Fady gjithashtu shëroi të gjitha llojet e sëmundjeve si p.sh.

1 DEKLARATË DASHURI

2 WIN S EX KTHIMI

3 FRUTA GOMS

4 NJOFTIM PËR PROMOCION

5 DEKLARATA E MBROJTJES.

6 MAGJIA E BIZNESIT.

7 DEKLARATË E MIRË PUNËS.

8 HIV SIDA.

9 NDALO MISKARRIMIN.

10 MIROOD NJOFTIM SHPALLJE.

11 SHKAK I TELEKINEZËS.

DREJTSHKRIMI I LOTARIVE DHE ÇËSHTJA GJYQËSORE Kontakt Kontaktoni atë për ndihmën tuaj nëpërmjet:
drfady720@gmail.com

WHATSAPP:+2349039422406

Reply



Pretty March 29, 2022 at 6:20 AM

M-am căsătorit cu dragul meu soț în ultimii 12 ani fără să rămân însărcinată, iar fibromul a fost problema. Am luat diferite medicamente prescrise, dar nu am putut să le vindec, dar soțul meu era atât de încrezător în mine și mă tot încuraja că într-o zi cineva mă va numi mamă. nu s-a odihnit căutând o soluție de la diferiți medici, tot ce au putut vedea a fost o intervenție chirurgicală și mi-a fost frică de asta, o prietenă din cabinetul meu mi-a prezentat doctorul DAWN ACUNA, ea a spus că Dawn acuna a ajutat-o când avea tubul blocat și a ajutat-o și ea. multe dintre prietenele ei să conceapă,

I-am scris imediat pe Whatsapp, mi-a promis ca ma ajuta dupa ce i-am explicat totul, mi-a dat niste instructiuni care am facut totul perfect conform instructiunilor, la 3 saptamani dupa tot am fost la spital si doctorul mi-a confirmat sarcina in 1 saptamana dar chiar acum am copilul meu frumos.

*Dacă vrei să tratezi infertilitatea.

*Daca vrei sa ramai insarcinata rapid.

*Dacă vrei să-ți întorci iubitul.

*Dacă vrei o căsătorie pașnică.

*Dacă doriți să tratați boala canceroasă.

Și mulți alții îl contactează pe Dr dawn acuna pe Whatsapp:+2348032246310

E-mail: dawnacuna314@gmail.com

Reply



weight loss pills for women and men April 14, 2022 at 6:13 AM

We are a reputed and secure online brand that delivers online medicines like cialis 10mg , percocet 10mg, cheap viagra 50mg, neurontin overnight delivery, blue bars b707 , yellow football xanax 1mg, verapamil 40 mg , oxycodone acetaminophen 10-325 en español and fioricet for tooth pain to your doorstep at very affordable rates.

Reply



Enter Comment



[View web version](#)

About Me



 **Tahmid**

I am Syed Tahmid Mahbub, from Dhaka, Bangladesh, born on August 1, 1994. Electronics is my passion and from class V, I have been learning electronics. I learnt and worked mostly on SMPS, power electronics, microcontrollers and integration of microcontrollers with SMPS and power electronics. I've used PIC and AVR microcontrollers - PIC 10F, 12F, 16F, 18F, 24F, dsPIC 30F, 33F, PIC32, ATmega and ATtiny, integrating them with various SMPS and power electronics circuits. I have completed my Bachelor's degree from Cornell University (Class of 2017) in Ithaca, New York, USA, majoring in Electrical and Computer Engineering (ECE). I am a member of the forum www.edaboard.com, where I am an "Advanced Member Level 5" (the highest level attainable) and also the forum allaboutcircuits.com, where I am a "Senior Member". I post to help solve electronics-related problems of engineers and engineering students from all over the world. I love watching and playing cricket and football (soccer), and listening to music. I am now a hardware engineer at Apple in Silicon Valley, California, USA.

[View my complete profile](#)

Powered by Blogger.