

USC700 Programmer User Guide

Introduction

About USC700 Programmer

The USC700 Programmer is a 32-bit Windows application used to configure the USC701 for use as a signal-conditioning module via the COA703 that provides an isolated connection to a personal computer USB port.

USC700 Programmer contains a "New Module Wizard", this wizard is used to create a USC program (module definition). Most common signal conditioning module types can easily be made using the new module wizard.

Previously made module definitions are stored in a single data base file on your PC and can be opened, closed and copied using the file/open new menu. Module definitions can also be exchanged between systems over the Internet or on floppy disk using the "File/Import/Export" commands.

USC700 Programmer has pre-made linearising tables that are selected automatically when measuring temperatures from thermocouples and RTD's however custom tables can easily be created and stored within the one USC Program.

The equation editor is used to customise the functionality of the USC701. Math functions, branching commands and control of when the equation is run create a powerful and versatile tool. While the equation editor creates complex functions usually there are one or two variables within the equation that you wish the end user to have easy access to. This is accomplished by creating a configurable user interface.

About the USC701

The Universal Signal Conditioner USC701 is a programmable signal conditioner that combines measurement and control functions in a single instrument and provides user selectable solutions for most signal conditioning applications.

The USC has the following functional blocks:

- Two analogue inputs that can be programmed to measure ac/dc current or voltage, resistance, potentiometer, strain gauge and temperature
- Two pulse (digital) inputs that can be programmed to measure frequency or phase and also be used as control inputs to perform hold, reset and enable.
- One isolated proportional output that can be configured as an analogue output or a pulse output.
- Two relay outputs.
- Excitation outputs.
- Digital (bus) outputs.
- Calibration & set point variables.
- Arithmetic function variables.

The configuration can be set up and altered by:

- The plug-on Access Module AM702 which has a 2 line by 8 character display and 5 push buttons and is menu structured for the selection of input/output/alarm variables and selection of calibration information
- The plug on Computer Adaptor COA703 that interfaces to the users PC that is loaded with the USC700 Programmer software. This Software offers a user-friendly interface for the configuration and setting of a USC.
- A Select switch and encoder are accessible on the top of the USC that allow the user to alter a number of calibration parameters: Output zero & span and relay set points.

About the AM702 Access Module

The plug-on Access Module AM702 is menu structured for the selection of input/output/alarm variables and selection of calibration information. The AM702 has four operational modes.

AM702 Run Mode

In run mode one selected variable will be continuously displayed.

- To change the number of significant digits displayed press the 'BACK' and 'NEXT' keys.
- To display the name of the program (Identity) stored in the USC701 press the two options buttons at the same time.
- To display the revision of the AM702 program press the  and  buttons at the same time.

AM702 Configure Mode

The AM702 allows the user to perform changes to the configuration of the USC. These changes include:

- The measurement type or function,
- The analogue output type,
- Ranging of the input and output,
- Setting the relay action, trip points and delays
- Setting of sample times and hold functions

AM702 Trim Mode

Trim Mode is used to change selected parameters for each previously programmed input, output and equation constant to an operating USC701 without disruption to output function. The "Trim Mode" can be used in conjunction with the "USC700 Programmer" program to create a customised user interface for each USC701.

AM702 Hot Key Mode

When Hot Keys have been enabled the USC701 will run alternate custom program branches that were written using "USC700 Programmer". Any combination of the first four AM702 keys can have a custom branch added to the USC701 program.

When hot keys are enabled the AM702 run mode commands are disabled. Hot Keys can be used in conjunction with the "USC700 Programmer" to create a customised user interface for each USC701.

AM702 Display only Function

When "Trim Mode", "Configuration Mode" and "Hot Keys" have been disabled by the "USC700 Programmer" PC program all functionality of the AM702 is disabled except for the run mode commands.

About the COA703 Computer Adaptor

The COA703 is a piece of hardware that clips on to the top of a USC701 to enable an isolated connection between a USC701 and the USB port on a personal computer.

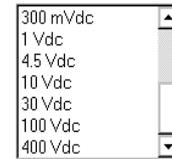
The USC Wizard and User Input

Take the time to read this before running the walk through. It may be illuminating.

Rule 1

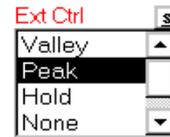
A Red caption over a list box without an accompanying button always indicates that an item in the list needs to be selected.

Channel 1



Rule 2

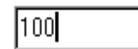
If a small command button appears over a list box with a red caption then additional settings are required to implement the selected option; pressing the small command button accesses these options.



Rule 3

A red caption over a text box indicates that the entered information is outside the limits or is the incorrect data type.

Min Value



Rule 4

Whenever an error occurs the wizard will stop on the faulty step and display a red caption over the problem. The helper box on the bottom of the screen will also display the problem in plain text.

Rule 5

The wizard only saves entered information as the user presses 'Next'. Changing information and then pressing 'Back' will lose the last entries

Communication and Operation Problems

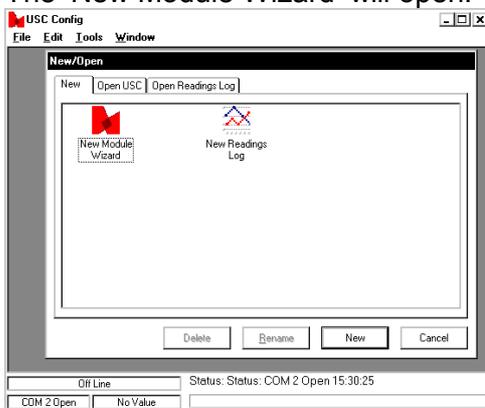
Problem	Solution
Power LED Blinks Periodically	<ol style="list-style-type: none">1. Check for correct supply voltage.2. Disconnect the input connector (not the power connector).3. Apply power to the USC701, if the power LED now remains permanently lit there is probably too much load on the "EXCIT" output for the voltage it has been programmed for, reprogram to a lower voltage or change load.
The Run LED Is Not Flashing	<ol style="list-style-type: none">1. Switch off supply and reapply, check if unit now operates.2. Check that suitable spike suppression is fitted across contactor coils operating from same power source.
Run LED only flashes for 5 seconds after switch on	<p>It is possible to create equations that will lock up the USC701. The lock can occur due to an improper equation or an unhappy circumstance that occurs during programming (The equation is in use while it is being changed). To return the USC701 to operating state a short simple program is downloaded during the power up period.</p> <ol style="list-style-type: none">1. Using the new module wizard make a signal isolator with one input one output and no relays.2. On the "Tools" menu select "Disable Protocol Interlocks"3. Connect the computer adaptor to the USC701 and switch off the power to the unit.4. Press program.5. Switch on the power to the USC701; the program should start downloading after a second.6. If process is unsuccessful power down the USC701 and try again.7. After the USC701 has recovered try suspect program again it may be successful or rewrite using an alternate structure.

My First USC701 Program

The computer adapter COA703 clips on to the top of a USC701 to provide an isolated connection between it and your personal computer. **NEVER** connect directly between any APCS signal conditioning module and the serial or USB port on your PC

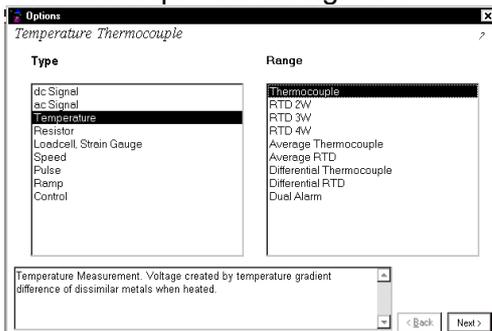
Start the USC700 Programmer

1. Start the program using the icon created when the program was installed.
2. After the start up screen and messages the program opens, as shown.
3. Double click on 'New Module Wizard' or, press the 'New' command button while 'New Module Wizard' is selected.
4. The 'New Module Wizard' will open.



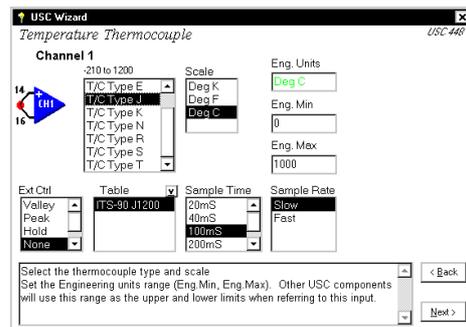
Running the New Module Wizard

1. Select a signal type in the left list, then select a range in the right list.
2. Select 'Temperature' on the left list, then 'Thermocouple' on the right list

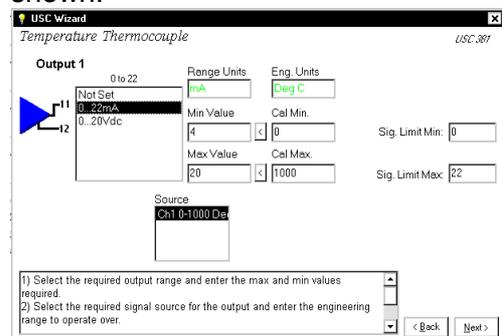


3. After selecting Thermocouple on the right list the range label will change from red, to the menu text colour. The 'Next' button will also have gone from disabled to enabled. Click 'Next' or press Alt + N

4. Select from the list the required input thermocouple type. When a range is selected the default properties for that range will be entered.

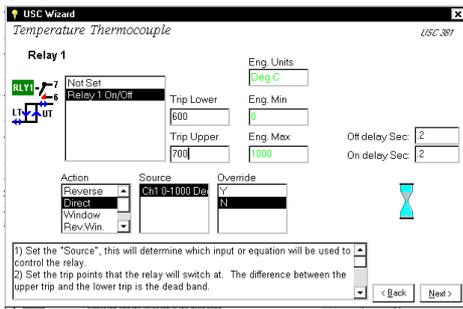


5. Leave entries in their default state, it is possible to change other settings at this stage but for reasons of clarity we will continue as they are. Click 'Next' or press Alt + N.
6. Click on the 22mA range, and the properties for the range will display as shown.



- The selected source is 'Ch1 0-1000 Deg C' the output from terminals 11 and 12 will be 4mA at 0°C and 20mA at 1000°C. The signal limiting is set outside the range unit's column and will therefore have no effect.

- Click 'Next' or press Alt + N. Select "Relay 1 On/Off" and enter the upper trip and lower trip points as 600 and 700 as shown.

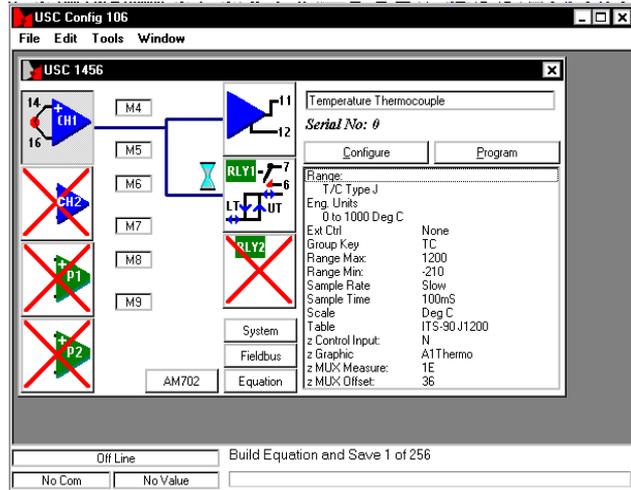


- With the relay set to direct action as shown it will switch on at 700 Deg C after the on delay (currently set at 200mSec) and will switch off at 600 Deg C after the off delay (currently set at 200mSec).

- Click 'Next' or press Alt + N. Select the System range. The properties displayed are various left over variables and are not specific to a particular section of the USC.
- Click 'Next' or press Alt + N. A message will appear regarding a 'Verify Wizard'. The "Verify Wizard" will always run at the end. Click 'Close' or press Alt + C the "Verify Wizard" will run and close to the screen shown next page.

The USC Properties Screen

- The first column of buttons represents the USC inputs.
- The second column of labels M4 to M9 represents the equation memory outputs; by moving the mouse pointer over them the expected output range will appear as a tool tip.
- The third column of pictorial buttons represents the USC outputs.

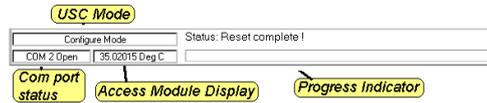


As the CH1 button is pressed the list box on the right is displaying CH1 properties. CH2, P1, P2 and RLY2 are all displayed as disabled, as they have not been programmed for use. The text box at the top is displaying 'Temperature Thermocouple', created from the previous selections made.

