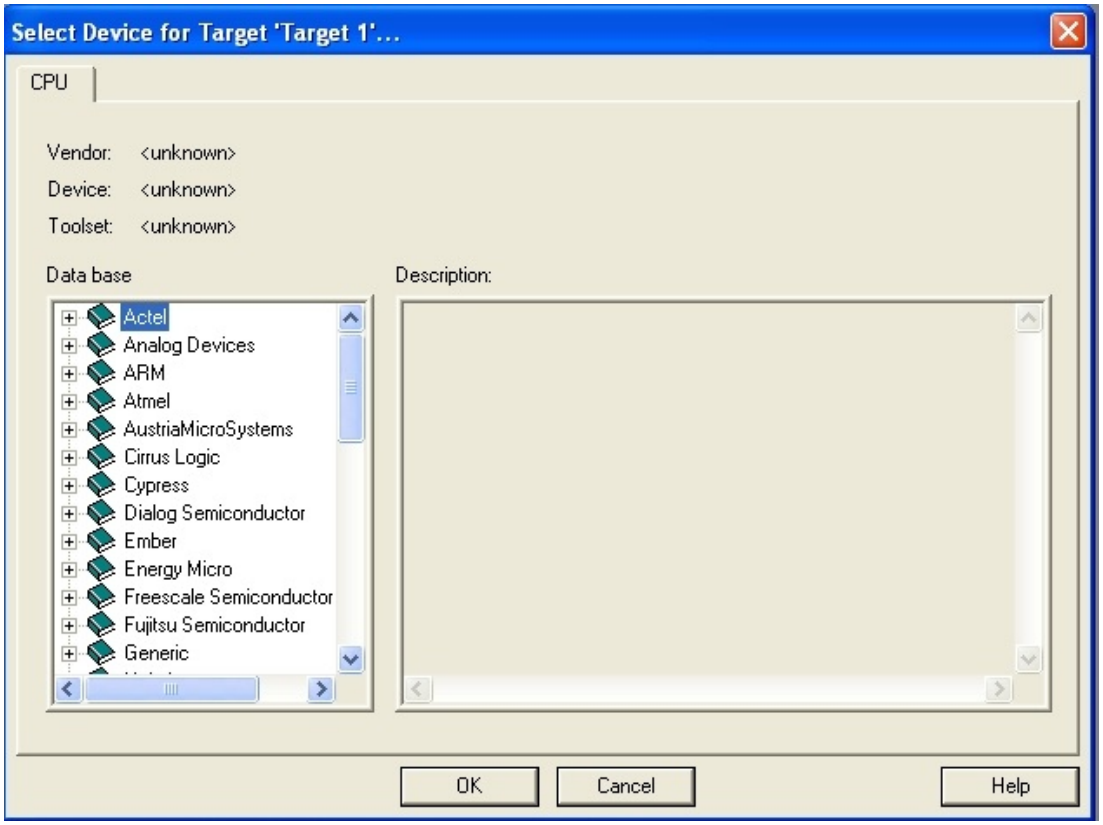
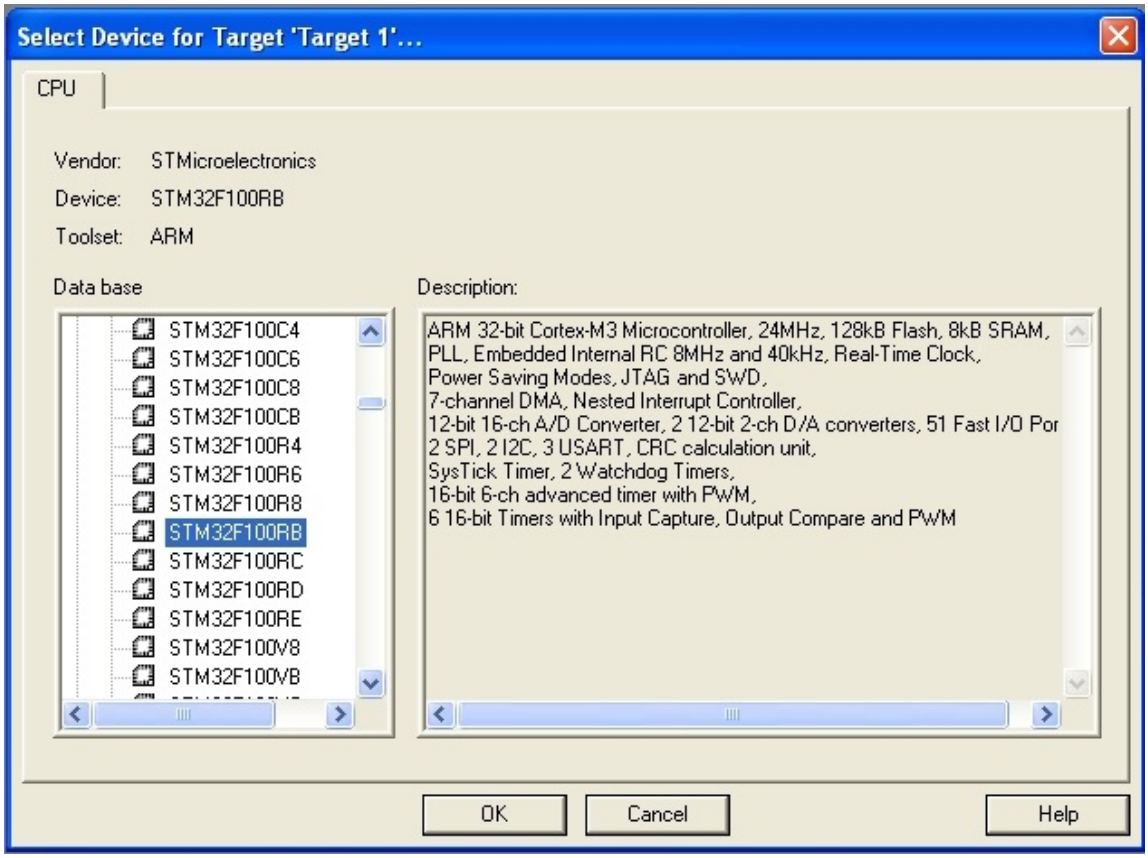
Procedure to create a new project using the Keil IDE ver 4

