

# Programmable High-Voltage DC to AC Inverter

Design and Performance Measurements

Ferdinand Keil  
FB IES  
TU Darmstadt

Release Date - May 15, 2013

# Contents

<b>I</b>	<b>System Overview</b>	<b>2</b>
<b>II</b>	<b>Boost Converter Circuit</b>	<b>2</b>
1	MAX1771 Boost Converter Controller IC	3
2	Design Process	3
3	Component Selection	4
<b>III</b>	<b>User Interface</b>	<b>4</b>
<b>IV</b>	<b>H-Bridge Board Version 10</b>	<b>4</b>
1	Micro-controller Circuit	5
2	H-Bridge and Voltage Regulator Circuit	5
3	Component Selection	5
4	Layout	6
<b>V</b>	<b>Software</b>	<b>6</b>
1	H-Bridge Board	7
2	User-Interface Board	7
<b>VI</b>	<b>Measurements</b>	<b>8</b>
<b>VII</b>	<b>Conclusion and Outlook</b>	<b>8</b>

## Part I

# System Overview

The design and construction of a programmable, modular, multi-channel high-voltage DC to AC inverter will be explained in the following report. These requirements for the system were established:

- a peak to peak voltage of 400V,
- an output frequency of at least 20kHz,
- a modular and extensible system,
- at least four output channels,
- and a way to easily change the output frequency and amplitude via either a PC or the included user interface.

The chosen architecture consists of three types of modules, where only one will be needed multiple times. The boost converter module generates the necessary high-voltage and can be replaced with an off-the-shelf lab power supply. The user interface module connects to the H-bridge modules via an I<sup>2</sup>C bus and to the PC via USB. It includes a display and buttons to change the parameters without a computer.

The H-bridge module is the core of the system. Multiple output channels can be implemented in a single module. The input high-voltage supply is regulated to the desired amplitude and then switched with an H-bridge. The generated square wave is accessible through a connector. A micro-controller generates the control signals for the H-bridge and communicates with the user interface module. The desired output frequency is generated by this micro-controller.

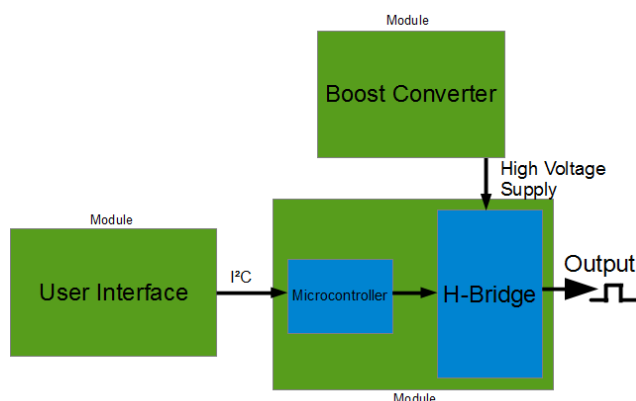


Figure 1: Block diagram

## Part II

# Boost Converter Circuit

## 1 MAX1771 Boost Converter Controller IC

The MAX1771 is a fully integrated Boost Converter Controller IC with an integrated gate driver. A maximum switching frequency of 300kHz enables the use of relatively small inductors even for high output voltages. The circuit can be configured to work in a bootstrapped or non-bootstrapped mode. An input range of 2V – 16.5V enables the use of high-power MOSFETs in non-bootstrapped mode (as  $V_{GS(th)} \geq 10V$  for most devices).

The output voltage can be adjusted via two resistors or is fixed at 12V. The MAX1771 is available in the SO8 and DIP8 package and therefore well suited for prototyping.

## 2 Design Process

The following specifications were chosen:  $V_F = 1V$ ,  $V_{out} = 220V$ ,  $V_{in} = 12V$ ,  $V_{sat} \approx 1V$  and  $I_{out(max)} = 150mA$ .

Using the following equations, the values for the inductor  $L$ , the maximum current flowing through the inductor, FET and diode  $I_{switch(peak)}$  were calculated.

$$t_{on}/t_{off} = \frac{V_{out} + V_F - V_{in}}{V_{in} - V_{sat}} = 19$$

$$I_{switch(peak)} = 2I_{out(max)}(t_{on}/t_{off} + 1) = 6A$$

$$L \geq \frac{V_{in} \cdot 2\mu s}{I_{LIM}} = 4\mu H$$

In the next step a matching MOSFET was chosen. The IRF740 was selected for its low price and easy availability. It has a  $V_{DS(max)} = 400V$  and is therefore suited for this application. The typical total gate charge for this device is given as  $Q_{g,TOT} = 41nC$ . This is small enough to be driven from the EXT pin of the MAX1771 but will require a larger bypass cap:

$$C_{bypass} \geq \frac{Q_g}{200mV} = 205nF$$

The integrated current-limit feature of the MAX1771 requires a shunt resistor between the MOSFET drain and ground. The required value was calculated as follows:

$$R_{sense} = \frac{85mV}{I_{LIM}} = \frac{85mV}{6A} = 14m\Omega$$

The maximum dissipated power in the sense resistor has to be calculated to select an appropriate package:

$$P_{Rsense} = R_{sense}I_{LIM}^2 = 0.504W$$

### 3 Component Selection

The components used have to be specified with the high output voltage in mind. The necessary voltage rating for each component on the output side of the circuit will be explained following the schematic from left to right, with the part designator given in brackets.