Program the Connected USC

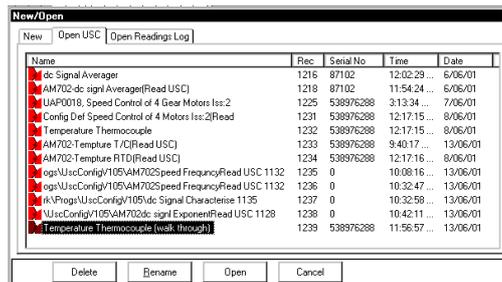
Connect the COA703 to the USC701 and press program. It takes between thirty seconds and two minutes to program a USC.

- To successfully program a USC the 'SELECT' switch on the top of the USC must be in position '0'.



Open an Existing USC Program

- If the 'New/Open' dialogue is not open then start it by clicking 'Open/New' on the 'File' menu.
- Select the 'Open USC' Tab as shown below and scroll down to the required existing file to open. The list can be resized and sorted by Name, Rec, Serial, Time and Date. The time and date will change every time the file is accessed.



- After selecting the required file the 'USC Properties' screen will open.

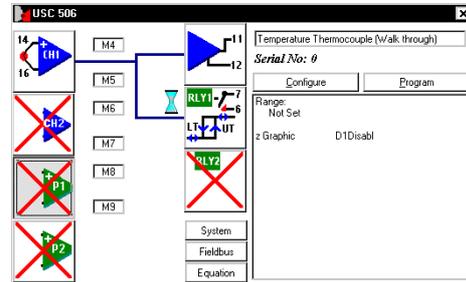
Extending My First Program

Adding Functions to an Existing USC Program

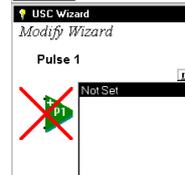
The following procedure will add a frequency measurement to input P1 and set RLY2 to switch on 50 m L/Sec and off at 45 m L/Sec of product flow.

The input will be scaled such that 0 to 20 Hz = 0 to 100 m L/Sec

1. Click the button representing P1 on the 'USC Properties Screen'. The list box on the right will indicate that P1 is currently 'NotSet'.



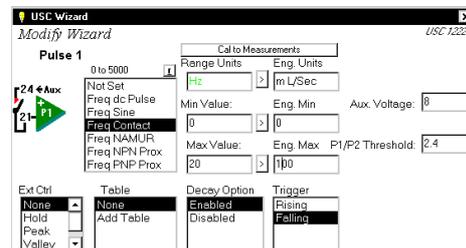
2. Click the 'Configure' button and the 'Modify Wizard' will step through to the pulse 1 range setting.



3. Click on the small 'r' button above the list to add the frequency ranges to the list box.
4. Select type 'Measurement' then range 'Frequency'. Press next to close the change range wizard.
5. Click on 'Freq Contact' to display the properties for setting pulse 1.

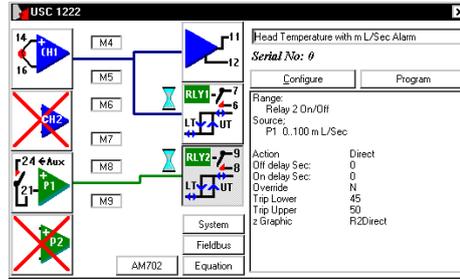


6. Enter 0 and 20 in the 'Range Units' column.
7. Enter 'm L/Sec', 0 and 100 in the 'Eng.Units' column.



8. Click 'Next' then 'Finish', the 'Modify Wizard' will finish and run the 'Verify Wizard' before the new properties will be displayed for pulse 1 on the 'USC Properties Screen'.
9. Click the button representing RLY2 on the 'USC Properties Screen'. The list box on the right will indicate that RLY2 is currently on range 'NotSet'.
10. Click the 'Configure' button and the 'Modify Wizard' will step through to the Relay 2 range setting.
11. Click on the small 'r' button above the list to add the other ranges to the list box.
12. Click on 'Relay 2 On/Off' to display the properties for setting relay 2.
13. Select the measurement source on the source list box, 'P1 0-100 m L/Sec'.
14. Select the trip action 'Direct'.
15. Enter the lower and upper points, 45 and 50.
16. Click 'Next' then 'Finish', the 'Modify Wizard' will finish and run the 'Verify Wizard' before the new properties will be displayed for pulse 1 on the 'USC Properties Screen'.

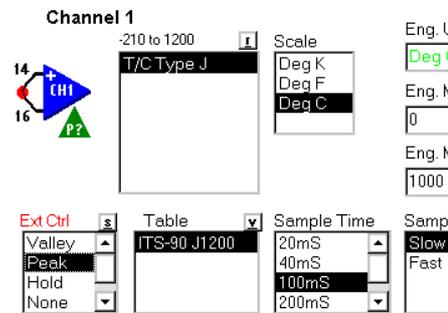
17. Edit the program name to represent something more meaningful enter, 'Head Temperature with m L/Sec Alarm'.



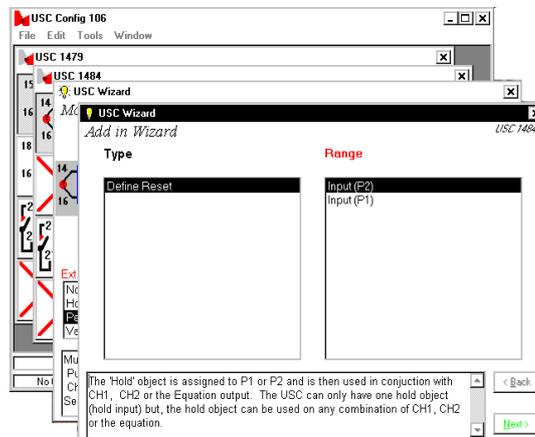
Adding Peak Measurement Function to CH1

The following procedure will take the 'Head Temperature with m L/Sec Alarm' program example and add a peak measurement function to CH1. This function could have been added at any time in any of the previous example steps, it has been separated here for clarity reasons only.

1. Open the required 'USC Properties Screen'.
2. Click on the CH1 button to show channel 1 properties in the list box.
3. Click the 'Configure' button and the 'Modify Wizard' will step through to the Channel 1 range setting.
4. Select 'Peak' in the 'Ext. Ctrl' list box as shown.
5. Click on the small 's' button above the list to add a 'Reset' input.



6. The 'Add in Wizard' will suggest using P2. Experiment, select P1 on the range list and take note of the error messages. The wizard will allow you to overwrite the function of P1 however the existing function will be lost. Select P2 and press 'Next'.
7. Select contact input set the trigger to required state when the 'Reset' function is active.



8. Press 'Next' then press 'Close' to close the 'Add In Wizard' and return to the 'Modify Wizard'
9. Click 'Finish', the 'Modify Wizard' will finish and run the 'Verify Wizard' before the new properties will be displayed for channel 1

Reading Programs out of the USC701

1. Connect the COA701 on to the top of the USC701.
2. On the USC700 Programmer file menu select "File", "Read Program from USC".

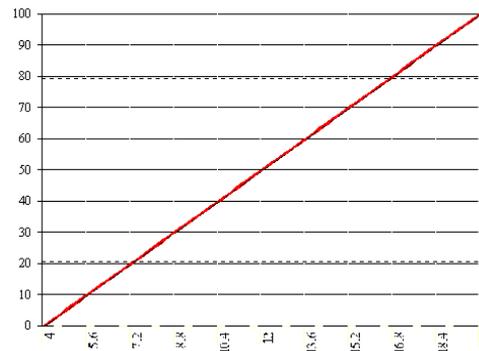
Using Look-Up Tables

About Look-Up Tables

The standard signal conditioner will produce a linear output in response to a linear input. This means that when plotting output against input on a line chart the result would be a straight line.

The USC700 Programmer program has output on the vertical axis and input on the horizontal axis.

Many measurement applications require the input output response to be tailored to correct errors in the linearity of the measured signal as in the following 'Shaft Angle Example'.



Shaft Angle Example

A potentiometer is used to measure an aperture opening from 0 to 100%. As the potentiometer is connected using long wires to the signal-conditioning module in a noisy factory environment the current flowing through the potentiometer (connected as a two-wire series resistance) is measured instead of the voltage.

PROBLEM

- This method of measurement is highly immune to noise but has a non-linear response.
- The aperture measurement of 0 to 100% does not use the full range of the potentiometer

Steps to Solving the Problem

- Run the 'New Module Wizard' ('dc Signal', 'Isolator', CH1 '30mA', Output 1 '0-22mA', Relay 1 'Not Set', 'System') and configure an input channel to measure input current and set the auxiliary voltage output to a level suitable for driving the potentiometer. (I used a 5k ohm potentiometer with an extra 150 ohm series resistor the auxiliary set to 8Vdc and CH1 set to the 30mA dc range)
- Set the USC to display the input channel (CH1) on the ACCESS module. On the USC Properties Screen change the program name from 'dc Signal Isolator' to 'Shaft Angle Example' and program the USC.
- Connect the USC into the system to be linearised.
- Move the aperture to different points and record the measurement displayed as shown in following table.
- Create a look-up table from measured points to linearise the measurement.
- Update the engineering unit for the new measured output.

Create A Look-Up Table From Measured Points

The following procedure will add a look-up table to CH1 where CH1 is currently set up as 30mA input. CH1 is connected to an external device (Aperture position) with measurements as shown.

Measured mA on CH1	Aperture position %
1.811	0
2.257	25
2.941	50
3.865	75
6.075	100

1. Open the 'USC Properties Screen' for the module definition to be changed.
2. Click the CH1 icon then the 'Configure' button. The 'Modify Wizard' will step through to the current CH1 range setting.
3. In the 'Table' list click on 'Add table'. Click the small 'u' button after it appears above the table list.
4. On the 'Enter Points for Table' screen click 'Create Table From Points'.
5. In the description box enter 'Aperture Position'.
6. Enter the known points on the grid.
7. After entering the points then press 'Ok' to create the table.

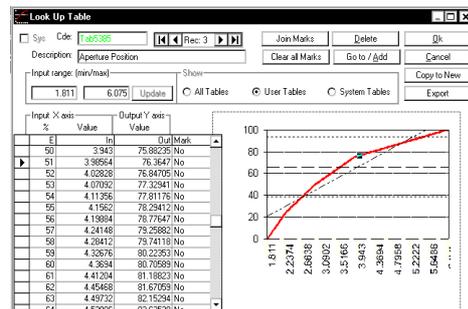
Two points will create a straight line.
Entries are sorted by the input value.

The "Create Table From Points" option will open with the values used to create the last table.
To add rows enter data in the last (empty) row. Press resort to display the values, as they will be used.
To delete rows click in the row, then select the whole row and press delete.

The first row and the last row will contain the maximum and minimum input values or the input range.
The input range entered will be divided into 101 even divisions for use by the USC. The input range can be changed after the table is created if required.

8. The 'Look Up Table' screen will open. I clicked on the chart line causing the data grid to move to E51. This point shows that input 3.98554 will have an output of 76.3647.

The input range consists of 101 even points on the x-axis the user cannot directly change these 'In' values.
The output is plotted in the vertical axis these 'Out' values can be directly changed by the user.

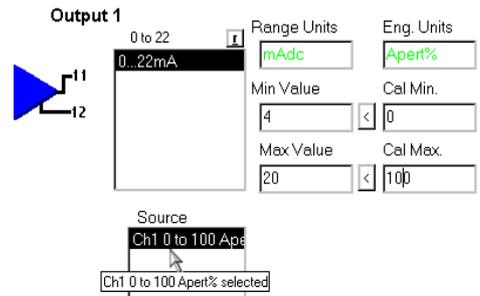


9. Click the 'Ok' button to accept the currently selected table for use.

The Min, Max Values and the Engineering Min and Max values are entered and locked by the table.
The Engineering units can be changed to reflect the input function.

10. Change the engineering units (Eng.Units) to 'Apert%' and press 'Next' until the modify wizard is on 'Output 1'.

11. Change the 'Cal Min.' and 'Cal Max' values to 0 and 100; this will cause a 4 to 20mA output for a 0 to 100% aperture measured at the input.



Editing an Existing Table

In this example we will copy the module definition made in the previous exercise ('Shaft Angle Example'), then change some points to further tailor the function.

