M5211DEMO

Demonstration Board for Freescale MCF5211

Axiom Manufacturing • 2813 Industrial Lane • Garland, TX 75041
Email: Sales@axman.com Web: http://www.axman.com

CONTENTS

FIGURES

TABLES

REVISION

Date	Rev	Comments
February 22, 2006	A	Initial Release.

CAUTIONARY NOTES

1) Electrostatic Discharge (ESD) prevention measures should be used when handling this product. ESD damage is not a warranty repair item.

2) Axiom Manufacturing does not assume any liability arising out of the application or use of any product or circuit described herein; neither does it convey any license under patent rights or the rights of others.

3) EMC Information on the M5211DEMO board:

   a) This product, as shipped from the factory with associated power supplies and cables, has been verified to meet with **FCC** requirements as a **CLASS A** product.

   b) This product is designed and intended for use as a development platform for hardware or software in an educational or professional laboratory.

   c) In a domestic environment, this product may cause radio interference in which case the user may be required to take adequate prevention measures.

   d) Attaching additional wiring to this product or modifying the product operation from the factory default as shipped may effect its performance and cause interference with other apparatus in the immediate vicinity. If such interference is detected, suitable mitigating measures should be taken.

TERMINOLOGY

This development board uses option selection jumpers. A jumper is a plastic shunt that connects 2 terminals electrically. Terminology for application of the option jumpers is as follows:

Jumper on, in, or installed - jumper is installed such that 2 pins are connected together.

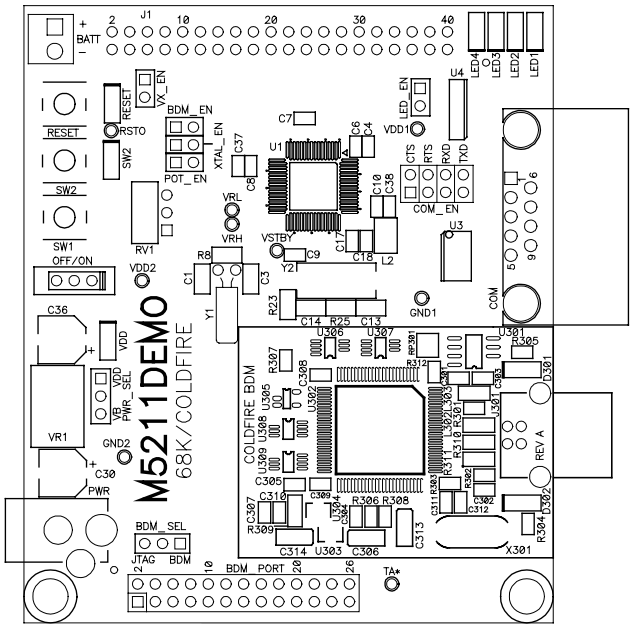
Jumper off, out, or idle - jumper is installed on 1 pin only. It is recommended that jumpers be idled by installing on 1 pin so they will not be lost.

FEATURES

The M5211DEMO is a low-cost development system for the Freescale MCF5211 ColdFire microcontroller. Application development is quick and easy with the included DB9 serial cable, and integrated BDM. CodeWarrior Development Tools are also provided to support application development and debug. The integrated BDM is compatible with popular hosting software and allows easy application development and debugging. An optional BDM port compatible with standard ColdFire BDM / JTAG interface cables and hosting software is provided but not installed,

Features:

- MCF5211 CPU, 64 pins
- 128 Byte Flash
- 16K Byte Ram
- 2K Byte Cache Ram
- DMA Controller w/ Timers
- Interrupt Controller
- 8 Channel 12 Bit A/D
- QSPI, IIC, and CAN Serial Ports
- 1 x UART Serial Ports with DMA capability
- Edge / Interrupt Port
- 8 PWM timers
- 4 GPT Timers
- BDM / JTAG Port
- 3.3V operation
- 66Mhz Bus
- 40 pin I/O port
- Integrated USB BDM port
- BDM / JTAG Port, 26 pins, development port (not installed)
- RS-232 Serial Port w/ DB9-S Connector
- Timing Options
- Internal Oscillator, 8 MHz
- External XTAL option (not installed)
- External clock options (not installed)
- RESET switch w/ indicator
- ON/OFF Power Switch w/ LED indicator
- Power Input Selection Jumper
- Power Input from USB BDM
- Power Input from on-board, +3.3V, regulator
- Power Input from terminal block
- Power from connector J1
- Optional Power output through connector J1
- User Features
- 4 User LED's w/ enable
- 2 User Push Switches
- 5k ohm POT w/ enable
- Option Jumpers
- Power Input Select
- Optional Power Output Enable
- BDM_EN
- XTAL_EN



