

Creating an ASM Project in CCS

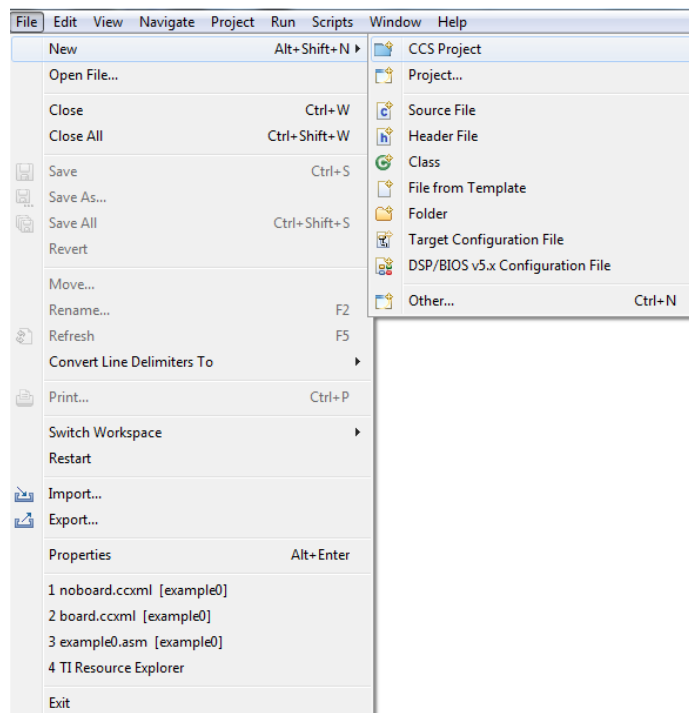
Introduction

The purpose of this document is to enable a student to quickly create a project under CCS for assembling, and linking an assembly file. To complete this tutorial you will need two additional files: **ex0.asm** and **KG_RAM_Link1.cmd**. After obtaining them from our class web page, copy the **KG_RAM_Link1.cmd** file into a folder called **c:\4744\ccs\projects\example0** and put the **ex0.asm** file on your desktop.

The file **ex0.asm** is an assembly file that illustrates various assembler directives and TMS320F28335 DSP assembly code. **KG_RAM_Link1.cmd** is a linker command file that is required to set the proper addresses for each code section and to instruct the linker how to build the final machine code output. This linker command file is an adaptation of Texas Instrument's linker command file "28335_RAM_Ink.cmd."