- The IRF740 (*Q1*) is rated at  $V_{DS,max} = 400\text{V}$  which gives more than enough headroom.
- The MUR440 rectifier diode (*D1*) is also rated at a working repetitive peak reverse voltage of 400V.
- Through-hole resistors (*R2*, *R11*, *R4*) and trimmer potentiometers (*R12*) are usually rated for voltages of at least 200V, so generic components can be used.
- To filter the ripple caused by the switching regulator high-quality capacitors are needed (*C10* ... *C14*). Their voltage rating should exceed 250V. Low-ESR types are preferable.

The LED at the output has two purposes: it indicates the presence of a high voltage and drains the output capacitors, thus increasing the safety of the device.

Components on the input side of the circuit are either less critical or were already specified.

## Part III

# User Interface

The user interface is the least complex part of the system. It is based on an Arduino UNO, which is an inexpensive and widely used micro-controller development platform. It comes with USB and I<sup>2</sup>C support built-in and is supported by a vast selection of software libraries and example code. It was chosen to shorten the development-cycle of the user interface firmware.

The Arduino system can be extended by add-on PCBs called shields. For this system a custom shield with the necessary display, buttons and connectors was developed. The choice of parts was mainly determined by their availability. So a graphical display from the DOG series by Electronic Assembly and an ALPS rotary encoder were chosen. Two generic switches, an LED, several connectors and some support circuitry completed the shield.

## Part IV

# H-Bridge Board Version 10

The H-bridge board is separated in two parts: the micro-controller circuit and the high-voltage switches and regulation. These parts are not isolated from each other as they share the same ground. Both circuits will be explained in detail in the following.

## 1 Micro-controller Circuit

The micro-controller circuit is based on the ATtiny861A from Atmel. This controller comes with I<sup>2</sup>C slave support and extensive timer features. The timers are used to generate the signals controlling the H-bridge. Two major features of the ATtinyX61A series make it especially suited for this application: the prescaler for Timer/Counter1 has 16 settings and the module integrates an independent dead time generator for each PWM channel. Using these features, the frequency linearity of the generated control signals is better than 1%.

The power for the ATtiny861A is provided by an MIC5205-5.0, which is a linear regulator with fixed 5.0V output in a tiny package. Furthermore a DIL switch, two status LEDs and the communication connector surround the micro-controller. The DIL switch is used to select the I<sup>2</sup>C slave address the controller listens to. This allows several otherwise identical modules to be configured independently.

To synchronize several modules, a master clock can be distributed via the communication connector. It is buffered locally by a single inverter gate. Together with the interrupt line present on the connector a programmable phase-shift between multiple PCBs can be implemented.

The signals controlling the H-bridge are also brought out on the connector, however this feature is not used anymore.

## 2 H-Bridge and Voltage Regulator Circuit

This part of the circuit does the actual high-voltage switching. To be able to generate signals of different amplitudes, the incoming high-voltage supply of about 210V is regulated before entering the switching transistors. Each of the two present output channels comes with a linear regulator able to output voltages in the range of 13.2 – 200V. For ease of implementation the LR8 high-voltage linear regulator from Supertex was chosen. It can withstand input voltages of up to 450V and output voltages in the range of 13.2 – 440V. It also comes with integrated current and junction temperature limiting, which prevents circuit failure under over-load conditions. As this regulator uses a resistor network for feedback, the output voltage has to be dialed in via a ten-turn potentiometer.

## 3 Component Selection

Only low voltages and currents are present on the micro-controller side of the circuit, therefore generic components can be used. This is not true for the H-bridge and voltage regulator circuit: because of high voltages special care has to be taken in specifying the components.

To begin with, the specifications of critical parts in one half-bridge will be explained. The part designator is again given in brackets.

- The gate resistors (*R10*, *R11*) have to be rated for the maximum current flowing when switching occurs. They have to be rated for 330mW.

