

# Photovoltaic Power Converter

Students:  
Thomas Carley  
Luke Ketcham  
Brendan Zimmer

Advisors:  
Dr. Woonki Na  
Dr. Brian Huggins

Bradley University  
Department of Electrical Engineering  
5/10/12

## Table of Contents

Abstract.....	iv
Project Summary.....	1
Overall System Block Diagram .....	1
DC Subsystem .....	1
DC Subsystem Requirements.....	1
Boost Converter Description.....	2
Boost Converter Design Equations .....	2
Boost Converter Key Components.....	3
Photovoltaic Panel .....	4
Gate Driver .....	4
Other Key Components.....	4
Circuit Layout .....	5
Boost Converter Simulations .....	6
Boost Converter Testing .....	7
Eliminating Voltage Spikes.....	8
Boost Converter Results.....	9
Maximum Power Point Tracking (MPPT) .....	11
Perturb and Observe (P&O) .....	11
Incremental Conductance (IC) .....	13
Implementing MPPT .....	13
eZdsp F2812 Features .....	13
Voltage Sensing.....	13
Current Sensing.....	14
Simulink Modeling .....	15

MPPT and Soft Start Results .....	17
AC Subsystem.....	17
AC Inverter General Overview and Theory .....	17
Controlling the Inverter with the eZdsp .....	19
Gate Drive .....	22
LC Filter .....	23
Experimental Results .....	23
Conclusion .....	23
References .....	24

## Abstract

Solar energy has become increasingly popular as the desire for clean energy has grown. This project explores a design method of solar power converter. The solar power converter incorporates a photovoltaic panel, boost converter, and inverter system to create 60 Hz, AC grid power. First, the DC voltage from a photovoltaic panel is stepped up using a boost converter. A Maximum Power Point Tracking (MPPT) algorithm known as Perturb and Observe method controls the duty cycle of the boost converter to ensure maximum possible power is drawn from the panel. The switching signal for the boost converter is generated by a Texas Instrument, 32 bit fixed point Digital Signal Processor (DSP), TMS320F2812. The stepped up voltage from the boost converter is then sent through a single phase inverter to output 60 Hz AC power. A Sinusoidal Pulse Width Modulation (SPWM) approach is used to control the inverter. The PWM signals are realized with the DSP. An LC filter removes unwanted switching harmonics from the inverter output. Matlab Simulink and Code Composer Studio were used to program, simulate, and debug the DSP based power converter.

## Project Summary

The Photovoltaic Power Converter system supplies both DC and AC power. The system utilizes Photovoltaic (PV) panels to generate DC power. Initially a DC-DC boost converter steps up the DC voltage output from the PV panel. A Maximum Power Point Tracking (MPPT) algorithm is implemented to ensure that maximum power from the solar panel is obtained. A current sensor, a voltage sensor, and a DSP board are used to implement the MPPT algorithm. The boosted DC voltage is then inverted to 60Hz AC Power. The inverter contains four IGBTs controlled by the same DSP board. An LC filter is used to smooth out the output.

## Overall System Block Diagram

The total system can be broken down into two subsystems, DC and AC. The overall functional block diagram is shown in Figure 1. The DC subsystem is shown on the left side of the block diagram, and the AC subsystem is shown on the right side.

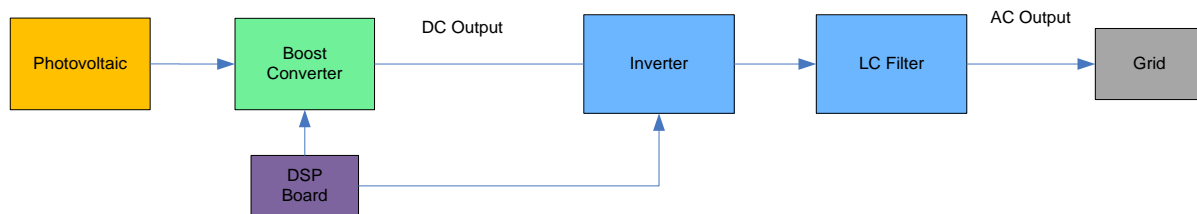


Figure 1 – Functional Block Diagram

## DC Subsystem

### DC Subsystem Requirements

- The boost converter shall accept a voltage from the photovoltaic cells
  - The input voltage shall be 48 V.
  - The average output shall be 200 V +/- 25 V
  - The voltage ripple shall be less than 20 V
  - The open loop boost converter shall operate above 65% efficiency
- The boost converter shall perform maximum power point tracking
  - The PWM of the boost converter shall be regulated based on current and voltage from the PV array
  - The efficiency of the MPPT system shall be above 80%

## Boost Converter Description

A boost converter is a switch mode power supply that has an output voltage higher than its input voltage. The switching in a boost converter is done through a MOSFET or IGBT. When the switch is closed the current flows in the first loop only, the current through the inductor grows. The switch then opens, and the voltage across the inductor and the input combine in series to charge up the output capacitor to a higher voltage than the input. The duty cycle of the switching signal determines the output voltage. The longer the switch is closed, the higher the output voltage will be expected. A sample Boost Converter is shown in Figure 2.

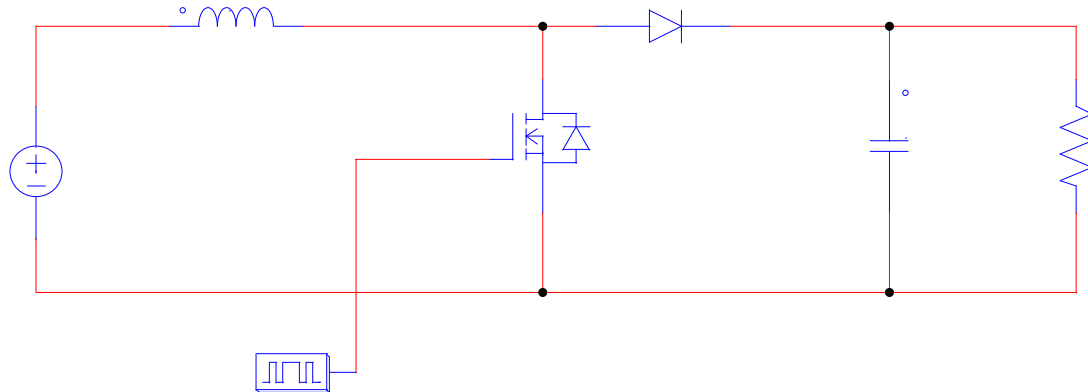


Figure 2 - Boost Converter

The purpose of the boost converter in this system is to take the input voltage from the photovoltaic panels and boost it to a voltage high enough for the inverter to convert the DC voltage to 120 V<sub>RMS</sub> AC. Originally the project was to have four photovoltaic panels. Each photovoltaic panel provided 50 watts of power and has a nominal voltage of 12 volts. This would have made the input voltage 48 volts nominal. However, we only had two panels available, and only one mounted and ready to use. The boost converter was designed to make use of this single photovoltaic panel, which could output up to 20V on a sunny day.

## Boost Converter Design Equations

One thing that must be calculated when designing a boost converter is the expected voltage output. The maximum power point tracking system would vary between 30% and 70% duty cycle. The expected maximum output voltage can be calculated by the equation below.

$$V_o = \frac{V_{in} - V_{trans} \cdot D}{1 - D} - V_d = \frac{20 - 1.3 \cdot .7}{1 - .7} - 1.3 = 62.33 \quad (1)$$

$V_{trans}$  is the voltage drop across the MOSFET, and  $V_d$  is the voltage drop across the diode in the system. This calculation shows that the output voltage should be around 62 volts with a 70% duty cycle.

An important consideration that must be made during the design of a boost converter is the size of the inductor. One way to determine an appropriate size for the inductor is to take into consideration the switching frequency, duty cycle, output voltage, and minimum load current. As can be seen from equation(2), at the lighter the load, the higher the inductance needed. This can even be applied to switching frequency.

$$L = \frac{V_{out} - V_i}{\min(I_{load}); f_s} \cdot (1-D) = \frac{66.66V - 20V + 1.3V \cdot (1-.7)}{(66V/1200\Omega) \cdot 30000Hz} = .0086H \quad (2)$$

The reason that the inductance calculated is so high is because the load is too small at 1.2k $\Omega$ , and the switching frequency is just 30kHz. This value of inductance is problematic since having 8.6 mH is quite large. However, there is another way to determine an appropriate inductor value for the boost converter.

It is important for the boost converter to remain in continuous conduction mode for control purposes. This means that the current through the inductor never reaches zero during a full switching cycle of the boost converter. If the boost converter is in discontinuous mode, the output voltage equation becomes a function of frequency, inductance, and output current, as well as duty cycle as seen equation (3).

$$\frac{V_o}{V_i} = 1 + \frac{V_i \cdot D^2 \cdot T}{2 \cdot L \cdot I_o} \quad (3)$$

By using equation (4) an appropriate resistance for the output load can be determined for the amount of available inductance. This equation shows that a load with a resistance of 1224.5 $\Omega$  or higher may not be used. If a higher resistance than  $R_{crit}$  is used, the load will be too light and the boost converter will enter into discontinuous mode.

$$R_{crit} = \frac{2 \cdot L \cdot f_s}{D(1-D)^2} = \frac{2 \cdot .0086H \cdot 30000Hz}{.3(1-.3)^2} = 1224.5 \quad (4)$$

## Boost Converter Key Components

The key components found in a boost converter are the MOSFET, inductor, diode, and gate driver. The boost converter also contains capacitors on the output in order to smooth out voltage ripples. Since this boost converter is boosting the voltage from a photovoltaic panel, it was also a key component to be considered when designing the boost converter.

