

# Digital to Analog Conversion Using Pulse Width Modulation

Samer El-Haj-Mahmoud  
Electronics Engineering Technology Program  
Texas A&M University

## Instructor's Portion

---

### Summary

The purpose of this lab is to provide the students with hand on experience in using the analog outputs of a National Instruments data acquisition card. The lab assumes that the students have prior exposure to LabVIEW, and that they know the basics of writing and debugging LabVIEW VIs (Virtual Instrument). The experiment will teach them how to apply their LabVIEW knowledge to generate an analog signal using Pulse Width Modulation (PWM) to control the analog output of the DAQ card. The students should have studied in class Digital to Analog conversion and PWM theory before coming to the lab.

The objective of this experiment is for the students to write the needed LabVIEW VI to generate a periodic pulse train using PWM. The instructions provided in the student's section describe a step-by-step approach to writing the basic structure of the VI. The rest of the implementation is left to the students. The objective is for them to learn how to write practical LabVIEW code to accomplish the D/A conversion. The VI provided is a sample implementation, and is intended for the instructor's reference use and *not* for the students. The diagram can be password protected in case the students want to look at the front panel and see how the VI they are writing should behave.

### Uses

This experiment applies to general instrumentation and electronics systems interfacing courses in electrical engineering or engineering technology programs. The experiment can also be useful in basic LabVIEW courses, teaching the concepts of Analog Output through data acquisition cards, in addition to other LabVIEW techniques such as the use of the Timers, Logical Select and Waveform Charts.

## Equipment List

- PC running MS Windows  
(visit [http://www.ni.com/labview/lv\\_sysreq.htm](http://www.ni.com/labview/lv_sysreq.htm) for requirements specific to your operating system)
- LabVIEW Full Development System
- PCI 1200 DAQ from National Instruments (part number 777386-01)
- CB-50LP I/O Connector Block from National Instruments (part number 777101-01)
- NBI Ribbon Cable from National Instruments (part number 180624-10)
- Tektronix TAS475 Four Channel Oscilloscope, or any general lab scope with 2 or more channels.
- Any laboratory digital Multimeter for measuring DC voltages.

## Setup

Follow the steps listed below to prepare the workstation for this experiment. The instructions assume you are using the equipment list shown previously.

**Note:** Most of the manuals that are referred to ship with National Instruments hardware and software. The lab manuals can be obtained from <http://www.ni.com/manuals>. If problems are encountered during setup, contact technical support at <http://www.ni.com/support>.

### Before the Day of the Lab

1. Install LabVIEW (see the *LabVIEW Release Notes* for your version of LabVIEW).
2. Install your PCI-1200 board (see the *PCI1200 User Manual*, or online at: <http://sine.ni.com/apps/we/nioc.vp?cid=5497&lang=US>).
3. Connect the ribbon cable to the PCI1200 card and to the I/O connector block.

### On the Day of the Lab

1. Power up the computer.
2. Start LabVIEW.
3. Power up the oscilloscope.

## References

- NI's web site: <http://www.ni.com>
- Schweber, William. Electronic Communication Systems. Upper Saddle River: Prentice-Hall, 1999; p 292-3.
- Burns, Mark and Roberts. An Introduction to Mixed-Signal IC Test and Measurement. Texas Instruments, 2000; p.174.

# Student's Portion

---

## Introduction

In this experiment, you will write a LabVIEW VI to generate a periodic pulse train on the analog output pins of the PCI 1200 Data Acquisition Card (DAQ). This lab requires background in digital to analog conversion and PWM theory, including the concepts of duty cycle and Nyquist theorem. These concepts are summarized in the Theory section of this experiment.

## Objective

- To learn how to use LabVIEW for analog output through a DAQ card.
- To learn about Digital to Analog conversion and how a periodic pulse train can be generated using PWM and analog output of a DAQ card.

