ADuC8XX UART Emulation

Revision Number: **V1.0**

Last Revision Date: **2008-01-17**

Author: **David Guo (david.guo@analog.com)**

Department: China Applications Support Team (CAST)

Revision History

Date	Revision Description	Who
2008-01-17	Draft	David

Background:

Many customers have puzzles about the application of the ADuC8XX UART Emulation. This document is a simplified document about the correct operation. One thing needed to be noticed is that: debugging by UART is a simple Method of evaluation, its function is very limited. For example, the UART which is used to communicate with PC and Timer 1 can't be used in the code. Also there are some other limitations, you will find more in the below descriptions.

So we suggest you use the JTAG Emulator to debug the code.

Conditions:

Hardware: EVAL-ADuC841
Software: uVison3 V3.53
(This is a Evaluation Software, can be downloaded at https://www.keil.com/c51/demo/eval/c51.htm. The version limits the code size 2K
)

Step:

1. Connect the PCB and PC by the serial cable, power on the hardware board; open the Keil Software and set up a project, and add your c file into the project.
2. Configure relative parameters.
1). Open the "Options for Target ", as Figure 1 shown, select the type of MCU in your PCB;

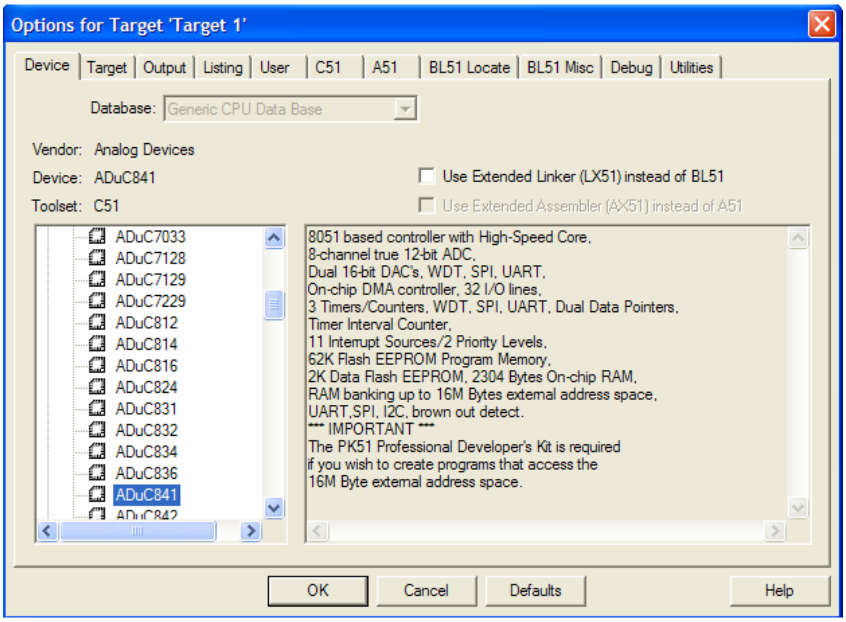

Figure 1

2). Configure the debug parameter. You should operate according to Figure 2. Pay attention to the places denoted by red lines.

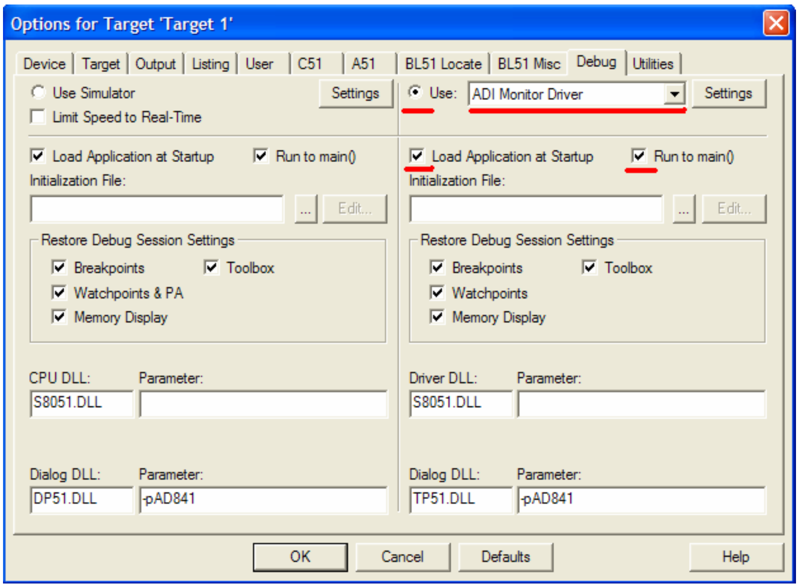

Figure 2

3. Translate and Build the target as Figure 3 shown.

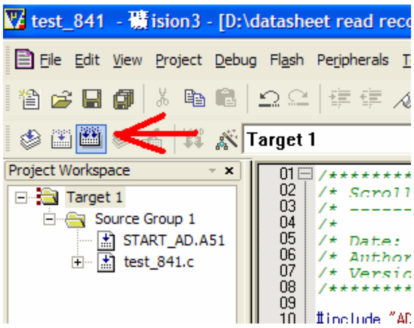

Figure 3

4. Now we can download the program and emulate the code. Before downloading, you need to press the Reset Key in your PCB. After the hardware reset, you can put down the "Debug" button as Figure 4 displays. Then you can enter into the debug mode.