## Photovoltaic Panel

The photovoltaic panel used in this project is the BP350J in Figure 3. This panel can provide up to 50 watts of power. It has a nominal voltage of 12 V. Upon testing the panel on a sunny day, it outputted 20 V at about 250 mA. This is only about 5 watts of power. The panel was not fully aligned to the sun and was behind a window. We did not expect to get maximum power out of the panels during testing.



Figure 3 – BP350J Panel

## Gate Driver

To drive the MOSFET, we chose the IR2110 gate driver. With this gate driver up to two MOSFETs can be driven simultaneously. As seen in Figure 4, only the low side output is being used in our boost converter. The high side driver may be needed for other applications, such as a full bridge inverter.

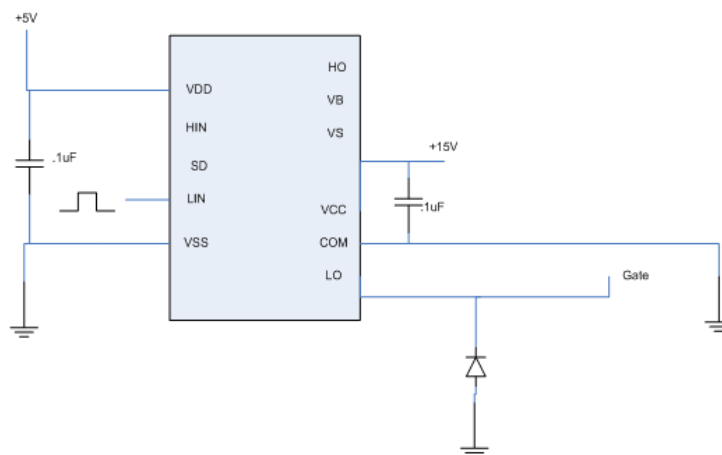


Figure 4 – IR2110 Gate Drive

## Other Key Components

The MOSFET, inductor, diode, and capacitor were all chosen with the consideration that the full-power system would be created. This meant that the MOSFET had to have a high breakdown voltage,



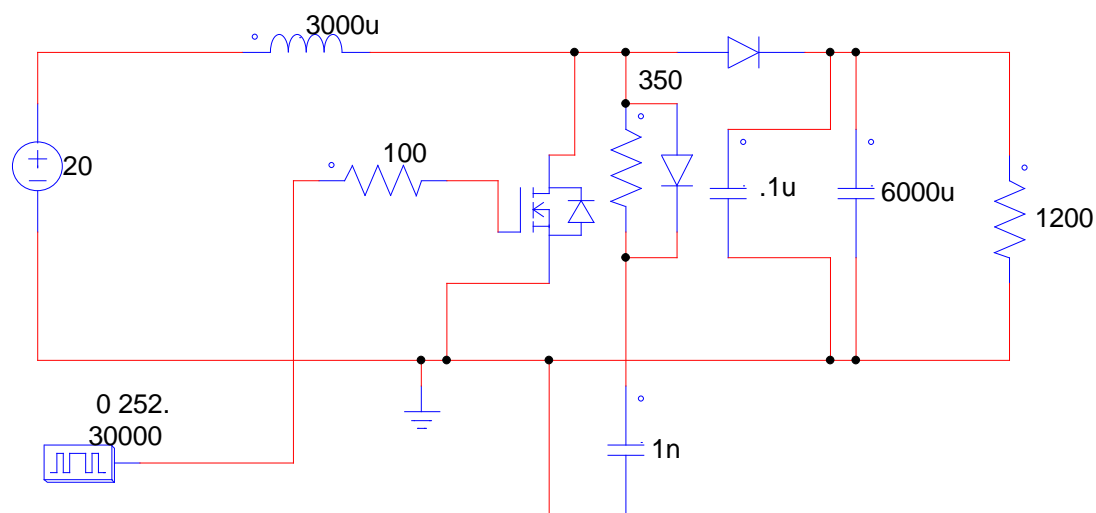
and be able to withstand high currents. The MOSFET chosen is the IRFP4768PbF, which has a breakdown voltage of 250 V and a max drain current of 93 A.

The diode chosen is the HFA50PA60C. This is an ultrafast diode with a reverse recovery time of 50ns. The reverse voltage is 600 V and forward current is 25 A. This diode should have no problems with the conditions of our full system.

The inductors and capacitors were chosen with voltage and current ratings that were well above the highest expected currents and voltages in our system. For more details on any of the components, please consult the datasheets.

## Circuit Layout

The circuit was designed in PSIM before laying out the components in hardware. As can be seen in Figure 5, there is a snubber circuit and gate resistor in addition to the standard boost converter layout. These extra components are to reduce large voltage spikes on the output.



**Figure 5 - Boost Converter PSIM Schematic**

All of the components were soldered to a perforated board, mounted to a heat sink, or mounted in a breadboard. The entire boost converter circuit can be seen in hardware in Figure 6.

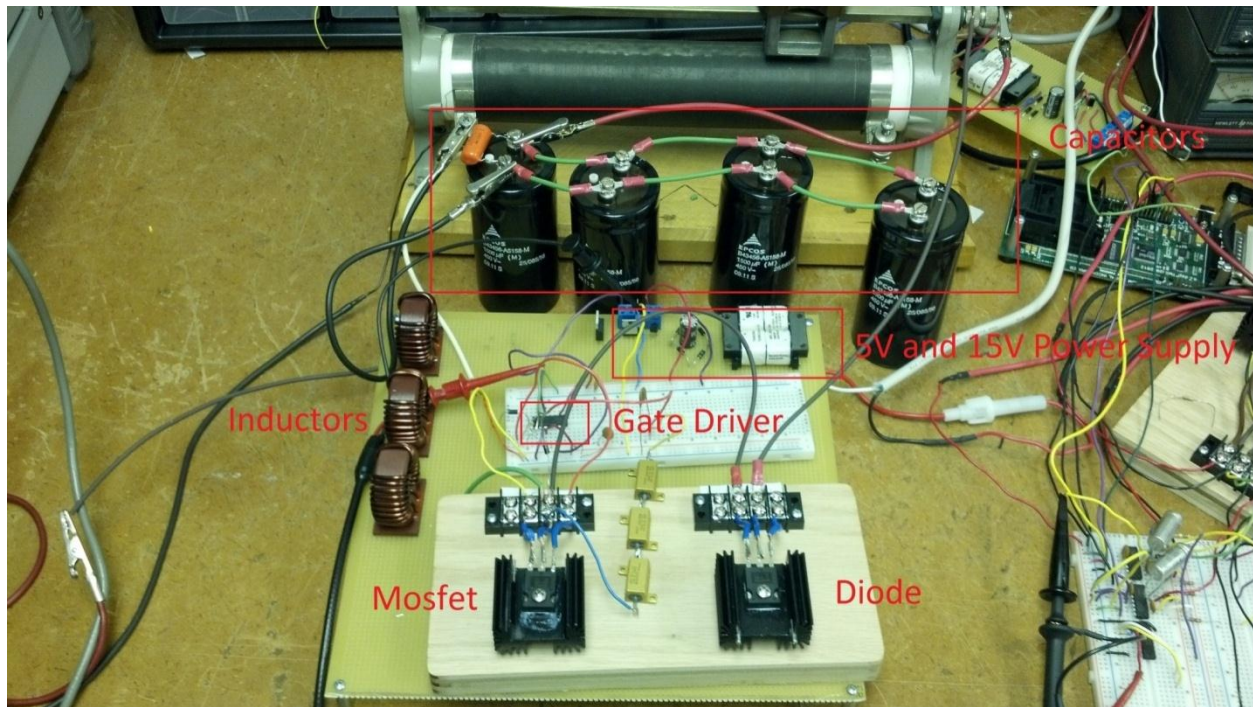


Figure 6 - Boost Converter

### Boost Converter Simulations

Before actually testing the boost converter in hardware, some simulations of the boost converter were done in PSIM. These tests were of the boost converter turning on to 70% duty cycle, boosting an input voltage from 20 V to 66V. In Figure 7 the voltage is seen to spike above the expected output voltage and then slowly settle. This is because of the large value of the output capacitance.

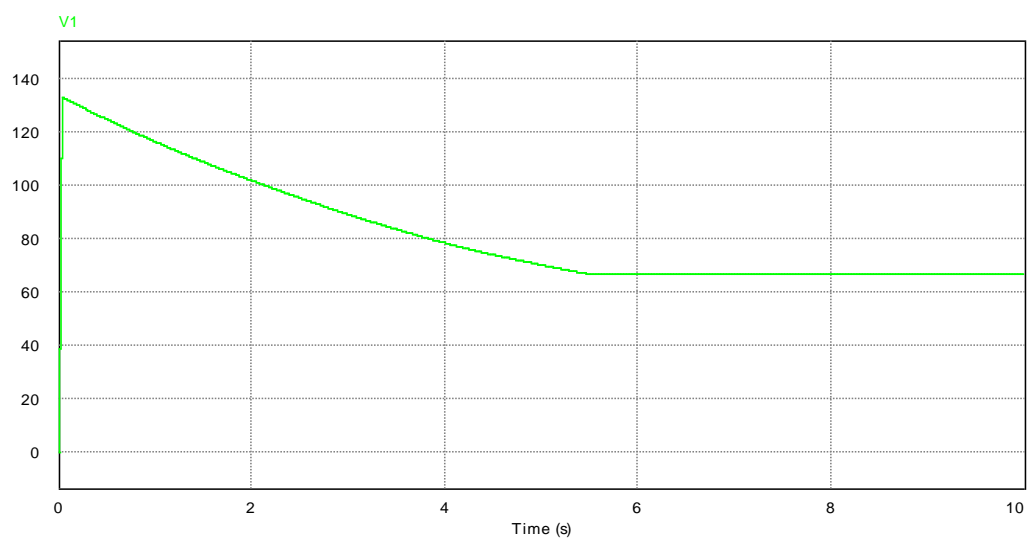


Figure 7 - Simulated Output Voltage

Much like the output voltage, there is a large spike in current as the boost converter turns on. Both of these spikes can be avoided by using a soft start technique to gradually turn on the boost converter.