0) Create a folder (directory) for your project
1) Download file keilstartup.zip from the class webpage to the project folder and unzip it.
2) Start the Keil IDE software. If it starts up with an open project, close the project..
3) Create a new uVision project. On the project tab select New uVision Project:
   Project > New uVision Project
4) A Create New Project window will open.
   Navigate so that the directory you created is shown in the Save In box at top of window..
   Enter the name you wish for the project in the File Name field at the bottom.
   Click Save.
5) A Select Device window will open with a long list of company names:

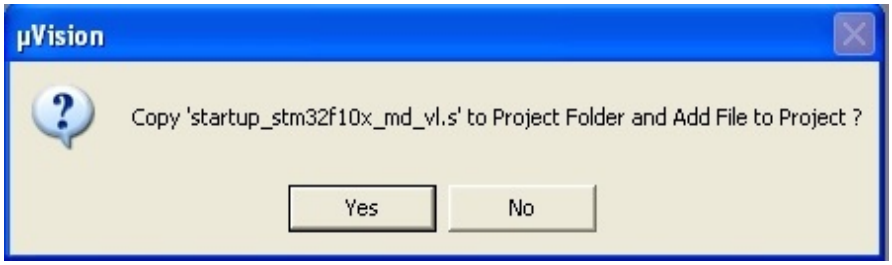


Scroll down to STMicroelectronics, click on the + symbol and then scroll down to select the STM32F100RB microcontroller:



Note that in the description window appears a summary of the resources that come with this microcontroller. Click OK.

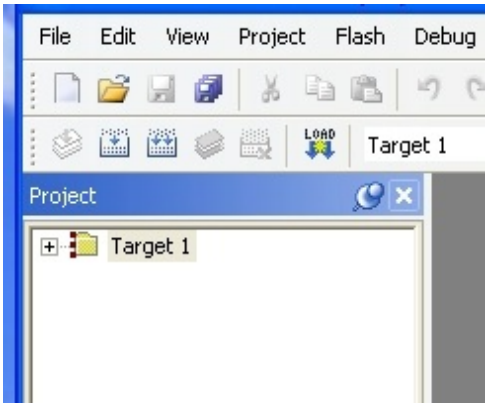
6) A prompt to add a start up file will appear:



Click Yes.

7) If you have a source file you have begun writing, such as a modified Blinky.c file, copy that file to the directory where the new project is but rename the file to match the name you have given to your new project. NOTE: project names and files must not have spaces in them.

8) On the left of the screen is a project box like this that shows the files associated with your project:

