User Guide

IRMCS1271 Quick Start Guide

Rev 1.0

By International Rectifier's iMotion Team

Table of Contents

User Guide 1
1 System Installation 2
1.1 Hardware Installation 2
1.1.1 Check All Hardware 2
1.1.2 Installation Procedure 2
1.2 Software Installation 3
1.2.1 Installing from the CD 3
1.2.2 MCEDesigner Startup 3
2 Running the System 5
2.1 System Power-On 5
2.2 Running a Motor with MCEDesigner 5
2.2.1 Opening a Motor Configuration File 5
2.2.2 Checking Communication Status 7
2.2.3 Configure Motor Function 7
2.2.4 Start/Stop Motor Function 8
2.2.5 Reference Speed 9
2.2.6 Drive Status 10
2.2.7 Emergency Stop 10
3 The Reference Design 11
4 Using MCEDesigner 13
4.1 Working with Functions 13
4.1.1 Creating a New Function 13
4.1.2 Modifying Functions 16
4.1.3 Executing Functions 20
4.2 Using the Monitor 20
4.2.1 Trace Setup 21
4.2.2 Activate the Trace 23
4.2.3 Analyze the Trace Results 24
4.2.4 AutoRepeat 26
4.2.5 Aborting a Trace Operation 27

1 System Installation

1.1 Hardware Installation

1.1.1 Check All Hardware

The following hardware pieces are needed to use the IRMCS1271 system.

- IRMCS1271 application board with integrated heat sink
- IRMCS-ISO V3.0 (or higher) isolation board

1.1.2 Installation Procedure

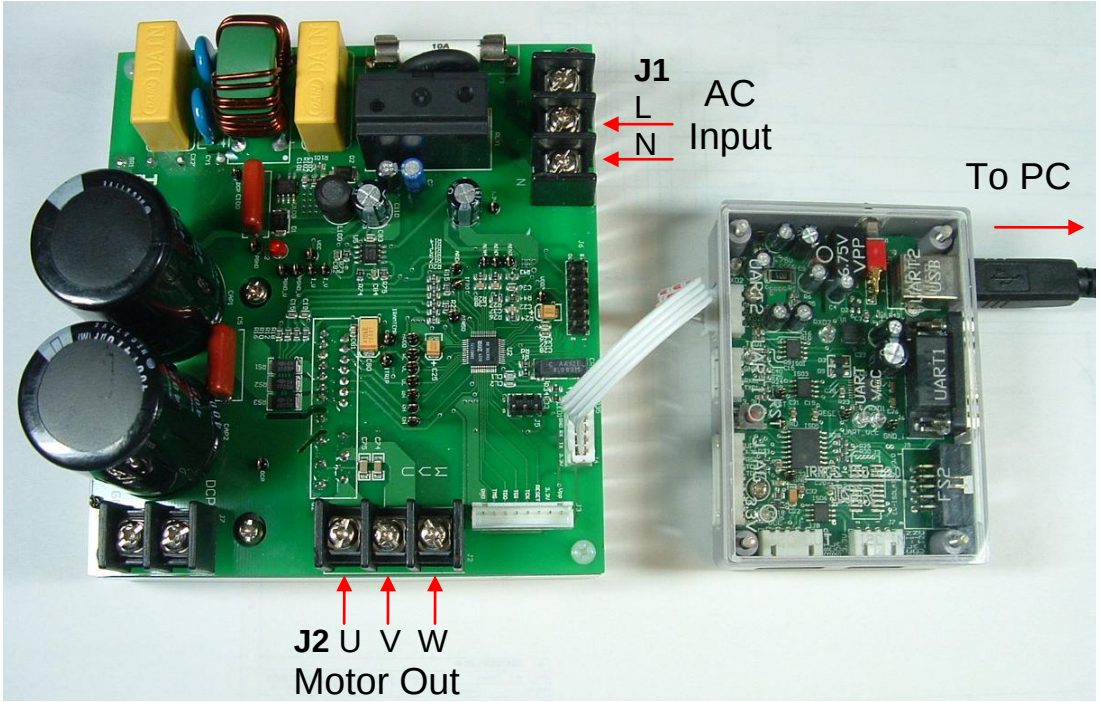


Figure 1—Board Connections

Step 1. Visual Inspection
Visually inspect the IRMCS1271 and IRMCS-ISO boards to check for loose wiring, loose or damaged components or other abnormalities. The IRMCS1271, IRMCS-ISO boards and important connections are shown in Figure 1.

Step 2. Communication Wiring
Connect the included cable from UART2 connector of IRMCS-ISO to J4 of IRMCS1271 as shown in Figure 1. Also, connect a USB cable from IRMCS-ISO to the PC to be used.

Step 3. Input Power Wiring
Prepare a power contactor switch rated at 250V/3A in series with AC power cables if necessary. Turn off or unplug the power and connect single-phase AC 115V to 230V power cables to L/N of J1. The Earth terminal can be left unconnected.

Step 4. Motor Wiring
Connect the motor wires to U/V/W of connector J2.

1.2 Software Installation

1.2.1 Installing from the CD

The MCEDesigner™ tool is distributed on the CD-ROM. Load the CD into the CD-ROM drive on a PC and double-click "100SeriesKitMCEInstallerV3.0.exe".

The automated procedure installs all necessary software on the PC, including the driver for the USB to UART adapter in IRMCS-ISO. The default location for the installation of the motor drive tools, documents and files is "C:\Program Files\iMOTION".

1.2.2 MCEDesigner Startup

The instructions in this section should be followed the first time MCEDesigner is executed. Refer to the MCEDesigner User's Guide for more information.

Step 1. RS232 Connection
MCEDesigner communicates with the IRMCS1271 using a COM port on the PC. Be sure that the USB cable is connected to IRMCS-ISO which is connected to J4 of IRMCS1271. The baud rate is fixed at 57.6Kbps and the signal format is 8 bits, no parity, 1 stop bit.

The first time MCEDesigner runs, a connection dialog box (shown in Figure 2) will appear with a list of available COM ports on the system. Select the COM port which is connected to the USB cable to the isolation board.

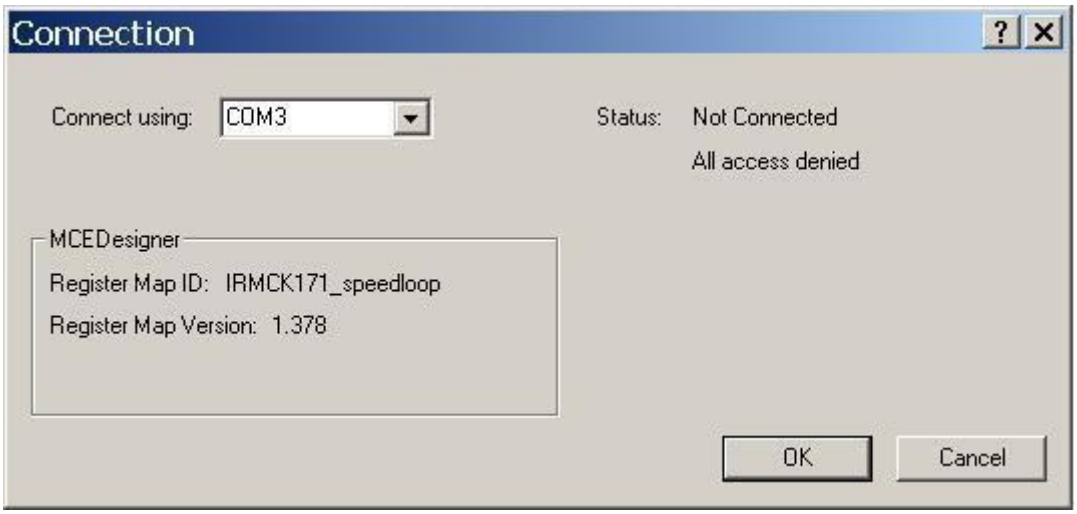


Figure 2—Connection Dialog

When you click OK in the Connection dialog, your selections are saved so they can be used the next time you start the application. (The dialog won't appear on startup again.) If you want to change your selections later on, you can access the Connection dialog through the Preferences menu.

The currently configured COM port is shown on the status bar at the bottom of MCEDesigner's main window.

Step 2. Numeric Format

The first time MCEDesigner runs, the Numeric Display Format dialog appears, as shown in Figure 3. This dialog allows user to select either decimal or hexadecimal format for numeric display. Click the blue button to switch between hexadecimal and decimal. After you click OK, MCEDesigner will save the setting so it can be used next time. (The dialog won't appear on startup again.) If you want to change the setting later on, you can access the Numeric Display Format dialog through the Preferences menu.