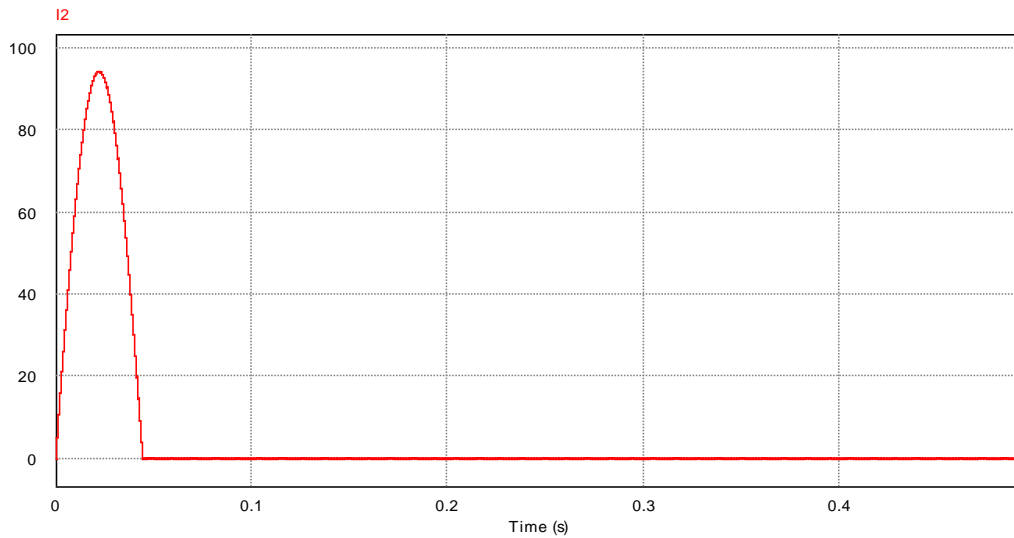


Figure 8 - Simulated Input Current

### Boost Converter Testing

When the boost converter was initially tested, there was an unexpected problem with the output, large voltage spikes were present at the turn on and turn off of the MOSFET.

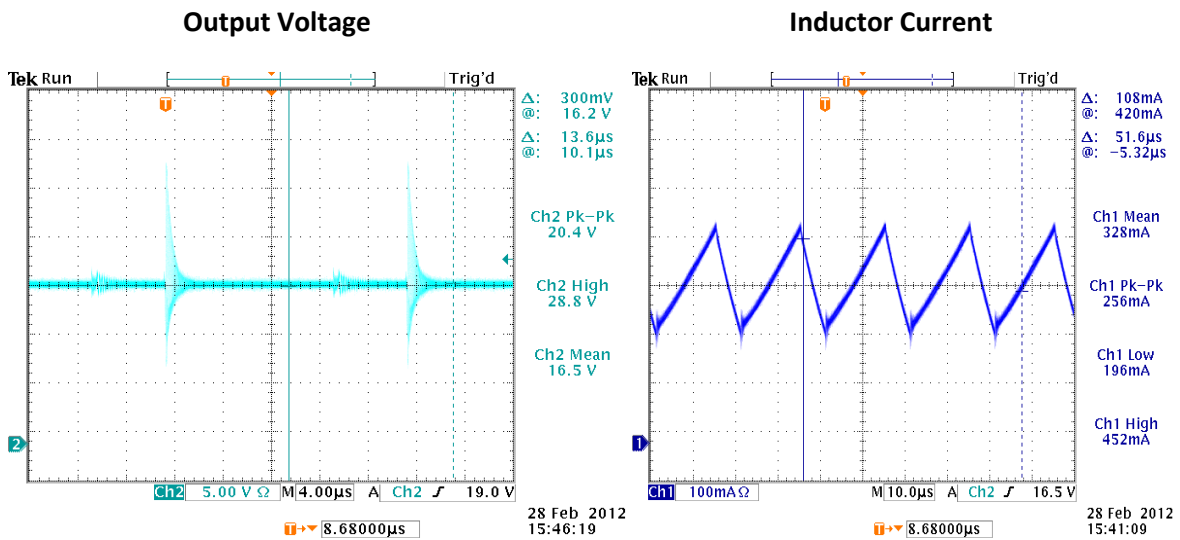


Fig 9 – Output Voltage and Inductor Current

The above simulation is only boosting 10 V to 16.5 V at a 40% duty cycle. There are 20V voltage spikes on the output. At higher voltage levels these spikes could rise above the breakdown voltage of

the MOSFET and damage the device. Therefore, an additional circuitry is needed to reduce these high voltage spikes in the output.

## Eliminating Voltage Spikes

There are a few reasons why there are large voltage spikes on the output of the boost converter. MOSFET switches have a parasitic capacitance and inductance associated with their switching, and diodes have a forward recovery time. When the MOSFET turns off, voltage builds up across the gate-source capacitance while the diode tries to conduct in the forward direction. If the diode turn on time is slow enough, or the switching time of the MOSFET is fast enough, a voltage could build up at the drain of the MOSFET, hence the spike occurs. Increasing the frequency of the boost converter makes the spikes even larger, as the MOSFET is switching faster. This is the reason why the boost converter is only switching at 30 KHz.

Fortunately there are ways to reduce the voltage spikes. Increasing the turn on and turn off time of the MOSFET lowers the voltage that builds up at the drain of the MOSFET. The idea here is that if the MOSFET takes longer to turn off, then the diode has more time to conduct. Additionally, an RC snubber circuit can be included to further dissipate the voltage build up at the drain. The capacitor in the RC snubber charges up when the switch opens and dissipates the power through the resistor. See Figure 6 for details of the circuitry.

By adding 100 $\Omega$  in series with the MOSFET gate the turn off time was increased significantly.

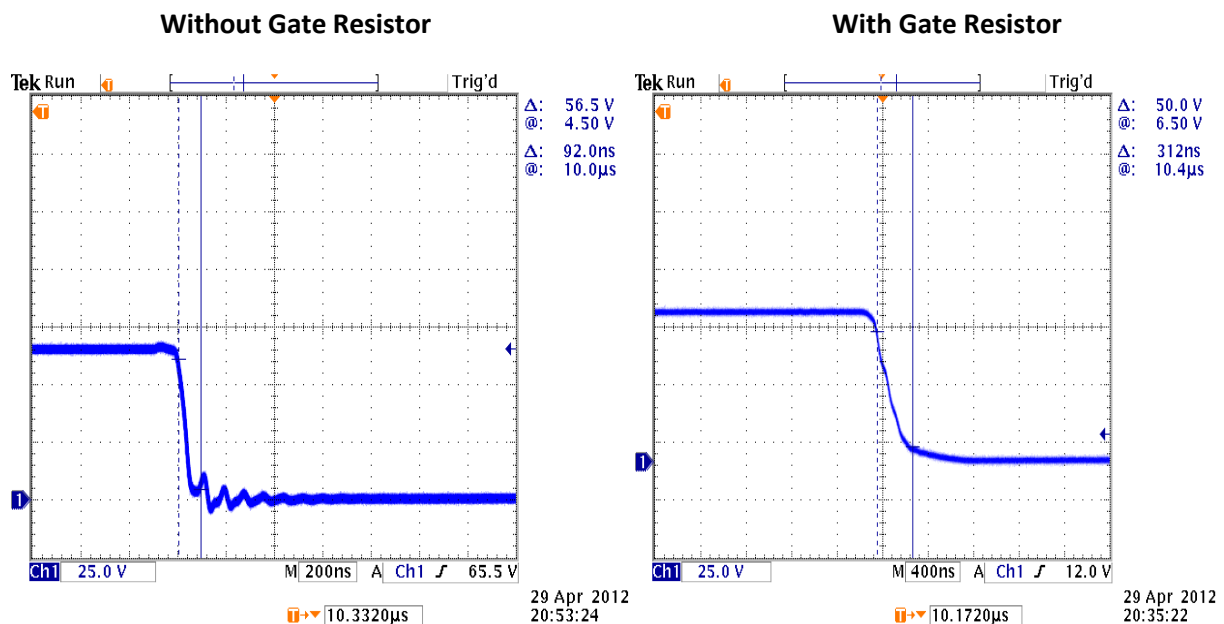


Fig 10 - Turn off time

The turn off time was increased from 92 ns to 312 ns by just adding the gate resistor, lowering the amplitude of the voltage spikes. Designing the RC snubber involved the use of a few design equations.

The value of the capacitor depends on the turn off time of the MOSFET,  $t_f$ , the maximum expected output voltage,  $V_{off}$  and maximum inductor current.

$$C_s = \frac{I_L \cdot t_f}{2V_{off}} = \frac{.3A \cdot 300 \mu s}{2 \cdot 70V} = .643 \mu F \quad (6)$$

The capacitor value is very low, and this is expected since the capacitor only has a very short amount of time to charge up. A standard value of 1nF was used in the circuit.