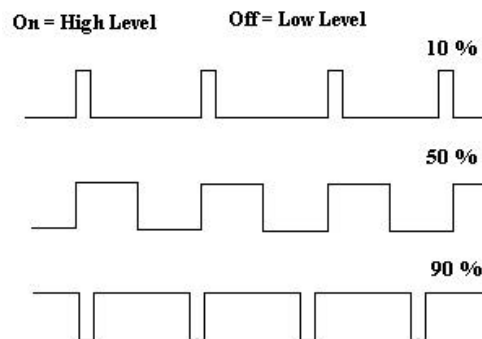
## Theory

The process of converting a number held in a digital register to an analog voltage or current is accomplished with a digital-to-analog converter (DAC). Analog signals can be generated from digital microprocessors with an accuracy and resolution determined and limited by the available analog to digital conversion process. Depending on the application requirements, differing degrees of resolution may be used for the analog to digital conversion.

The minimum voltage that can be converted using the digital to analog conversion process depends on the bit width of the digital to analog converter and the supply voltages. For example, if we are using an 8 bits digital to analog convertor supplied at +/- 5 volts, the converted analog voltage is in steps of  $10/2^8 = 10/256 = 0.04$  V

Pulse width modulation (PWM) is a powerful technique for controlling analog circuits with a microprocessor's digital outputs. It is a way of digitally encoding analog signal levels. Through the use of high-resolution counters, the duty cycle of a pulse train is modulated to encode a specific analog signal level. The PWM signal is still "digital" because, at any given instant of time, the full DC supply is either fully on or fully off. The voltage or current source is supplied to the analog load by means of a repeating series of on and off pulses. The on-time is the time during which the DC supply is applied to the load, and the off-time is the period during which that supply is switched off. Given a sufficient bandwidth, any analog value can be encoded with PWM.

Figure 2 shows three different PWM signals. The first one is a PWM output at a 10% duty cycle. That is, the signal is on for 10% of the period and off the other 90%. The second and third signals show PWM outputs at 50% and 90% duty cycles, respectively. A duty cycle is defined as the time the signal is high divided by the time it is low (in each period). These three PWM outputs encode three different analog signal values, at 10%, 50%, and 90% of the full strength. If, for example, the supply is 10V and the duty cycle is 10%, a 1 V analog signal results, after filtering. A low pass filter is necessary to remove the “high” frequency component of the signal, leaving the DC average value.



**Figure 2. Pulse Width Modulation (PWM) with different duty cycles**

The following equations define the duty cycle and the period:

$$\text{Period} = 1 / \text{Frequency}$$

$$\% \text{ Duty Cycle} = (\text{High Time} / \text{Period}) * 100$$

## Pre-Lab Preparation

Read this experiment before coming to the lab. You should understand the idea of digital to analog conversion and PWM theory, including the concepts of duty cycle and Nyquist theorem.

Bring the following to lab with you:

- This experiment.
- Your lab notebook and pencil.
- 2 virus-free formatted 3.5-inch floppy disks (always make a backup copy of your code on the second disk).

Answer the following questions in the datasheet provided at the end of this experiment *before* coming to the lab. Remember to include these answers in your lab report too.

1. How many Analog Outputs can you take from PCI-1200 board and what are the pin numbers on the output board? Also give the DAC (Digital-to-Analog Converter) chip number?

2. Why are Digital-to-Analog converters used to output analog voltage and enumerate the errors associated with the output analog voltage? What is the smallest voltage, greater than zero, which can be output through the analog outputs?
3. Which VI will you use for the Analog Output voltage (Functions Menu → Data Acquisition → Analog Write)? And Why?
4. How will you use the mod-N sub-vi (divide by N) to get a required time period?

## Workstation Details

Your workstation should have the following items:

- Computer with National Instruments LabVIEW software
- National Instruments DAQ board (inside the computer)
- National Instruments DAQ board terminal block
- Tektronix Oscilloscope.