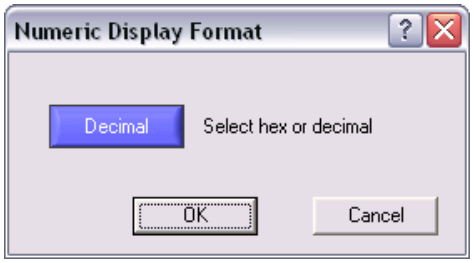


Figure 3—Numeric Display Format Dialog

Important Note: Regardless of which display option you choose, you can always enter values in decimal or hexadecimal. MCEDesigner interprets a value you enter as hexadecimal if it begins with "0x" (0x222), and otherwise as decimal (222).

2 Running the System

2.1 System Power-On

Apply AC power 115V to 230V (recommended voltage range) to the system (IRMCS1071).

Immediately after power-on, the power LED (LED2) will turn red indicating DC bus voltage is established. Also surface mount type LED (LED1) will blink red indicating successful boot load of the MCE program from the IRMCx171 internal memory. Refer to Figure 1 for the locations of the indicator LED's.

2.2 Running a Motor with MCEDesigner

2.2.1 Opening a Motor Configuration File

For a quick start with preconfigured parameters, the following motor is supported with a preconfigured IRMCS1271_R1_2.irc file:

- GK6040-6AC31-WE

If any other motor is used, reconfiguration is required. Please refer to the IRMCx100 Application Developer's Guide for more information.

Open the "IRMCS1271_R1_2.irc" (found in My Documents\iMotion) configuration file from MCEDesigner.

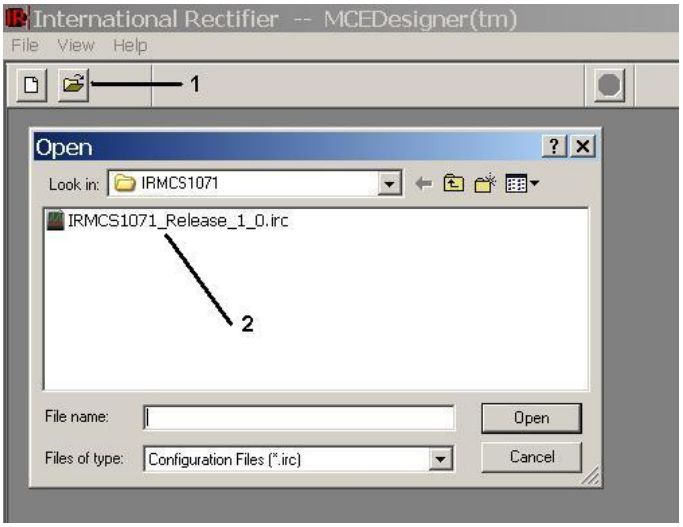


Figure 4—Opening a Configuration File

Then, you will see two windows: System, Motor1. Move your mouse to the "System" window first and go to the "Window" menu, choose "Tile", then you will see a clear view of the two windows

(Figure 5). Basically you can move your mouse between these two windows, to configure and operate each control element.

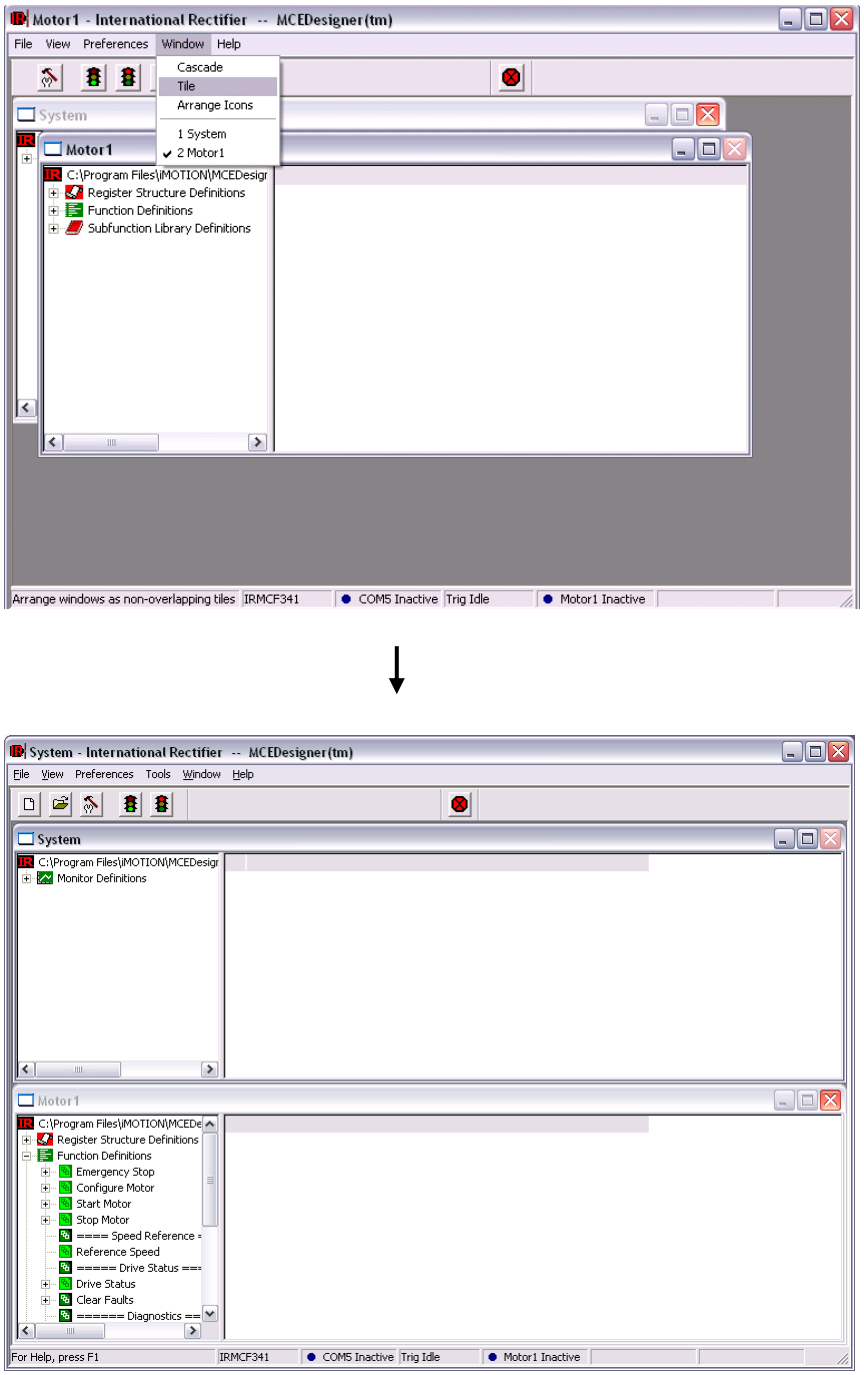


Figure 5—Two Windows in MCEDesigner

2.2.2 Checking Communication Status

When you open a configuration file, MCEDesigner attempts to establish contact with the IRMCS1271. If MCEDesigner receives a reply to its handshake query, the COM port status shows “Up” (Green). If no reply is received, the status stays “Inactive” (Blue). If the status is inactive, it usually means the reference platform is not powered on, the RS232 cable is not connected, or the cable is connected to the wrong COM port. If the COM port does not come up, first try changing the COM port by clicking on the System window, then selecting menu Preferences → Connection.

Figure 6 shows how communication status should appear before you continue your testing.