The resistor value is calculated using two equations. One equation uses the duty cycle and switching time combined with the capacitor value to determine an appropriate resistance. The other equation determines a resistor value that will ensure that the voltage across the resistor does not exceed the breakdown voltage of the MOSFET. The resistor value should be below this calculated value.

$$R_s < \frac{V_{br}}{I_L} = \frac{250V}{.3} = 833.33 \Omega \quad (7)$$

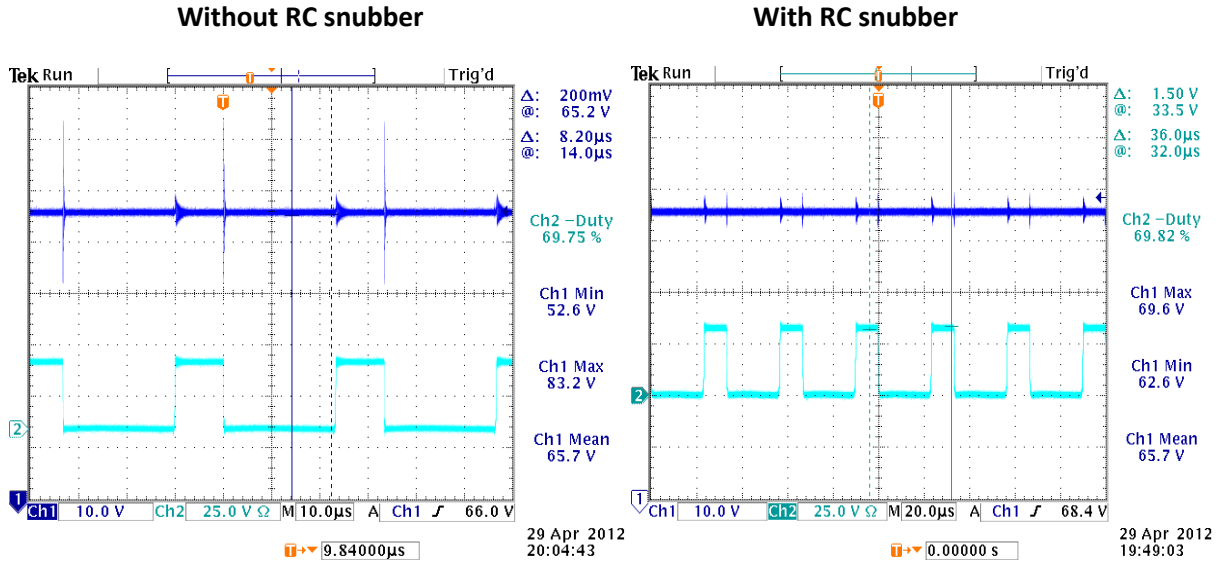
$$R_s < \frac{D_{min} \cdot T_s}{3 \cdot C_s} = \frac{.3 \cdot .000033 s}{3 \cdot .643 \mu F} = 5185 \Omega \quad (8)$$

Where  $V_{br}$  is the breakdown voltage of the diode and  $D_{min}$  is the minimum duty ratio as 0.3.

Since the calculated value was 833.33 $\Omega$ , a 450 $\Omega$  resistor was used.

## Boost Converter Results

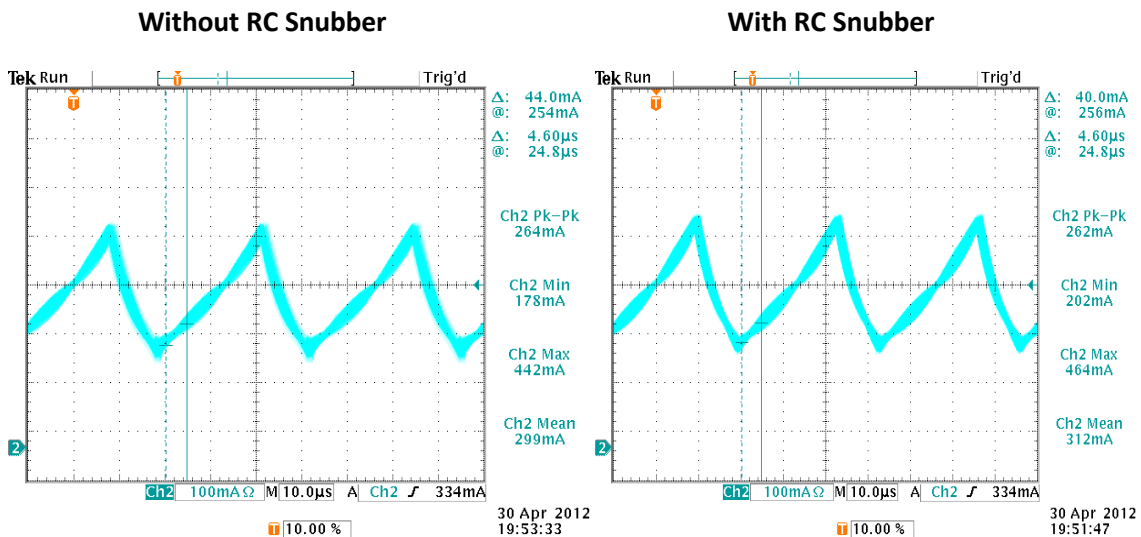
Adding the RC snubber and gate resistor successfully reduced the output voltage spikes. Below are oscilloscope screens of the boost converter boosting 20 V to 66 V at 70% duty cycle and 30 KHz.



**Fig 11 - Boost Converter Output Voltage RC Snubber Results**

The peak to peak value of the voltage spikes reduced from 30.6 V to just 7 V. Further optimizations could be made to the RC snubber design and gate resistor value to further reduce the spikes. Further reduction of the spikes might allow a much higher frequency to be used with the boost converter, which may improve efficiency and allow less inductance to be used.

Using the RC snubber did slightly decrease efficiency slightly. The oscilloscope plot on the left in Figure 12 shows the inductor current without the RC snubber and the plot on the right shows it with the snubber implemented. The efficiency decreases from 60.7% to 58.1% when the snubber is added.



**Fig 12 - Boost Converter Inductor Current RC Snubber Results**

The boost converter functions as expected, although it has a low efficiency. The efficiency can be improved by shortening leads in the circuit, optimizing the RC snubber circuitry, and increasing switching frequency. The boost converter had no problems receiving an input voltage from a photovoltaic panel. This section of the project is ready to be connected to the MPPT software and inverter circuitry.

## Maximum Power Point Tracking (MPPT)

Maximum Power Point Tracking (MPPT) is a method of ensuring that the Solar Panels are always delivering the maximum power regardless of any conditions. As insolation and temperature change, the photovoltaic panel will not be able to supply the same maximum power. Figure 13 shows a photovoltaic voltage versus current curve, and Figure 14 shows an example of a photovoltaic voltage versus power curve[1,2]. The peak is where the panel is operating at max power. This is the point we want to be at all the time.

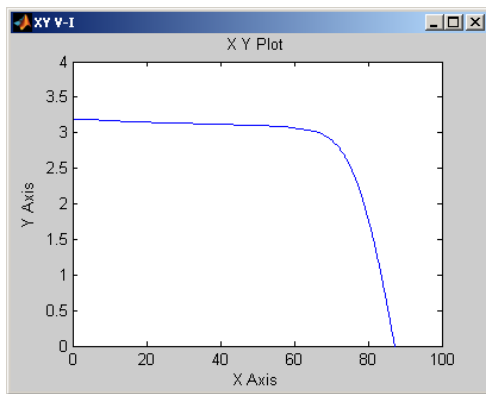


Figure 13 – PV Voltage (X) v. Current (Y)

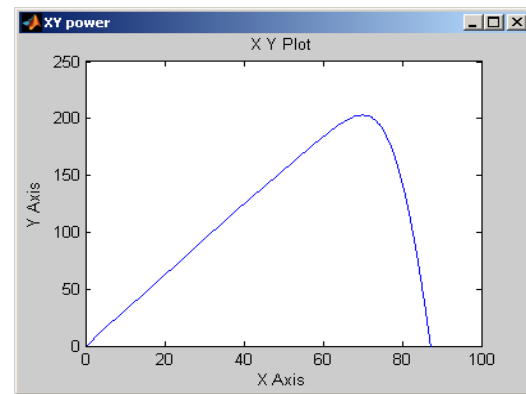


Figure 14 – PV Voltage (X) v. Power (Y)

If the duty cycle of the boost converter remains the same, we will not be getting maximum power from the solar panel. By measuring the input voltage and input current to the boost converter from the photovoltaic panel, we can compare current power and past power to the current boost converter duty cycle and past boost converter duty cycle. This algorithm is called Perturb and Observe.

## Perturb and Observe (P&O)

There are many algorithms for MPPT[4]. Two algorithms studied this year were Perturb and Observe (P&O) and Incremental Conductance (IC). Perturb and Observe introduces an initial perturbation to the boost converter voltage by changing the gate signal duty cycle and then observations are made using sensing circuitry to change the gate signal duty cycle to move closer to the Maximum Power Point (MPP). Perturb and Observe uses voltage and current measurements to calculate change in power over a change in time ( $\Delta P$ ) and change in the duty cycle ( $\Delta D$ ) of the signal sent to the gate of the switch in the boost converter.

Table 1 – P&amp;O Cases

$\Delta P$	$\Delta D$	Next Duty Cycle
+	+	+
+	-	-
-	+	-
-	-	+

Given that  $\Delta P$  and  $\Delta D$  can be each either positive or negative, there are four cases to determine whether the duty cycle of the gate signal should be increased or decreased. The four cases are shown in Table 1. The first case, when both power and the duty cycle has increased, the duty cycle should continue to increase toward the MPP. Case two is similar except the duty cycle should continue to decrease toward the MPP. Cases three and four occur when the power has decreased, so the duty cycle has moved the PV voltage away from the MPP. The duty cycle is then reversed. It is decreased in case three and increased in case four. The flowchart in Figure 15 explicitly explains the P&O algorithm.

