MC9S08QG8 Project Board
MC9S08QE8 Project Board

This printed circuit board is designed to be a hardware platform for evaluating Freescale's MC9S08QG8 microcontroller. Several interesting and useful projects using this board are on my website but many others are possible that I haven't even thought of.

If you are reading this you are likely to be aware of the characteristics of this MPU chip. The highlights are: 8K bytes of flash, 512 bytes of RAM, high speed, many peripherals, 16 pin DIP package. For more information you must read the MC9S08QG8 Data Sheet from Freescale. The newer MC9S08QE8 is pin compatible and should also work. The 'QE8 has several advantages over the 'QG but may be more difficult to find in the distribution chain for awhile.

Please refer to the schematic diagram. I will describe some of the features of the board.

You should supply power in the range of 6 to 9 Volts DC to the header H1 in the upper right corner of the board. Polarity is unimportant because it is fed thru the full wave bridge rectifier, D1. A filter capacitor may be soldered at C1, or not, depending on your power supply. The voltage is regulated to 5 Volts by the LM7805 or LM340 at IC1. This one amp regulator was chosen, not because the board required that much current but some peripherals may. The MCU requires a 1.8 to 3.3 volt power supply so IC2 is used to provide that voltage. The regulator listed is a LM2936Z-3.3 but there are other parts that could be used. Be careful that the pinout is the same.

IC3 is the MCU, a MC9S08QG8CPBE (or MC9S08QE8CPG). Freescale recommends putting a .1 uF and a 10 uF next to the power pins. They are C6 & C5.

Communication to the outside world is via a Serial Communication Interface to a RS-232 transceiver. IC4 is the transceiver which is a SP3232E because it is 3.3 volt compatible. The output is to a header H2 and/or a 9 pin female D-Sub socket X2.

The processor has a Serial Peripheral Interface the signals of which are fed to the 6 pin header, H3, located at the lower left side of the board. An I/O signal, which may be used as an Enable is provided on the header as well as +5V and ground. A SIP resistor network, RN1, may be installed for pull-up. Diodes D3, D4 and D5 along with the pull-ups allow the MCU to connect to 5 volt peripherals. This SPI header is used to connect to several of the projects described on my website.

Located above the header on the board is a LED and a 6-pin mini-DIN socket which is intended for a keyboard interface project but may be used for other purposes with suitable programming. The two I/O lines that would connect to a keyboard are bidirectionally level shifted with a N-MOSFEF so that you can use a regular 5 volt keyboard.

On the far lower right side of the board is a three pin header intended for a Dallas Semiconductor one-wire interface. On my web pages is a project using the DS18B20 temperature sensor. Other one-wire devices may also be used.

Header SV2 is for the HCS08 background debug controller (BDC). Using the BDC and CodeWarrior you can debug, trace, and load programs. You would need a background debug module to connect between your PC and

the Project Board. P&E sells one for about $100. For less than half the cost there is the Open Source BDM (OSBDM08). You can buy an assembled unit from Witztronics. There is a minor problem in using the OSBDM08, however. With a brand new, blank 9S08 chip you cannot establish a connection because the BKGD must be held low during power-up. There is provision for doing this on the (Rev. 1) board with IC5. Unfortunately, that should only be used to establish connection with a blank chip; it will prevent proper operation of the MCU once it is programmed. Therefore, IC5 should only be connected to the BKGD pin of the MPU when programming a blank chip. There is a jumper, JP1, which should be closed when using a blank chip and open otherwise. If you never intend to program a blank chip you can ignore IC5 and JP1.

There is another way to load your program code into a 9S08 and that is thru the Serial Communication Interface RS-232 port. The chip has to have the Developer's Serial Boot-loader installed first. Any chips that you buy from me will have the bootloader. See Freescale app note AN2295 for more details.

The (Rev. 1) project board also has provision for a serial EEPROM of the AT24CP type which is connected thru the Inter-Integrated Circuit Module (IIC).

There are pads along the right side of the MCU that connect to pins 12 thru 16. The pads connected to pin 12 (RxD) and pin 16 (TMPCH0) were added in Rev. 1 of the board to facilitate using a program to trim the timer of the MCU should that be necessary using a method based on reading the RS-232 port. See AN2496.

The project board has a prototype area for the user. The top right three pads are connected to Ground and the top left three pads are connected to + 3.3 Volts.

The project board is 3.925" x 1.875" and is sized to exactly fit the cute Serpac A-20 enclosure. Four of the six mounting holes in the board align with the mounting bosses in the enclosure. The two other holes align with mounting bosses in the Pactec CM5-125.

Roger Schaefer
Roger@RASmicro.com

http://www.rasmicro.com/