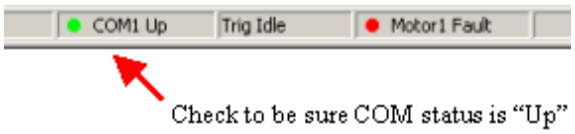
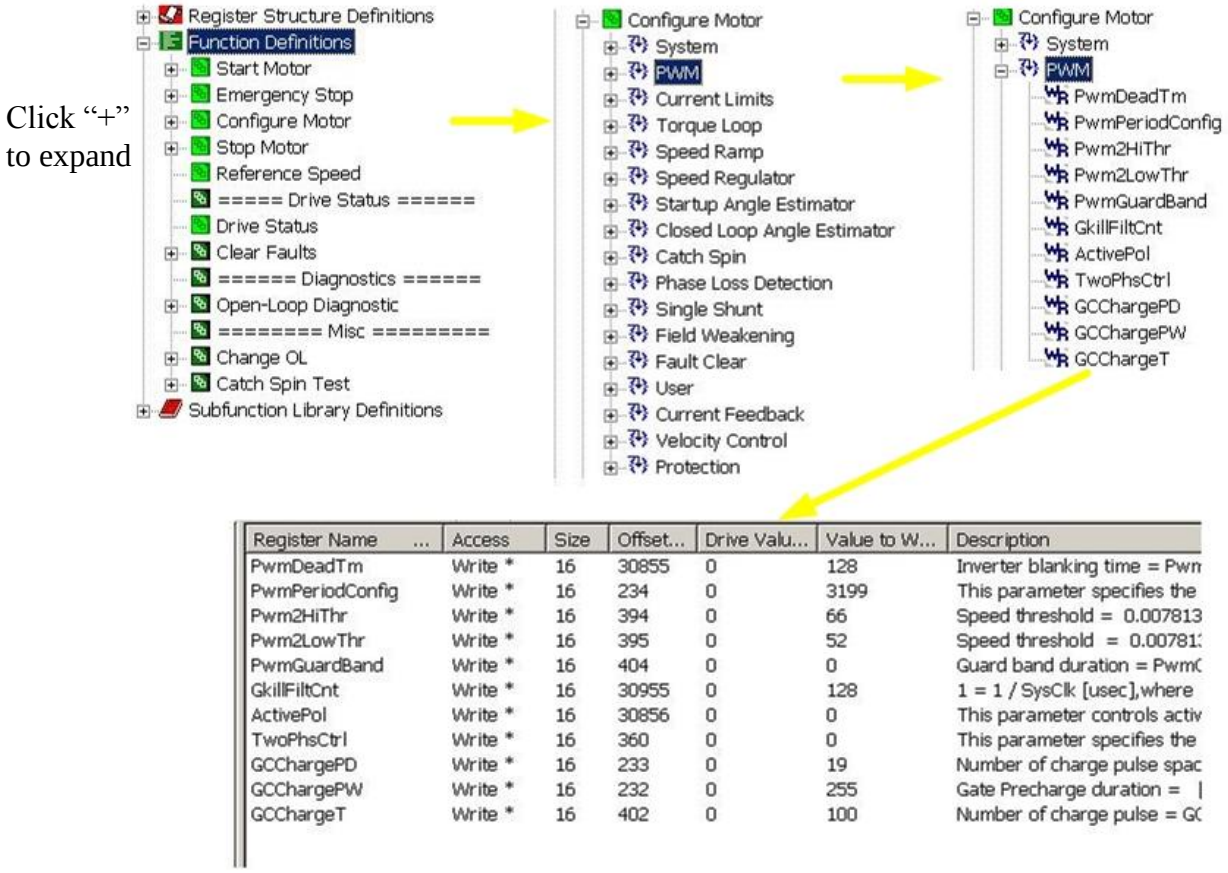


Figure 6—Communication Status Indicator

If MCEDesigner loses communication with IRMCS1271 after COM port has been “Up”, the status will change to “Down” (Yellow) instead of “Inactive”.

2.2.3 Configure Motor Function

The Configure Motor function is one of the pre-defined operations in MCEDesigner’s Function Definitions section, in the “Motor 1” window. This function initializes the host registers for normal operation. Move your mouse to the Motor 1. If you click the Configure Motor entry in the tree view, a list of the sub-functions and registers that are written when this operation is executed is displayed in the right pane of the main window.



Register Name ...	Access	Size	Offset...	Drive Valu...	Value to W...	Description
PwmDeadTm	Write *	16	30855	0	128	Inverter blanking time = Pwm
PwmPeriodConfig	Write *	16	234	0	3199	This parameter specifies the
Pwm2HiThr	Write *	16	394	0	66	Speed threshold = 0.007813
Pwm2LowThr	Write *	16	395	0	52	Speed threshold = 0.00781
PwmGuardBand	Write *	16	404	0	0	Guard band duration = PwmC
GkillFiltCnt	Write *	16	30955	0	128	1 = 1 / SysClk [usec],where
ActivePol	Write *	16	30856	0	0	This parameter controls activ
TwoPhsCtrl	Write *	16	360	0	0	This parameter specifies the
GCChargePD	Write *	16	233	0	19	Number of charge pulse spac
GCChargePW	Write *	16	232	0	255	Gate Precharge duration =
GCChargeT	Write *	16	402	0	100	Number of charge pulse = GC

Figure 7—Configure Motor Function

The "Value to Write" column shows the value that will be written to each register. You can click the "+" symbol to the left of the Configure Motor function to access detailed information about each of the registers.

To execute *only* the Configure Motor function, click in the Motor 1 window and then click the Configure Motor toolbar button (the icon shows a hammer and wrench), or double-click on the Configure Motor entry in the tree view of the Motor 1 window.

Once this function is executed correctly, the LED2 will turn to blinking green from solid red. And the Motor 1 status indicator will change from "Motor1 Fault" to "Motor1 Stop" as figure below.

2.2.4 Start/Stop Motor Function

Start Motor and Stop Motor functions are also pre-defined Function Definitions, in the Motor 1 window.

To start the motor, click the Start Motor toolbar button (the green traffic signal) or click in the Motor 1 window and then double-click the Start Motor entry in the tree view.

To stop the motor, click the Stop Motor toolbar button (the red traffic signal) or click in the Motor 1 window and then double-click the Stop Motor entry in the tree view. When you execute the Stop Motor function, the motor is stopped immediately and if any function is still executing it is halted.

When the motor is running, the far right hand status bar pane should show drive status “Run” with a green indicator.

When the motor is stopped, the drive status should be “Stop” with a yellow indicator. If a drive fault occurs, the status changes to “Fault” with a red indicator. The status is “Inactive” when the COM port status is “Down” or “Inactive.”

2.2.5 Reference Speed

MCEDesigner provides a special built-in function that allows you to easily control the motor’s reference speed and direction. To modify the settings, see Figure 8. First, locate the “Reference Speed” entry in the Function Definitions section of the tree view. Right-click on the entry and select Properties. In the Properties window, you can enter the desired speed in RPM.

After entering a speed, click OK in the window. Click “Start Motor” if it’s not running and then **double-click the “Reference Speed”** entry in the tree view to change the speed. MCEDesigner calculates the appropriate value to be written to the host register (based on your specified speed) and performs the write operation to the “TargetSpeed” register.

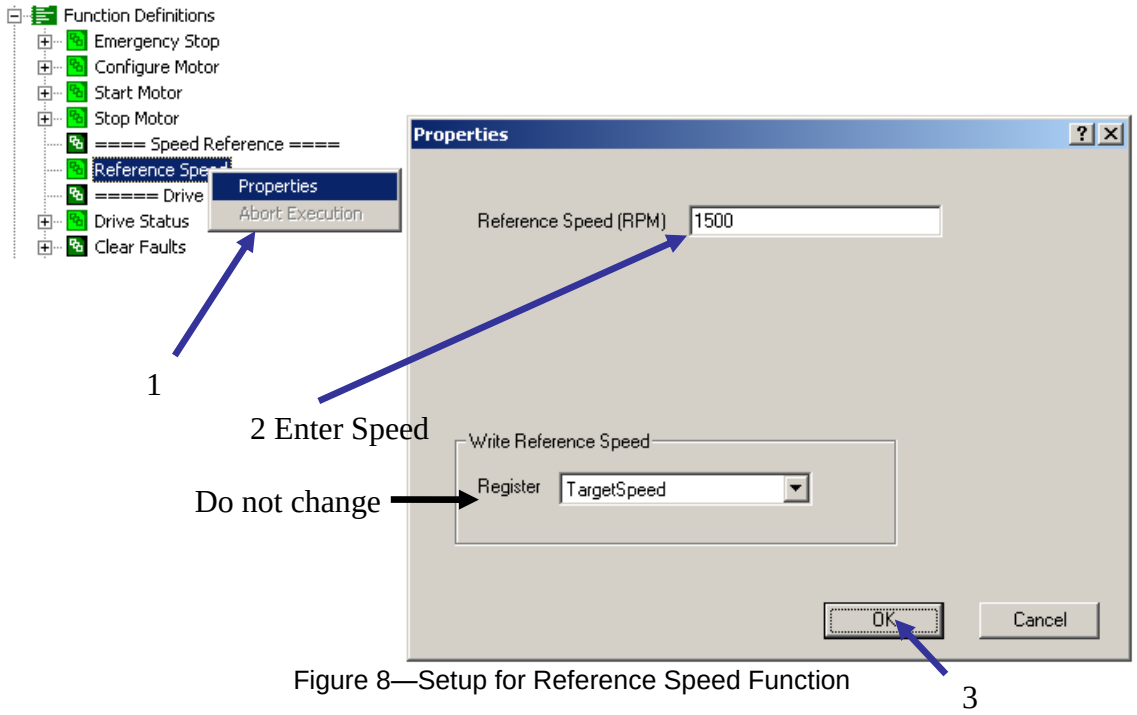


Figure 8—Setup for Reference Speed Function

2.2.6 Drive Status

The Info button on the toolbar (letter “i”) executes the pre-defined Drive Status function, which reads a group of host registers associated with the function.

To get drive status, click the Info button and then click on “Drive Status” in the tree view. In the right pane of the main window, the values read from the Drive Status registers are shown in the Current Value column. You may need to refresh the right pane of the main window by selecting any other entry in the tree view and then clicking on “Drive Status” in the tree view again.

2.2.7 Emergency Stop

The emergency stop function immediately stops all components. It is mainly for multi-window system such as two motor drive plus PFC control. When you click the Emergency Stop toolbar button, the Emergency Stop function is executed for all components regardless of which window is currently active. If you double-click the Emergency Stop entry in the tree view of any one of the component windows, the Emergency Stop function is executed for all components.