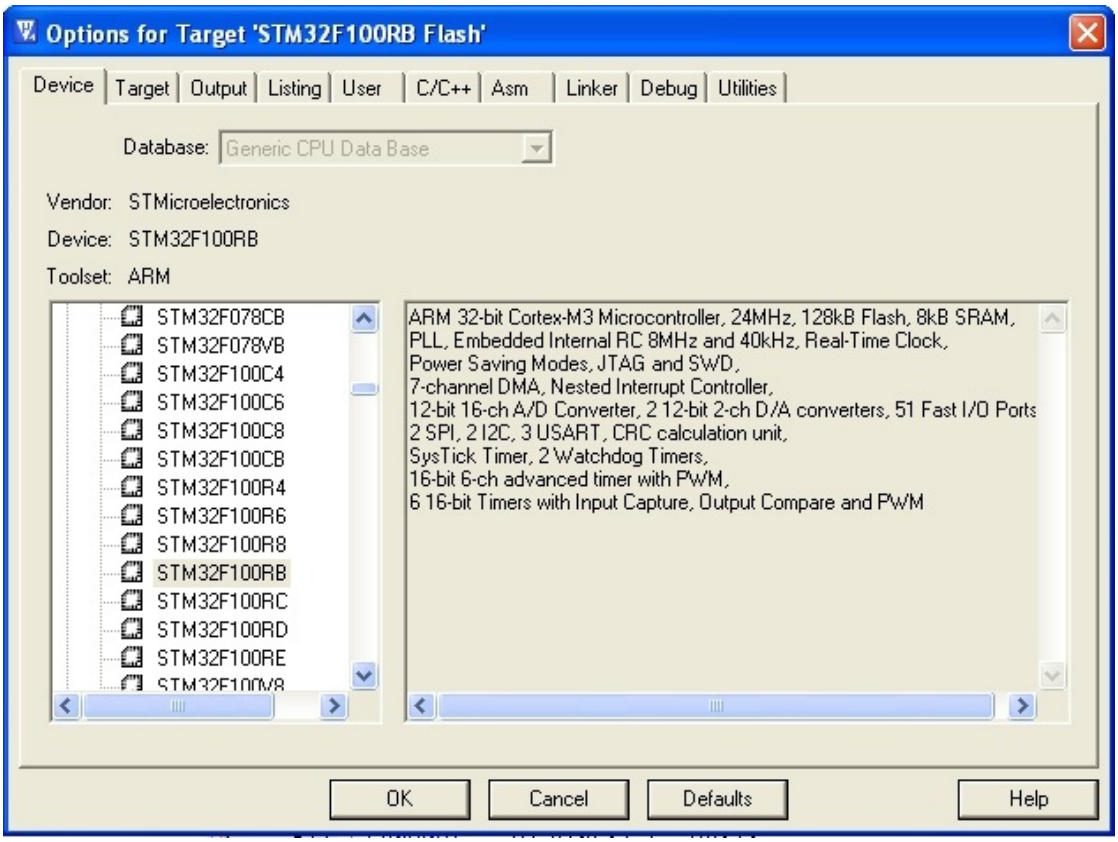


Click on the + just left the word Target 1 to get this:

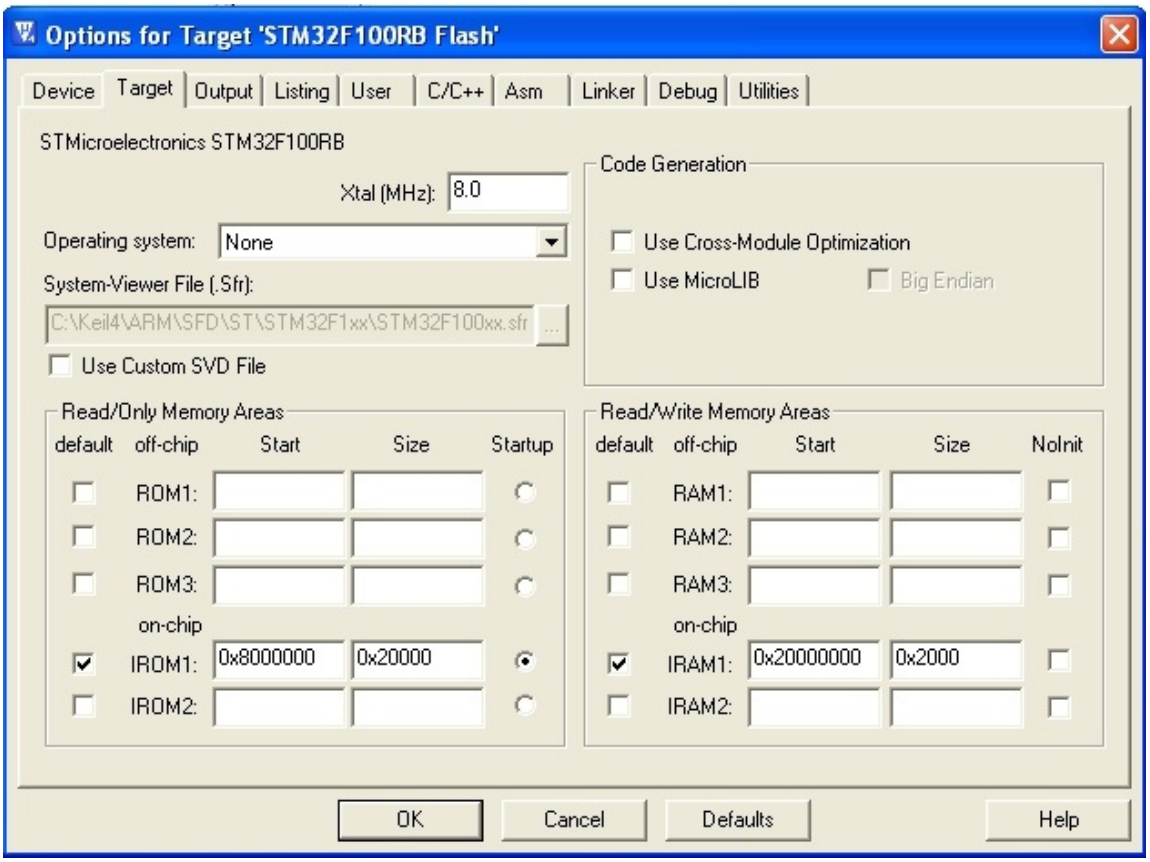
Right click on the line Source Group 1
Left click on Add Existing Files to Group ‘Source Group 1'

