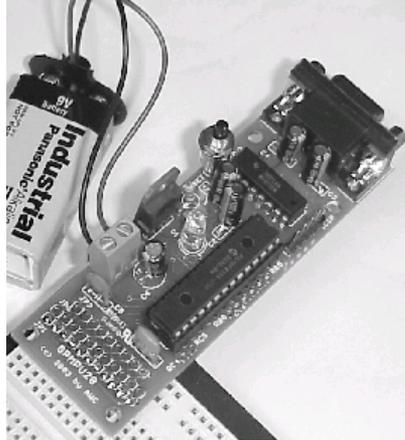


AWC

GPMPU28 PIC Development Board Data Sheet

© 2002-2003 by AWC



AWC
310 Ivy Glen
League City, TX 77573
(281) 334-4341

<http://www.al-williams.com/awce.htm>

V1.1 30 Aug 2003

Table of Contents

Overview	1
If You Need Help	1
Planning	1
Board Assembly and Modifications	2
Testing	4
Customization	5
Powering the GPMPU	6
Bill of Materials.....	7
About the RS232 Interface	8
Using an ICD	8
Additional I/O	9
Figure 1. Circuit Diagram.....	10
Connections to JP1	11
Connection to ICSP Header	11
Figure 2. Board Layout.....	12

Overview

The GPMPU kit is an excellent prototyping and development platform for 28 pin Microchip PIC parts (such as the 28-pin PIC16F87x chips or the 18F252). The board has many features, including:

- Optional MAX232/MAX232A RS232 converter with DB-9 connector.
- Optional 7805-compatible regulator circuit.
- In circuit programming connector.
- Space for a ceramic resonator.
- An edge connector that can plug into a solderless breadboard or connect to other circuitry.
- A unique system that allow you to customize the board connectors quickly and easily.

If You Need Help

If you require assistance, please feel free to contact us. The best way to get support is via e-mail (stamp@al-williams.com). However, you may also call between 9AM - 4PM Central Time at (281) 334-4341. You can also fax to (281) 754-4462. Be sure to check out our Web page for updates at www.al-williams.com/awce.

Planning

If your GPMPU board is part of another kit, refer to that kit's instructions to determine what options you must build and any modifications you may need to make to the board. If you are building the GPMPU for your own circuit, consider these options you can exercise when building:

- CPU and clock choice. The board can accommodate many PIC 28 pin devices including the PIC16F870, PIC16F873, PIC16F876, PIC18F242, PIC18F248, and many other 28-pin PICs.
- You can power the board from a 5V regulated supply, or build the onboard 5V supply using a 7805 (or other pin compatible regulator such as a TI PT78ST105 switching regulator or an LM2940-CT5.0 low drop out regulator).
- For RS232 connections you can install a MAX232 or MAX232A IC and a right-angle female DB9.
- For TTL serial, you may jumper across the MAX232 to connect the microprocessor directly to the DB9.
- The board is designed to accept standard resonators with capacitors built-in. However, it is possible to build the board with a crystal or an RC oscillator.
- If you install pins in the JP1 position, the board can easily plug into a breadboard or connect to a prototyping PC board. You may also leave these pins open to connect wires or components.

Board Assembly and Modifications

Once you've decided which components to install, start with the shortest components first. Try not to install the tall components (IC sockets, pins, LEDs, connectors) until all of the smaller components are in place. Installing tall components first will make it more difficult to fit the smaller parts in the board for soldering, although this is not strictly necessary – you can install the components in any order you wish.

If you are building the board to use with a solderless breadboard, you should install the header at JP1 on the bottom of the board and solder it at the top of the board. This allows you to plug the GPMPU28 directly into a solderless breadboard.

Selecting C1-C4: If you are installing a MAX232, you need 1uF capacitors for C1-C4. If you are installing a MAX232A, then you need .1uF capacitors instead. If you are not installing IC2, there is no need to install C1-C4.

Special note about electrolytic capacitors: If you use electrolytic capacitors for C1-C4, you must insert them correctly. Use this guide:

- C1 – The + lead is nearest the reset pins.
- C2 – The + lead is nearest pin 1 of IC1.
- C3 – The + lead is nearest the center of the board.
- C4 – The + lead is nearest the J1 mark near the board's edge.