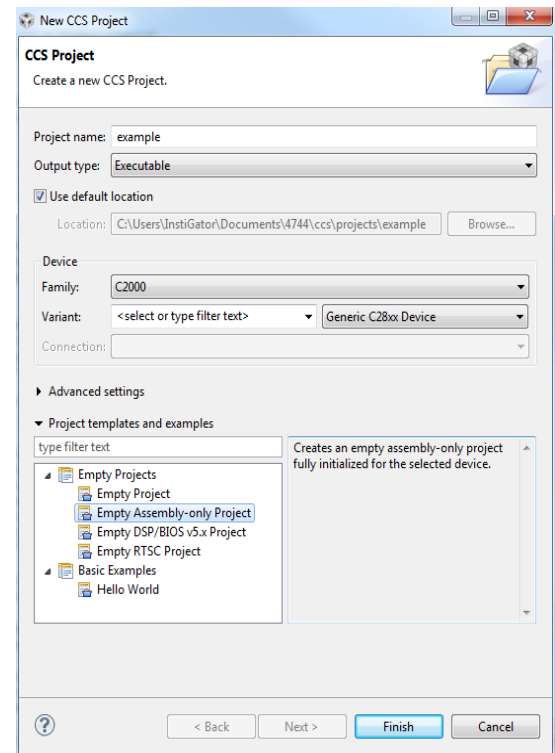
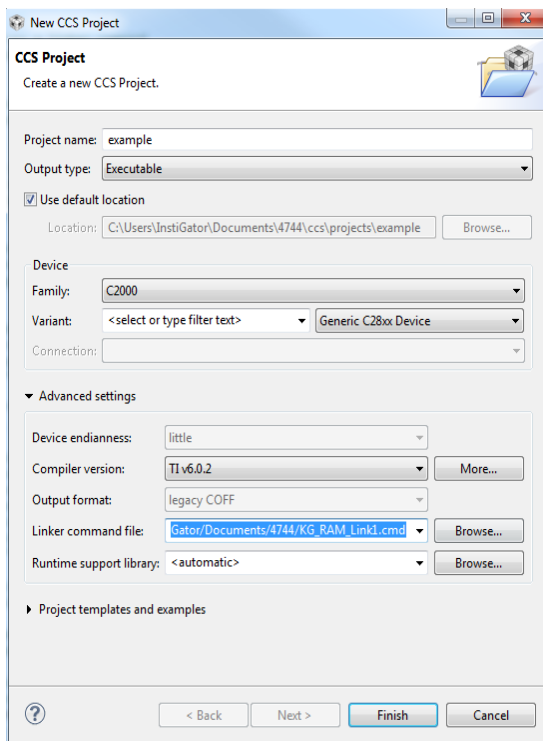
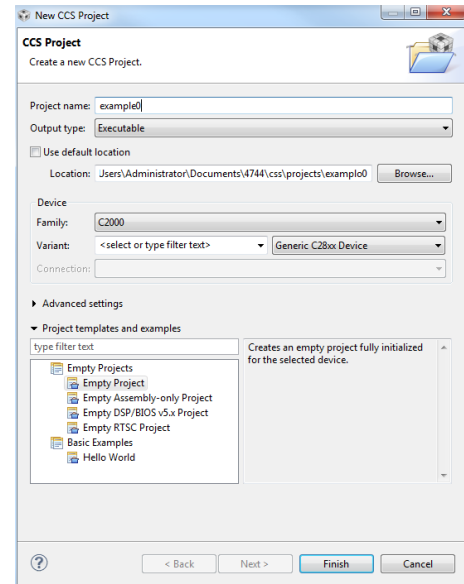
Procedure

1. This tutorial assumes that you already have Code Composer Studio installed and have set your workspace folder location on your hard disk. For more information on this topic see the tutorial [CCS_Installation_Instructions.pdf](http://mil.ufl.edu/4744/docs/CCS_Installation_Instructions.pdf) (at http://mil.ufl.edu/4744/docs/CCS_Installation_Instructions.pdf).
2. Open CCS and create a new project via the following commands (as shown below): **File → New → CCS Project**.



Creating an ASM Project in CCS

- Type in your Project Name, e.g., **example0**. Next, **un-check** the box “Use default location” to place the project in another location other than your workspace folder. It is recommended that you create a new folder called **projects** underneath the folder you created in the CCS installation tutorial, *CCS_Installation_Instructions.pdf*. In this folder you should then create a new folder for every project, e.g., **c:\4744\ccs\projects\example0**. Note: CCS does not create a specific file to contain the project information but instead looks for files under a particular name in a given folder. This will be discussed further later; the important rule is to create a folder for every new project (examples, lab code, experimentation, etc.). See the screen snapshot to the right. Select “Next >”.
- Select the ‘Advanced settings’ tab. This should drop down a new menu for you to select options. Click the drop down box for the Linker command file. Point to the *KG_RAM_Link1.cmd* file. This file is located on the website and in the *example0.zip* file. You will use this linker file for the entire semester. Next, select the ‘Project templates and examples’ tab and then highlight the *Empty Assembly-only Project*. Click finish. See the screenshots below.



Creating an ASM Project in CCS

We now have created the project dependencies (libraries to be used for assembly and memory map for the machine code output) and will now create the actual assembly file.