- The switching transistors (*FET7*, *FET8*) were chosen for their fast switching time. Of course they have to withstand a drain-source voltage of at least 250V.
- As the IR2110 uses a bootstrap circuit to switch the upper n-FET the supporting parts (*D4*, *C23*) had to be specified according to the data-sheet. The diode has to have a peak reverse of at least  $V_{HV} + V_{DD} = 200V + 12V = 212V$ . The ES2G is specified for  $V_{RRM} = 400V$  and should therefore work without a problem.

## 4 Layout



Figure 2: PCB layout - top

Critical parts of the PCB layout will be explained in the following section.

The circuit is divided in two parts: the low-voltage part in the lower half and the high-voltage part above it. Both share the same ground, but are connected by a small trace only. This is done to prevent high-frequency switching noise from propagating across the whole circuit. The high-voltage circuit part is not flood-filled with a copper pour to direct the ground currents in certain paths.

The high-voltage switching circuitry is grouped together as close as possible to minimize track inductance and shorten current loops.

All surface-mount components are placed on the bottom side of the PCB (with the exception of the LEDs and the reset switch). This prevents users from accidentally touching parts carrying high voltage, as leaded parts can easily be isolated, e.g. using shrink tubing.

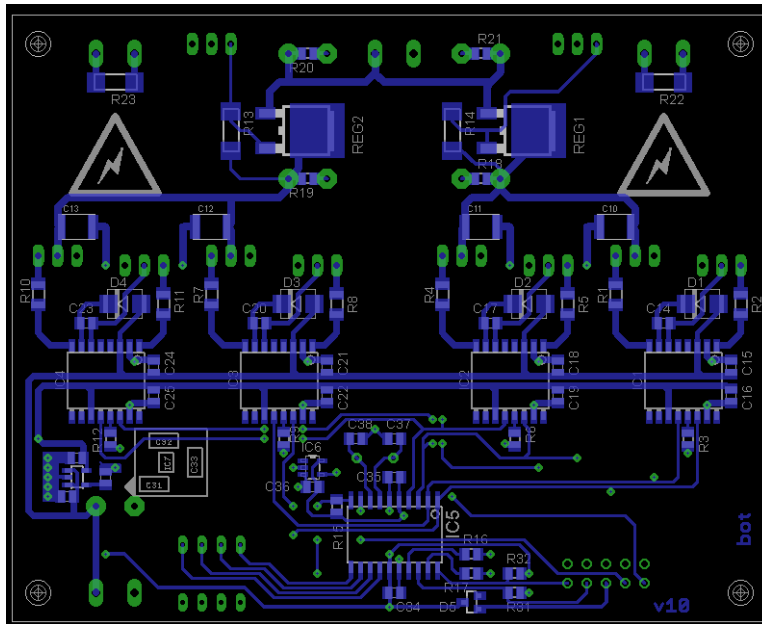


Figure 3: PCB layout - bottom

## Part V

# Software

The firmware for the micro-controllers on the H-bridge board and user-interface board was developed separately by two students. The controllers communicate over the I<sup>2</sup>C bus, where the user-interface board acts as the master and each H-bridge board as a slave node.

## 1 H-Bridge Board

The micro-controller on the H-bridge board is mainly concerned with generating the requested waveforms. Therefore the advanced features of the integrated hardware timer are used. To support communication via I<sup>2</sup>C the integrated two-wire hardware module is used.

The firmware for this board was written by Ferdinand Keil.

## 2 User-Interface Board

The user-interface board is based on the Arduino development platform. This eases the re-usability of available code and accelerates the development cycle. It also comes with USB support, which is used by the firmware to connect to LabView on a PC. The board can get its commands either via USB from a host PC or directly via the on-board display and encoder. The settings are forwarded over I<sup>2</sup>C to the slave nodes.



The firmware for this board was written by Manuel Fuentes.

## Part VI

# Measurements

As the measurements in Figure 4 show, the options offered by the system make it possible to generate complex waveform patterns. To generate these signals the following parameters were varied:

- amplitude,
- frequency
- and phase.

At low frequencies ( $\leq 10\text{kHz}$ ) the output signals reach the full amplitude of  $400V_{peak-peak}$ . However, at higher frequencies the amplitude starts to drop due to thermal overload of the used linear regulator. Apart from this problem all major functions were verified and are functional.