Special note about D1: The LED, D1, is a 5V LED. Do not attempt to fit an ordinary LED into these holes since there is no provision for a dropping resistor. Although you can fabricate your own 5V LED by cutting one lead of the LED short and soldering a suitable resistor (220 to 470 ohms) in series with that lead, you can also obtain 5V LEDs from a variety of sources including AWC and Radio Shack (#276-0208). Many blue LEDs will work directly on 5V also (for example, Radio Shack #276-311). Of course, if you don't want an LED, omit it and you can use the hole to make a connection to RC3.

The flat part of the diode (which is normally next to the shortest lead) should line up with the flat part of the LED marking on the board.

Testing

It is possible to build the board in sections, testing as you go (although this will certainly cause you to violate the tall components last rule). The suggested order of construction is:

- Power supply – install IC3, C5 and C6. Assemble the heat sink on IC3. Connect a DC supply (unregulated) that supplies 8 to 12 volts between the Vin and Ground pins. If this does not work, check your connections and make sure IC3 is not backwards.
- RS232 section – Install J1, IC2, and C1-C4. You can test this section using a PC with Hyperterminal and an appropriate cable. First start Hyperterminal and create a connection to the port you are using. Set the serial parameters to 19200 baud, 8 bits, no parity, and one stop bit. Set handshaking to none (very important). You'll also want to disable local echo (in the ASCII setup options).

The next step is to make sure you have Hyperterminal working properly. Connect the cable to your PC but **not** the GPMPU. Using a test jumper, short pin 2 and 3 of the cable together. When you type into Hyperterminal, you should see what you are typing when the jumper is present and not see anything when you remove the jumper. If you see each character twice, you have local echo enabled (turn it off, or remember that you have it on, it doesn't really matter). If you don't see any characters then you have the wrong port, or a port conflict with another serial port. You must fix this before you can test the board (or use it with the serial port, for that matter).

Now remove the test jumper from the serial cable. Using a test lead, short pins 11 and 12 of IC2. Plug the serial cable into the GPMPU and apply power to the board (either via the power supply or, if you don't have the power supply, with a 5V regulated supply). When you type in Hyperterminal, you should see what

you are typing if everything is functioning (unless you have local echo on, in which case you should see each character *twice*). If this does not work, check for proper insertion of IC2. You should read about +/- 8V on IC2 pins 2 and 6, respectively. If using electrolytic capacitors for C1-C4, check the polarity carefully.

- Processor section – There isn't much you can do to test the processor other than use it for something. It is a good idea to program a very simple test program to blink D1 until you are certain the board is assembled properly (you can find a program that does this on our Web site). Common errors include setting the wrong oscillator fuse bits, or using an unprogrammed PIC.

Customization

There are three methods used to customize the board. First, do not install pins at JP2 and JP3. These holes are shorted together along with the corresponding JP1 pin. So if you do nothing, you have a processor signal at each pin of JP1 (except the edges which carry power and ground).

However, if you want to customize the signals on JP1, you can use a sharp hobby knife to cut the trace that runs between the corresponding holes of JP2 and JP3. This leaves you with a signal free on JP3, and a connection to the JP1 pin at JP2. You can run a wire jumper to another uncommitted hole, or underneath the board to solder to a pad. In this way you can bring any 9 I/O pins out to JP1. In some case, you may be instructed to cut certain pads to use for extra components, as well. For example, you might solder a resistor into a 5V hole and solder it into one of the JP2 or JP3 holes to create a pull up.

A third method for customizing the board involves replacing components with other components. For example, suppose you don't want to install a MAX232, and you need to do TTL-level serial output. You can fit a wire between IC2 pins 12 and 13 along with another wire between pins 11 and 14. It is also possible in

some cases to directly drive a PC's input (short pins 11 and 14) and use a 22K resistor on the PC's output (place a 22K resistor between pins 12 and 13). Note, however, that for this resistor trick to work you can't use the PIC's UART – you'll have to use a "bit banging" UART program that can invert the sense of the serial data. On the other hand, a wire and a resistor is cheaper than a MAX232 and 4 capacitors.

You can also fit different components in the resonator holes (X1). For example, a two pin resonator will fit and you can solder the loading capacitors (around 15pF; check your resonator's data) on the bottom of the board from the resonator's pins to the center ground hole.

There are also many substitutions you can make for IC3. For example, you may wish to use a low dropout regulator (a LM2940-CT5.0) or a 78L05 in these holes. IC3 faces the inside of the board so that you may fit a heat sink or heat sink the chip to a metal case, if desired.