- POT_EN
- COM_EN
- LED_EN
- Connectors
- DB9 Serial Connector
- Type B USB connector
- 2.0mm Barrel Power Input
- 2pos, screw type, terminal block
- Supplied with DB9 Serial Cable, USB cable, Utility / Support CD, and CodeWarrior Development Studio
- Manuals and Wall Adapter Power supply.

Specifications:
Board Size 3.5" x 5.5"
Power Input: +5 - +20VDC, 9VDC typical
Current Consumption: 100ma typical @ 9VDC input

REFERENCES

Reference documents are provided on the support CD in Acrobat Reader format.

M5211DEMO_UG.pdf	M5211DEMO User Guide (this document)
M5211DEMO_SCH_E.pdf	M5211DEMO Schematic, Rev E
M5211DEMO_STARTUP.pdf	M5211DEMO Quick Start Guide
MCF5213RM.pdf	MCF5213 Reference Manual
MCF5213DE.pdf	MCF5213 Device Errata
CFPRM.pdf	ColdFire Programmers Reference Manual
M5211DEMO.zip	CodeWarrior Demo Application for MCF5211
MCF5213SC_V1B.zip	CodeWarrior Demo Application for MCF5213

GETTING STARTED

To get started quickly, please refer to the M5211 Quick Start Guide included with this development kit.

The M5211DEMO single board computer is a fully assembled, fully functional development board for the Freescale MCF5211 microcontroller. An included serial cable, USB cable, and support software will allow the user to get started quickly.

The purpose of this development board is to assist the user in quickly developing an application in a known working environment, to provide an evaluation platform, or as a control module for an embedded system. Users should be familiar with memory mapping, memory types, and embedded software design for successful application development.

The user should have familiarity with the hardware and software operation of the target MCF5211 device before beginning. If necessary, refer to the MF5213 Integrated Microcontroller Reference Manual, included on the support CD, for details on the operation of the MCF5211.

SOFTWARE DEVELOPMENT

Software development requires the use of a ColdFire assembler or compiler and a host PC running a ColdFire BDM interface. CodeWarrior 6.1 Special Edition, which is supplied with this board, allows the user to develop and debug application code and to program flash.

One method to ensure successful application development is to load and execute the application from RAM. After the application has been completely debugged fully functional, it can be ported to FLASH memory. When programmed into FLASH memory, the application will execute from Power-On or RESET.

OPERATING MODES

The M5211DEMO board operates in two basic modes Run Mode, or Debug Mode. Run Mode executes user application code from Power-On or Reset. Debug Mode supports the development and debug of applications via the integrated USB BDM. An optional BDM_PORT is provided but not installed. See the related sections below for quickly starting the board in the desired mode of operation.

The board has been preloaded with a demonstration program that operates in the Run Mode. Refer to the M5211DEMO Quick Start Guide included with this kit for details on the loaded application.

RUN Mode

Run mode allows the user application to execute out of FLASH when power is applied to the board or the RESET button is pressed. Use the following settings to configure the M5211DEMO for RUN Mode using the USB bus to power the board. See the POWER section below for details on configuring the board for alternate power input.

1. Connect a serial cable between the board and a host PC if needed.

2. Connect auxiliary equipment to board if needed.

3. Configure the board option jumpers as shown.

Table 1: Run Mode Setup

PWR_SEL	Set to VB
BDM_EN	OFF
COM_EN	ALL ON (if required)
VX_EN	ON (if required)

Connect the USB cable to an open USB port on the host PC and attach to the USB port on the target board. LED's D301 and D302 located adjacent to the USB connector, and the VDD LED will light and the loaded application will begin to execute.