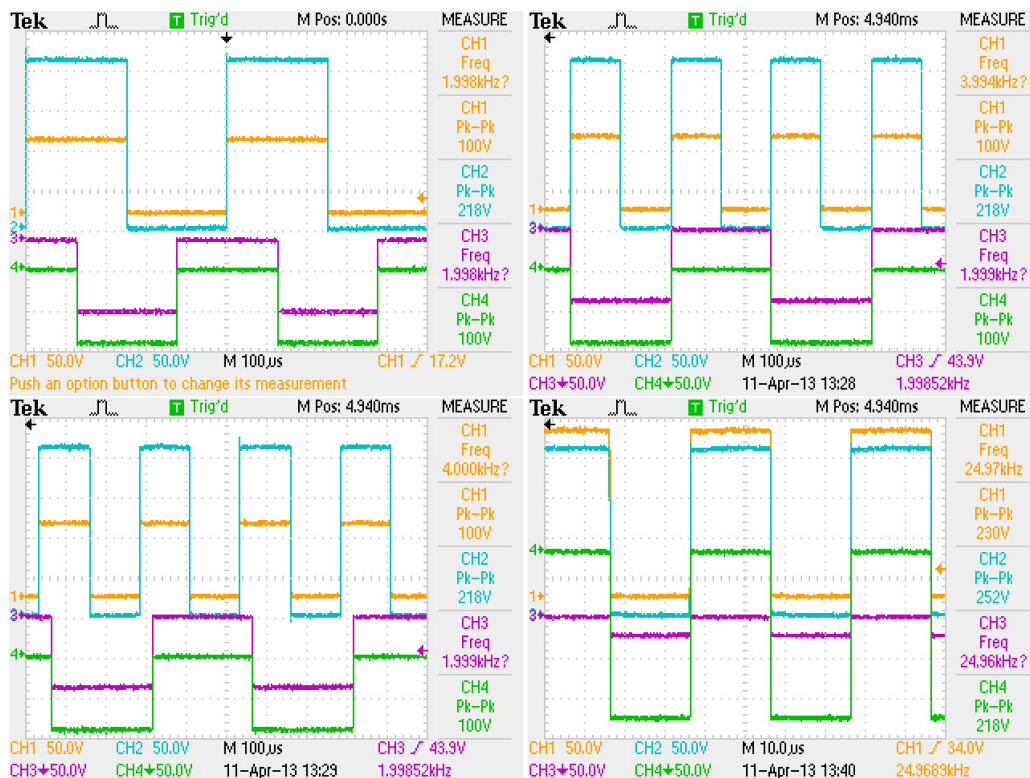


Figure 4: Measurements - from top left to bottom right: basic signals; signals of different frequencies; arbitrary phase shift between channels; high frequency signals.

## Part VII

# Conclusion and Outlook

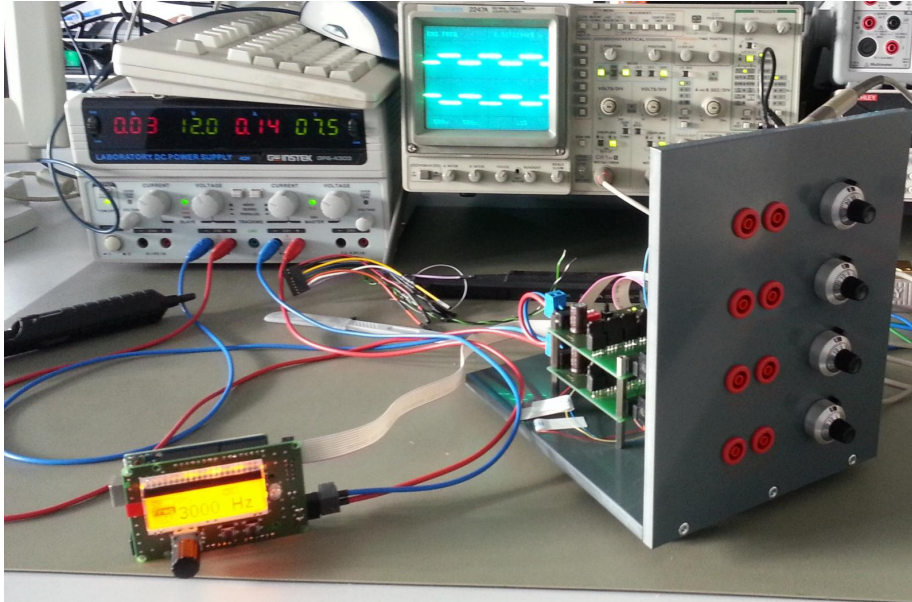


Figure 5: The completed circuit

It was shown with the first two board revisions, that the chosen system design is up to the given task. All things considered, the required specifications could be satisfied.

But, as mentioned in Measurements, some problems with the circuit still remain:

- The output amplitude drops at high switching frequencies. This problem is somewhat difficult to solve, as the current used by the H-bridge can not be reduced drastically because of switching losses.
- Communication between the user interface board and the H-bridge boards is unstable at high switching frequencies. This can be solved with error correction in software and better PCB layout.
- The overall efficiency of the circuit could be improved by using separate step-up converters for each channel.
- To be able to change all parameters via the PC interface the output amplitude needs to be programmable from the micro-controller.

Implementing and verifying these changes will take at least one additional board revision.