PROBLEM

It was found that after using the 'Shaft Angle Example' that there is a mechanical dead spot in the middle where the shaft moves but the aperture does not.

When the input is at 30% of the range or 3.0902mA the aperture stays at 54.0368% until the input is at 34% or 3.26076 when the aperture starts moving again and returns to the original curve at 36% input.

1. Open the 'USC Properties Screen' for the module definition to be copied (Select 'Shaft Angle Example' in list) and select 'Open'.
2. From the 'Edit' menu select 'Copy' and change the program to 'Shaft Angle Example MK2'.
3. Click the CH1 icon then the 'Configure' button. The 'Modify Wizard' will step through to the current CH1 range setting.
4. Click on the small 'u' command to edit the user-defined table.

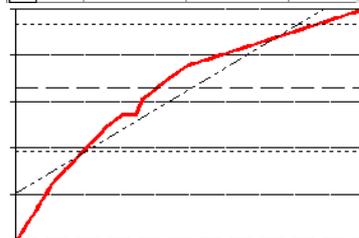


5. On the 'Look Up Table' screen click 'Copy to New' to retain the original table. Change the description from 'Aperture position Copy' to 'Aperture position MK2'.
6. On the grid move to row E30 and select inside the 'Mark' column. Click on the dropdown button to mark the point.

Input X axis		Output Y axis		
%	Value	Value		Mark
E	In		Out	Mark
29	3.04756	52.88312	No	
30	3.0902	54.0368	Yes	
31	3.13284	55.19048	No	
32	3.17548	56.34415	No	
33	3.21812	57.49783	No	
34	3.26076	58.65151	No	
35	3.3034	59.80519	No	
36	3.34604	60.95887	No	
37	3.38868	62.11255	No	
38	3.43132	63.26623	No	

7. On the grid move to row E34 and select inside the 'Out' values column and change the value to 54.036. Now select inside the 'Mark' column. Click on the dropdown button to mark the point.
8. On the grid move to row E36 and select inside the 'Mark' column. Click on the dropdown button to mark the point.
9. You will now have three points marked, click 'Join Marks' to update the points in-between, click the 'Ok' button to accept the currently selected table for use.
10. Press 'Finish' to close the 'Modify Wizard' and 'Program' to download to new module definition to the currently connected USC.

Input X axis		Output Y axis		
%	Value	Value		Mark
E	In		Out	Mark
29	3.04756	52.88312	No	
30	3.0902	54.0368	Yes	
31	3.13284	55.19048	No	
32	3.17548	56.34415	No	
33	3.21812	57.49783	No	
34	3.26076	54.0368	Yes	
35	3.3034	59.80519	No	
36	3.34604	60.95887	Yes	
37	3.38868	62.11255	No	
38	3.43132	63.26623	No	



Import / Export Data

When the 'USC700 Programmer xxx' program is installed its default location is

'C:\Program Files\USC700 Programmer 1xx\' where xxx is the sub version number of the internal USC software that the program is written for.

Files installed in the program directory are:

UscCfg.exe Executable

UscRun.mdb Run database containing object definitions

UscData.mdb Saved setting database containing
module definitions, look-up tables, recording sessions and script files.

Reasons to Export Data

1. All work that you do is saved in the 'UscData.mdb' file. If the 'USC700 Programmer program' is uninstalled (deleted) or copied over then any work you wish to keep may be lost.
2. Exporting module definitions that use look-up tables will export the tables as well as the module definition providing insurance against accidental table change.
3. Exported module definitions are small and can be easily emailed.
4. Recording sessions can be exported to a 'TXT' file that can easily be read into an EXCEL spread sheet.

Export A Module Definition

1. Open the module definition required for export and display on the 'USC Properties Screen' as the active window.
2. On the 'File' menu select 'Export USC'. In the 'Save As' dialogue change the file name and save location as required and press 'Save'.

Import A Module Definition

1. On the 'File' menu select 'Import Module Definition'. In the 'Open' dialogue change the 'Look in' location as required and select a "*.USC" file.
2. Press 'Open', a new module definition number is created. If the module definition used user-defined look up tables then new table definitions codes will also be created.

Export Recording Session Data

1. Open a reading log that contains some recorded data and display as the active window.
2. On the 'File' menu select 'Export Readings'. In the 'Save As' dialogue change the file name and save location as required and press 'Save'.

Import Readings to Excel

1. Open a blank Excel sheet. On the 'File' menu select 'Open'. In the 'Open' dialogue change the 'Look in' location as required and type "*.TXT" in the file name box.
2. Select the required file name (created in export session) and press open.
3. On the text import wizard selected 'Delimited' and press next.
4. In the 'Delimiters' frame select the comma and un-select the other types. Set the text qualifier on ' and press finish.

Using the Equation Editor

Equation outputs are saved in memory location M4 to M9 or Mem4 to Mem9. The “modify wizard” is then used to connect M4 to M9 to the relays, analogue or pulse output.

The equation can also directly manipulate the state of the relays.

The equation editor can perform both floating point and integer maths, when using M4 to M9 to control USC701 output components ensure that the result is in floating point format.

The equation editor is used to customise the functionality of the USC701. Math functions, branching commands and control of when the equation is run create a powerful and versatile tool.

Typical applications include:

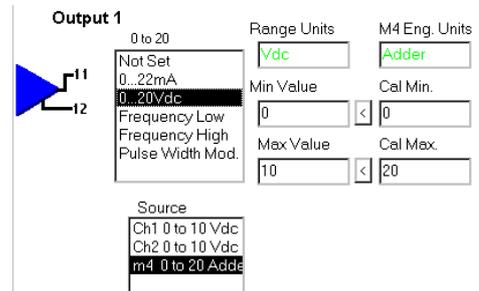
- Mathematical manipulation of a single input signal.
- Mathematical manipulation based on the values read from multiple inputs.
- Conditional processing of signals based on the state of other inputs.
- Filtering of input signals over long periods by providing a moving average of input signal while using the fixed equation update time option.
- PID control of a process variable.
- Pulse width modulation of the relays.

Following is a series of exercises designed to familiarise with the user while building some typical working applications.

- The “Create a Simple Adder using the New Module Wizard” is a simple exercise that should take about three minutes to run and impart entry-level knowledge. A later exercise “Review the Simple Adder Wizard Equation” has a full listing to explain the reverse polish program list and how to use the step tool to simulate the equation function.
- The “Creatw a Simple Divider using the New Module Wizaed” is similar to the previous example except that it demonstrates some error conditions and how to correct or ignore them. The “Evaluate Equation Outputs” screen is demonstrated as well as one reason to disable “Auto Evaluate” feature.
- The “Make an Adder with Peak Output and Low Limit” demonstrates the branching functions and uses a control input, branch labels and a constant.

Create a Simple Adder using the New Module Wizard

1. Run the 'New Module Wizard' and select 'dc Signal', 'Adder' then 'Next'.
2. Select channel 1 as 10Vdc then 'Next'.
3. Select channel 2 as 10Vdc then 'Next'.
4. Select the '0..22mA' output on the range list.
5. Select 'm4 0..20 Adder' ([m4] [Min]..[Max] [Engineering Unit]) ON the source list then click 'Next'.
6. Select 'Next' until the wizard closes.



The new module wizard added a simple equation (read CH1, read CH2, add, save Mem4) to the settings of the module definition that you are creating.

When the USC wizard moved from the input components to the output components the "Auto Evaluator" option calculated the expected output range of the equation based on the minimum and maximum engineering range of the inputs used.

In this case: $0+0=0$, $0+10=10$, $10+0=10$, $10+10=20$

Therefore the maximum expected output range in memory 4 is 0 to 20.

This calculated range is used by the wizard to check limits and set the sensitivity of the 'ADJ.' encoder on the top of the USC with this range.

When the new module ran it also added the engineering unit for Mem4, "Adder". To change the engineering unit of the adder to a different value you will need to use the equation editor.

7. Select the 'Equation' command button in the 'USC Properties' screen. The M4 engineering units will be displayed at the top of the properties list.
8. Click the 'Configure' button; the 'Special Function Equation Editor' will open.

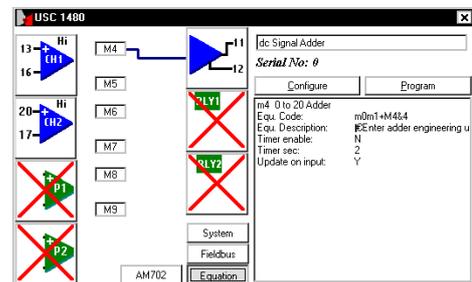
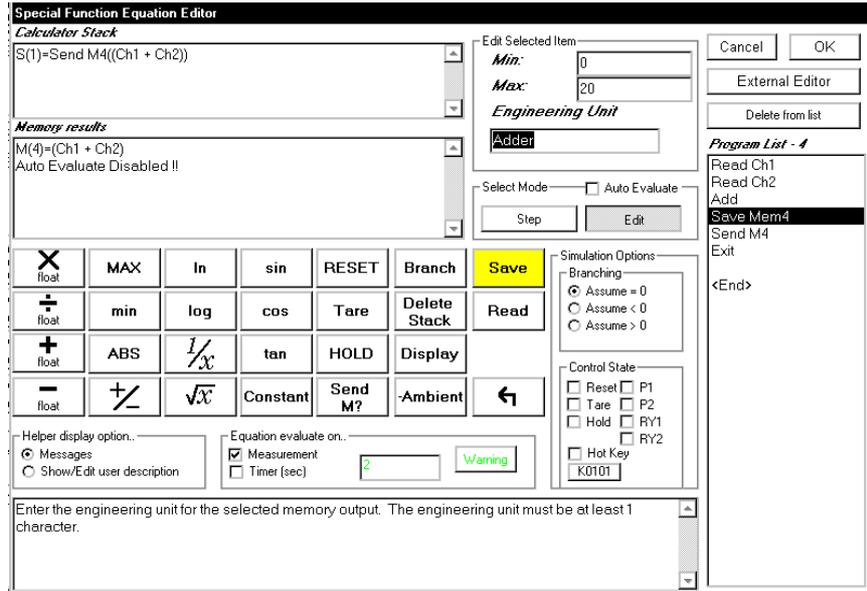


Figure 28 Changing Mem4 Engineering Units With Special Function Equation Editor.

- Click in the program list on "Save Mem4" and type in the engineering unit in the box provided.



The memory results window displays the equivalent equation made from the program list $M(4)=(Ch1 + Ch2)$ and also displays "Auto Evaluate Disabled !!". With "Auto Evaluate Disabled!" as shown in figure 28 the expected output range for the equation must be entered by the user in the Min and Max boxes provided.

- In the "Simulation Options" check "Auto Evaluate", the memory results window will now display $M(4)=(Ch1 + Ch2)$!!Auto Evaluate!! $M(4)=0$ to 20. With this option enabled if you change the inputs the expected range from memory 4 will also automatically update.
- Press Ok to close the equation editor.

For a full explanation of the simple adder please refer to "Review the Simple Adder Wizard Equation"

Create a Simple Divider using the New Module Wizard

1. Run the 'New Module Wizard' and select 'dc Signal', 'Divider' then 'Next'.
2. Select channel 1 as 10Vdc then 'Next'.
3. Select channel 2 as 10Vdc then 'Next'.

The new module wizard added a simple equation (read CH1, read CH2, divide, save Mem4) to the settings to module definition that you are creating.

When the USC wizard moved from the input components to the output components the "Auto Evaluator" option calculated the expected output range of the equation based on the minimum and maximum engineering range of the inputs used.

In this case: $0/0$ =error, $0/10=0$, $10/0$ =error, $10/10=1$

Therefore the maximum expected output range in cannot be determined.

4. The "Evaluate Equation Outputs" screen opens due to an equation evaluation error as shown.

Figure 29 "Evaluate Equation Outputs" screen opened due to evaluation error.