Debug Mode

Debug Mode supports application development and debug using the ColdFire background debug mode (BDM). Background mode is accessible using either the integrated USB-BDM or an external ColdFire BDM cable. Use of the integrated BDM requires a host PC with an available USB port and an A/B USB cable and appropriate hosting software. The USB cable used must be USB 2.0 compliant. A 26-pin BDM_PORT header supports the use of an external BDM cable. This header is not installed in default configurations. The steps below describe using the integrated USB-BDM. See the POWER section below for details on configuring the board for alternate power input.

1. Connect a serial cable between the board and a host PC if needed.

2. Connect auxiliary equipment to board if needed.

3. Install and launch CodeWarrior 6.1 Special Edition, or other software capable of communicating with the ColdFire MCU.

4. Configure the board option jumpers as shown.

Table 2: Debug Mode Setup

PWR_SEL	Set to VB
BDM_EN	ON
COM_EN	ALL ON (if required)
VX_EN	ON (if required)

Connect the supplied USB cable between an available USB port on the host PC and the USB connector on the board.

Hosting development software will establish background communication.

MEMORY MAP

Refer to the MCF5213 Integrated Microcontroller Reference Manual for details.

DEVELOPMENT SUPPORT

Application development and debug for the target MCF5211 is supported through the BDM interface. The debug interface consists of an integrated USB-BDM debugger and an optional 26-pin header (BDM_PORT). The BDM_PORT header is not installed in default configuration and may be installed by the user if needed.

Integrated BDM_PORT

The M5211DEMO board features an integrated USB-BDM debugger from P&E Microcomputer Systems. The integrated debugger supports application development and debugging via the background debug mode. A USB, type B, connector provides connectivity between the target board to the host PC.

The integrated debugger provides power and ground to the target, thereby eliminating the need to power the board externally. When used, power from the USB-BDM circuit is derived from the USB bus and total current consumption is limited by the USB specification. Total current consumption for the target board, and all connected circuitry, must not exceed **500mA**. Excessive current drain will violate the USB specification causing the bus to disconnect. This will force a target POR.

CAUTION: Violating the USB specification will cause the USB bus to disconnect forcing the target to reset.

Use of the integrated USB BDM requires the use of CodeWarrior 6.1 Special Edition or development tools from P&E Microcomputer Systems.

BDM_PORT Header

A ColdFire BDM cable may be attached to the 26-pin BDM_PORT port header. This header is not installed in default configuration and its use requires the user to install a 2x13, 0.1" center, BERG header. Refer to the BDM documentation, in the MCF5213 User Manual, for details on the use of the BDM cable. The BDM_PORT header pin-out is shown below.

Figure 1: BDM_PORT

NC	1	2	BKPT*
GND	3	4	DSCLK
GND	5	6	TCLK (JTAG)
RSTI*	7	8	DSI
VDD	9	10	DSO
GND	11	12	ALLPST
ALLPST	13	14	ALLPST
ALLPST	15	16	NC
NC	17	18	NC
NC	19	20	GND
NC	21	22	NC
GND	23	24	TCLK (BDM)
VDD	25	26	TA*

Refer to the Debug Module chapter of the MCF5213 Microcontroller Reference Manual for complete details on the BDM_PORT usage.

NOTE: This header is not installed in default configuration.

POWER

The M5211DEMO is designed to be powered from the USB_BDM during application development. A 2.0mm barrel connector and a 2-position, screw-type, terminal block (BATT)

has been applied to support stand-alone operation. Additionally, the board may be powered through connector J1. The board may also be configured to supply power through connector J1 to external circuitry. An OFF/ON switch allows the user to quickly and easily turn the board on and off.

When using the integrated USB-BDM, the board draws power from the USB bus. Excessive current drain will violate the USB specification causing the USB bus to disconnect forcing a POR. Total current consumption of the board and connected circuitry, therefore, must be limited to less than **500mA**.

> **CAUTION:** Violating the USB specification will cause the USB bus to disconnect. This will force a hard reset on the target.

The installed barrel connector accepts a center-positive, 2.1m barrel plug. The terminal block accepts wire sizes ranging from 28ga to 16ga. Voltage input must be in the range between +5V and +20V. At no time should input voltage exceed +20V as damage to the board may result. The terminal block input is connected directly to the upper voltage rail. Input protection is not applied on this voltage input. The user must exercise caution when using the terminal block to input power to the board.