3 The Reference Design

The IRMCS1271 controls one sensorless motor drive. The MCE Simulink design (IRMCx171_R210.mdl) contains a single motor control loop, as shown in Figure 9. The MCE design is pre-loaded to the IRMCS1271. Below a few important sections of the Simulink design are highlighted.

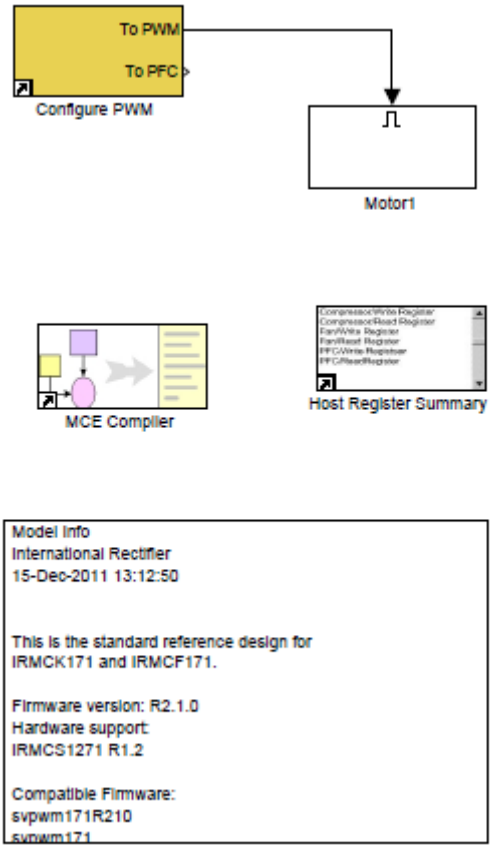


Figure 9—Top Level of the IRMCS1271 Reference Design in Simulink

Figure 10 is a block diagram of the MCE design of the motor 1 control loop. More detail on the MCE block diagram and how to modify this design can be found in the IRMCx100 Application Developer's Guide.

- Angle estimator and reluctance torque control for interior permanent magnet motor is implemented in the SENSORLESS_FOC block.
- Speed loop control is implemented in this Simulink design file (IRMCx171_R210.mdl) by using PI, LIMIT, LOWPASS_FILT, RAMP and some other blocks.
- As shown in the diagram, the lower left section of the design contains the logic for field weakening control. It generates optimal D-axis current command based on voltage margin for current regulator.
- The motoring and regeneration current limits for forward and reverse directions are implemented in the top right section of the design.

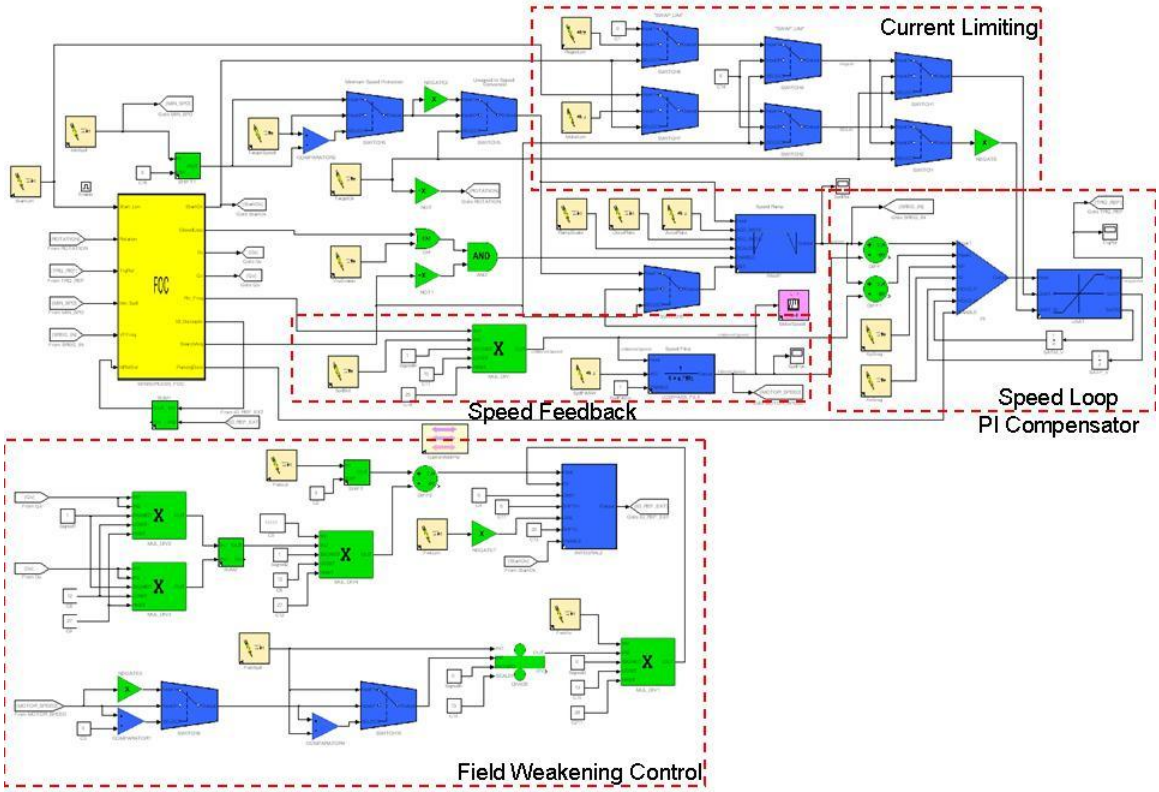


Figure 10—MCE Simulink Design of the Motor Control Loop

4 Using MCEDesigner

As the user continues to work with the IRMCS1271 platform, MCEDesigner will be the primary tool to start, stop, control, and configure the motor. This section gives instructions on how to use select features of MCEDesigner: Creating and modifying functions and Using the trace. It is recommended that the user review the "MCEDesigner User's Guide" section of the IRMCx100 Reference Manual.

4.1 Working with Functions

4.1.1 Creating a New Function

To create a custom function definition, right click on Function Definitions in the tree view and select Create New Function. A properties dialog appears, which allows you to enter a function name and optional description. An example is shown in Figure 11.

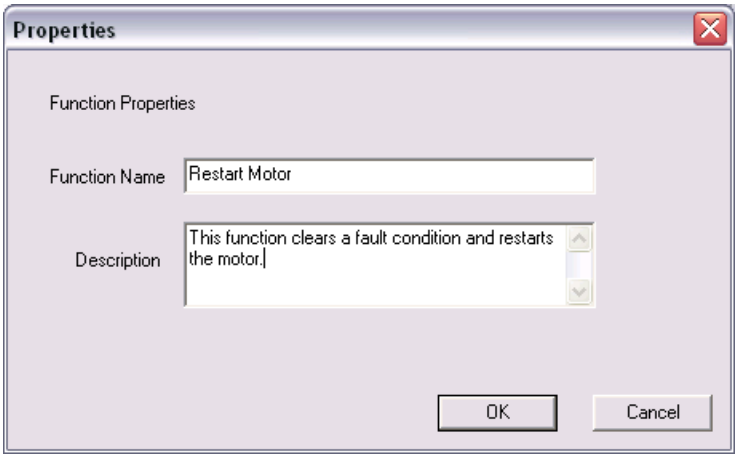


Figure 11—The Function Properties Dialog

Once you have created a new function, you need to define the sequence of operations you want to perform when the function is executed. You can add a delay between register accesses, and you can group operations into a "repeat" loop so a sequence can be repeated some number of times. You can also add a subfunction from the Subfunction Library. (See MCEDesigner User's Guide for more about subfunctions.) Figure 12 shows the right-click menu options for a function.

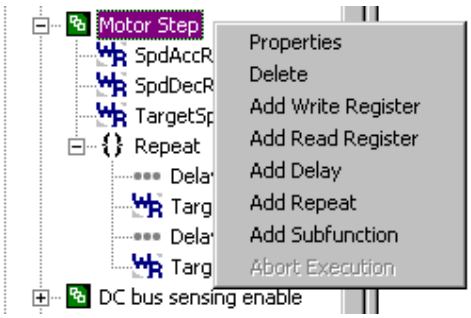


Figure 12—Right-Click Menu for a Function

Add Registers to a Function

To add a register to a function, right click on the function name and select Add Write Register or Add Read Register. A dialog appears that allows you to select a register from a pull-down list and enter an optional description. An example of the dialog is shown in Figure 13. If you selected Add Write Register, the pull-down list includes all the registers defined under Write Registers in the Register Structure Definitions section of the active Component window's tree view. If you selected Add Read Registers, the pull-down list includes all the read registers instead. You can only add read and write registers from the active window. A function cannot contain registers defined in another Component window.