Figure 4

5. As Figure 5 displays, we can only debug the code in the Disassembly window. The yellow arrow is the beginning of program.

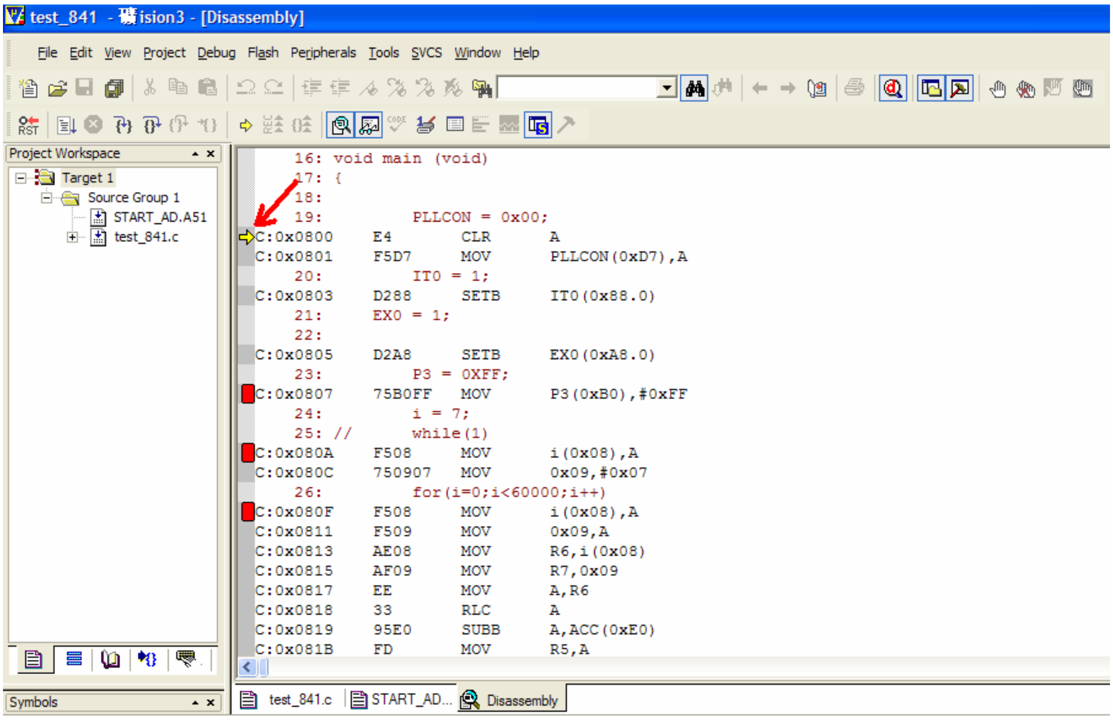

Figure 5

1). You can set the breakpoint at places where you hope the program stops. You can press the right key of the mouse at the place like Figure 6, and click the "Insert/Remove Breakpoint", then there will be a red point at the head of the current row.

2). Also, you can observe the variables you are interested to. In the "View" Menu, click the "Watch & Call Stack Window", then the Window like Figure 7 appears. You can input the names of the variables in the window.

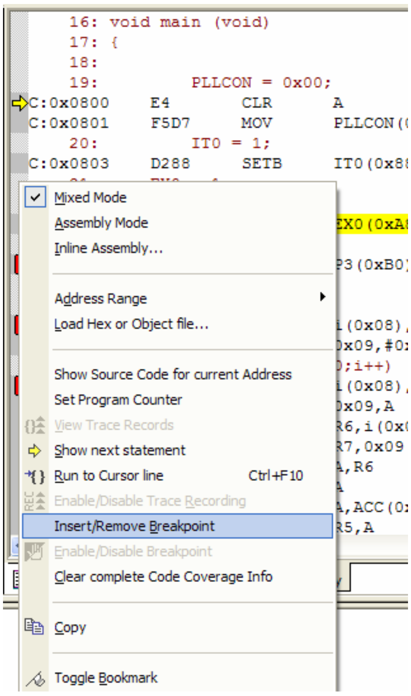

Figure 6

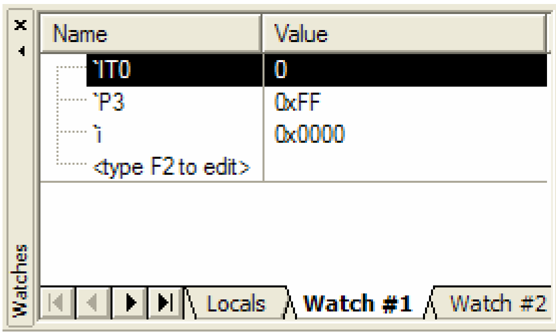

Figure 7

3). Now we can run the program step by step. Click the button as Figure 8 shown.