Normally, the reset jumper is simply two pins and a jumper cap. However, it is possible to insert a small switch (for example, Radio Shack 275-1571) in these holes to create a reset button.

Powering the GPMPU

If you have the power supply installed, you simply feed DC voltage into the Vin (+) and Ground (-) terminals. This must not be AC voltage. With a standard 7805, you should feed at least 8V to produce a stable 5V supply. A 12 to 15V supply is ideal. Do not exceed the voltage rating of C6 (it is OK to replace C6 with a higher voltage capacitor).

Be aware that common wall transformers often put out much more than their rated voltage at low loads. A 12V 1A transformer may put out 18V or more at 100mA. Conversely, you may be able to

use a transformer rated at a lower voltage (say, 6V) if you are drawing less current than the transformer expects to supply.

With a 12V input and a standard 7805, you should be able to draw at least 1A (although you will probably need to add a heat sink). The basic GPMPU card draws only a fraction of this (depending on the speed you operate your PIC, and which PIC you are using). The more input voltage you provide, the less current the regulator will provide.

If you don't build the power supply, you can also feed **regulated** 5V directly to the board's power buss. In particular, you can feed +5V into JP1 pin 11 and ground into JP1 pin 1. You can also feed +5V into the hole for IC3 pin 3 and the ground connection to IC3 pin 2.

Even if you aren't using the onboard power supply, you should install C5. This filters noise from the power supply. You can also install a jumper from IC3 pin 1 to IC3 pin 3 and use C6 as a larger filter capacitor if you aren't using IC3.

Bill of Materials

Note: Not all parts may be in your kit (see the specific GP kit manual for details). Many parts are optional depending on the kit you've purchased:

C1, C2, C3, C4 – If IC2 is a MAX232, then 1uF; if IC2 is a MAX232A, then .1uF, 16V minimum

C5 – .1uF capacitor, 16V minimum

C6 – 220uF (or larger) capacitor, 35V minimum

D1 – 5V LED (5mm)

IC1 – 28-pin Microchip flash processor (typically, 16F873)

IC2 – Maxim MAX232 or MAX232A

IC3 – 7805 regulator (or equivalent, see text)

J1 – Right angle female DB9 (short reach style)

R1 – 10K to 22K resistor

X1 – Ceramic resonator with capacitors

Optional:

Header pins or connectors for interface holes

Heat sink and hardware for IC3

Socket for IC1

About the RS232 Interface

The DB9 connector is wired by default as DCE and connected to the PIC's UART pins. Because the connector is wired DCE, you can connect to a PC directly with a straight cable. If you wish to connect to another DCE device (like a modem) you will need to use a null modem (or cross) cable.

RB2 of the processor is connected to CTS and RB1 is connected to RTS. These are conditioned by IC2 to RS232 voltage levels.

Using an ICD

It is possible to use the Microchip ICD to debug programs on the board. You can connect directly to the ICSP header (RB3 is not used by the ICD; leave it disconnected). You can also use a 28 pin header by placing a 28 pin socket under it first. This will lift the header high enough to clear components on the board.

Additional I/O

It is possible to use the holes on the edge of the board to access Port C bits 4 and 5. You can also access Port B, bits 0, 3, 4, and 5. If you are not using the ICSP header, Port B bits 6 and 7 are available there. The serial port uses Port B bits 1 and 2 as well as Port C bits 6 and 7. If you don't require RTS and CTS on the serial port, it is possible to cut the traces from Port B bits 1 and 2 and use them for another purpose.

Any pins that are not connected to anything should be set to outputs to avoid problems associated with floating inputs.

Connections to JP1

Pin 1 of JP1 is closest to the JP1 marking on the board:

JP1 pin	Signal	IC1 Pin
1	Ground	
2	RC2	13
3	RC1	12
4	RC0	11
5	RA5	7
6	RA4	6
7	RA3	5
8	RA2	4
9	RA1	3
10	RA0	2
11	5V	

Connection to ICSP Header

You can use the ICSP header to connect to an in circuit programmer or in circuit debugger. Pin 1 is the pin closest to the DB9 (J1).

Pin	Signal	IC1 Pin
1	MCLR	1
2	5V	
3	Ground	
4	RB7	28
5	RB6	27

Figure 2. Board Layout