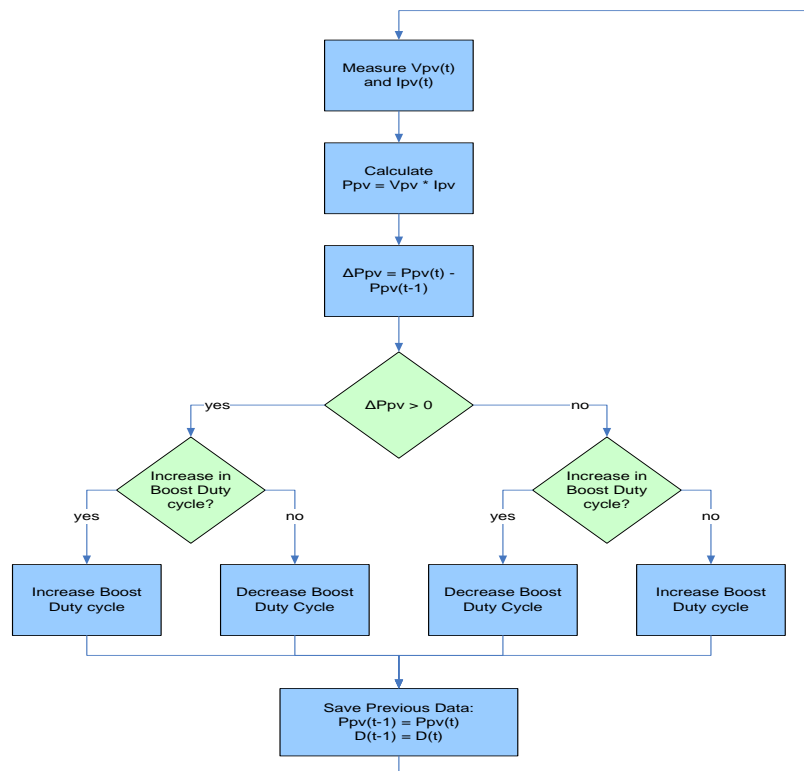


Figure 15 - P&amp;O MPPT Algorithm Flowchart



## Incremental Conductance (IC)

Incremental Conductance (IC) is the second MPPT algorithm possibility for this project. The details of IC will not be discussed in this paper, but here are a few advantages and disadvantages of IC and P&O. Unlike P&O, IC is able to track a more rapidly changing MPP. However IC has increased susceptibility to noise and has increased complexity compared to P&O. P&O is a very popular and widely accepted MPPT algorithm and simpler to implement than IC. One drawback of P&O is the slight power loss from the perturbation of the duty cycle if it moved away from the MPP. This power loss is necessary because the perturbation is required to determine if the duty cycle needs to be increased or decreased to find the MPP. With all this taken into consideration P&O was chosen for implementing into this project.

## Implementing MPPT

To implement the P&O MPPT method, software was chosen for controlling the duty cycle, and voltage and current sensing circuits were designed. The Spectrum Digital eZdsp F2812 microcontroller development board was chosen because we already possessed this piece of equipment. Using Matlab, Simulink, and Code Composer Studio, the MPPT algorithm can easily be programmed and debugged in a real time environment.

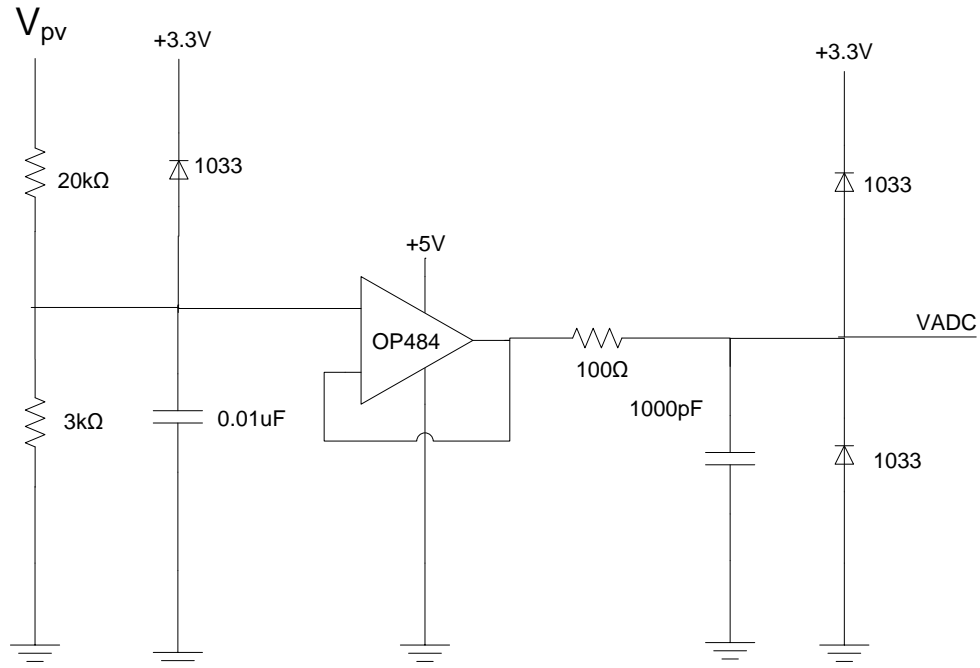
### eZdsp F2812 Features

The eZdsp F2812 has the following important features:

- Texas Instruments TMS320F2812 chip
- 32-bit DSP Core – 150 MIPS
- 18K + 64K RAM
- 128K Flash
- 30 MHz clock
- 12 PWM outputs
- 16 ADC 12 bit inputs
- 60 ns conversion time

### Voltage Sensing

The voltage sensing circuit designed and build in lab contains a voltage divider and two diodes at the output for overvoltage protection for the ADC input of the eZdsp. Any voltage above +3.3V can cause damage to the ADC. The circuit can be seen in Figure 16 below.

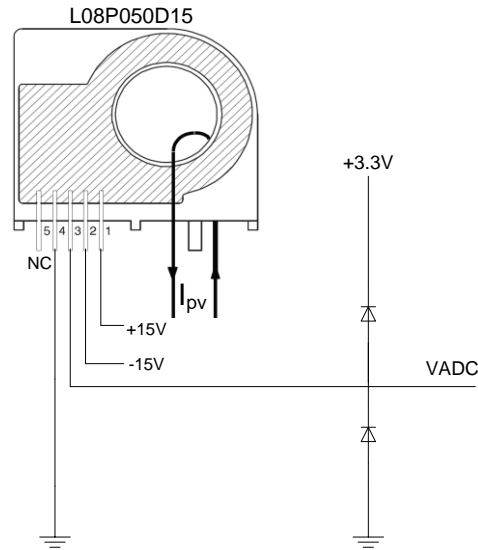


**Figure 16 - Voltage Sensing Circuit**

This voltage sensing circuit is designed for an input voltage,  $V_{pv}$  from 0 to 24V. This voltage range was set based on receiving power from one BP350J PV panel. The output voltage ranges from 0 to 3.3V. To correct for the divider multiply by  $24/3.3$  or 7.2727.

### Current Sensing

The current sensing circuit designed and built in lab contains a L08P050D15 current sensor and like the voltage sensing circuit, two diodes for overvoltage protection for the ADC of the eZdsp. The circuit can be seen in Figure 17 below.



**Figure 17 - Current Sensing Circuit**

The current sensor measures up to 50A of current giving an output voltage of 4V. The output voltage is estimated to be linear so to convert to Amps in software multiply by 50/4 or 12.5. In the future the wire carrying the current being measured could be wound multiple times through the sensor for increased accuracy since currents approaching 50A will not be seen from the PV panels.

### Simulink Modeling

Modeling for the MPPT algorithm started with basic testing of the ADC inputs. Voltages from the voltage sensing circuit were read into the eZdsp and were validated on the scope functionality in Code Composer Studio. Testing of the current sensing circuit and an ADC input was also done. The ADC is set to read in measurements every  $100\mu\text{s}$ . Once the sensing voltages are read a windowed average of 1s is taken to increase the accuracy of the measurements. Gain blocks are used to account for the voltage division of the voltage sensor and to convert from Volts to Amps for the current sensor. A product, unit delay, and sum blocks are used to find the change in power over the averaged 1sec measurement. Similarly two unit delay blocks and a sum block are used to find change in duty cycle over the 1sec time period. A product block is used to determine if the duty cycle will increase or decrease. A subprocess block was made in Simulink to increase or decrease the duty by 1%. If the input to the block is less than zero the output is -1, if the input to the block is greater than zero the output is +1, and if the input to the block is equal to zero the output is +0.01. In the future, the increase or decrease in duty cycle can be tuned using a gain to allow for how much the duty cycle should be changed based on the magnitude of the change in power. A soft start was also designed in the Simulink model to slowly increase the duty cycle from 0 to 30% before switching to MPPT duty cycle control. The soft start is needed to protect hardware of the boost converter from large inrush current. A simulation of the P&O MPPT and soft start model with an input of random numbers for the PV voltage and current is shown in Figure 18. The simulated scope output showing the combined soft start and MPPT duty cycle is shown in Figure 19.