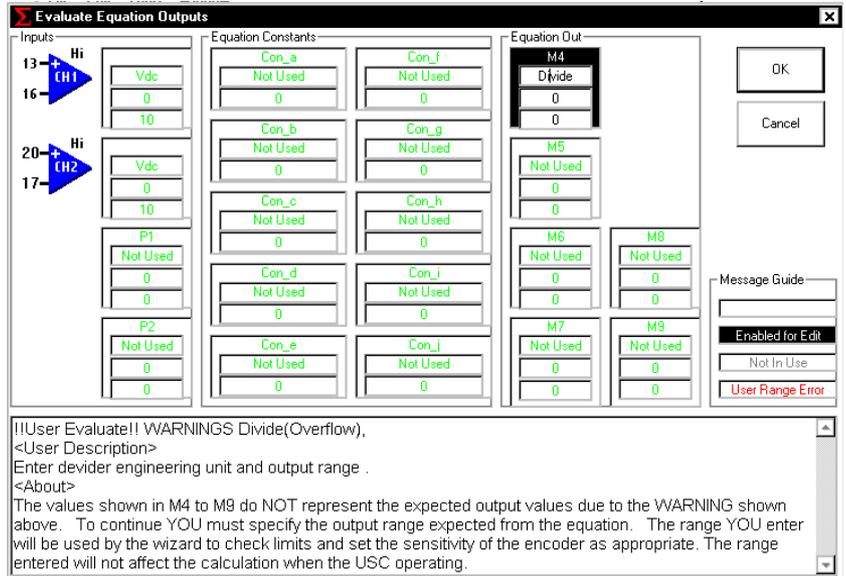
5. Either use

a. "Go Back and Avoid the Evaluation Error"

or

b. "Enter Expected Equation Output Range"

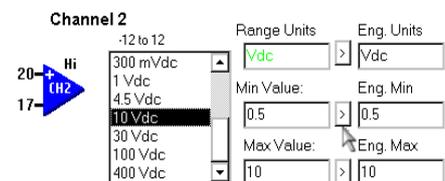
to overcome this problem.



a. Go Back and Avoid the Evaluation Error

- a1. Click cancel on the "Evaluate Equation Outputs" screen, an information screen will open warning that the output range for the memories has not been updated.
- a2. Click 'Ok' to close the information screen. The program will return to the "dc Signal Divider" wizard.
- a3. Click "Back" until the "dc Signal Divider" wizard is on the Channel 2 screen.
- a4. Change the 'Min Value' from 0 to 0.5 and press the copy across button to change the 'Eng. Min' to 0.5

This change does not affect the measured value.



The divider wizard created a simple equation CH1/CH2, changing the minimum value of CH2 to 1 prevents the divide by zero error.

$0/1=0$, $0/10=0$, $10/1=10$, $10/10=1$

Therefore the maximum expected output range is 0 to 10.

- a5. Click next, the wizard will now move to the output components without stopping.

b. Enter Expected Equation Output Range

The wizard will stop on the "Evaluate Equation Outputs" screen if;

- > the auto evaluate option is switched on and the equation simulator cannot determine the output range;
- > the engineering range for a used memory has never been entered;
- > the trim menu has been enabled, giving the operator the opportunity to change the values of some equation constants.

If the equation output does not auto evaluate you must enter the output range for each memory.

In the divider example CH1/CH2 will produce a larger number as CH2 approaches zero.

With this in mind I chose some numbers as follows;

Minimum: $0 / [\text{any number}] = 0$

Maximum: $10 / 0 = \infty$ however if CH2 never goes below 0.5 the maximum output range is 0 to 20 (conditional).

b1. On the "Evaluate Equation Outputs" screen enter the values that you have determined for M4 minimum and maximum.

The wizard checks limits and set the sensitivity of the encoder using the range YOU enter.

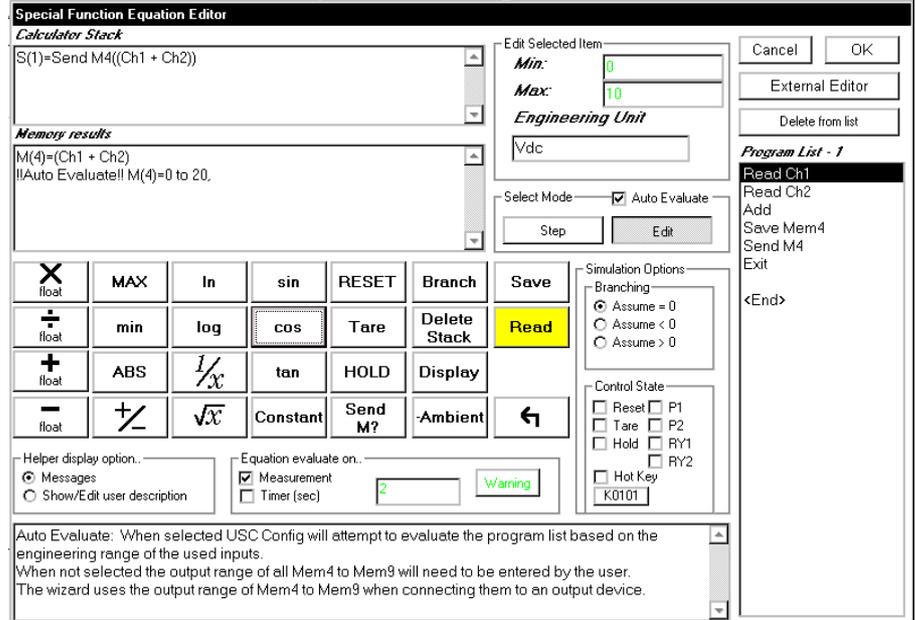
The range entered will not affect the calculation when the USC operating.

b2. Click next, the wizard will now move to the output components.

Review the Simple Adder Wizard Equation

1. Create a module definition file using the instructions in “Create a Simple Adder using the New Module Wizard”.
2. Select the ‘Equation’ command button in the ‘USC Properties’ screen then click the ‘Configure’ button; the ‘Special Function Equation Editor’ will open.
3. Check the ‘Auto Evaluate’ option.

Figure 32 Special Function Equation Editor With A Simple Adder Program.



The ‘Program List’ displays the equation commands in the order that they are used.

To enter ‘(4 + 5) x (3 + 4)’ into the equation editor in reverse polish; 4, Enter, 5, +, 3, Enter, 4, Enter, +, x. In ‘Edit’ mode the ‘Calculator Stack’ and ‘Memory results’ windows display program execution based on the control state and branching options (don’t get too up tight about what this means).

The memory results is displaying a simulation of the equation that produces an output at memory 4’

4. Click on the first item in the ‘Program List’, ‘Read CH1’ the engineering unit and range appear for CH1.
5. Click on the second item in the ‘Program List’, ‘Read CH2’ the engineering unit and range appear for CH2.
6. Click on the fourth item in the ‘Program List’, ‘Save M4’ the engineering unit and range for M4 can now be changed if required.
7. Select Mode = Step.
8. As the “Step >” button is pressed equation operation is simulated as shown. Pressing “Step >” six times will move the simulation to the exit command where it will stop.

Note that the memory results window is empty until the program reaches a save command.

Program List	Calculator Stack	Memory Results
Read CH1	S(1)=CH1	
Read CH2	S(2)=CH2	
Add	S(1)=CH1 S(1)=(CH1 + CH2)	
Save M4	S(1)=(CH1 + CH2)	M(4)=(CH1 + CH2)
Send M4	S(1)=(CH1 + CH2)	M(4)=(CH1 + CH2)
Exit	S(1)=(CH1 + CH2)	M(4)=(CH1 + CH2)

9. Figure 33 shows the simple adder in step mode evaluated to 6 meaning that the equation evaluator performed six steps to get to the current screen.

Figure 33 Simple Adder In Step Mode

The value of all memory results are only reported if they are different from their corresponding initial values.

10. By changing the initial value of CH2 to 5 as shown will instantly simulate the outputs expected up to the current step number.

Remember the calculated result of M4 was not shown in step 9 as the calculated result was the same as the initial value.

Make an Adder with Peak Output and Low Limit

This exercise will take the previous input adder example and add a peak measurement output that is reset to a minimum output level (4) when a digital input is active.

Create some program documentation.

11. Enter the equation screen and click on the 'Show/Edit user description' option.
12. In the 'Helper' window type in following information to describe the variable use in the equation to be written.
Adder with Peak Output And Low Limit
Mem 4 = Adder Output
Mem 5 = Peak Output
Const b = reset value for peak output
P2 = reset the peak output to Const b

Create a block of commands that saves memory 4 in memory 5

13. In 'Edit' mode click on the <End> label, click 'Branch' on the keypad and insert an 'Exit' command.
14. While still on the <End> label, click 'Branch' on the keypad and insert a 'Line Label'. When prompted enter the name of the line label as "SavePk".
15. While still on the <End> label, click 'Read' on the keypad and insert a 'Read M4' command.
16. While still on the <End> label, click 'Save' on the keypad and insert a 'Save M5' command.
17. While still on the <End> label, click 'Send M?' on the keypad and insert a 'Send M5' command.

Create a block that saves constant 'b' in memory 5

18. While still on the <End> label, click 'Branch' on the keypad and insert a 'Exit' command.
19. While still on the <End> label, click 'Branch' on the keypad and insert a 'Line Label'. When prompted enter the name of the line label as "ResetPk".
20. While still on the <End> label (shift button up ) , click the 'Constant' button on the keypad and insert 'Constant b'.
When prompted enter the value as 0 and the name as "ResetVal".
21. While still on the <End> label, click 'Save' on the keypad and insert a 'Save M5' command.
22. While still on the <End> label, click 'Send M?' on the keypad and insert a 'Send M5' command.

Insert a branch that saves constant b in memory 5 when P2 is pressed.

23. Click on the first exit command in the program list.
24. Click 'Branch' on the keypad and insert 'If P2 Goto' command. Select line "ResetPk" when prompted.

Insert a branch that saves memory 4 in memory 5 when memory 4 is greater than memory 5.

25. Click on the first exit command in the program list.
26. Click 'Read' on the keypad and insert a 'Read M5' command.
27. Click  on the keypad.
28. Click 'Branch' on the keypad and insert 'If > 0 Goto' command. Select line "SavePk" when prompted.

Save results so far.

29. Press 'Ok' to exit then 'Ok' to close and save changes, the following message will appear.
P2 has been used in the equation editor but it has not been enabled for use.
Please enable when the 'modify wizard' opens.
Pressing cancel will disable all further checks of the equation in this pass.
30. Press 'Ok', when the P2 not set screen appears press the small range button 'r' and select 'Logical' then 'Settings' on the change range wizard. Press Next to return to the modify wizard.

31. Select 'Contact' from the list. Change the trigger to 'High' so that the P2 function occurs when the button is pressed.
32. Press "Next" then "Finish". When the 'USC Properties' screen appears press 'Configure' to re-enter the special function equation editor.
33. In the 'Select Mode' group select 'Step'.
34. Progressively press the "Step >" (step next) button. The function of the equation will be simulated as shown in figure 36.

Figure 36 Adder with Peak Output and Low Limit Equation List

Program List	Calculator Stack	Memory Results
Read CH1	S(1)=CH1	
Read CH2	S(2)=CH2 S(1)=CH1	
Add	S(1)=(CH1 + CH2)	
Save Mem4	S(1)=(CH1 + CH2)	M(4)=(CH1 + CH2)
Send M4	S(1)=(CH1 + CH2)	M(4)=(CH1 + CH2)
If P2 Goto ResetPk	S(1)=(CH1 + CH2)	M(4)=(CH1 + CH2)
Read Mem5	S(2)=(MEM5 S(1)=(CH1 + CH2)	M(4)=(CH1 + CH2)
Subtract	S(1) =(CH1 + CH2) – MEM5	M(4)=(CH1 + CH2)
If> 0 Goto SavePk	S(1) =(CH1 + CH2) – MEM5	M(4)=(CH1 + CH2)
Exit		M5 remains the same
LAB: SavePk	S(1) =(CH1 + CH2) – MEM5	M(4)=(CH1 + CH2)
Read Mem4	S(2) = MEM4 S(1) =(CH1 + CH2) – MEM5	M(4)=(CH1 + CH2)
Save Mem5	S(2) = MEM4 S(1) =(CH1 + CH2) – MEM5	M(5)=MEM4 M(4)=(CH1 + CH2)
Send M5	S(2) = MEM4 S(1) =(CH1 + CH2) – MEM5	M(5)=MEM4 M(4)=(CH1 + CH2)
Exit		
LAB: ResetPk	S(1)=(CH1 + CH2)	M(4)=(CH1 + CH2)
Con_b	S(2)=Con_b S(1)=(CH1 + CH2)	M(4)=(CH1 + CH2)
Save Mem5	S(2)=Con_b S(1)=(CH1 + CH2)	M(4)=(CH1 + CH2) M(5)=Con_b
Send M5	S(2)=Con_b S(1)=(CH1 + CH2)	M(4)=(CH1 + CH2) M(5)=Con_b
Exit		

Interfacing with the Access Module

This manual applies to:

AM702-02

Plug on configuration module

“USC700 Programmer” PC based configuration software

Compatibility with the AM702

1. Any USC701 module programmed with an AM702-02 in configure mode can be read into the PC using the “Read Program from USC” command.
2. When USC700 programs a USC701 it tests the data:
 - If the program could have been made by the AM702 then the various flags are set within the USC701 for the configure mode to operate on the pre-set menu items.
 - If the program could not have been made by the AM702 then the “configure mode” flags are set to PC Special.
3. USC701 modules programmed as PC Special can still be modified using the AM702-02 in Trim Mode.