## Lab Procedure

1. Start a new VI in LabVIEW. You should then add the following controls and indicators to the front panel:
  - a. Start/Stop Switch: This is the main On/Off switch for the VI.
  - b. Output Signal Duty Cycle Digital Control: A knob to control the duty cycle of the generated output square signal. You should be able to control the duty cycle between 0% and 100%.
  - c. Output Signal Amplitude Digital Control: By using this function the user can specify the amplitude of the output square signal. The range should be between 0 and 10 volts (assuming that you have configured the analog output channel to work in that range).
  - d. Output Signal Frequency Digital Control: This knob will set the frequency of the generated square wave between 0 Hz and 1000 Hz.
  - e. Waveform Chart of the Output Signal: This chart will display the generated signal. It should match the signal obtained on the oscilloscope when measured at the analog out channel.
2. After adding those controls, start wiring the VI diagrams with the controls to implement the functionality of your program. You might find the following VIs useful in your code:

- a. A main loop, best implemented as a For Loop structure, which will be controlled by the Start/Stop switch.
  - b. Two Tick Counts VIs and a Subtract VI. These will be used to generate the needed timing to generate the signal. Each Tick Count returns each time it is accessed the number of milliseconds since a fixed reference point of time. We will use two tick counts in this design, one placed outside the loop and the other inside it. The Tick Count outside the loop will be accessed only once, holding a constant value of milliseconds when the loop was entered. The second Tick Count, however, will be accessed every time the loop is repeated, giving an increasing value in milliseconds each time. The Subtract VI will subtract the output of the *outside* Tick Count from the *inside* one. This gives the normalized time, or the number of milliseconds since the loop has been entered.
  - c. To obtain the “period”, invert the Frequency (in Hz) using a  $1/x$  VI, then divide that by 1000. The output will be the period in milliseconds.
  - d. To obtain the “Hi Time”, divide the duty cycle by 100 (in %) and multiply the result with the obtained period in milliseconds. The result of this multiplication is the Hi Time in milliseconds.
  - e. You will also need a Mod VI (Quotient and Remainder) to calculate when one period has elapsed. Do this by dividing the time progressed by the period, yielding a remainder with a range from 0 to period size. When the remainder is zero, you are at the start of a new period.
  - f. You will also need a comparison VI (less than or equal) which compares the remainder obtained in “e” with the Hi Time. The output of this comparator is a logical value used to switch a Select VI between Amplitude and 0.
  - g. The output of the Select VI feeds both a waveform chart and goes to the Analog output channel number 0. Use an Analog Out (one point) VI from Data Acquisitions → Analog Write. The device number is 1 because there is only one DAQ in the PC.
3. These are the basic VIs that you will need to have a working program. Wire the entire diagram correctly and debug any errors you have. You may want to add more features, or implement the logic in a different way. If you have any questions about writing the code, ask your lab instructor.
  4. After finishing your code save it to the floppy disks. Next, you need to connect the DAQ Analog Out Channel 0 (ACH0) and Analog Ground (AGND) to the scope.

5. Test your code by generating square waves of different duty cycles and frequencies and amplitude. Compare the result you get on the waveform chart with the one displayed on the scope. Measure the output signal on a digital multimeter to get its RMS value. Comment on your reading. Use a low pass filter on the output and repeat your measurement. Comment on the new readings. What is the theoretical limit on the frequency you can achieve? What is the actual maximum frequency you can successfully achieve? What happens at higher frequencies? Note down any observations you have.
6. On your own, write a new VI to generate a sine wave output varying its amplitude between  $-5$  and  $+5$  volts and the frequency between 0 and 1000 Hz. Use the ready made VIs in LabVIEW (look at example VIs if you want). Study the effect of quantization by displaying the sine wave on the oscilloscope and a waveform chart and comparing between the two plots. What is the maximum frequency you can successfully achieve?

## Lab Report

Your report must be typed and of professional quality. You should include the following in your report:

1. All schematics, which must be computer-generated. Components must be labeled with circuit references and values. The schematics must be fully compliant with standard engineering practices for circuit depiction.
2. Front panel and block diagram of all VI's and Sub VI's that you wrote or used.
3. A Theory section describing the theory behind the concepts used in this lab.
4. An implementation section describing the details of your VI code and the logic flow.
5. The tabulated test results and observations.
6. The answers to the Pre-lab questions in the provided datasheet.
7. Selected sections of the datasheet of the PCI 1200.

## Data Sheet

1. Answer : .....  
.....

2. Answer : .....  
.....

3. Answer : .....  
.....

4. Answer : .....  
.....