Figure 13—The Add Register to Function Dialog

Add a Delay to a Function

To add a delay in function execution (between register accesses), right click on the function name and select Add Delay. A dialog appears that allows you to specify the delay in milliseconds. An example of the dialog is shown in Figure 14.

Delays can range from 0 to 65,535 msec. The function delay is implemented with a granularity of 10 msec, so the delay you enter must be rounded up to the nearest increment of 10 msec to determine the actual delay. For example, if you enter a delay of 23 msec, your actual delay will be 30 msec. Please note that the function delay is implemented on the host PC, and since the Windows operating systems are not real time, the exact duration of the delay is non-deterministic. You should not depend on the function delay to implement a time-critical sequence of operations.

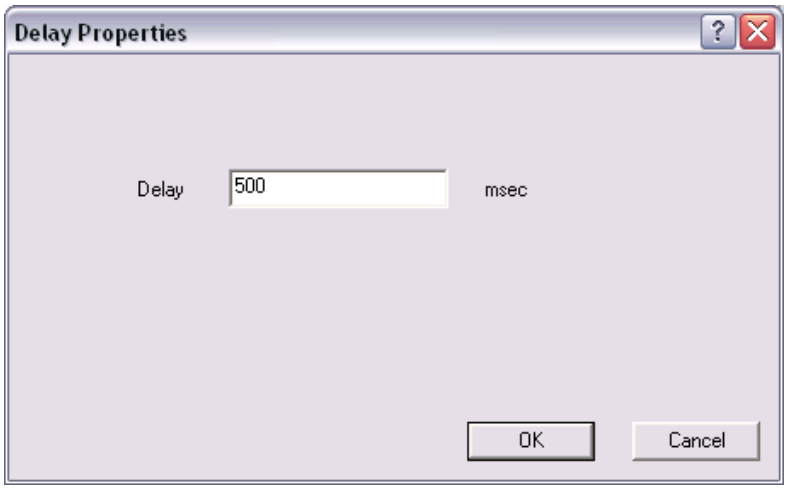


Figure 14—Delay Dialog

Add a Subfunction to a Function
To add a subfunction to a function, right click on the function name and select Add Subfunction. A dialog appears that allows you to select a subfunction from a pull-down list. An example of the dialog is shown in Figure 15. The pull-down list includes all the subfunctions defined under the Subfunction Library Definitions section of the active Component window's tree view. You can only add subfunctions from the active window. A function cannot contain subfunctions defined in another Component window. When you add a subfunction to a function, the entire content of the subfunction is automatically included.

Once you have added the subfunction to your function, you can modify its register values, delay durations and repeat counts, but you can't change the structure of the subfunction (by adding or deleting registers, for example). The subfunction structure can only be modified in the Subfunction Library Definitions section of the tree.

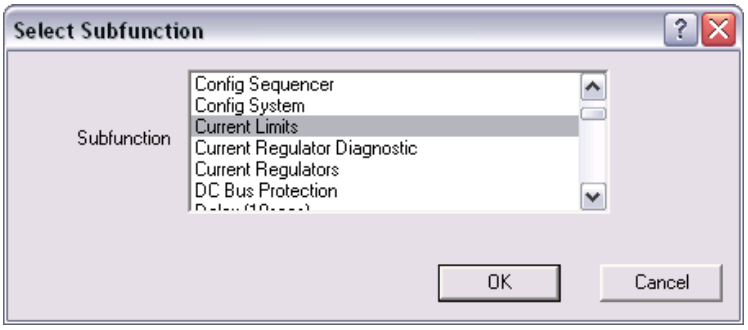


Figure 15—Subfunction Dialog

Add a Repeat to a Function
To add a sequence of operations that you want to execute more than once, you must first add a repeat loop and then add registers, delays, and/or subfunctions to the loop.

To add a repeat loop to a function, right click on the function name and select Add Repeat. A dialog appears (shown in Figure 16) that allows you to specify the repeat count. The count can range from 0 to 4,294,967,295. If you specify a repeat count of 0, the sequence inside the loop is

skipped (not executed at all) when you execute the function. If you want to repeat a sequence "forever," just use a very large repeat count.

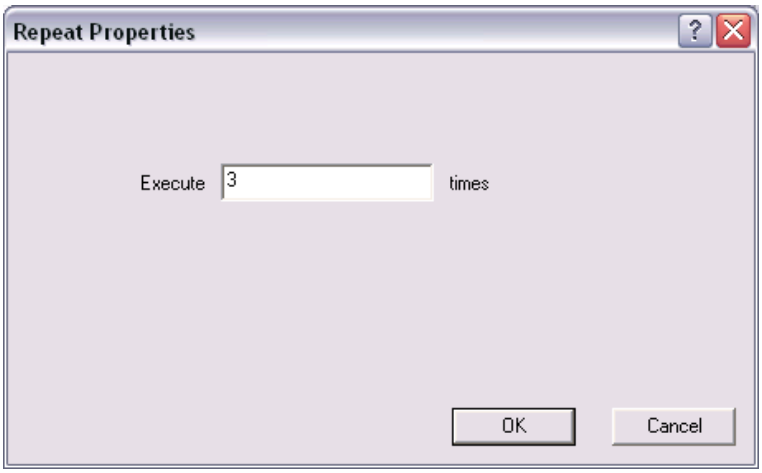


Figure 16—Repeat Dialog

To add a register, delay or subfunction inside a repeat loop, right click on the Repeat icon in the tree view and select Add Write Register, Add Read Register, Add Delay or Add Subfunction. You cannot add another repeat loop inside a repeat loop. Figure 17 shows an example of the tree view display for a function containing a repeat loop. The repeat loop contains two write registers (TargetSpd and SysCtrl) and a delay.

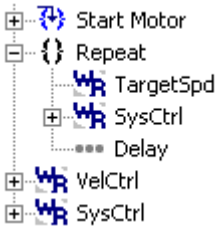


Figure 17—Repeat Loop Example

4.1.2 Modifying Functions

Once you have created a function with an initial sequence of operations, you can modify the following items:

- function name and description
- subfunction descriptions
- register, delay and repeat values

In addition, you can add, insert and delete registers, delays, repeat loops and subfunctions.

Note:

- If you modify a register definition (other than value and description) in the Register Structure Definitions section of the tree view, the change is reflected anywhere you use that register/field in the Function Definitions and Subfunction Library Definitions sections.

- If you delete a register from the Register Structure Definitions section of the tree view, it is also deleted from any functions and subfunctions that reference it.
- If you modify a subfunction definition in the Subfunction Library Definitions section of the tree view, the change is reflected anywhere you use that subfunction in the Function Definitions section.
- If you delete a subfunction from the Subfunction Library Definitions section of the tree view, it is also deleted from any functions that reference it.

Modify a Function Definition
To modify a function definition, right click on the function name and select Properties. The dialog shown in Figure 11 appears so you can modify the function name and description.

Modify a Subfunction Description
To modify a subfunction description, right click on the subfunction name and select Properties. The dialog that appears (shown in Figure 18) allows you to modify only the subfunction description.

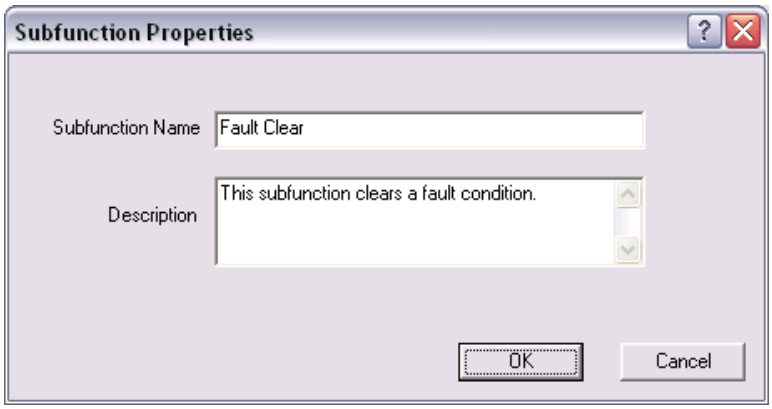


Figure 18—Subfunction Properties in a Function

Modify Function Register Properties
Inside a function definition, or inside a subfunction within a function definition, you can modify the value of a register or the register's description, or set advanced properties for the register. (For a read register, you can only modify the register's description.) To modify register properties, right click on the register name and select Properties. The dialog that appears (shown in Figure 19) allows you to modify only the register value and the description, which are specific to the function definition. The register name and offset can only be modified in the Register Structure Definitions section of the tree view.