Another window will open showing files in the project.
Select your source file (.c file) and click Add.
Select the **system_stm32f10x.c** file and click Add
Close the window.

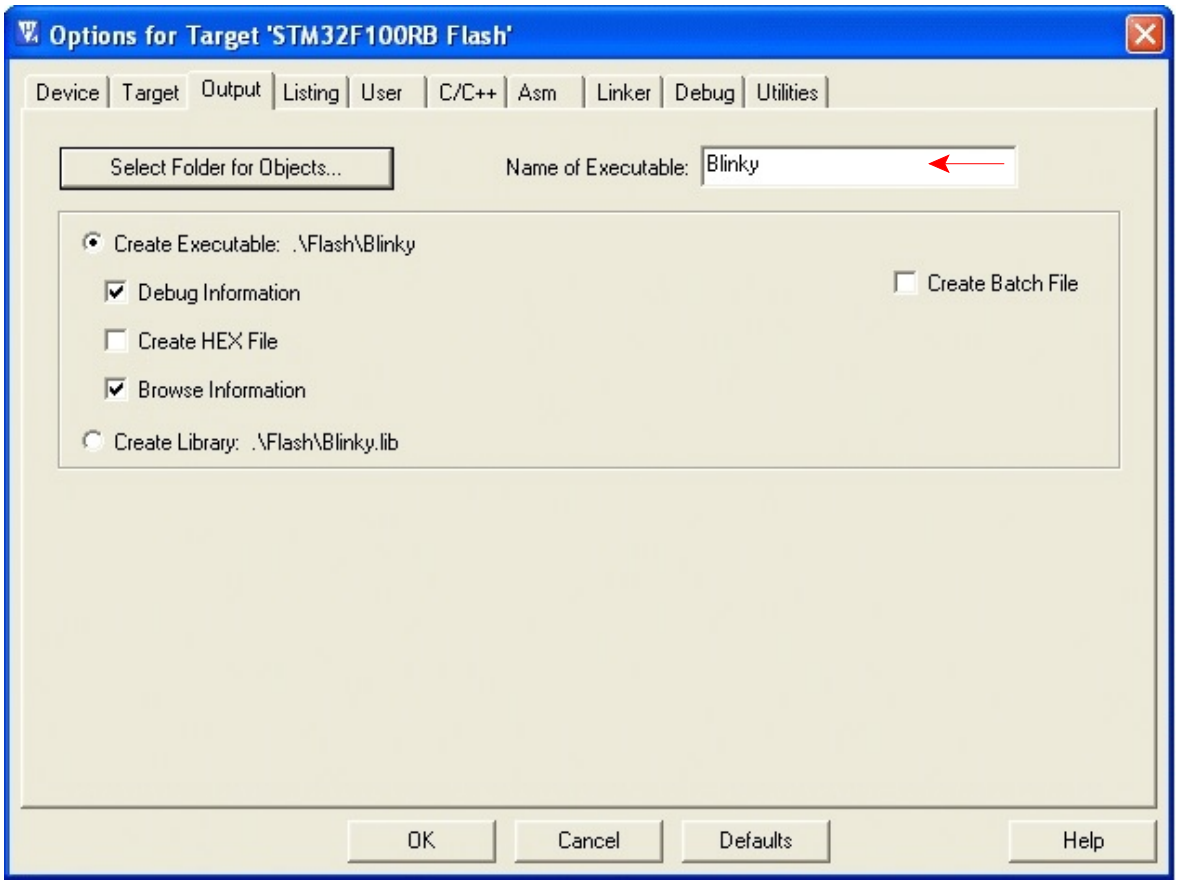
9) On the Flash tab select Configure Flash tools (**Flash > Configure Flash Tools**). A window like this should open (it is possible that it will open with a different tab active. Click the left tab which is labeled Device to get this screen). STM32F100RB should be shown.