Voltage supplied through connector J1 is also connected directly to the board voltage rails. No protection is applied on this input and the user must exercise caution when powering the board from connector J1.

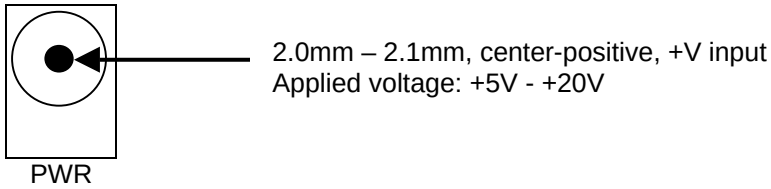
> **CAUTION:** Input protection is not applied to the J1 or BATT power inputs. Excessive input voltage or current will damage the board.

POWER INPUT

POWER JACK

The PWR power jack consists of a 2.1mm, center-positive, barrel connector. Voltage applied at this connector should range between +5V and +20V.

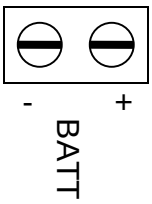
Figure 2: PWR Jack



Terminal Block

The BATT terminal block is a 3.5mm, screw-type terminal block connected directly to the VDD voltage rail. Caution must be used when using this input since input protection is not applied. Use of this input requires a regulated +3.3V voltage source.

Figure 3: BATT Terminal Block



Accepts wire size 28AWG – 16AWG
Applied voltage must be +3.3V.

Connector J1

See the schematic for details on using this connection to supply power to the board or source power from the board. Use of this input requires a regulated +3.3V voltage source.

CAUTION: Do not over drive either the J1 or BATT inputs as damage to the board may result.

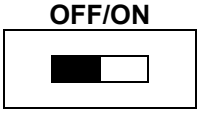
VDD LED

The VDD LED indicates the state of power applied to the development board. This LED is on when power is applied to the board. If no power is applied or the OFF/ON switch is in the OFF position, this LED be off. The VDD LED is located after the OFF/ON switch and will indicate the state of the switch if the board is powered from the on-board regulator, the USB circuit, or the BATT terminal block input.

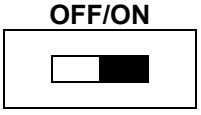
POWER SWITCH

The OFF/ON switch connects and disconnects all input voltage sources to the board. In the OFF position, the board is unpowered and no voltage is present on the upper voltage rail. In the ON position, the input voltage source is connected to the upper voltage rail.

Figure 4: OFF/ON Switch



CONFIGURATION
OFF - Input power source disconnected from upper voltage rail



ON - Input power source connected to upper voltage rail

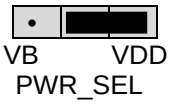
POWER SELECT

Configuration of applied input power is controlled using 2 option headers. The PWR_SEL header selects between the on-board voltage regulator and the USB voltage input. The VX_EN header connects J1-1 directly to the upper voltage rail. The illustrations below show the different configuration for each option header.

PWR_SEL

Figure 5: PWR_SEL Option Header

Select power input from USB-BDM



Select power input from VR1

Power from the integrated BDM is drawn from the USB bus and is limited to **500mA**. Excessive current drain will violate the USB specification causing the USB bus to disconnect.

CAUTION: Violating the USB specification will cause the USB bus to disconnect. This will cause the board to reset.

The on-board voltage regulator (VR1) accepts power input through a 2.1mm barrel connector (PWR). Input voltage may range from +5V to +20V. VR1 provides a +3.3V fixed output limited to 800mA. Over-temperature and over-current limit built into the voltage regulator provides limited protection from damage due to excessive stresses.

The user should consider the maximum output current of the selected power source when attempting to power off-board circuitry through connector J1.

VX_EN

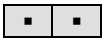
The VX_EN option header is a 2-pin jumper that connects the target-board voltage rail to J1-1. J1-3 is connected directly to the ground plane. Use of this input requires a regulated +3.3V voltage source. This power input is decoupled to minimize noise input but is not regulated. Also, no protection is applied on this input and damage to the target board may result if over-driven. Do not attempt to power the target board through this connector while also applying power through the USB-BDM or the PWR connector; damage to the board may result.