Configurable User Interface

The USC700 Programmer can create customised USC701 configurations far beyond the capabilities of the AM702 Configure Mode.

USC700 Programmer can flag variables that you want the operator to change using the AM702-02 access modules. Variables and settings that you wish to protect from user scrutiny can be disabled. The USC700 Programmer can also enable or disable the main categories in the trim menu. Disabled menus, settings and constants do not appear when scrolling through the AM702 trim menu.

As the customised interface is stored within the USC701 the same AM702 can be used to trim many different applications.

The “Create a Divider with Trim-able CH1 Minimum Value” example will:

- Create an input divider CH1/CH2.
- Add a trim-able constant will be added to set the minimum output out of CH2.
- Set the AM702 interfaces to only allow the trim menu.
- Set the trim menu categories such that only constant d and RLY1 trip points can be adjusted.
- Add a helper prompt that will appear on the “Evaluate Equation Outputs” screen.

After completing the “Create a Divider with Trim-able CH1 Minimum Value” example hot keys will be added to allow the user to display more than one memory variable.

The “Add Hot Keys to Control Branching” example will:

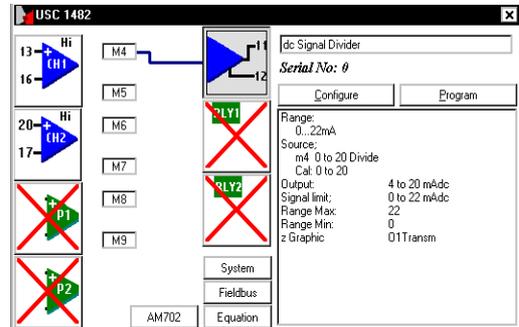
- Allow the user to display CH1 when keys  and  are pressed.
- Allow the user to display CH2 when keys  and  are pressed.
- Allow the user to display M4 when keys  and  are pressed.

The example shown is used to control the displayed variable however they could be used for running self test procedures, automatic input calibration or switch to complete alternate programs.

Create a Divider with Trim-able CH1 Minimum Value

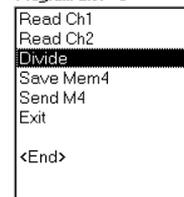
The first step is to create the USC program.

1. Follow previous example "Create a Simple Divider using the New Module Wizard"
2. When the error message occurs due to the divide by zero follow option "b. Enter Expected Equation Output Range" to overcome this problem.
3. After the module definition has been created select the equation button on the bottom of the properties screen then select configure.

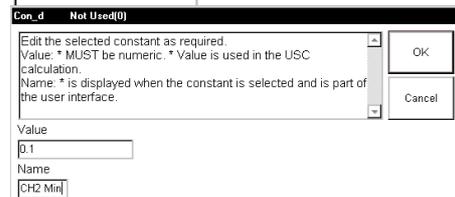


4. Now in the equation editor click on the divide command in the program list as shown.

Program List - 3



5. Click the 'Constant' button on the keypad and insert 'Constant d'.
When prompted enter the value as 0.1 and the name as "CH2 Min" and press ok.



6. With the divide command in the program list still selected click the 'MAX' button on the keypad. The resulting program is shown.
7. In the simulation options enable 'Auto Evaluate'.

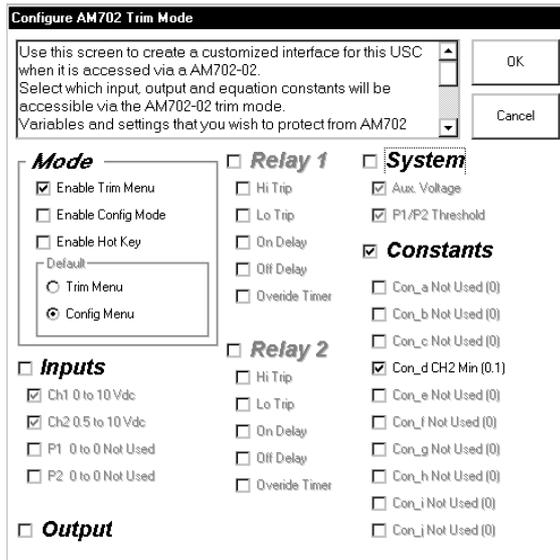
Program List - 5



8. Click on "Show/Edit user description" and replace the description in the helper window with "Enter the minimum value for CH2, it must be greater than zero."
9. Press ok to close the equation editor.
10. The modify wizard may stop on the output components due to out of range errors, correct these and continue the USC properties screen.

The second step is to enable and disable parameters to be trimmed.

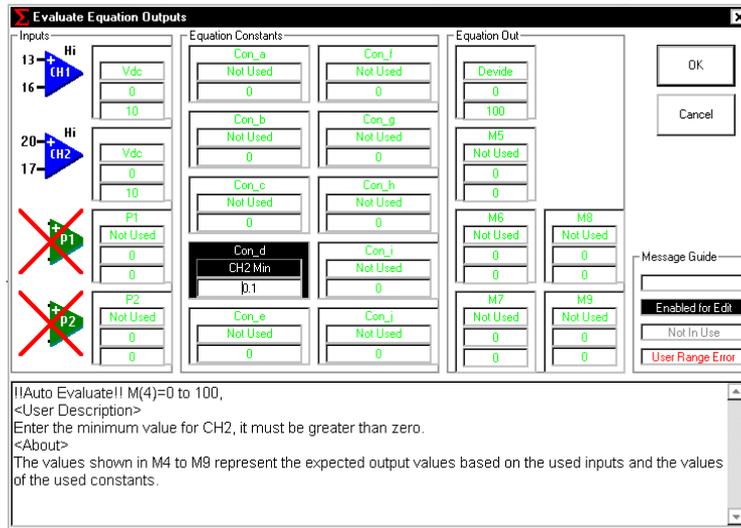
11. Select the AM702 button and press configure, the Configure AM702 Trim Mode Screen will appear.
12. Un-tick all main options except "Enable Trim Menu", "Constants" and 'Con_d'. The resulting screen should look the figure opposite.
13. Press OK to close.



The third step is to test the constant trim function using the "Evaluate Equation Outputs" screen.

14. On the USC properties screen select the output button and press configure.

When the USC wizard moved from the input components to the output components the AM702 trim constants option is checked causing the "Evaluate Equation Outputs" screen to open.



15. Constant d and its name "CH2 Min" appears and is enables for edit. As the value is changed the auto evaluate option determines the output range based on the engineering range of CH1 and CH2. Also the helper text "Enter the minimum value for CH2, it must be greater than zero." Appears in the <User Description > section of the helper window.

The fourth step is to program the USC701 with the appropriate inputs connected.

16. After programming the USC701 remove the COA703 from the top of the USC701 and fit a AM702-02.
17. The AM702-02 will continuously display the selected variable while in run mode.
18. Press enter on the AM702, the trim menu will load and display 'Trim Constant'.
19. Press enter the AM702 will display 'CH2 Min' and its value.
20. Press enter again to edit the value of constant d then press enter twice to return to run mode, the effect of the change you made can be seen when the input to CH2 is at zero volts.

Add Hot Keys to Control Branching

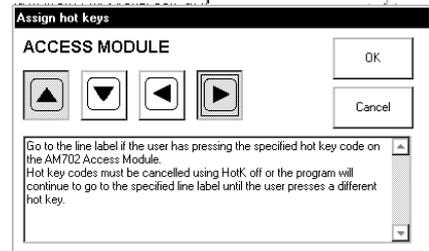
- The AM702 will display CH1 when keys  and  are pressed.
 - The AM702 will display CH2 when keys  and  are pressed.
 - The AM702 will display M4 when keys  and  are pressed.
1. This example follows the previous example “Create a Divider with Trimable CH1 Minimum Value” however the same logic can be applied to any program. The complete program listing from the previous example is shown.
 2. Select line <End>, press “Branch” then “Insert Line Label” and type “ShowCh1” when prompted.
 3. While still on <End> press “Display” then select display CH1.
 4. While still on <End> press “Branch” then select “HotK off”.
 5. While still on <End> press “Branch” then select “Exit”.
 6. Select the first exit command in the list as shown above and press “Branch” then select “Insert If HK1000-”
 7. Select the required key code on the assign hot keys dialogue (CH1 = HK1001, CH2 = HK0101, M4 = HK0011). The correct key code for channel 1 is displayed on the diagram opposite.
 8. Press OK and select the line label to jump to when the hot key is pressed.
 9. Repeat steps 2 to 8 substituting the line label names and key codes for each. The new lines should look like the listing opposite.
 10. Press OK to save and exit the equation.
 21. On the USC properties screen select the AM702 button and press configure. On the ‘Configure AM702 Trim Mode’ screen select ‘Enable Hot Key’ as shown.
 22. Program the USC701 then fit an AM702-02.
 23. The AM702 will start in run mode, by pressing the predefined key combinations the display will change to the appropriate variable.
 24. Pressing enter will load the trim menu and still operate according to the previous. After exiting the trim menu hot keys will resume control.

Program List - 10

```
Read Ch1
Read Ch2
Con_d
MAX
Divide
Save Mem4
Send M4
Exit
<End>
```

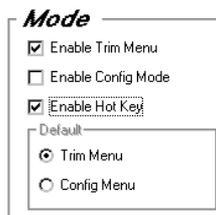
Program List - 8

```
Read Ch1
Read Ch2
Con_d
MAX
Divide
Save Mem4
Send M4
Exit
LAB: ShowCH1
Display Ch1
HotK off
Exit
```



Program List - 1

```
Read Ch1
Read Ch2
Con_d
MAX
Divide
Save Mem4
Send M4
If HK1001-ShowCH1
If HK0101-ShowCH2
If HK0011-ShowM4
Exit
LAB: ShowCH1
Display Ch1
HotK off
Exit
LAB: ShowCH2
Display Ch2
HotK off
Exit
LAB: ShowM4
Display Mem4
HotK off
Exit
<End>
```



TIP the engineering unit for both CH1 and CH2 are both “Vdc” this may be confusing. Changing channel 1 to “CH1 Vdc” and channel 2 to “CH2 Vdc” will provide visual feedback to which channel is displayed

Using Integer Maths

The USC701 has four memories CH1, CH2, P1 and P2 (M0 to M3) that contain input measurements, these measurements are floating point numbers.

The output devices O1 (mA, V, PWM, Frequency), RLY1 and RLY2 can be controlled from M0 to M9 (memory's).

The values in M0 to M9 must be floating point numbers to control the output devices correctly.

There are two problems that occur when using floating-point maths in custom applications:

- Floating point arithmetic requires more cycles of the CPU and therefore takes more time to complete. Each floating-point addition / subtraction takes about 2mS while each floating-point multiplication / division takes about 5mS.
- Testing for a particular value can be risky as simply starting at 0 and adding 1 progressively will not result in exactly whole numbers in every case, some will be ever so slightly lower than the whole number while others will be ever so slightly higher than the whole number.

The equation editor has commands to branch on the condition of P1 and P2 making it possible to implement different processing based on the condition of the input. By setting the equation update time to a known value this same technique can be used for counting and period measurement functions with resolution and speed limitations.

When using the user equation at the maximum update speed of 5mS care must be taken to ensure the equation can be processed within that time. Obviously floating-point division and multiplication are out of the question while one or two floating-point addition and subtractions are possible.

Another problem that can occur at fast update times is excessive use of the send memory command. The send memory command is not required for USC701 output components to use the values in memory. If M4 was connected to the milliamp output and the equation contained no send M4 commands the milliamp output will still be updated each time the equation is run.