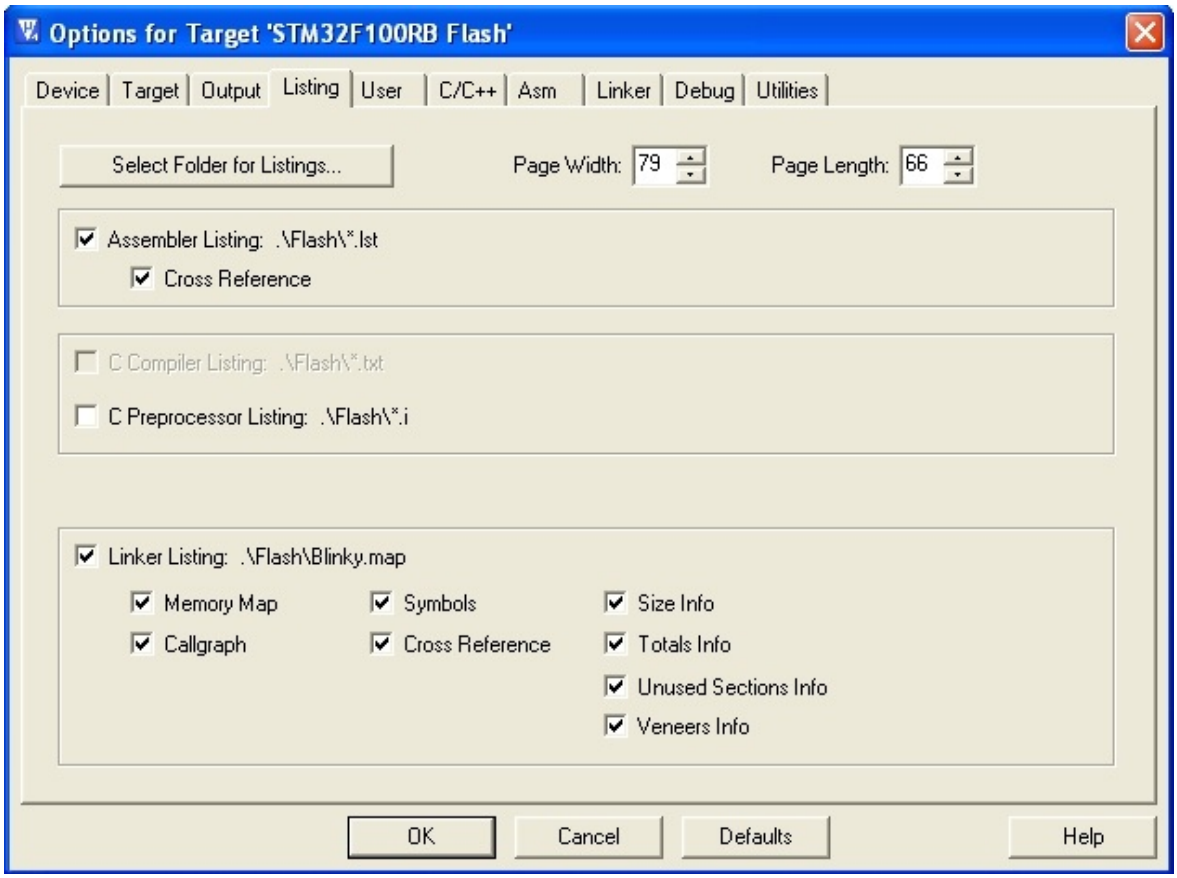
Clicking the Target tab give this (default values should be ok):