Power may be sourced to off-board circuitry through the J1 connector. The current limitation of the USB bus or the on-board regulator must be considered when attempting to source power to external circuitry. Excessive current drain may damage the target board, the host PC USB hub, or the on-board regulator. The figure below details the VX_EN option header connections.

Figure 6. VX_EN Option Header

ON Enable power connection to connector J1

VX_EN



OFF Disable power connection to connector J1

VX_EN

CAUTION: Do not apply power to connector J1 while also sourcing power from either the PWR connector or the USB-BDM circuit. Damage to the board may result.

NOTE: Do not exceed available current supply from USB-BDM cable or on-board regulator when sourcing power through connector J1 to external circuitry.

RESET

The MCF5211 can be reset several ways. A RESET switch applies an external reset to the device. An internal low-voltage detect forces the part into RESET when voltage falls too low. The part may also be reset by applying a logic low on the RSTI* signal. The RSTI* signal is used by a connected BDM to force the part into reset when necessary. A reset indicator shows when the part is in reset.

RESET SWITCH

The RESET switch provides a method to apply an asynchronous reset to the MCU and is connected directly to the RSTI* input on the MCU. Pressing the RESET switch forces the MCU into reset until the switch is released. An external pull-up on the RSTI* line prevents spurious resets and allowing normal operation.

LOW VOLTAGE DETECT

The MCF5211 utilizes an internal Low Voltage Detect (LVD) to protect against under-voltage conditions. The LVD is enabled out of RESET. Consult the MCF5211Integrated Microcontroller Reference Manual for details on configuring LVD operation.

RESET INDICATOR

The RESET LED turns ON when the MCU is in RESET and remains on for the duration of a valid RSTO* signal.

LOW-POWER MODES

The MCF5211 supports several operational modes designed to reduce power consumption. Low-power modes include Wait, Doze, Stop, and Halt. Refer to the MCF5213 Microcontroller Family Hardware Specification and the MCF5213 Integrated Microcontroller Reference Manual for details on configuring and using the various low-power modes.

TIMING

Timing for the M5211DEMO is provided by an on-chip, 8 MHz relaxation oscillator in default configuration. Resistor R23 causes the MCU to select the internal oscillator as the timing

source out of reset. Refer to the M5213 Integrated Microcontroller Reference Manual for details on use and configuration of internal timing.

Pad locations for an external crystal oscillator have been provided but not populated in default configurations. The external oscillator is connected to the XTAL and EXTAL inputs. To implement an external crystal oscillator simply remove R23, install the oscillator and related circuitry. The XTAL_EN shunt must be installed to enable external timing operation.

Figure 7: OSC_EN Option Header

ON Enables external crystal oscillator input to MCU

XTAL_EN

OFF

XTAL_EN

NOTE: Remove resistor R23 to use the external crystal oscillator.

Pad locations for an optional 32.768 kHz have also been provided but not populated. The 32.768 kHz timing input is connected to GPT0 and GPT1 inputs. To implement the this timing input simply install the timing source and related circuitry.

COMMUNICATIONS

The M5211DEMO board provides 2 Serial Communications Interface (SCI) ports, one Serial Peripheral Interface (SPI) port, and one Inter-Integrated Controller (IIC) port. RS-232 communications are supported through a DB9 connector and connector J1. SPI and IIC communications are supported through connector J1. The COM_EN option header enables SCI0 operation on the board.

SCI Port

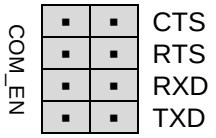
An RS-232 transceiver provides RS-232 to TTL/CMOS logic level translation between the COM connector and the MCU. The COM connector is a 9-pin Dsub, right-angle connector. A ferrite bead on shield ground provides conducted immunity protection. Communication signals TXD0 and RXD0 are routed from the transceiver to the MCU. These signals are also available on connector J1. Hardware flow control signals RTS0 and CTS0are also routed from the transceiver to the MCU.

Serial communications signals RXD1, TXD1, RTS1, and CTS1 are available on connector J1 and route directly between the MCU and connector J1. No RS-232 logic-level translation is provided on the SCI1 port.

COM_EN

The COM_EN option header individually connects and disconnects SCI0 signals between the MCU and the SCI transceiver. Removing a shunt disconnects the associated signal. Installing a shunt connects the associated signal.