If using a field bus card or logging results with the COA703 the values in M4 to M9 do require the send memory to occur. For applications running at up date times faster than 40mS then extra branching should be added to send values say once every eighth time or 40mS as shown in the following example.

The following example also features:

- Integer maths is used to calculate values for use in the equation at power up. This technique can be used to free up the equation constants for use in the user interface.
- Integer variables are directly controlling program branching. Three logical decisions are available after the variable is read into the stack, “=Int0”, “>Int0” and “<Int0”.
- Integer counter variables are used to generate the output values due to their very fast calculating time.

Integer Maths and Branching Demonstration

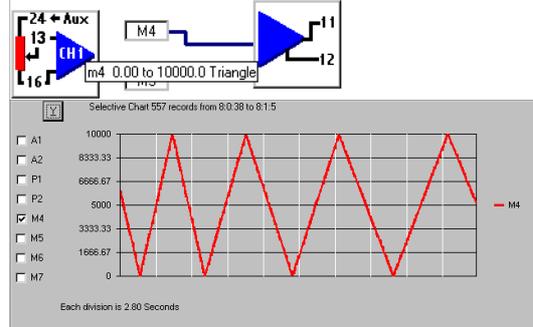
To follow this application import UAP1052 "Variable Period Triangle Generator" from the USC centre on the APCS web site.

Complete understanding of the provided application is not necessary as the following is a guide to using the step mode to test program branches independently.

Channel 1 is connected to a potentiometer and has been scaled to produce 1 to 10 (period in seconds) when moved from 0 to 100%

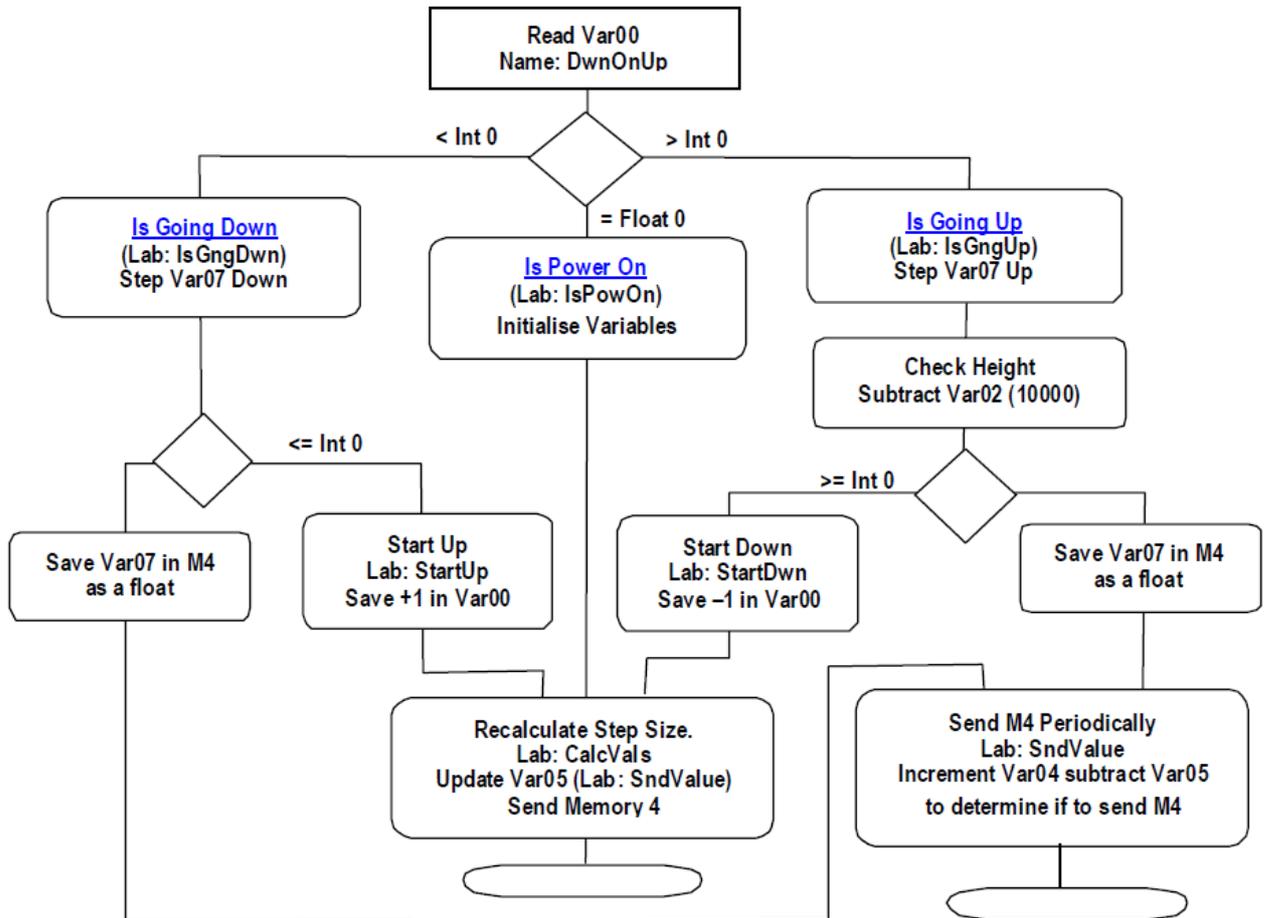
Memory 4 drives the milliamp output and is scaled to produce 4 to 20mA out for an input of 0 to 10000.

The setting of channel 1 was moved every 4 seconds during the recording resulting in changes to the triangle period.



Referring to the flow chart the program is made up of three main branches that are called base on the value of variable 00.

Figure 47 Flow Chart Integer Maths and Branching Demonstration

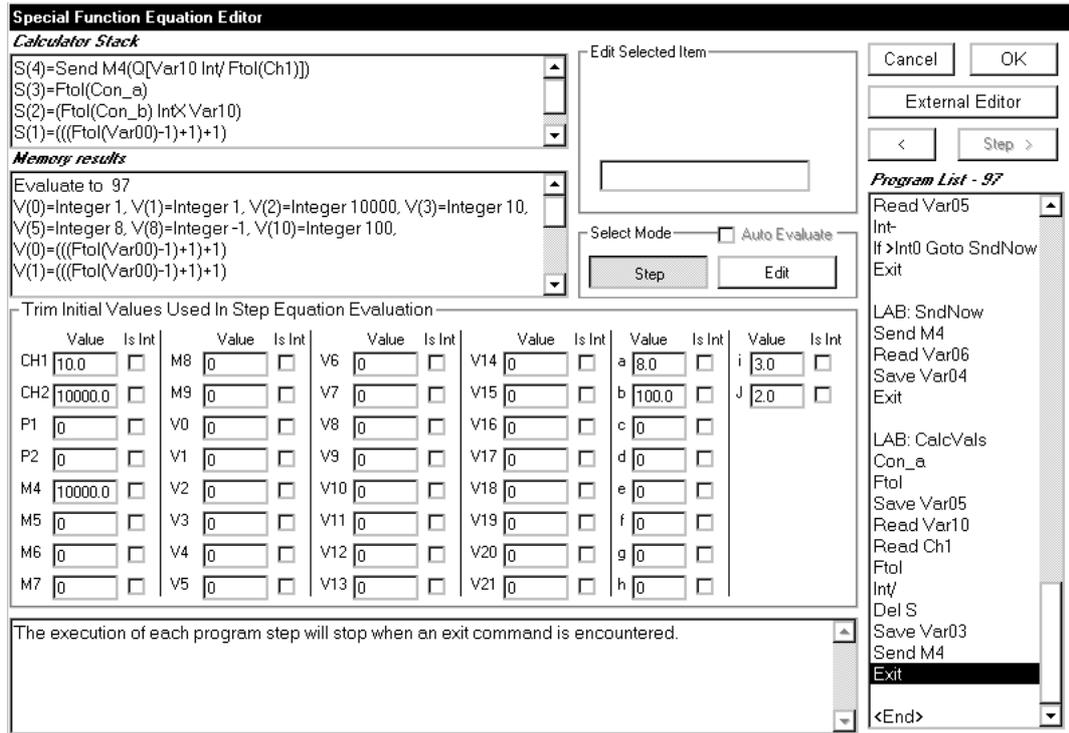


Simulating The Power On Branch

1. Open UAP1052 click on "Equation" then "Configure".
2. In the "Select Mode" frame click step.
3. Click the "Step >"
4. When asked if Var00 =0 press "Yes"
5. The program list will jump to the beginning of the "IsPowOn" branch.
6. Press "Step >" until the "Step >" button will no longer operate.
7. The program will have completed the "IsPowOn" branch and the "CalcVals" branch as shown in figure 48.



Figure 48 Results Of the Power On Branch



At switch on the value in Var00 is float 0, therefore when the power is first applied to the USC701 the "IsPowOn" branch will run. From Var0 other values are calculated for use in the other branches of the equation.

Power On Branch Calculations (IsPowOn)

Name	Label	Use	Initial Value	Calculated
Var00	DwnOnUp	Main branch flag	Float 0	Int +1
Var01	=Int 1	Fixed value after power on.	Float 0	Int +1
Var02	H=Int10k	Fixed value after power on.	Float 0	Int 10000
Var06	= Int 0	Fixed value after power on.	Float 0	Int 0
Var08	= Int -1	Fixed value after power on.	Float 0	Int -1
Var10	=Int 100	Fixed value after power on.	Float 0	Int 100

Before continuing to the 'Is Going Up' branch remain in the current step position (evaluate to 95) and type in the initial values plus data type to be used for V1, V2, V3, V5, V6, V8, V10 as shown in figure 49, leave V0 as float 0 for a moment.

Figure 49 Only Values That Are Different To The Initial Values Are Reported

Memory results

Evaluate to 95
 V(0)=Integer 1,
 V(0)=(((Ftol(Var00)-1)+1)+1)
 V(1)=(((Ftol(Var00)-1)+1)+1)
 V(2)=(Ftol(Con_b) IntX Var1 0)

Select Mode Auto Evaluate

Step Edit

Trim Initial Values Used In Step Equation Evaluation

	Value	Is Int		Value	Is Int		Value	Is Int		Value	Is Int		Value	Is Int		Value	Is Int
CH1	10	<input type="checkbox"/>	M8	0	<input type="checkbox"/>	V6	0	<input checked="" type="checkbox"/>	V14	0	<input type="checkbox"/>	a	8.0	<input type="checkbox"/>	i	3.0	<input type="checkbox"/>
CH2	10	<input type="checkbox"/>	M9	0	<input type="checkbox"/>	V7	0	<input type="checkbox"/>	V15	0	<input type="checkbox"/>	b	100.0	<input type="checkbox"/>	J	2.0	<input type="checkbox"/>
P1	0	<input type="checkbox"/>	V0	0	<input type="checkbox"/>	V8	-1	<input checked="" type="checkbox"/>	V16	0	<input type="checkbox"/>	c	0	<input type="checkbox"/>			
P2	0	<input type="checkbox"/>	V1	1	<input checked="" type="checkbox"/>	V9	0	<input type="checkbox"/>	V17	0	<input type="checkbox"/>	d	0	<input type="checkbox"/>			
M4	10000.0	<input type="checkbox"/>	V2	10000	<input checked="" type="checkbox"/>	V10	100	<input checked="" type="checkbox"/>	V18	0	<input type="checkbox"/>	e	0	<input type="checkbox"/>			
M5	0	<input type="checkbox"/>	V3	10	<input checked="" type="checkbox"/>	V11	0	<input type="checkbox"/>	V19	0	<input type="checkbox"/>	f	0	<input type="checkbox"/>			
M6	0	<input type="checkbox"/>	V4	0	<input type="checkbox"/>	V12	0	<input type="checkbox"/>	V20	0	<input type="checkbox"/>	g	0	<input type="checkbox"/>			
M7	0	<input type="checkbox"/>	V5	8	<input checked="" type="checkbox"/>	V13	0	<input type="checkbox"/>	V21	0	<input type="checkbox"/>	h	0	<input type="checkbox"/>			

It can be seen that only V0 is now reported, as all other values are the same as the initial value.

Now change V0 to integer 1 the error message is reported as the simulator is running the power on branch with an incorrect data type for the Var00 calculations.