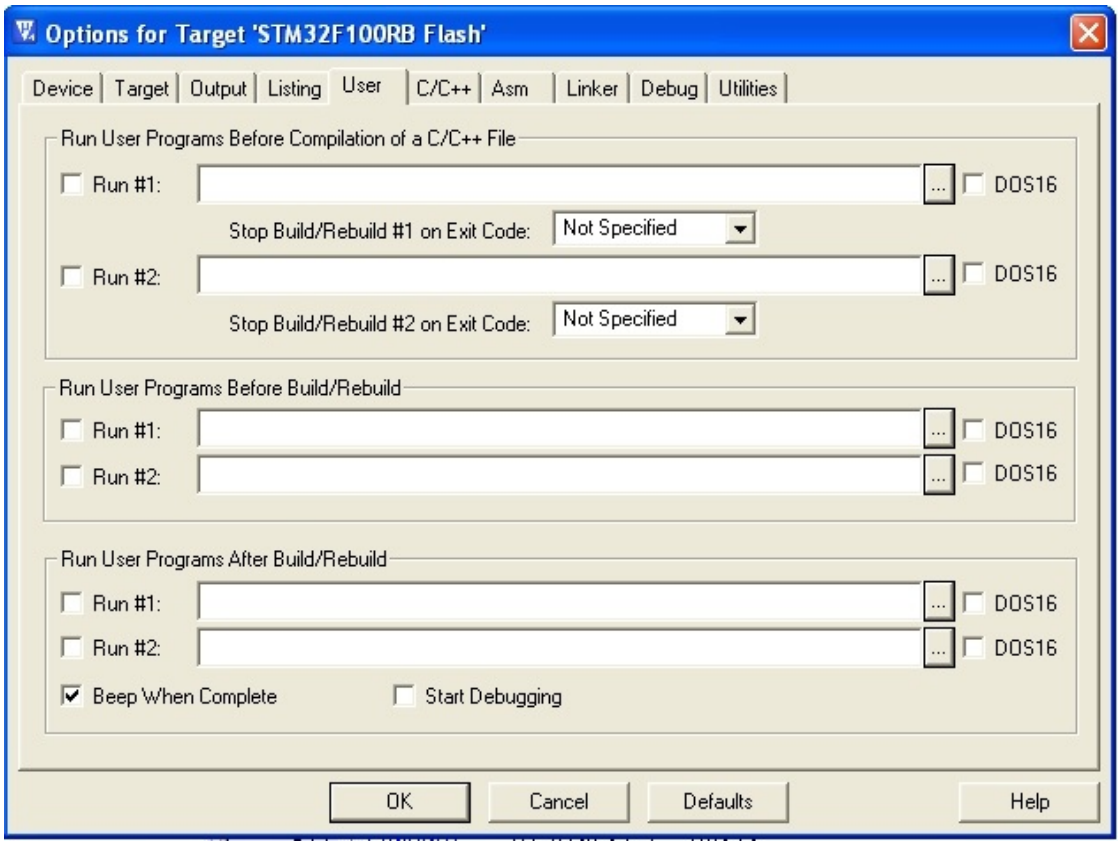
Clicking the Output tab should reveal this window. By default the name shown should be the name of your project (Blinky happened to be the project used to get this screen shot):



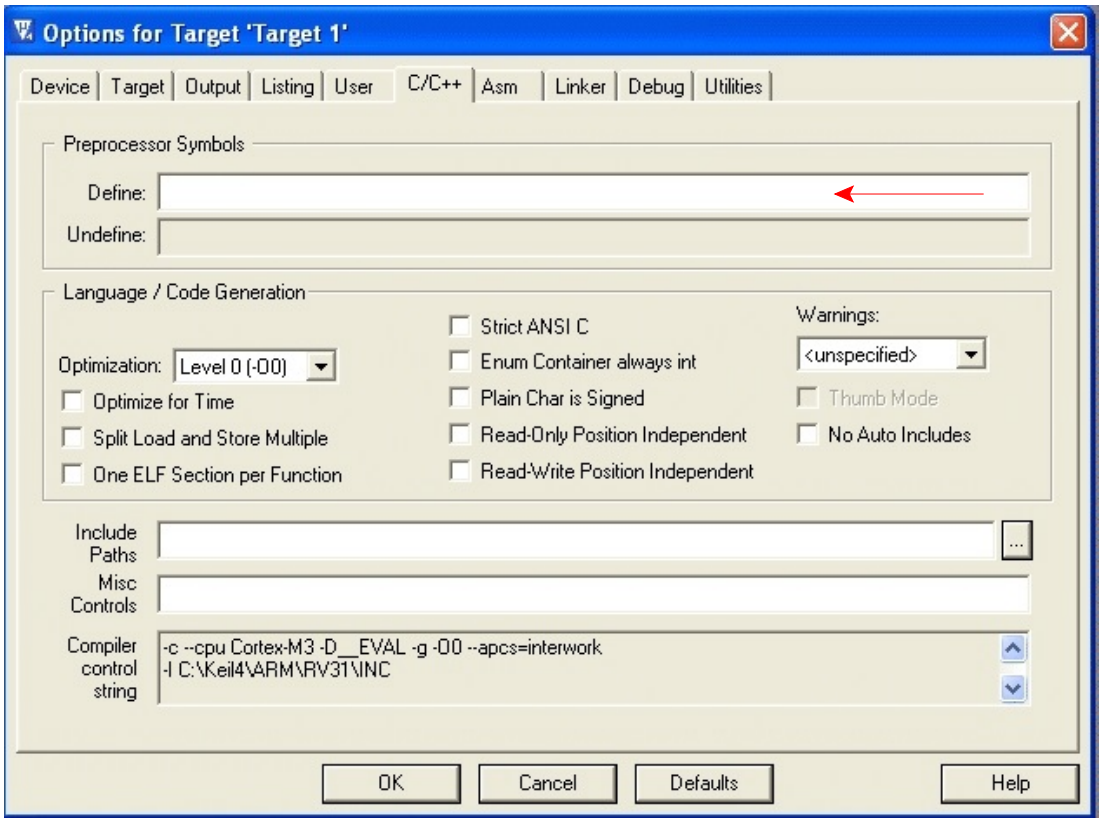
The Listing tab has this (default selections should be ok):