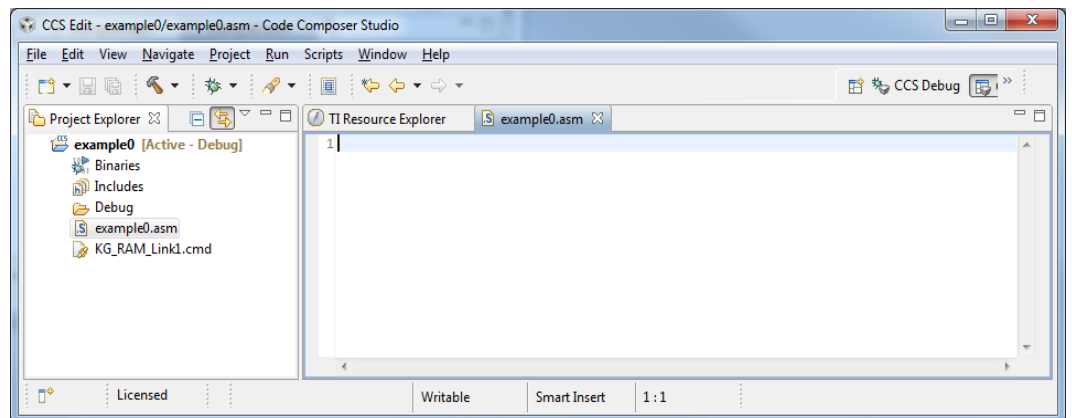
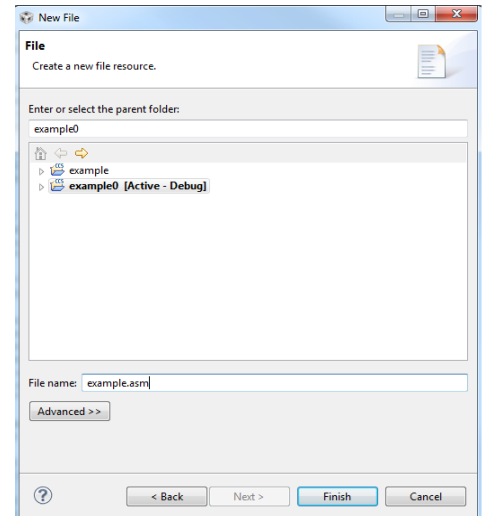
5. To create a new ASM file press: Right click the project in the Project Explorer window and then go to **New** → **File**. This can also be done by selecting **File** → **New** → **Other** → **General** and then selecting File. For this example, I used the filename example0.asm.

Note: Make sure to write the .asm extension following your file name. Failure to do this will generate an error when trying to compile your code. If you type the .asm extension correctly, a line number “1” will appear next to the cursor in the new file (window) as show below.

6. You should now see this created project. Note the included files in the left most window. The new asm window has a “1” indicating line number. At this point you can type in new ASM code in the center window. However, since you probably are not proficient at writing F28335 DSP assembly yet, you should copy

in the assembly code from our example code file that you previously copied to your desktop: **ex0.asm**. To do this press **File** → **Open** **File** → and then select **ex0.asm** from the directory where you originally placed the file.

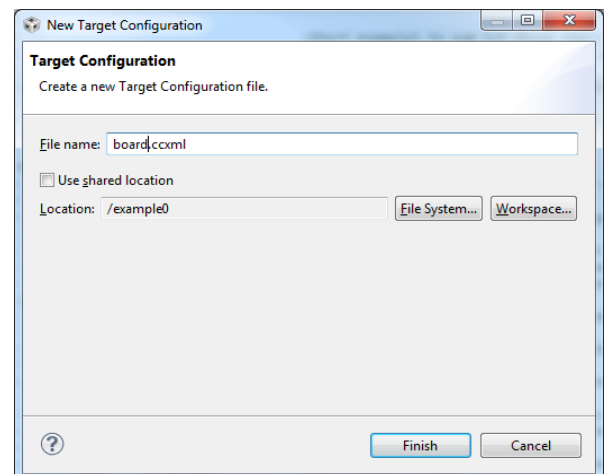
When **ex0.asm** is open, copy the contents using **Edit** → **Select All** and then **Edit** → **Copy** in the **ex0.asm** window to copy the code to your new **ex0.asm** window with **Edit** → **Paste**



We are now almost ready to compile the code but must create a target configuration file to tell CCS what emulator/programmer you will be using to download the code.

7. Create a new Target Configuration File by pressing **File** → **New** → **Target Configuration**. The target configuration file will identify the type of emulator CCS will use for programming and emulating the DSP board, i.e., it tells CCS where to load the program. In our case the programmer/emulator hardware is actually on our lab board.