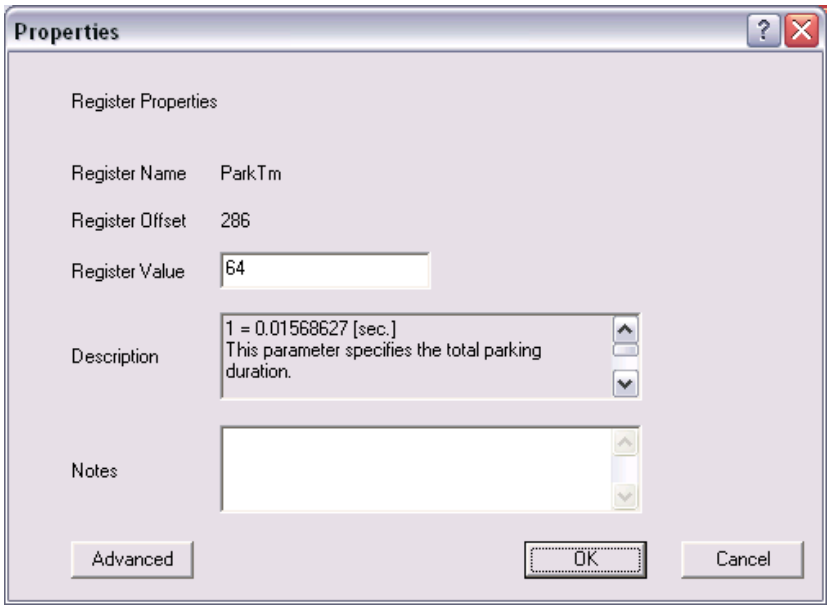


Figure 19—The Register Properties Dialog for a Function

To modify the register's advanced properties (for a write register only), click the Advanced button in the Properties dialog. The Advanced Register Properties dialog shown in Figure 20 appears. The options that can be set in this dialog are used when a register in a function or subfunction should not be modified under normal circumstances. If the "Do not update on Import Drive Parameters" checkbox is checked, the register value, for this particular use of the register, is not modified when the Import Drive Parameters operation is performed, even if the imported text file specifies a value for the register. If the "Show warning when value is modified" checkbox is checked, a warning message is displayed when you click OK after modifying the value of the register in the Properties dialog. (The message is only a warning and does not prevent the register value from being modified.)

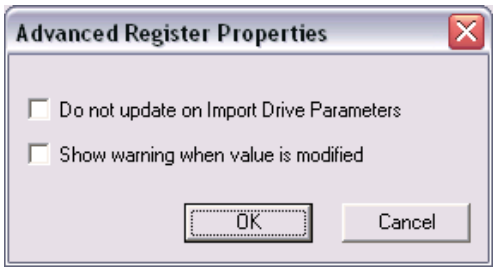


Figure 20—Advanced Register Properties Dialog

Modify the Sequence of Operations in a Function

To modify the sequence of operations within the main part of a function or in a repeat loop, you can add an item to the end of the sequence, insert an item above an exiting entry, delete an item, or drag an item to a new location. Figure 21 shows the right-click menu for a register inside a function.

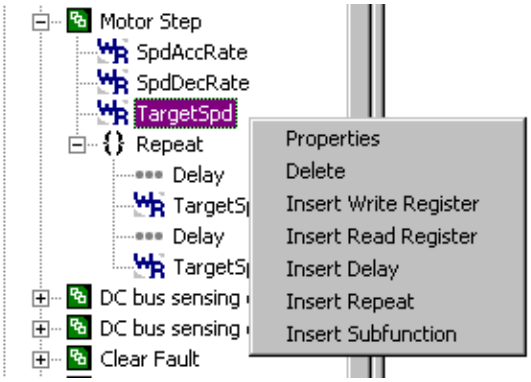


Figure 21—Right-Click Menu for a Register in a Function

For the main function sequence:

- To add an item at the end of a function sequence, right click on the function name and select Add Write Register, Add Read Register, Add Delay, Add Repeat or Add Subfunction.
- To insert an item before (above) an existing item in a function sequence, right click on the existing item and select Insert Write Register, Insert Read Register, Insert Delay, Insert Repeat or Insert Subfunction.
- To remove an item from a function sequence, right click on the item you want to remove and select Delete.
- To move an item within a function sequence, click and hold your left mouse button and move the mouse to drag the item. With the cursor positioned over another item in the list, release the mouse button and the item you're dragging will drop into the location *following* the item you dropped it on. To move an item to the beginning of the list, drop it over the function name. You cannot drag an item into another function, into a repeat loop or into another window.

For a repeat loop:

- To add an item at the end of a repeat loop sequence, right click on the Repeat icon and select Add Write Register, Add Read Register or Add Delay.
- To insert an item before an existing item in a repeat loop, right click on the existing item and select Insert Write Register, Insert Read Register or Insert Delay.
- To remove an item from a repeat loop, right click on the item you want to remove and select Delete.
- To move an item within a repeat loop, click and hold your left mouse button and move the mouse to drag the item. With the cursor positioned over another item in the loop, release the mouse button and the item you're dragging will drop into the location *following* the item you dropped it on. To move an item to the beginning of the loop, drop it over the "Repeat" entry. You cannot drag an item out of a repeat loop.

Note: As shown in Figure 22, when you right click on a Repeat icon, Add, Insert and Delete options are available. The Add options add items *inside* the repeat loop (after the last existing item), while the Insert options add items *before* (outside) the loop. The Delete option removes the entire repeat loop.

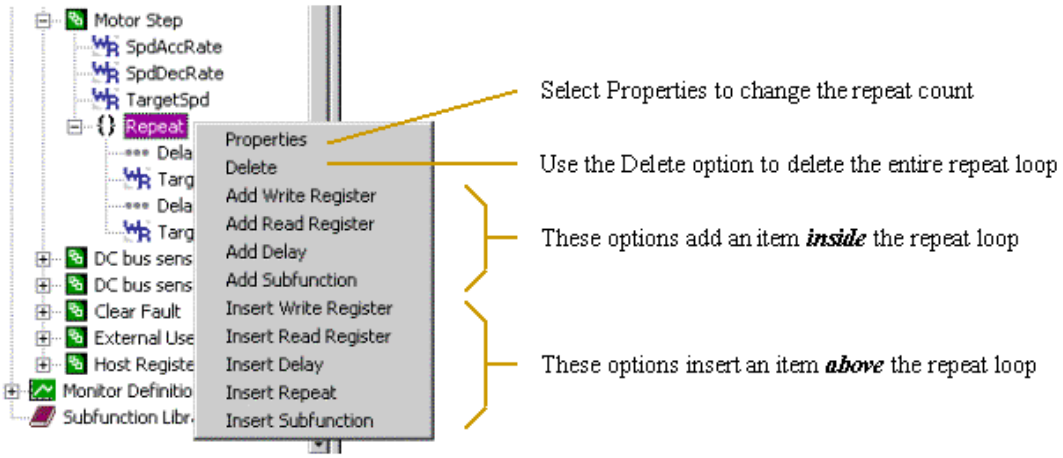


Figure 22—Right-Click Menu for a Repeat

Delete a Function
To delete an entire custom function definition, right click on the function name and select Delete. The built-in functions cannot be deleted.

Reorder Functions
You can reorder your function definitions using drag-and-drop. To do this, click on the name of the function you want to move. Holding the left mouse button down, move the mouse to drag the function. With the cursor positioned over another function name, release the mouse button and the function you're dragging will drop into the location *following* the function you dropped it on. To move a function to the beginning of the list, drop it over the "Function Definitions" entry. You cannot move a function out of the Function Definitions section of the tree view or into another window.

4.1.3 Executing Functions

To execute a custom function, first click on the function name in the tree view. The right pane of the main window shows all the registers defined for the function. The column labeled Value to Write shows the value that will be written for each register when the function is executed. (For a read register, the column shows "N/A".) To modify the value to be written for a register, right click on the register name beneath the function in the tree and select Properties. Modify the register value (see Figure 19) and click OK. When all registers show the correct values for your function, double click the function name in the tree view to execute the function.

4.2 Using the Monitor

The IRMCx series reference designs include a diagnostic trace feature. MCEDesigner provides a method of capturing data from the trace buffer, saving it to a text file and displaying it in graphic form. The text file can be imported into any standard spreadsheet program, such as Microsoft Excel, for evaluation of the data. To use the trace, click on the System window and then click the Monitor Definitions section of the tree view.

When the trace is activated, data is sampled from any two of 16 different data sources and stored in a buffer in RAM on the IRMCx100 Series IC. The buffer holds 256 samples for each of the two

selected data sources. When the trace begins and terminates depends on the mode selected and the user-configurable "trigger locator", as follows:

- In **Force Trigger** mode the trace terminates when the buffer is filled.
- In **Trigger on Level** mode the trace terminates after a configurable trigger condition is met. When the trace buffer is filled, the buffer "wraps" and old samples are overwritten as new samples are acquired. After either the trigger condition or a fault condition occurs, up to 256 additional samples are collected, depending on the configured trigger location and product type.
- In **Trigger on Fault** mode the trace terminates only after a fault condition occurs. When the fault occurs, up to 256 additional samples are collected, depending on the configured trigger location and product type.
- In **Auto Repeat Level** mode the trace terminates as described for Trigger on Level mode, above, but the trigger is rearmed as soon as the trace data is collected and the sequence (arm, trigger, collect, display) continually repeats.