Table 3: COM_EN Option Header

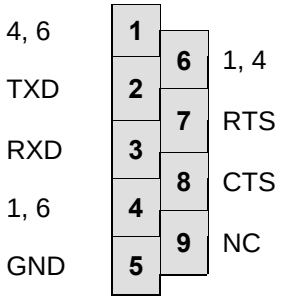


	Shunt	
	On	**Off**
CTS	Enabled	Disabled
RTS	Enabled	Disabled
RXD	Enabled	Disabled
TXD	Enabled	Disabled

COM Connector

A standard 9-pin Dsub connector provides external connections for the SCI0 port. The Dsub shell is connected to board ground through a ferrite bead. The ferrite bead provides noise isolation on the RS-232 connection. The DB9 connector pin-out is shown below.

Figure 8: COM Connector



Female DB9 connector that interfaces to the ColdFire internal SCI1 serial port via the RS232 transceiver.

Pins 1, 4, and 6 are connected together.

SPI Port

SPI signaling connects directly between connector J1 and the MCU. Refer to the MCF5213 Integrated Microcontroller Reference Manual for details on using the SPI interface.

IIC Port

IIC signaling connects directly between connector J1 and the MCU. Refer to the MCF5213 Integrated Microcontroller Reference Manual for details on using the IIC interface.

USER OPTIONS

The M5211DEMO includes various input and output devices to aid application development. User I/O devices include 2 momentary pushbutton switches, 4 green LEDs, and 1 potentiometer.

Pushbutton Switches

Two push button switches provide momentary, active-low input, for user applications. Pull-ups internal to the MCU must be enabled to provide error free switch operation. Pushbutton switches SW1 and SW2 are connected to MCU I/O ports IRQ4* and IRQ7* respectively.

LED Indicators

Indicators LED1 through LED4 are enabled by the LED_EN option header. Each LED is active-high and illuminates when a logic high signal is driven from the respective MCU I/O port. A 3S buffer between the MCU port and the user LED's provides the drive current necessary to control the LED's.

Figure 9: LED_EN Option Header

ON User LED's enabled active-high

OFF User LED's disabled

A SW2_LED pad has been provided but not installed. If installed, this LED will light on SW2 press and will remain lit for approximately 2ms after SW2 release.

Potentiometer

A 5k ohm, thumb-wheel type, potentiometer at RV1 provides continuous, variable resistance input for user applications. The output is the result of a voltage divider that changes as the thumb-wheel is turned. The potentiometer is connected between VDD and GND with the center tap providing the divider output. This center tap is connected to the MCU on signal AN0. The potentiometer may be disconnected from AN0 by means of the POT_EN option header.

Figure 10: POT_EN Option Header

ON User POT enabled

OFF User POT disabled

The table below summarizes user option connections on the development board.

Table 4: User Option Connections

OPTION	MCU PORT	MCU PIN
SW1	IRQ4	57
SW2	IRQ7	58
LED1	DTIN0	22
LED2	DTIN1	23
LED3	DTIN2	18
LED4	DTIN3	19
RV1	AN0	25

I/O PORT CONNECTOR

Connector J1 provides access to M5211DEMO I/O signals. Signal positions not shown are not connected (NC).

Figure 11: Connector J1

VDD	**1**	**2**	IRQ1*
GND	**3**	**4**	RSTI*
UTXD1	**5**	**6**	RSTO*
URXD1	**7**	**8**	
URTS1*	**9**	**10**	ANO
UCTS1*	**11**	**12**	AN1
GPT0	**13**	**14**	AN2
GPT1	**15**	**16**	AN3
QSPI_DOUT	**17**	**18**	AN4
QSPI_DIN	**19**	**20**	AN5
QSPI_SCLK	**21**	**22**	AN6
QSPO_CS0	**23**	**24**	AN7
UTXD0	**25**	**26**	SCL
TRXD0	**27**	**28**	SDA
URTS0*	**29**	**30**	GPT2
UCTS0*	**31**	**32**	GPT3
IRQ4*	**33**	**34**	DTIN0
VRH	**35**	**36**	DTIN1
VRL	**37**	**38**	DTIN2
IRQ7*	**39**	**40**	DTIN3