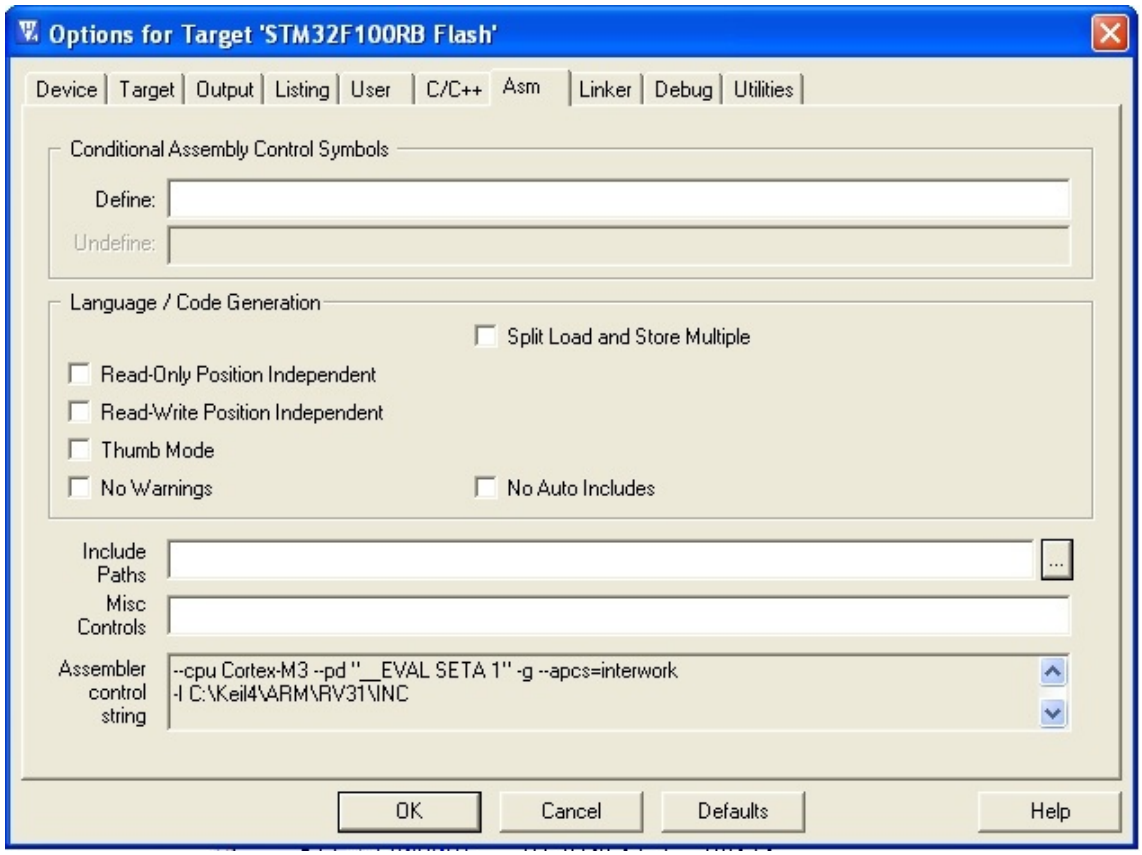
Here is the User tab contents (the default, blank, fields are ok):