In any of the four modes, when data sampling is complete, MCEDesigner collects the complete buffer of data from the IRMCx100 IC, stores the samples in the specified text file for evaluation, and displays a graph of the collected data. In the text file, the samples are stored two per line; on each line, the first column is the sample from channel 1 and the second column is the sample from channel 2. The first line in the file contains the oldest samples, and the last line contains the last samples collected. In Auto Repeat Level mode, the file is overwritten each time the trigger occurs.

4.2.1 Trace Setup

Before you can capture trace data, you must define the parameters for the trace function. MCEDesigner allows you to create and save any number of trace definitions so you can reuse them as needed during your development. The default configuration files include an example trace definition named Monitor Setup.

To create a new trace definition, right click on Monitor Definitions in the tree view and select Create New Monitor Group. An example of the Properties dialog, which allows you to enter a monitor group name and description, is shown in Figure 23.

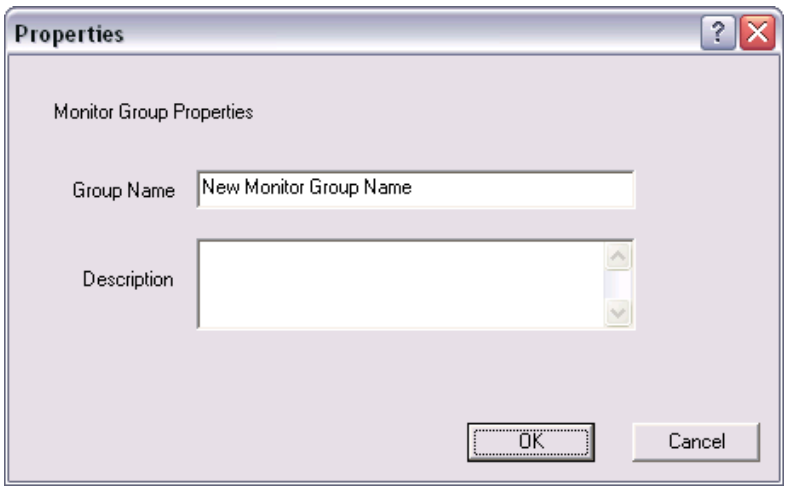


Figure 23—Monitor Group Properties Dialog

After you have created a new monitor group, right click on the monitor group name in the tree view and select Define Trace. An example of the Trace Setup dialog is shown in Figure 24.

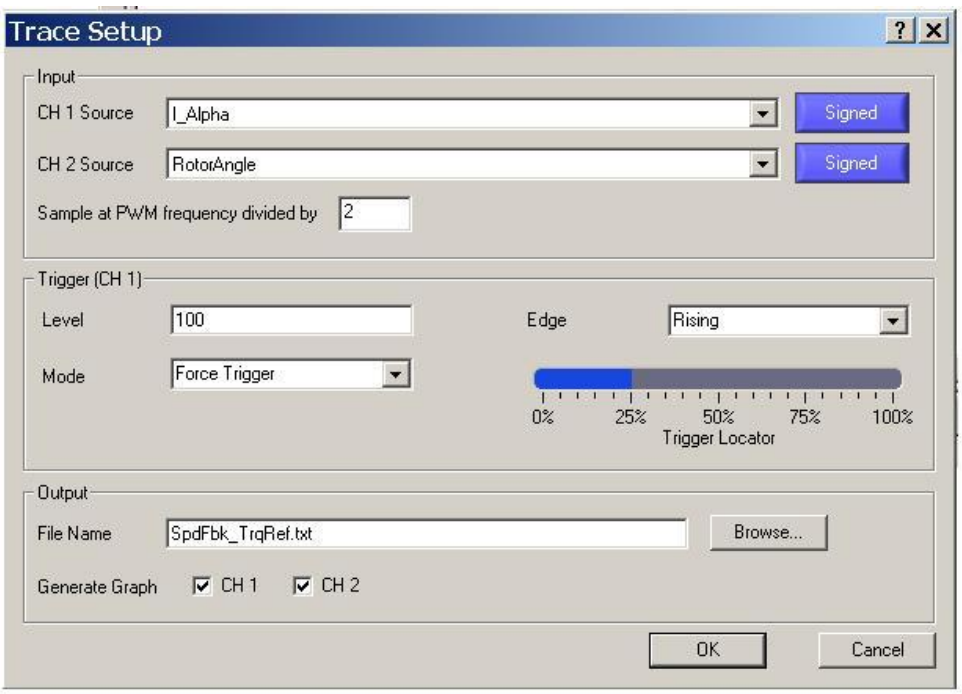


Figure 24—Trace Definition Dialog

The Trace Definition dialog is divided into three sections: Input, Trigger and Output. The Input section allows you to define a data item to be captured for each of two channels. For each channel, choose an item from the pull-down menu. Click the button to the right of the pull-down menu to specify whether MCEDesigner should interpret the data as signed or unsigned for the purposes of trigger level comparison and data plotting. The “Sample at PWM frequency divided by” field allows you to select how often data will be sampled and stored in the trace buffer. The sampling rate is always based on the PWM frequency of the data item selected for channel 1. If you enter 1 in the “divide by” field, data for both channels is sampled on every PWM cycle. If you enter 2, the data is sampled on every other cycle, and so on. The maximum value for this field is 255.

The Trigger section of the Trace Definition dialog allows you to select a capture mode for the trace. Four Mode options can be selected from a pull-down list. They are Force Trigger, Trigger on Level, Trigger on Fault and Auto Repeat Level. The Level and Edge entries in the Trigger section are used only in Trigger on Level and Auto Repeat Level modes and are used to define the trigger event. The trigger is always associated with the data source selected for channel 1, and occurs when that data value passes through a particular level in either the rising or falling direction. The Level entry specifies the level at which the trigger is to occur and the Edge entry specifies whether the trigger is to occur when the level is rising or falling. Enter an integer value for Level. The pull-down list associated with the Edge entry allows you to specify whether the trigger should occur on the rising or falling edge of the channel 1 data source.

The slider bar labeled “Trigger Locator” in the Trigger section of the dialog allows you to configure the percentage of the trace buffer used for samples collected prior to the trigger point. For example, if you set the Trigger Locator to 25%, the trace data will consist of 64 samples preceding the trigger and 192 samples following the trigger. The trigger location is marked on the

graphical display, but not in the trace data file. The Trigger Locator is not used in Force Trigger mode. (In Force Trigger mode, the buffer simply contains 256 samples collected after the trace was activated, and the trigger location is not marked on the graphical display.)

The Output section of the dialog allows you to select processing options for the captured data. Data is always written to a text file and can be optionally displayed in graphical form. The File Name entry specifies the file to which the captured data is to be written. You can type the pathname of a file or click the Browse button to browse for the desired output file. If you want the data to be plotted, be sure to click the CH1 and/or CH2 checkboxes.

When you are finished defining the trace, click OK to save your setup. This does not activate the trace.

4.2.2 Activate the Trace

Once you have created a new monitor group and set up the trace parameters for the monitor group, you'll see two new entries in the Monitor Definitions section of the tree view. Under Monitor Definitions, there's an entry with the name you assigned to your monitor group. Under your monitor group, there's an entry named Trace Setup. (The Trace Setup entry is not present until you've clicked OK in the Trace Definition dialog.) When you are ready to activate the trace, double click the Trace Setup entry in the tree view.

When the trace buffer is full (in Force Trigger mode) or the trigger event or fault occurs, MCEDesigner reads the entire trace buffer from the IRMCx100 IC and writes the collected data to the output file. After the data has been saved to the output file, a Trace Results dialog showing the data in graphical form is displayed (if configured). An example is shown in Figure 25. In Auto Repeat Level mode, the progress bar is not displayed, and the Trace Results dialog is updated each time the trace triggers and the trace data has been collected.