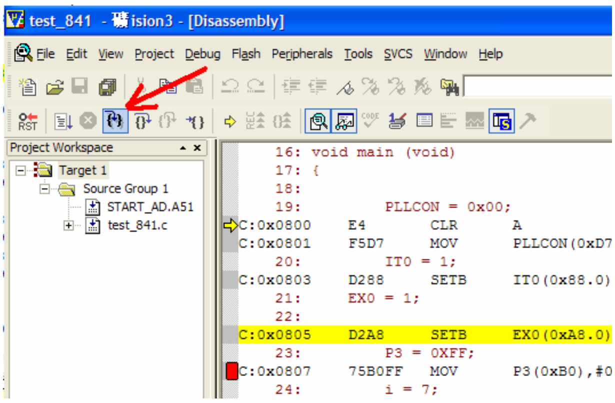

Figure 8

6. In the Figure 9, the yellow arrow denotes the current place of program. You can observe the change of the variable's value. Here the IT0 is 1, the variable "i" is 7.

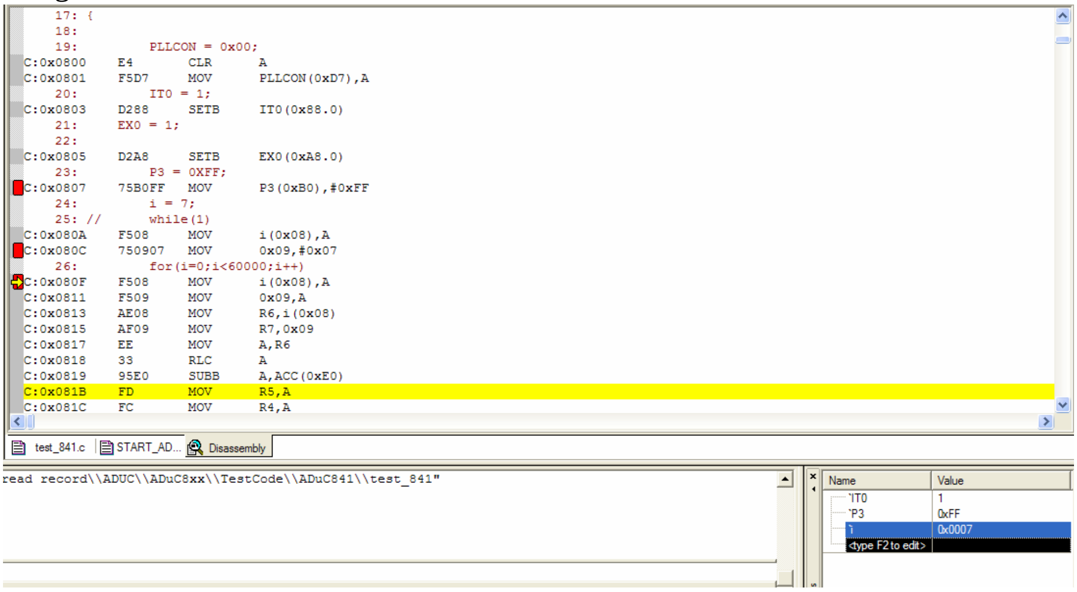

Figure 9

7. For UART Emulation, you can't set breakpoints in a cycle, for example, the "for" cycle and the "while" cycle. And you can't stop the program when the program is running, if you try to stop the program, then a warning happens as Figure 10.

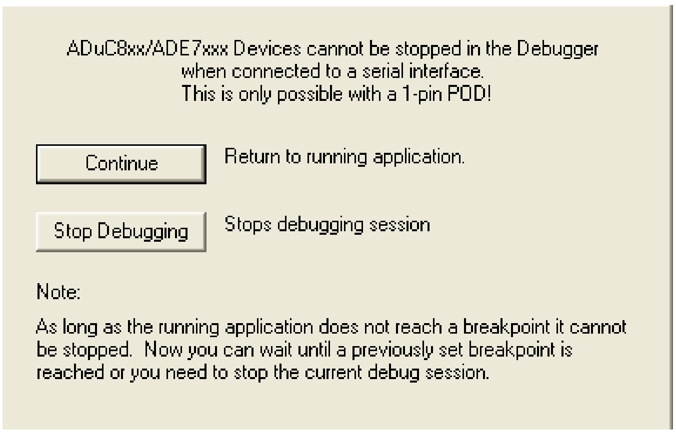

Figure 10