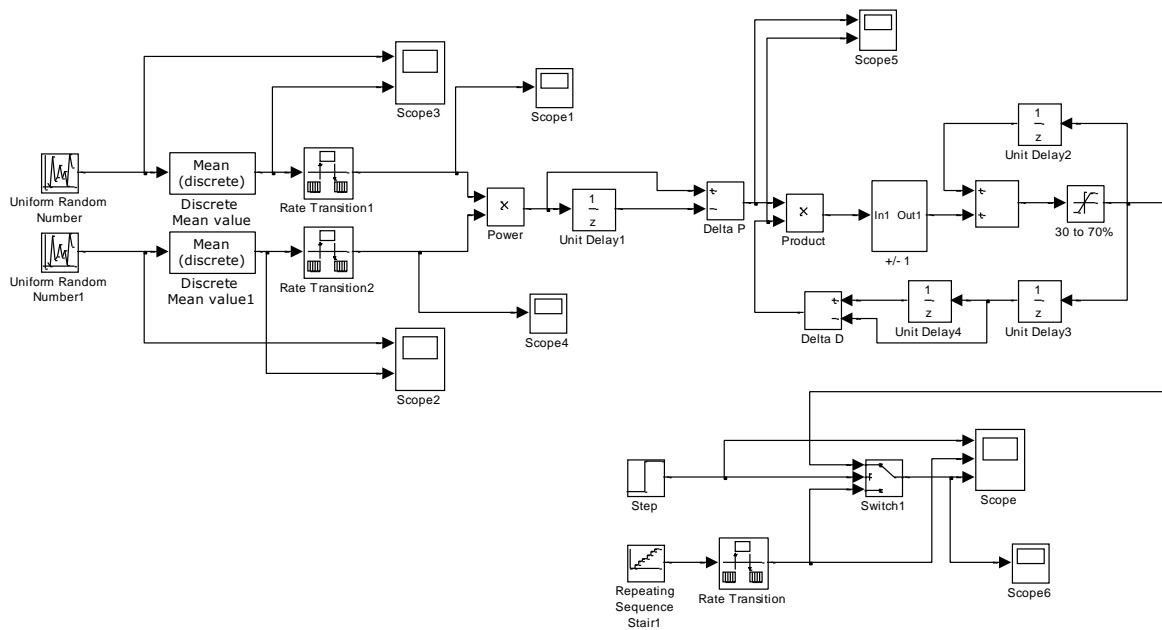


Figure 18 - P&O MatLab Simulink Simulation (mppt\_PO\_simulation.mdl)

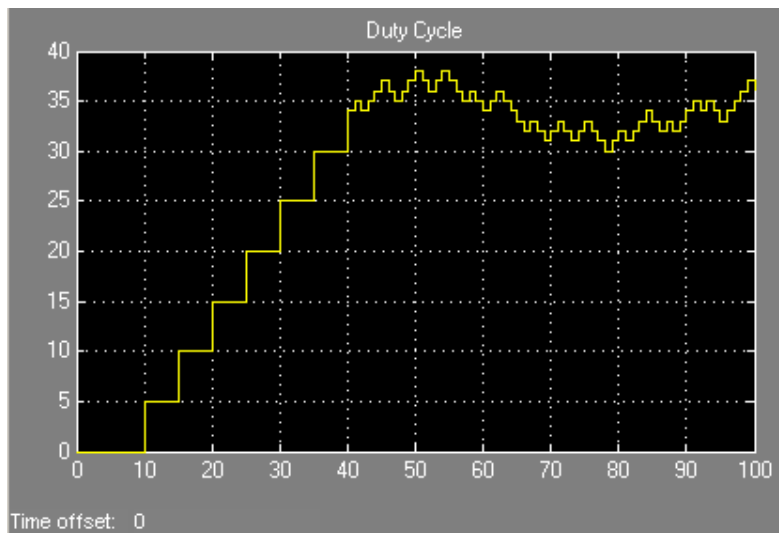
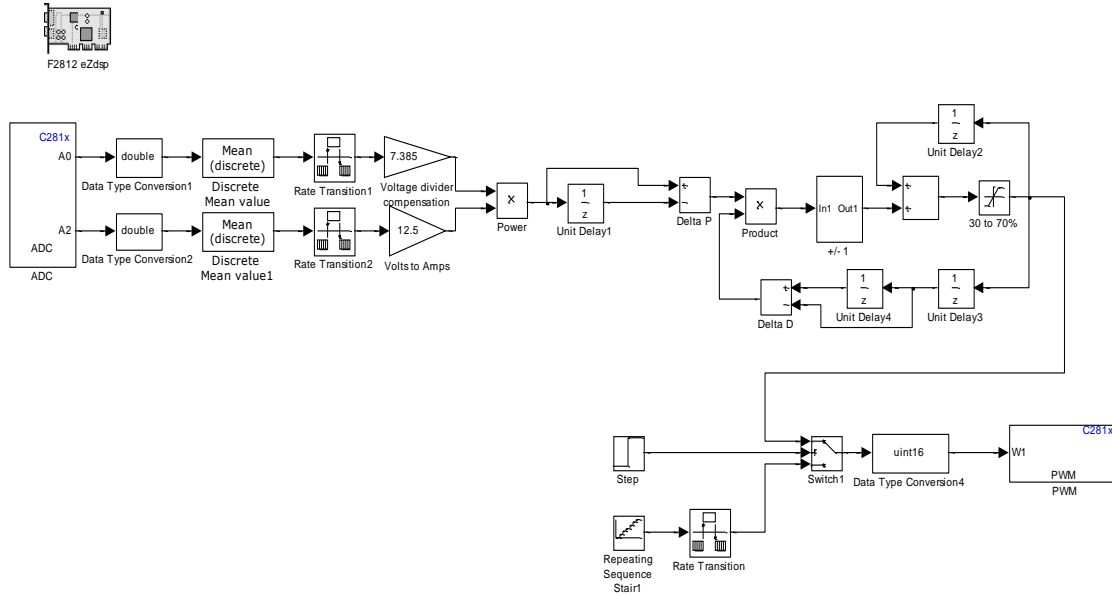


Figure 19 - P&O Simulation Duty Cycle Output with Soft Start

As shown in Figure 19, the soft start increases the duty cycle by 5% every 5 seconds until 40 seconds when the duty cycle is at 30%. The duty cycle is then controlled by the MPPT algorithm which is updated every second. Once the MPPT algorithm was tested in simulation the model in Figure 20 was made to program the DSP board.



**Figure 20 - P&O Simulink Model (mppt\_PO\_eZdsp.mdl) for eZdsp with Soft Start**

The F2812 eZdsp block in the top left configures the programming of the eZdsp in Figure 20. The ADC and PWM blocks are used to input and output to the eZdsp.

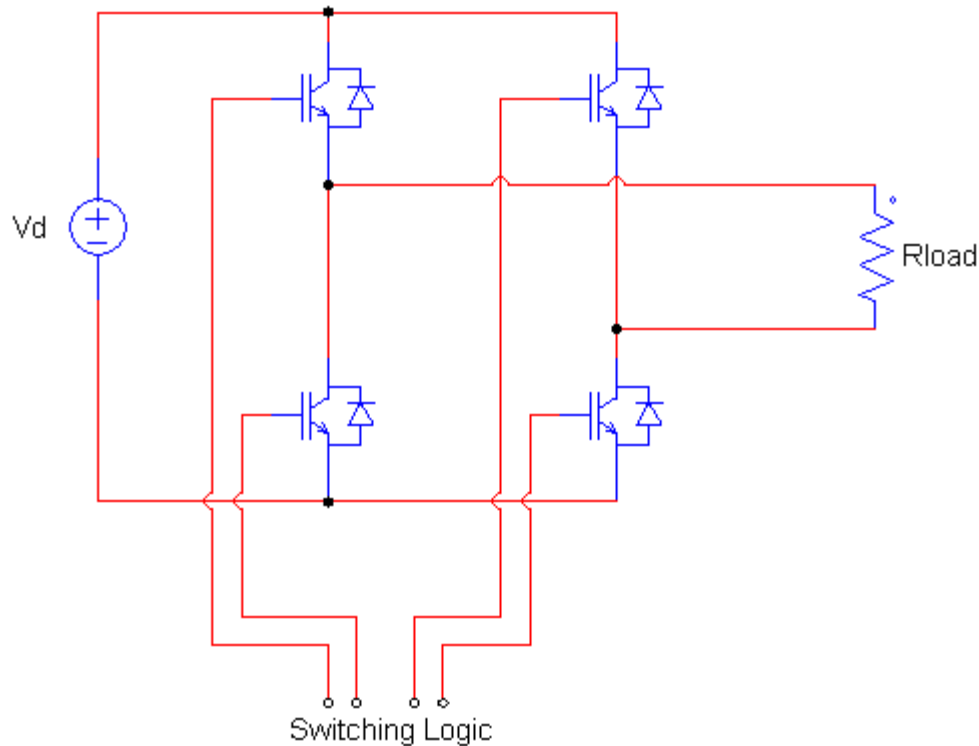
### MPPT and Soft Start Results

Using a DC power supply to provide both voltage and current to sense, the Simulink model was tested in lab. After programming the eZdsp, the duty cycle of the PWM output signal was observed with the positive duty cycle measurement on an oscilloscope. The soft start performed the same as simulation, and the MPPT changed as desired when the voltage of the DC power supply was increased and decreased. Future testing of the model needs to be done the boost converter when it is connected to the BP350J PV panel. Time and weather constraints prevented this testing from occurring this year.

## AC Subsystem

### AC Inverter General Overview and Theory

The AC inverter takes the DC output of the boost converter and turns it into AC power. The inverter uses a classical H-Bridge topology shown in Figure 21.