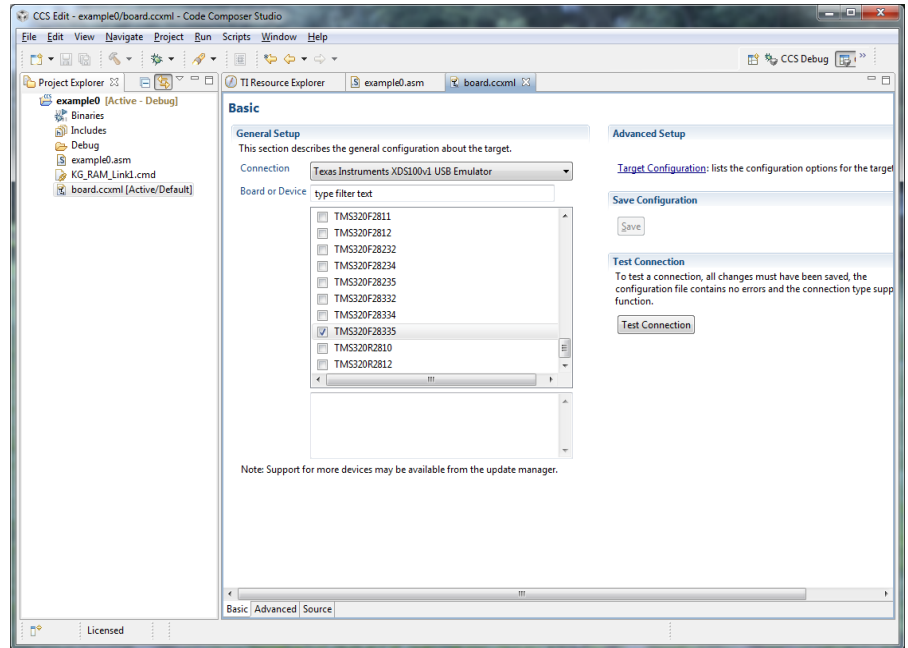
Type in a filename **board** to match target’s location; the **cxxml** file extension is automatically added. Next, uncheck the **Use shared location** box and then press



Creating an ASM Project in CCS

Workspace to select the current project location. If the current project location shows up in the “Location:” field (i.e., /example0), you can leave **Use shared location** checked and just press **Finish**. We want to make sure we add the target configuration file to our current project files. Click **Finish** after the location and filename have been set.

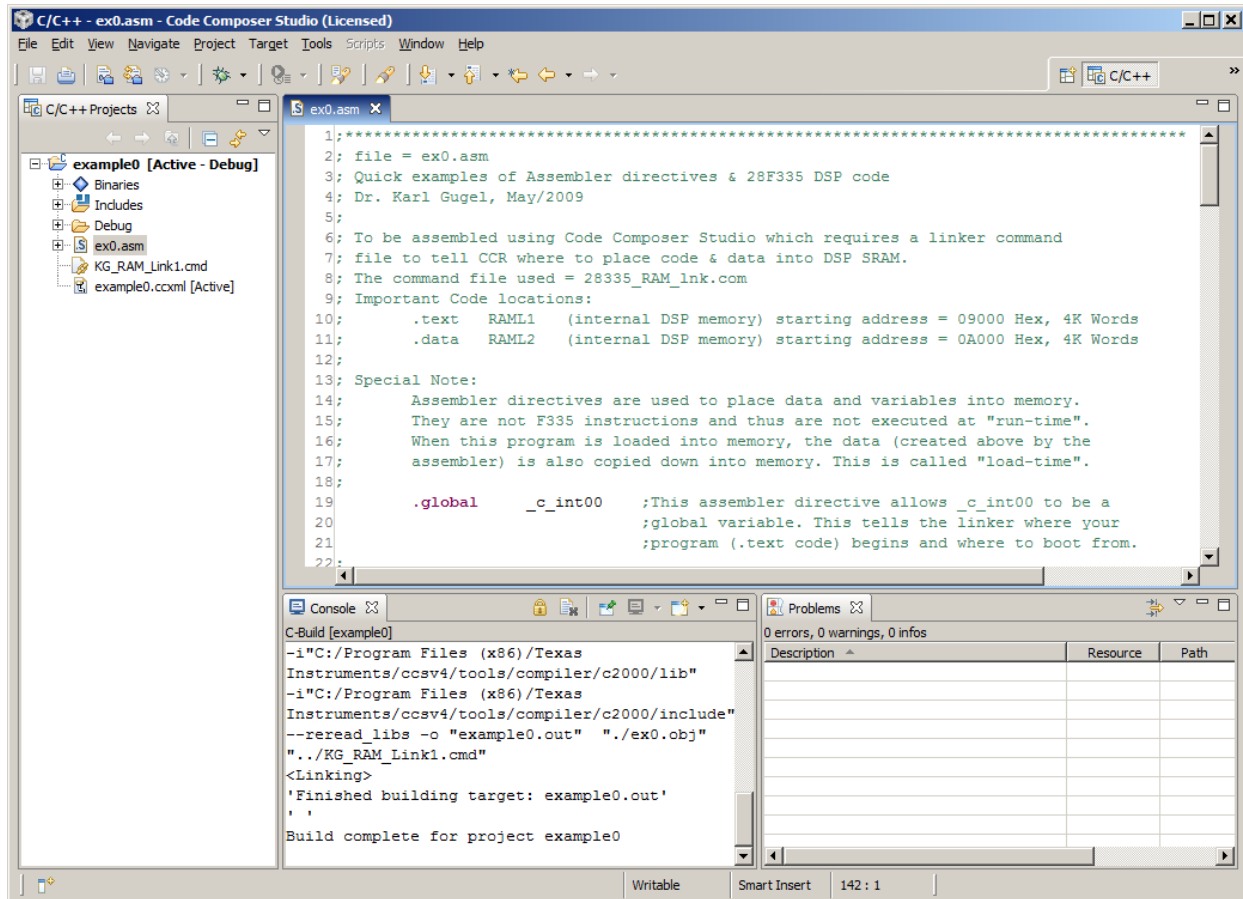
- For the board.ccxml file look under “Basic” for “General Setup” and then for “Connection.” Select **Texas Instruments XDS100v1 USB Emulator** in the **Connection** pull-down. Next check the **TMS320F28335** device in the **Device** scrollbar area (near the bottom). See the correct settings to the right. Our target DSP is the F28335 and the emulator circuitry we have placed on the lab board is equivalent to that used by the XDS100 originally designed as a stand-alone programmer by TI.