Memory results

FAILED at 52 Stack not a float:Ftol

V(0)=(((Ftol(Var00)-1)+1)+1)
 V(1)=(((Ftol(Var00)-1)+1)+1)
 V(2)=(Ftol(Con_b) IntX Var1 0)

Trim Initial Values Used In Step Equation Evaluation

	Value	Is Int		Value	Is Int		Value	Is Int		Value	Is Int
CH1	10	<input type="checkbox"/>	M8	0	<input type="checkbox"/>	V6	0	<input checked="" type="checkbox"/>	V14	0	<input type="checkbox"/>
CH2	10	<input type="checkbox"/>	M9	0	<input type="checkbox"/>	V7	0	<input type="checkbox"/>	V15	0	<input type="checkbox"/>
P1	0	<input type="checkbox"/>	V0	1	<input checked="" type="checkbox"/>	V8	-1	<input checked="" type="checkbox"/>	V16	0	<input type="checkbox"/>

Simulating The Up Branch

If still in step mode from previous procedure “Simulating Power On Branch” then reset the step counter by;

- > Selecting “Edit” in the select mode frame
- > Reselecting “Step” in the “Select Mode”, if prompted to “Move To Last Step Position” select “Cancel”.

8. Enter step mode end type in the power on values V1= Int 1, V2 = Int 10000, V6 = Int 0, V7 = Int 0, V8 = Int -1, V10 = Int 100
9. Press step, when asked at program list 2 “Is Var00=0” select no.
10. Press step, when asked at program list 3 “Is Var00>0” select yes
11. .Press step until the program list has reached line 31, these steps will read the last value of Var07 (Integer value representing the output) and increase it by Var03 (required step size for current input setting).
12. The memory results screen shows the calculated values to this point.
13. Each time the equation runs the “IsGngUp” branch will continue to run until Var07 reaches the value of Var02 at which time the “StartDwn” branch will run.
14. To simulate multiple operation of the equation to the current point, change the value of V7 in the “Initial Values Used In Step Equation Evaluation”. V(7) in the evaluation results screen will increase by the step size.

```

Calculator Stack
S(2)=(Var07 Int+ Var03)
S(1)=Var00

Memory results
Evaluate to 31
V(7)=Integer 10.
V(7)=(Var07 Int+ Var03)
    
```

```

Program List - 31
Exit
LAB: StartUp
ltoF
Save Mem4
Read Var01
Save Var00
Goto CalcVals
Exit
LAB: IsGngUp
Read Var07
Read Var03
Int+
Save Var07
Read Var02
Int-
If =Int0 Goto StartDwn
If >Int0 Goto StartDwn
Read Var07
ltoF
Save Mem4
Goto SndValue
Exit
    
```

Each time the equation is run the resulting value in Var07 is converted to a floating-point number and saved in memory 4 for use by the USC701 output components.

Simulating The Down Branch

If still in step mode from previous procedure "Simulating The Up Branch" then reset the step counter by;

- > Selecting "Edit" in the select mode frame
- > Reselecting "Step" in the "Select Mode", if prompted to "Move To Last Step Position" select "Cancel".

15. Enter step mode and type in the power on values V1= Int -1, V2 = Int 10000, V6 = Int 0, V7 = Int 10000, V8 = Int -1, V10 = Int 100
16. Press step, when asked at program list 2 "Is Var00=0" select no.
17. Press step, when asked at program list 3 "Is Var00>0" select no.
18. Press step, when asked at program list 4 "Is Var00<0" select yes.
19. Press step until the program list has reached line 11, these steps will read the last value of Var07 (Integer value representing the output) and decrease it by Var03 (required step size for current input setting).

The memory results screen shows the calculated values to this point, note that the equation has been evaluated to step 11, this is not the program list number but how many equation operations have occurred to get to the current point.

Each time the equation runs the "IsGngDwn" branch will continue to run until Var07 <= 0 at which time the "StartUp" branch will run.

To simulate multiple operation of the equation to the current point, change the value of V7 in the "Initial Values Used In Step Equation Evaluation". V(7) in the evaluation results screen will decrease by the step size.

Calculator Stack

S(2)=(Var07 Int- Var03)
S(1)=Var00

Memory results

Evaluate to 11
V(7)=Integer 9990.
V(7)=(Var07 Int- Var03)

```

Program List - 11
Read Var00
If=0 Goto IsPowOn
If >Int0 Goto IsGngUp
If <Int0 Goto IsGngDw
Exit

LAB: IsGngDwn
Read Var07
Read Var03
Int-
Save Var07
If =Int0 Goto StartUp
If <Int0 Goto StartUp
IttoF
Save Mem4
Goto SndValue
Exit

LAB: StartUp
IttoF
Save Mem4
Read Var01
Save Var00
Goto CalcVals
Exit
    
```

Recording Readings

In general there are three possible methods to record readings coming from the USC701:

1. The optional field bus card can be configured for 'Modbus'. A 'Modbus' controller could then periodically query the USC701 memories and record the records.
2. The optional field bus card can be configured for 'Serial Talk Only'. Under this mode user specified USC701 memories are sent out the field bus serial port at a user specified interval. No controller is necessary; any product capable of capturing serial data could store the results.
3. The third method works with all standard USC701 modules this is recording data from the COA703 connection using 'USC700 Programmer'.

USC700 Programmer can record readings from the USC701 and is intended for use when fault finding. It does require full time use of a windows pc so is probably not appropriate as a permanent solution. Some enhancements have been made in version 106 as there has been a lot of interest in using the 'Readings Log'.

The readings log screen although intended to record data is more commonly used to view all channels of the USC (inputs and memory's) at the same time.

Creating a New Readings Log

The following example is recordings values from a USC701 programmed with UAP1052. A potentiometer is connected to CH1 and the mA output O1 is connected back to the mA input on CH2.

1. On the 'Open/New' screen double click on 'New Readings Log' or, press the 'New' command button while 'New Readings Log' is selected. The record properties screen will appear as shown.
2. In the name of recording box enter a description that you would use to later recall the results if required.
3. Enter the maximum number of records to be recorded = 500 (use between 10 and 99999).
4. Enter how often a record is added = 50mS, this recording time should be set while considering how fast the measured value is expected to change. Multiply this time by 'Max No Records' to determine how long the recording will take to complete.
5. Select 'Stop on maximum no of records'. The continuous option will overwrite the oldest readings and therefore never stop recording.
6. Press 'Record Replace' and adjust the potentiometer on CH1 to read '2.5 Per S'.

Store Readings

Readings are stored in recording sessions. Each session records CH1, CH2, P1, P2, Mem4, Mem5, Mem6 and Mem7. USC channels not sending values are recorded as 0. To receive messages from Mem4 to Mem7 you must add 'Send M?' commands for each memory location in the equation editor.

Name of Recording Session:

This description is used to find the record set for the current recording 'Session'. Multiple recording 'Sessions' are possible.

Recording Options:

Max No Records: Continuous Loop

Record every: Stop on maximum no of records

Enter the maximum number of readings to store and the time between each reading. Readings can be stored in a circular file where oldest readings are overwritten by new readings when the maximum number of records has been added or the option to stop recording may be selected.

Buttons: Record Replace, Open, Cancel

Recording Options:

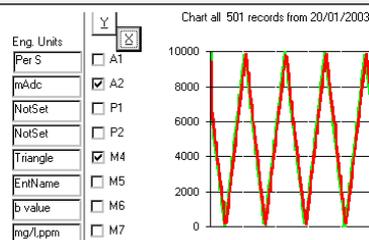
Max No Records: Continuous Loop

Record every: Stop on maximum no of records

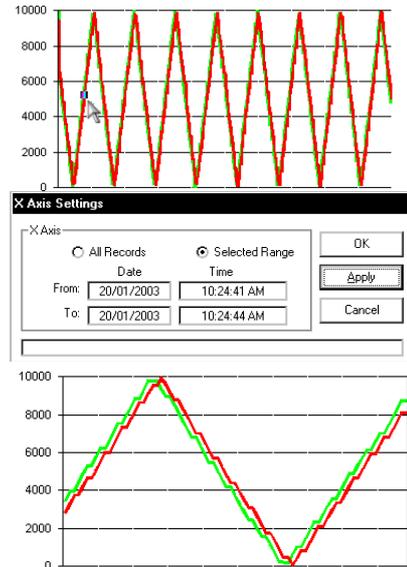
Enter the maximum number of readings to store and the time between each reading. Readings can be stored in a circular file where oldest readings are overwritten by new readings when the maximum number of records has been added or the option to stop recording may be selected.

Because I am recording values from UAP1052 the output is expected to change every 5mS however this is beyond the measurement speed of the USC701 20mS and of the recording equipment of 50mS. This means that when the application is set at maximum speed some data will be lost.

7. After 25 seconds the recording will be complete, select 'Chart Readings' and check A2 and M4 for display only as shown.
8. The X and Y buttons are used to zoom in on particular areas of the chart.



9. The X-axis zoom can also be achieved using point and click. Using the mouse left click the first point of interest (you must click on a charted line).
10. Move along to the next point of interest, hold down the shift key and left click again. The 'X Axis Settings' dialogue will appear with the required settings entered, press ok to continue.
11. The resulting chart shows a delay between M4 and CH2, this is the combined response time of the output and CH2.



PID Control

USC700 contains a wizard to create programs for the USC701. These programs are easy to make and enable technicians with no programming experience to create a large variety of signal conditioning modules from just one module.

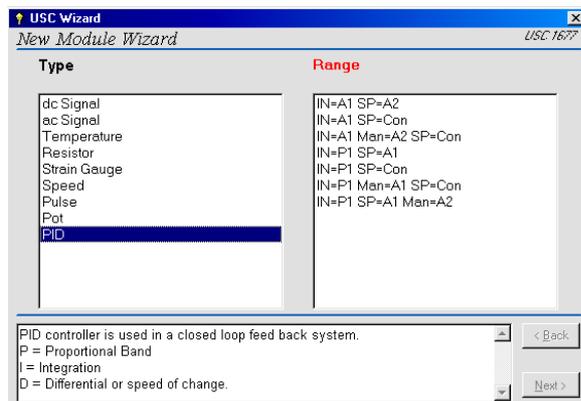
This guide is intended for end users and technicians to enable them to create a standard USC701 PID controller and tune the PID variables.

This process contains the following stages:

1. Use the PID wizard.
2. Use the modify wizard to change the input calibration to any special sensor types.
3. Program the USC701
4. Create a readings log for use while tuning.
5. Run the plant or process and tune PID values.
6. Modify the created program to add extra functions that may be required.

Step 1 Use the PID Wizard

1. Start the new module wizard in 'USC Config 106' by selecting 'File' > 'Open/New' > 'New Module Wizard'

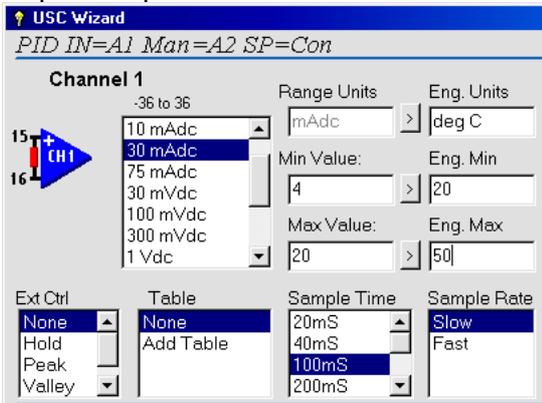


2. Select PID on the type list then select the required inputs on the range list.
3. In this example I will select the third item to create the following example.
 - a) CH1 = Measurement input Pt100 20 to 50 deg range.

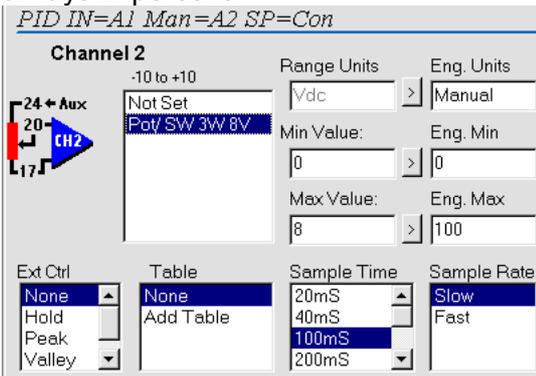
- b) CH2 = Manul control of output heating from 0 to 100%.
- c) Controller setpoint is entered into a constant using an AM702 in trim mode or 'USC Config 106' in trim mode.
- d) Input P2 is used to switch from auto to manual control.