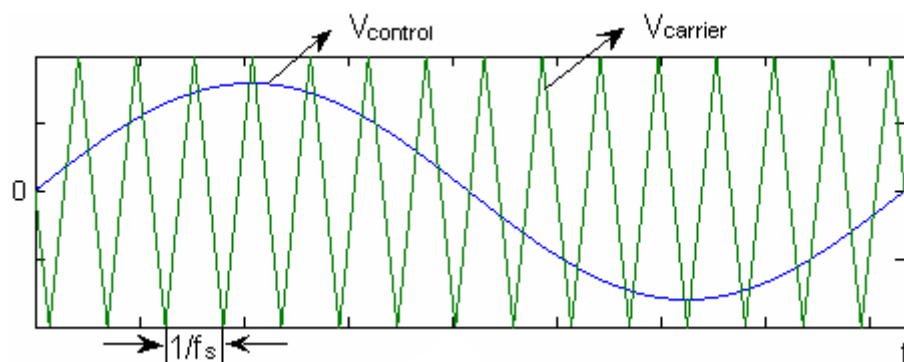


**Figure 21 – H-Bridge Topology**

The H-bridge is comprised of a voltage source ( $V_d$ ) and four power switches. A purely resistive load ( $R_{load}$ ) is also shown.

To produce AC power at the load a method known as Sinusoidal PWM (SPWM) is used. An excellent description of SPWM is given in the Zhou and Tian papers[4,5] located in the References section.

To generate an AC output with frequency  $f_1$ , a sinusoidal control signal  $V_{control}$  at frequency  $f_1$  is compared with a triangular waveform  $V_{carrier}$  as shown in Figure 22. When  $V_{control}$  is greater than  $V_{carrier}$  the PWM output is HIGH, otherwise the PWM output is LOW. The frequency of the carrier ( $f_{carrier}$ ) determines the switching frequency of the inverter ( $f_s$ ).



**Figure 22 – Bipolar SPWM Description ( $m_i = 0.8$ ,  $m_f = 15$ )**

An important ratio called the modulation index ( $m_i$ ) can be described as:

$$m_i = \frac{\hat{V}_{control}}{\hat{V}_{triangl}} \quad (9)$$

Also, the modulation ratio ( $m_f$ ) can be defined as:

$$m_f = \frac{f_{carrier}}{f_{control}} \quad (10)$$

An important feature of SPWM is that it allows for the control of the output frequency and the control of the output voltage amplitude. The output frequency and output amplitude are governed by the following equations:

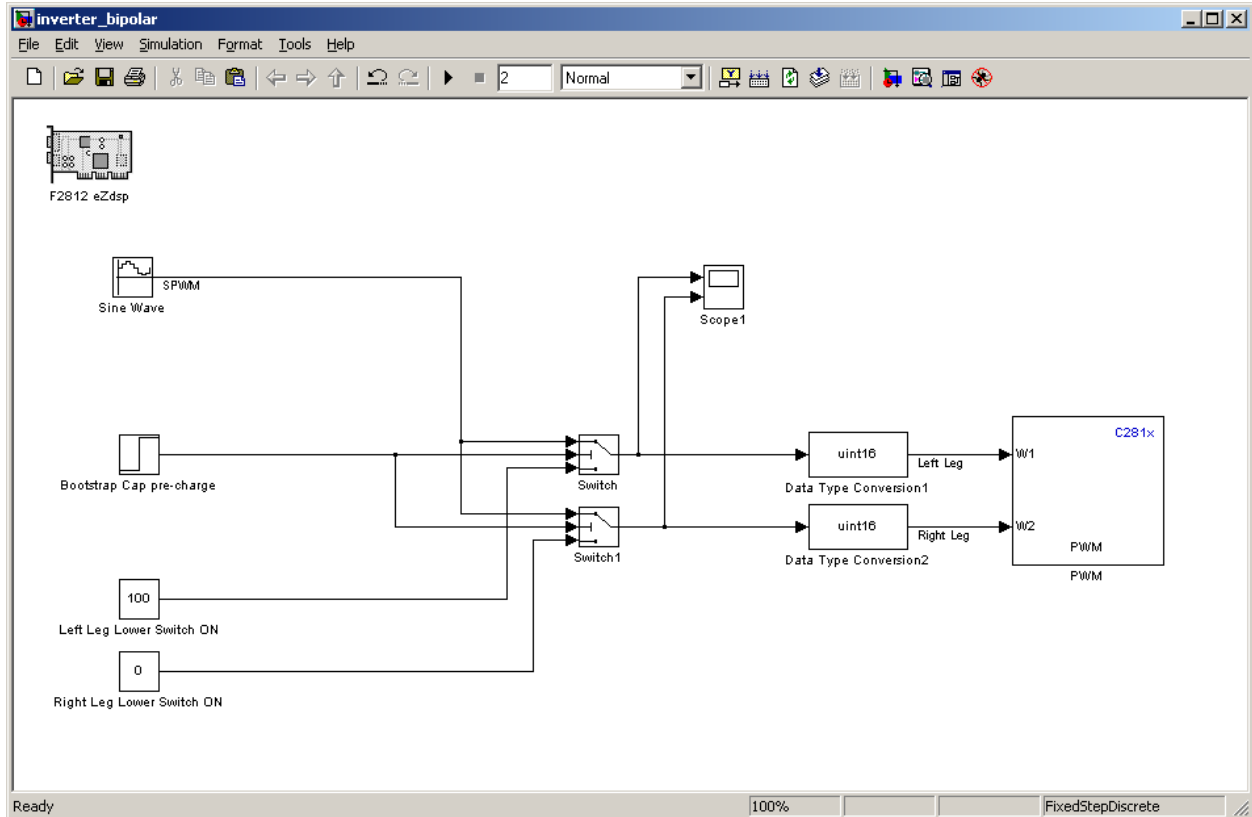
$$f_1 = f_{control} \quad (11)$$

$$(\hat{V}_{out})_1 = m_i * V_d \quad (12)$$

These equations (9)~(12) mean that the output voltage frequency are the same as the control signal frequency, and the peak value of the fundamental output component can be controlled by varying the modulation index  $m_i$ .

### Controlling the Inverter with the eZdsp

To create the SPWM signals a TMS320F2812 DSP based eZdsp prototyping board was used. A Simulink block diagram and the auto-code generation tool was used to create a program for the DSP board. Software called Code Composer Studio sends the program to the DSP board. Code Composer Studio also allows for real time debugging of the DSP chip. Dr. Dempsey had created several good tutorials on the basics of using Simulink and Code Composer Studio to program the DSP board. Software to create the Bipolar SPWM signals is included in a file called inverter\_bipolar.mdl shown in Figure 23.



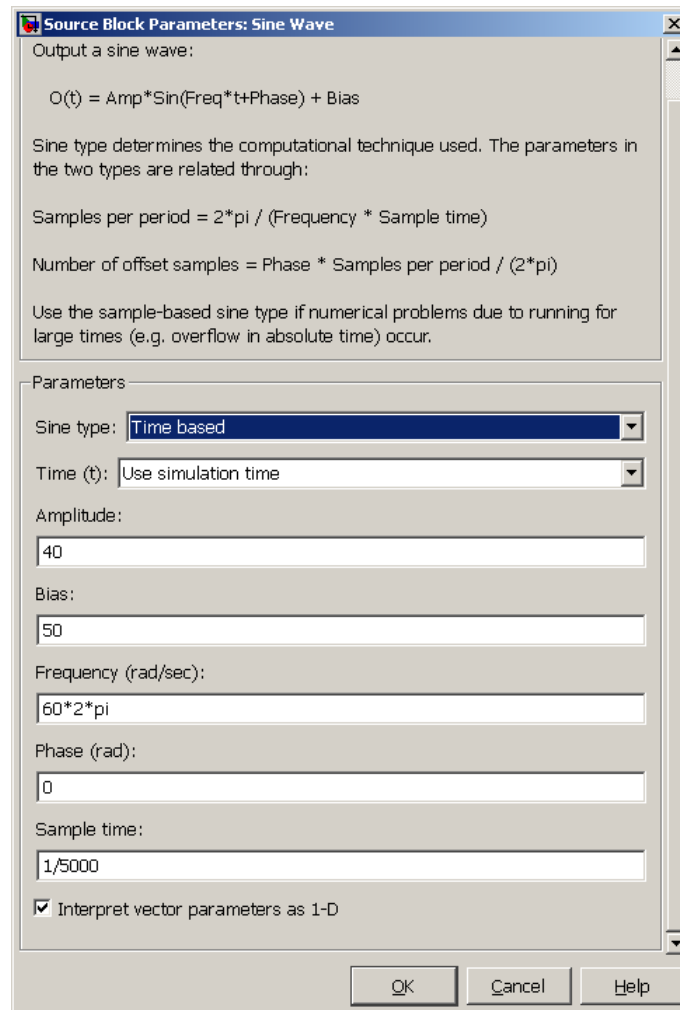
**Figure 23 – Screenshot of inverter\_bipolar.mdl**

A novel way to create the SPWM signals can be achieved by directly feeding a specialized sine wave into a PWM block. The PWM block accepts values between 0 and 100. The values correspond to the negative duty (in percent). For example, to create a 20% positive duty cycle, a value of 80 should be input to the PWM block.

In Figure 23 the PWM output pairs 1, 2 (W1) and 3, 4 (W2) are used. The input to W1 creates the output on PWM output 1. PWM output 2 is automatically created as the logical inverse of PWM output 1. Likewise W2 corresponds to PWM output 3, and PWM output 4 is automatically created as the inverse of PWM output 3.



The settings for the sine wave block will now be given.