Select “Save” under “Save Configuration.” You can now close this window since no future modifications will be made to it when you are writing new code or debugging machine code.

- Finally we are ready to build (assemble & link) the project. Select **Project → Build Project** or right click the project in the Project Explorer window and click Build Project. You should see zero errors and warnings when you build the project containing ex0.asm (see below) and you should also see that a machine code output file is generated called example0.out (See “Finished building target: example0.out” in the console window.)

Creating an ASM Project in CCS



This ends the project creation/build tutorial. Refer to the debug tutorial to run & test machine code Example0.out. Also, see the Frequently Asked Questions section on the next page for more information relating to time saving time when performing CCS code development.

FAQ Relating to ASM Project Creation & Build

1. Do I always have to save my ASM file every time I make a change before building?

No, every time you press the Build Active Project icon, your ASM file is automatically saved first.

2. I want to start writing & debugging code right away. How do I start the debugger?

*Create a new target file with a new name. i.e. NoBoard.ccxml In this new target file, select **Connection: TI Simulator** and then click F283x CPU Cycle Accurate Simulator. Next close this window to save the information. You can now select between the two target files by right clicking one and setting it "active". Now press the little bug icon. (See the tutorial: Debug_ASM_CCS_Project.pdf for more details on this process.)*

3. Do I need to create a new workspace for every lab?

No, you can (and should) create all you projects or labs in the same workspace. You can just hide the projects you are not using by pressing the "-" symbol to the left of the project name.

4. Do I need a different Linker Command File for every project?

Creating an ASM Project in CCS

No, instead point the linker to the same file `KG_RAM_Link1.cmd` used in your first project. Or simply copy the `KG_RAM_Link1.cmd` file into each new project directory and point the linker to it.

5. Can I create two or more projects in the same directory?

No, you need to save each project in a different directory. If you try to save two projects in the same folder, an error message indicating that your new project overlaps the location of another project will be shown in the screen. It is recommended that you create a directory for each lab (i.e. labs 0-9) at the beginning of the semester so that when creating a new project, all you have to do is select the specific directory created for that project.

6. When I have multiple projects open, how do I choose which project is active?

*This is performed by right clicking on the project that you want to make active and select **Set as active project**. All other projects will be ignored when you build the active one.*