◆ When selecting items on the range list the above information is listed in the helper screen.

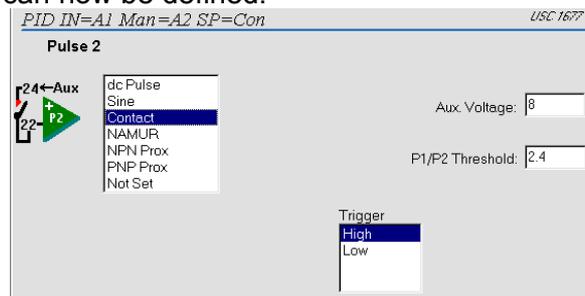
- Press next, the measurement input is presented first, however the Pt100 range is not available at this stage, this is changed later. Select the 30mA range and change the engineering units column to reflect the required input calibration as shown.



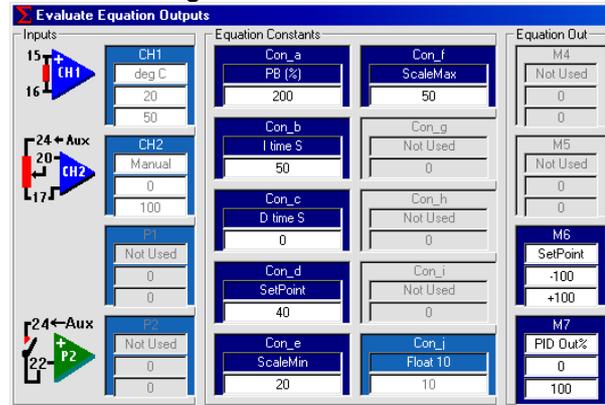
- Press next; the manual input is now presented. Only the pot input range is available, this is changed later. The engineering units of the manual control are always in percent.



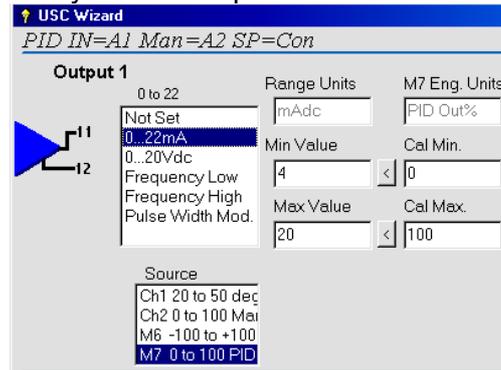
- Press next; the Auto/Manual control switch can now be defined.



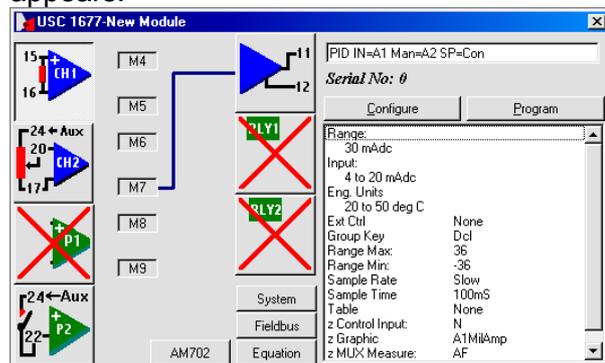
- Press next; the evaluate equation outputs screen is presented with instructions in the window below. ScaleMin and ScaleMax must now be changed to roughly match the output control range. As rule of thumb enter in the same range as the measurement input in this case 20 and 50 as shown. The setpoint and initial PID valued can also be set at this stage.



- Press OK and select the required controller output. The PID equation output is M7 and is always scaled in percent.

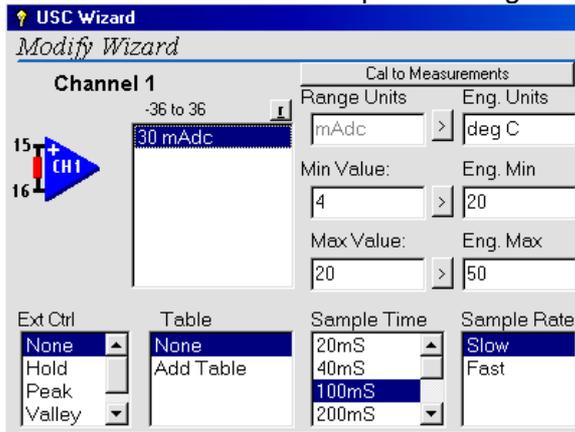


- Press next until the main prosperities screen appears.



Step 2 The Modify Wizard

10. Select CH1 as shown then press configure.



11. Click the small 'r' button to change the range on channel 1.
 12. Select type = temperature, range = RTD 2W then next
 13. Click on the required input range as shown.

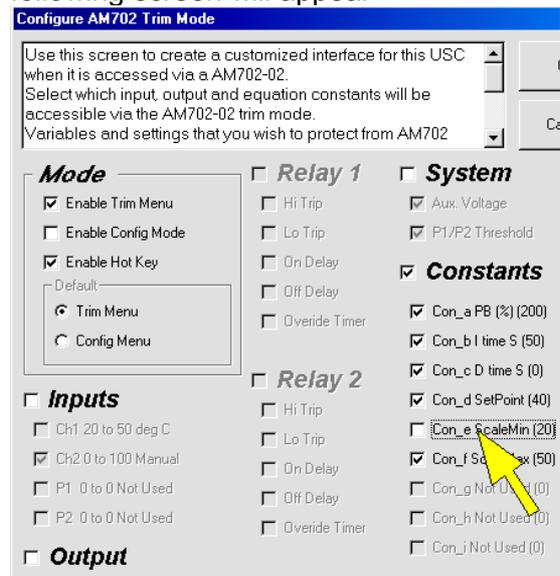


14. Press next until the main prosperities screen appears.

Step 3 Program the USC701

17. Program the USC701 with initial PID parameters Set 'PB (%)' = 200, 'I time S' = 50, 'D time S' = 0

15. Click on the AM702 button then configure, the following screen will appear



16. Uncheck ScaleMax and ScaleMin to prevent them from being changed during the tuning process.

◆ All the PID wizards define hot keys for use with the AM702. The keys are defined as the following where appropriate.

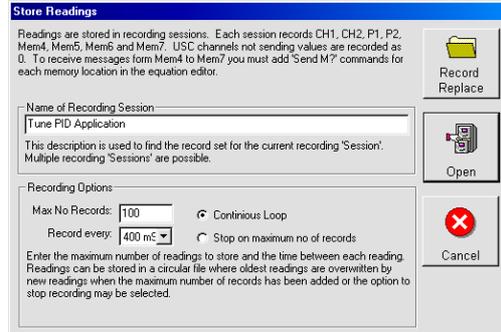
KEY	Description
UP	Display measurement input
DOWN	Display setpoint.
BACK	PID Out%
NEXT	Display position of manual input in percent.
ENTER	Goes to trim menu to adjust required flow rate and Trim menu.

Step 4 Create Readings Log

- ◆ The PID controller can be tuned without using the readings log. Also if the correct or close parameters are known for the plant no tuning is necessary.
- ◆ The readings log screen can chart the last 100 readings while still collecting new readings.

18. Start a recording session in USC700 by selecting 'File' > 'Open/New' > 'New Readings Log'

19. On the store readings screen adjust the 'Record every' setting so that the expected response of the plant can be viewed in 100 readings. I selected 400mS so that a plot of the last 100 readings will cover the last 40 seconds. Leave on continuous loop and press



record.

20. Press record replace.

- ◆ If attempting to tune a very slow system with a response oscillation of an hour or more it may be more practical to record more readings and set to stop on maximum number of records.

Step 5 PID Tuning

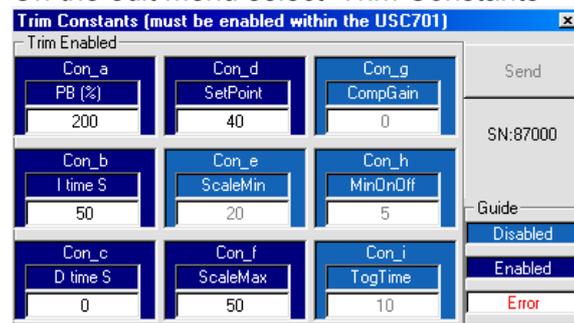
21. Run plant and reduce PB% until control oscillates or increase until control stops oscillating. If the plant has a manual control mode run in manual control and check that the measurement input is reading the correct values. The current values stored in the PID controller can be viewed on the readings log screen.

Eng. Units	Measurement		
deg C	27.99996	1:36:51 PM	A1
Manual	0.00615	1:36:51 PM	A2
NotSet			P1
LOGIC			P2
NotSet	53.2699	1:36:12 PM	M4
NotSet	20.0	1:36:12 PM	M5
SetPoint	40.0	1:36:51 PM	M6
PID Out%	33.08617	1:36:51 PM	M7

22. Click the chart readings tab to chart progress



23. On the edit menu select 'Trim Constants'

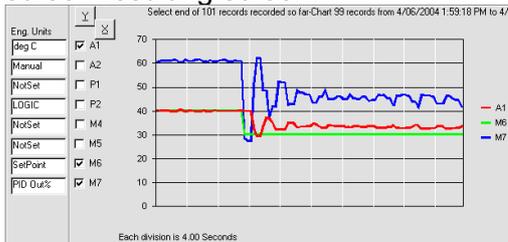


This screen will allow only the constants that have been enabled for the AM702 trim menu to be changed.

24. As my plant is responding slowly with no hint of oscillation I will progressively reduce PB% until oscillation occurs.

◆ *When changing the PID parameters the setpoint should also be changed. This ensures that the true affect of changing the parameters is apparent.*

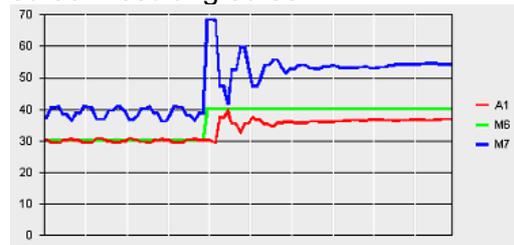
25. I changed by PB to 100 and setpoint to 30 then pressed send on the trim constants screen resulting screen.



26. It can be seen that the plant quickly adjusted to near the setpoint with some oscillation.

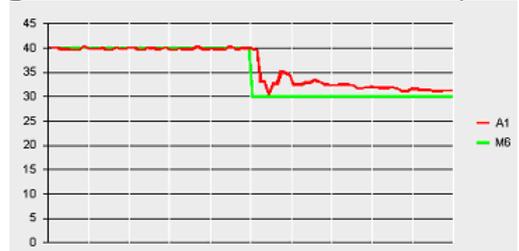
◆ *Tuning from here on depends on the type of response you want, quick and hard with overshoot, slow and soft no overshoot or somewhere in-between.*

27. I changed by PB to 120 and setpoint to 40 then pressed send on the trim constants screen resulting screen.



◆ *I now un-clicked the M7 check box to remove the PID output trace.*

28. I changed the 'I time S' to 10 and the setpoint



to 30.

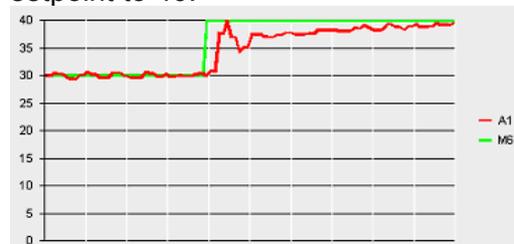
The reduction in the 'I time S' value caused the integration effect to correct to the setpoint quicker

◆ *Do NOT make 'I time S' = 0.*

29. I now did a rule of thumb 'D time S' setting

from the equation $D \approx \frac{1}{4 \times i}$

where 'I time S' = i and 'D time S' = D. I set 'D time S' to 0.025 and changed the setpoint to 40.



Notes

When 'PB (%)' = 100 the gain in the system is = 1

When 'PB (%)' = 50 the gain in the system is = 2

When 'PB (%)' = 200 the gain in the system is = 0.5

The proportional band ('PB (%)') will affect how savage the control action is. If slower more gentle control action is required then higher numbers should be entered into 'PB (%)'.

To prevent long correction times during start up the 'I time S' value is reduced by a factor of 10 during large proportional errors, this can result in too much spring during start up in responsive control systems. Increasing the 'I time S' by as little as 0.5 can overcome this problem.