**Figure 24 – Sine Wave Block Parameters**

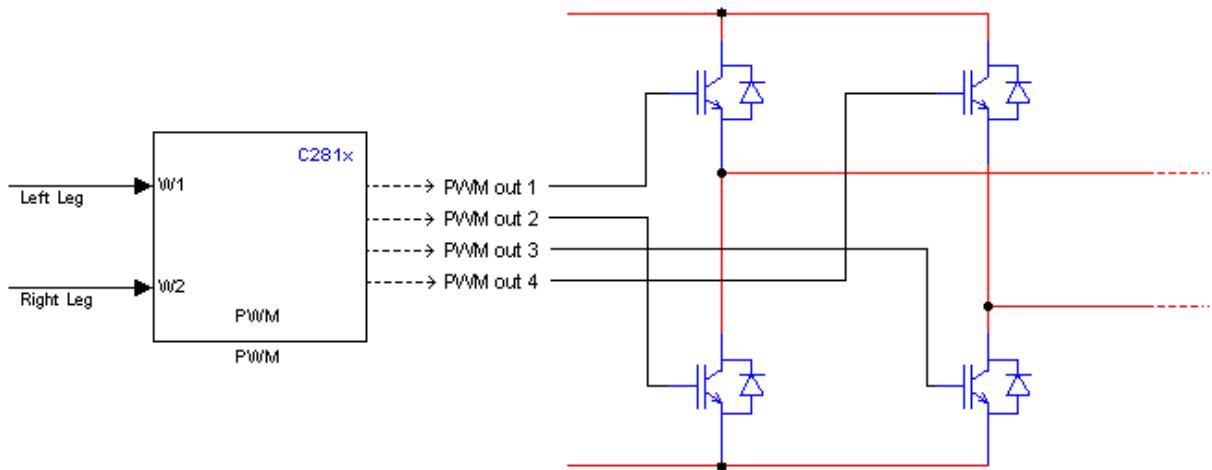
The modulation index is a function of the `Amplitude` and `Bias` parameters. `Bias` should always be set to 50. The equation for `Amplitude` is given as:

$$\text{Amplitude} = 100 / 2 * m_i = 50 * m_i \quad (13)$$

For example: for a modulation index of 0.7, set `Bias` to 50 and `Amplitude` to 35. An alternative way to set amplitude of the sine wave (hence setting the modulation index) would be set to `Amplitude` and `Bias` to 50 and use a gain block at the output of the sine block. In this case the gain would directly correspond to the modulation index.

The frequency of the reference sinusoid can be set by entering the desired frequency (in radians/sec) into the `Frequency` parameter. The `Sample time` field dictates the frequency of the carrier,  $f_{\text{carrier}}$ .

Figure 25 shows the connection diagram between the DSP PWM outputs and the individual IGBTs. Note that the gate drives have not been shown.



**Figure 25 – Connection between DSP and H-Bridge**

## Gate Drive

Gate drivers are necessary for proper operation of the IGBTs in the H-bridge. To properly switch on the IGBT the gate voltage must be 12 V higher than the source voltage. If the IGBTs are not turned fully ON (which can be done by supplying a gate voltage of 12 V or less) the devices will be damaged. To allow for a margin of error, the gate voltage is usually driven at 15 V higher than the source. The maximum gate to source voltage is 20 V.

Driving the gate is easy for the lower side IGBT because the source is grounded and generating 15 volts higher than ground voltage is easily done. Activating the high side switches is more difficult because when they are conducting the source voltage is going to be at  $V_d$ , and  $V_d+15$  is needed at the gates to completely turn on the IGBTs.

To achieve this, the IR2110 gate drive is used. This device can drive a high and low side IGBT. Two IR2110s are needed for the full H-bridge. In order to drive a high side IGBT the IR2110 uses an external bootstrap capacitor. The bootstrap capacitor is charged to 15 volts and when the high side device is to be turned on the IR2110 references the capacitor voltage to the source voltage of the high side IGBT. As the high side IGBT turns on the capacitor voltage keeps the gate of the IGBT 15 volts higher than the source, which is now at  $V_d$ .

It is important to initially charge the bootstrap capacitors before the inverter is operated. The bootstrap capacitors are only charged when the high side device is off and the low side device is on. It takes on the order of several milliseconds to charge the bootstrap capacitors.

It is also important not to leave the high side device ON indefinitely. The bootstrap capacitor will leak current over time and eventually the charge will not be sufficient to keep the high side device ON.

A germanium diode has also been placed at the low side output of the IR2110. The reason to do this is to ensure that the low side output voltage does not go negative, otherwise it could damage the gate drive.

## LC Filter

The goal of the LC filter is to average the output of the H-bridge to an AC signal. Figure 26 shows the LC filter used in this project.

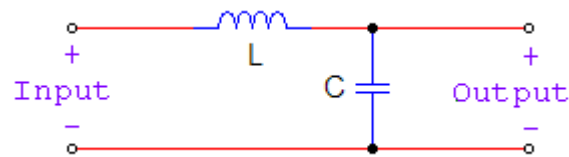


Figure 26 – LC Filter

The values of L and C should be chosen so that the cutoff frequency of the filter is less than the switching frequency ( $f_s$ ) and greater than the output frequency ( $f_1$ ). In our solar inverter the values of 1mH and 100uF were chosen for the inductor and capacitor respectively.

## Experimental Results

To test the small-scale operation of the solar inverter system a DC power supply was connected to the boost converter to mimic the DC output of a solar panel. The output waveform of the filter was then recorded with an oscilloscope, as shown in Figure 27. FFT analysis was performed on the signal to check the output frequency.

Operating parameters:

$$V_d = 10 \text{ V}$$

$$f_{\text{carrier}} = 60 \text{ Hz}$$

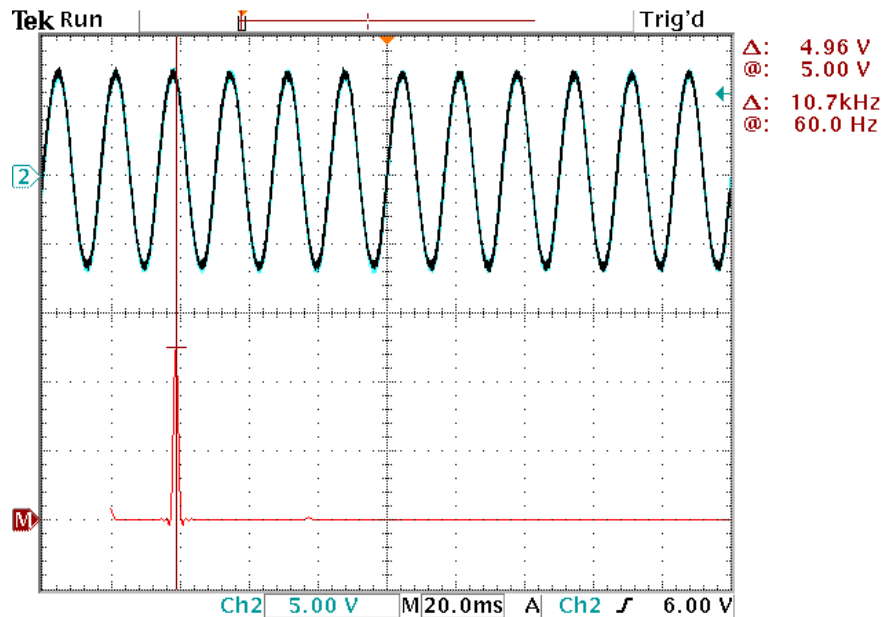
$$m_i = 0.8$$

$$m_f = 83$$

$$L_{\text{filter}} = 1\text{mH}$$

$$C_{\text{filter}} = 100\mu\text{F}$$

$$R_{\text{load}} = 500\Omega$$



**Figure 27 – Experimental Output**

Figure 27 shows a sinusoidal output of the LC filter in the upper half of the plot. The lower half of the plot is the frequency spectrum of the output. The output signal is  $15V_{pp}$ . The large DC component at the fundamental frequency, 60Hz in the FFT analysis is shown because the output is operating at 60Hz. The almost complete absence of spikes at other frequencies indicates there is little harmonic distortion at the output.

## Conclusion

Over the course of year we have learned a basic PV power converter topology and built a 100W power converter consisting of a boost converter followed by a single phase AC inverter. If this system was to be connected to the power grid electrical isolation, a Phase Locking Loop (PLL), and Power Factor Correction (PFC) must be considered[2,3]. Due to time constraints, we have not implemented the MPPT algorithm in our boost converter. However, our work would be a good starting point for future senior students who are interested in PV power converters.

## References

- [1] "PV Module Simulink Models." ECEN2060. University of Colorado Boulder.
- [2] Rockett, Angus, Robert W. Scott. "Evaluation of the Potential for Photovoltaic Power Generation in Illinois" 22 Nov. 2011. <[www.isws.illinois.edu/pubdoc/IEM/ISWSIEM2006-05.pdf](http://www.isws.illinois.edu/pubdoc/IEM/ISWSIEM2006-05.pdf)>.
- [3] Rozenblat, Lazar. "A Grid Tie Inverter for Solar Systems." Grid Tie Inverter Schematic and Principles of Operation. 6 Oct. 2011. <<http://solar.smps.us/grid-tie-inverter-schematic.html>>.
- [4] Tafticht, T., K. Agbossou, M. Doumbia, and A. Cheriti. "An Improved Maximum Power Point Tracking Method for Photovoltaic Systems." *Renewable Energy* 33.7 (2008): 1508-516.
- [4] Tian, Yi. ANALYSIS, SIMULATION AND DSP BASED IMPLEMENTATION OF ASYMMETRIC THREE-LEVEL SINGLE-PHASE INVERTER IN SOLAR POWER SYSTEM. Thesis. Florida State University, 2007.
- [5] Zhou, Lining. EVALUATION AND DSP BASED IMPLEMENTATION OF PWM APPROACHES FOR SINGLE-PHASE DC-AC CONVERTERS. Thesis. Florida State University, 2005.