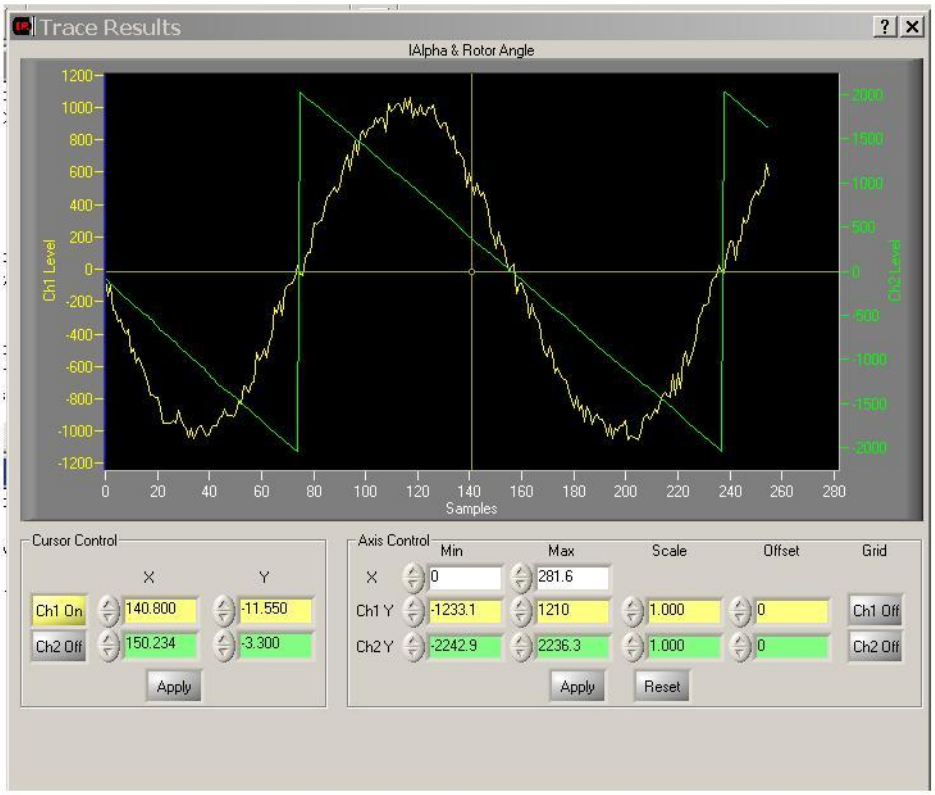


Figure 25—Trace Results Dialog

4.2.3 Analyze the Trace Results

On the graph, data samples are displayed along the X axis. The Y axis on the left shows the level of the selected data item for Channel 1. The Y axis on the right shows the level for Channel 2. Channel 1 data is shown in yellow and Channel 2 data is shown in green. A vertical line in blue shows the point at which the trigger occurred.

At the bottom of the Trace Results dialog are a number of features that allow you to control the appearance of the plotted data. You can adjust the ranges of the X and Y axes, display cursors for locating precise data points, and display a background grid to aid in evaluating the data.

Important Note

When the Trace Results dialog is displayed, the settings that adjust the appearance of the data are initialized to the same values you entered last time you used the dialog. If the ranges of the data are different this time, the plot might not be readable with these settings. To quickly adjust the display to show all the data, simply click the Reset button in the Axis Control section of the dialog. (See Section 0 for more information.)

X-Axis Scaling
The X-axis of the Trace Results graph is labeled Samples. The timing of the samples is based on the Trace Definition input, "Sample at PWM frequency divided by" (N) and the actual PWM frequency. The time scaling is calculated as follows:

1 Sample Period = N / PWM_Freq

For example, each if N = 50 and the PWM frequency is 20kHz, then a trace data sample is taken every 2.5ms and the entire trace graph represents 2.56 seconds.

Cursor Control
The Ch1 On/Off button in the Cursor Control box turns on or off a yellow **cursor** for the Channel 1 data. The Ch2 On/Off button selects a green cursor for Channel 2. The cursors are both off by default. In the example shown in Figure 25, the Channel 1 cursor is off and the Channel 2 cursor has been turned on.

The cursors are crosshairs that can be used to exactly determine or locate any position on the graph. With either cursor, you can click and drag either the vertical or horizontal line, or drag both (horizontal and vertical) at the same time by clicking on the intersection of the lines. The edit boxes in the Cursor Control box show the current location of the cursor as you move it. The yellow boxes show the position of the Channel 1 cursor and the green boxes show the position of the Channel 2 cursor. To set the cursor to an exact location, you can type in the edit boxes (or use the up/down arrows) and then click the Apply button in the Cursor Control box.

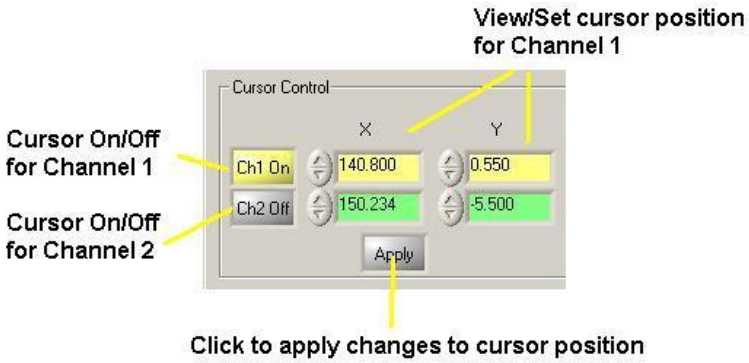


Figure 26—Trace Results Cursor Controls

Axis Control
You can adjust the **ranges** of the X and Y axes by typing (or using the up/down arrows) in the Min and Max edit boxes in the Axis Control section of the dialog. The white boxes control the range of the X axis. The yellow and green boxes control the ranges of the Channel 1 and Channel 2 Y axes, respectively. This feature can be used to "zoom in" on a particular area of the plot. Click Apply in the Axis Control box to redraw the graph with the new ranges.

You can change the **scale** of data by entering a multiplication factor in the yellow and green (Channel 1 and Channel 2) Scale edit boxes. For example, a scale factor of 2.00 in the yellow (Channel 1) Scale box multiplies all Channel 1 data values by 2. Click Apply in the Axis Control box to redraw the graph with the new scaling.

You can add an **offset** to the data values by entering an offset in the yellow and green (Channel 1 and Channel 2) Offset edit boxes. For example, an offset of 500 in the yellow (Channel 1) Scale box adds 500 to all Channel 1 data values. Click Apply in the Axis Control box to redraw the graph with the new offsets.

Note: If you supply both a scale factor and an offset, the scaling is applied *before* the offset is added.

The Ch1 On/Off button in the Axis Control box turns on or off a red background **grid** for the Channel 1 data. The Ch2 On/Off button selects a red grid for Channel 2. You cannot turn both grids on at the same time. In the example shown in Figure 25, the Channel 1 grid has been turned on.

The Reset button in the Axis Control box resets all axes to automatically calculated ranges with scales set to 1.0 and offsets of 0. MCEDesigner selects these settings to best display all data points on the plot.

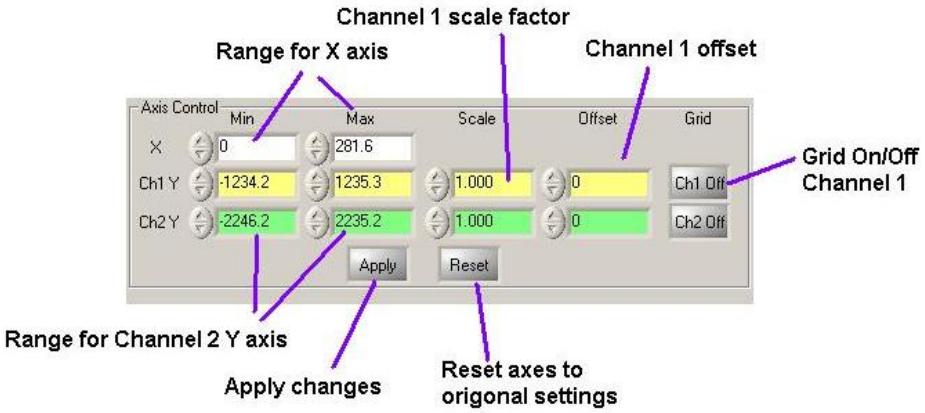


Figure 27—Trace Results Axis Controls

4.2.4 AutoRepeat

In Auto Repeat Level mode, a Pause button at the bottom of the Trace Results dialog allows you to pause the auto repeat feature. When you click the Pause button, it changes to a Resume button, as shown in Figure 28. Once you've clicked the Pause button, the graph won't be updated to display new trace data until you click the Resume button. The Pause/Resume button is only displayed in Auto Repeat Level mode.

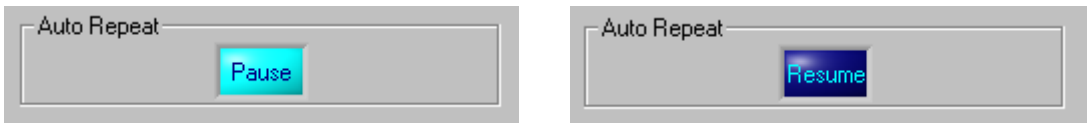


Figure 28—Trace Results Auto Repeat Control

4.2.5 Aborting a Trace Operation

You can create multiple monitor groups, each with its own trace definition, but you can only activate one trace at a time. If you've activated a trace and it has not yet triggered, you can abort the operation by right clicking on *any* Trace Definition entry in the tree view and selecting Abort Trace. If the trace has already triggered and data collection/output is in progress, you must wait for the operation to complete before you can start another trace.

In Auto Repeat Level mode, the trace sequence (arm, trigger, collect, display) is repeated as long as the Trace Results dialog is displayed (although you can pause the sequence as described in Section 4.2.4). When you exit the Trace Results dialog, the Auto Repeat sequence terminates.