

Multiple CPUs with CCS

Suppose you are working with the RT-IHU, which has two processors. You have two debug probes (XDS-110 equivalent) and you plug first one, then the other into the host computer. But a) you'd kind of like not to be pulling USB connectors in and out due to limited cycle count, and b) sometimes you forget, leave both plugged in, and if it asks you to choose one you are screwed with a ccxml file syntax error. (Thanks TI!) Here are several levels of solution to that.

How to fix the "syntax error"

What IS the error?

TI in their infinite wisdom decided to require that a serial number specified in the ccxml file be 8 characters long, while by default the XDS-110 has only 5 characters. And then TI in their infinite wisdom, when you choose a particular device, does not CHECK that the s/n is long enough. It just edits the ccxml file with the short serial number. And then TIITIW do not give a reasonable error message, but just says "Syntax Error". Thanks guys.

What do you do about it?

Make a hardware fix for a software stupidity. Change the XDS-110 serial number. You do this as follows on Linux, (and similarly on Windows):

You need a program called xdsdfu which in Linux is in `~/ti/ccsv8/ccs_base/common/uscif/xds110`. It is very similar in Windows, but with the stupid back slashes rather than normal ones. You also need a terminal in either Windows or Linux. I'll use Linux syntax below, but the commands are identical.

```
./xdsdfu -e !This will give you info about the debug probe you have plugged in, including its current serial number.  
./xdsdfu -m !Put the probe in firmware update mode  
./xdsdfu -n 111 -r ! Add 111 to the end of the s/n giving it 8 character total
```

Do that to both probes. Now when CCS asks you to choose a probe, it will work. Except that after you have chosen one, it won't ask again. There are two solutions to that:

1. Go into 'debug configurations' (the pulldown beside the little debug bug in CCS), select your current configuration, and push the button labelled "Reset Debug Probe Connect S/N". Now it will ask you again the next time.
2. Fix up multiple debug configurations (in a later section)

But I don't use CCS! I'm a purist!

I bet when you load your code, you somehow specify a ccxml file. See below about adding multiple ccxml files and just choose the right one. Or perhaps you will get the same message as I described above? I don't know.

The simpler to use, harder to do solution

The plan here is to create a separate ccxml file, aka "target configurations", for each debug probe. One advantage is that it might be possible (but believe me, I have not tried) to run a debugger on both processors simultaneously. Aggh!

A new target configuration (or ccxml file)

You might be able to simply create a new ccxml file and edit each one to have the right s/n in it. In fact that seems to be the best way to do it with CCS V9. See the instructions below for that. But to get mine working on CCS8, I followed the instructions from the TI Community helpers, which is more or less as follows (again this is all using CCS)

1. Under the "View" menu, choose "Target Configurations"
2. In that view, hit the "New Target Configuration File" button.
3. Give it a name, and then select XDS110 USB Debug Probe for 'connection' and TMS570LS0914 for Board or Device.
4. Now select the "Advanced" tab at the bottom and select XDS 110 etc (the top line).
5. On the right should appear 'connection properties'. For debug probe selection, choose 'select by serial number'
6. Enter the serial number (all 8 characters)
7. Leave everything else at the default and save
8. Now in the Target Configurations window again, your new config is under User Defined. Open that folder, right click on the new

config and select "Launch Selected Configuration"

Ok, now you will see the beginning of a startup, but it will stop without loading any code. Don't panic as I did. You just have to create a debug configuration that tells it to load. <Sheesh>

A new debug configuration

If you go back to "Debug Configurations..." you should see a new one with the same name as the target config you just made. Edit it as follows:

- Under the Program tab, in the Project box, click on Workspace and select your project. Now under program it should say "\${build_artifact:projectName}"
- Below that, select "load program" if it is not already checked. (You can create another Debug Config with "Load Symbols Only". This allows you to connect to a running system without reloading code and starting from the beginning.)
- Under "Target" tab, and "Auto Run and Launch Options", I said "Run to symbol startup". "Main", the default, is not very interesting. I also have "connect to the target on debugger startup". That may happen automatically.
- Under "Flash Settings" if you scroll down you see "Erase Options". The default previously was "Necessary Sectors Only" and that is faster. It may be the right thing later when we have a boot loader too.

Wash Rinse Repeat

Now you have to do the SAME thing for the other serial number.

Alternate method for CCS9 (and probably for 8 as well)

- Use the default program loader with both XDS110's plugged in. It will ask you which one you want to use. Be sure you know both serial numbers and then choose one. I assume below that you chose the primary processor. Now that ccxml file will be rewritten to specify the particular serial number.
- Under the "view" menu choose "target configurations" and you should be able to find the default ccxml file that just got rewritten. It is probably located in ~/ti/CCSTargetConfigurations. Go there and make 4 copies of that file named something like "Primary Load", "Primary Noload", "Secondary Load" and "Secondary Noload". Edit the two 'secondary' ccxml files and replace the primary serial number with the secondary one.
- Go back to the target configurations view, and for each file, do the following:
 1. Right click the file and choose "Launch Selected Configurations".
 2. The launch will fail. Close the launch and go to the "bug" icon pulldown and select "debug configurations"
 3. For each of the new config files, go to the "program" tab, and beside "project" hit the 'workspace' button and choose the project name. Apply and close.
 4. Now you should be able to choose one of the 4 debug configurations on the 'debug' pulldown when you want to load or debug. Don't forget to change the symbol or the erase options as mentioned above if you wish.

AND YET ANOTHER Alternate Method for CCS10

I don't know whether something has changed, or it is just my head, but in fact this was the ONLY way I could make it work in CCS10. This is just a variant of the CCS9 method above. In fact, you do the above except for the following:

- You want the ccxml file to be in a file folder local to your project. In Golf-Tee, this is in "targetConfigs".
- Go to "Debug Configurations..." in the pulldown beside the green bug icon on the top row.
- (BTW, you will see "Reset Debug Probe Connection S/N" under the "Main" tab near the bottom. If you push this it forces it to ask which one you want to use.)
- Make sure that "Use Default Target Configuration" is **not** checked. Then in target configuration click "Workspace" (not file system) and find your ccxml file and select it. It should come out in this format \${workspace_loc:/Golf-Tee/targetConfigs/RT-IHU-Primary.ccxml}. Having that "workspace_loc" format seems to be important. It does not work if you go to 'file system' and give it a full path.
- Set up with multiple devices serial numbers and load or don't load code as above.

VOILA!!!

That should do it. Now when you debug with CCS, you chose which XDS-110 you want, and you can actually name the debug configuration primary and secondary or whatever you like.