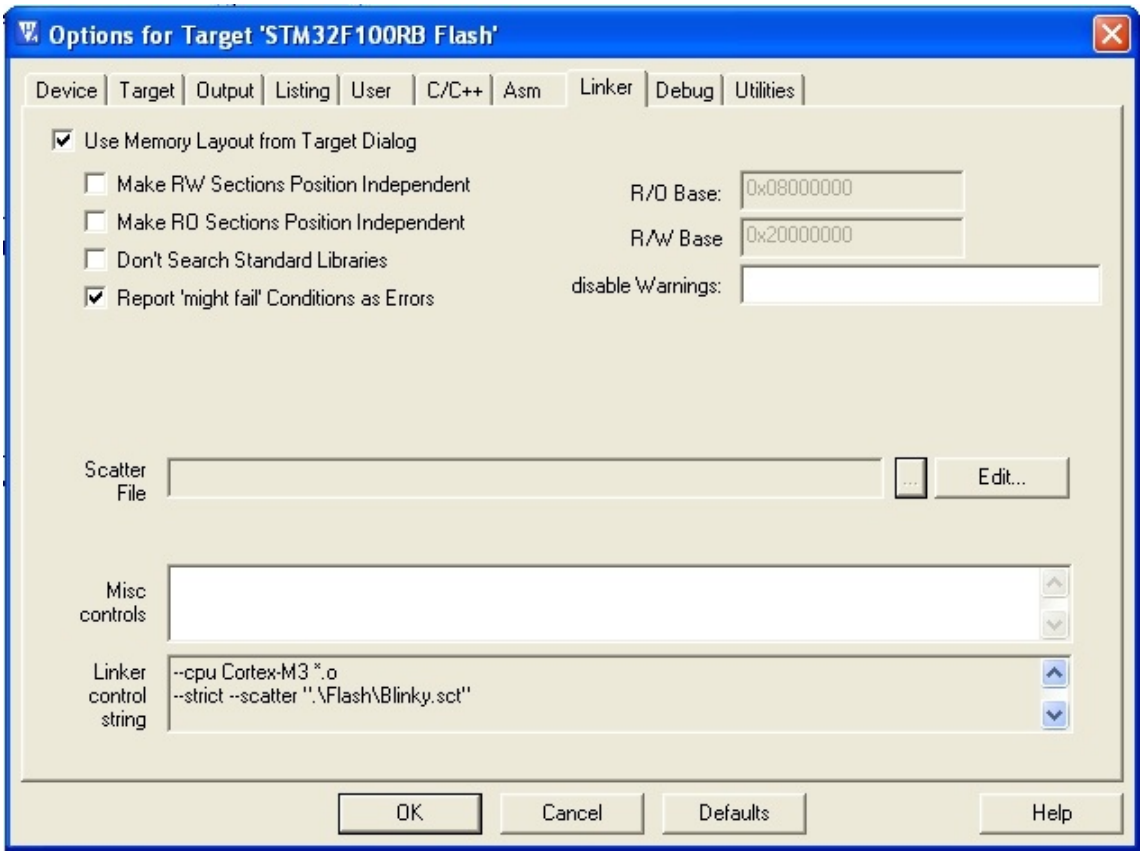
The C/C++ tab does need your attenton. In the Define box the processor being used must be defined. Enter STM32F10X_MD_VL.



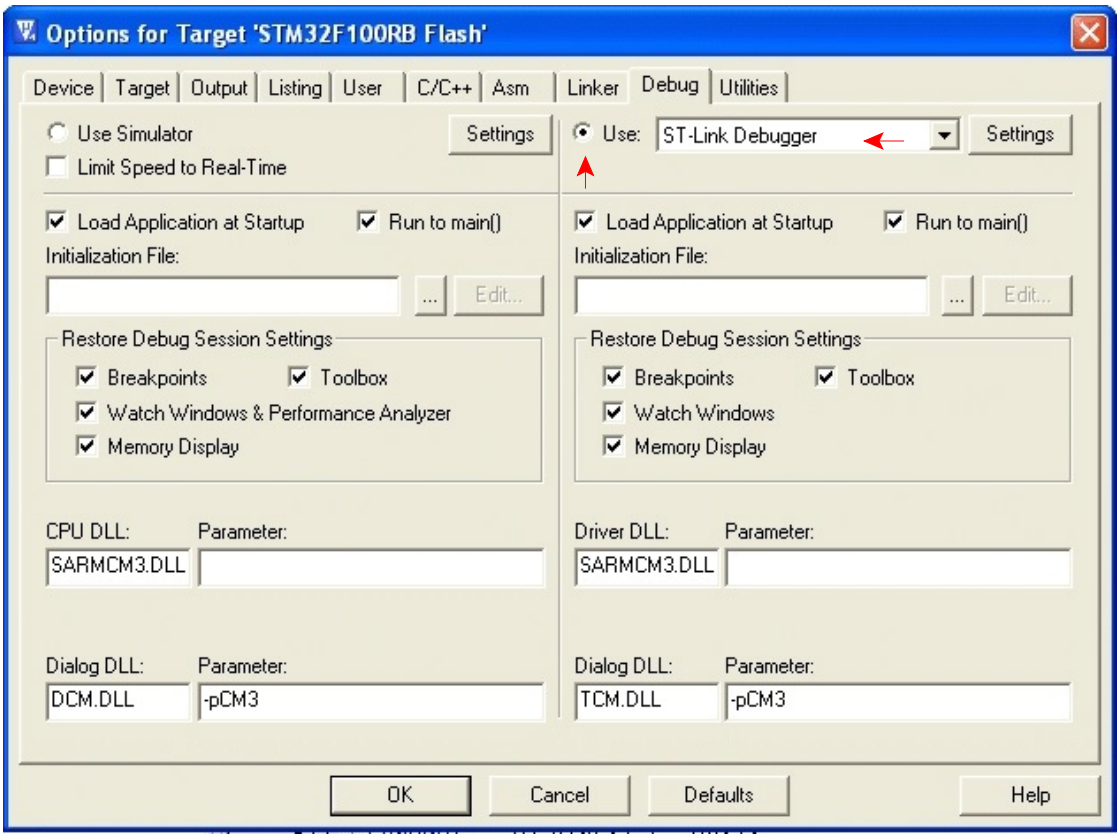
Below is the window opened when Asm is selected (default values are ok):



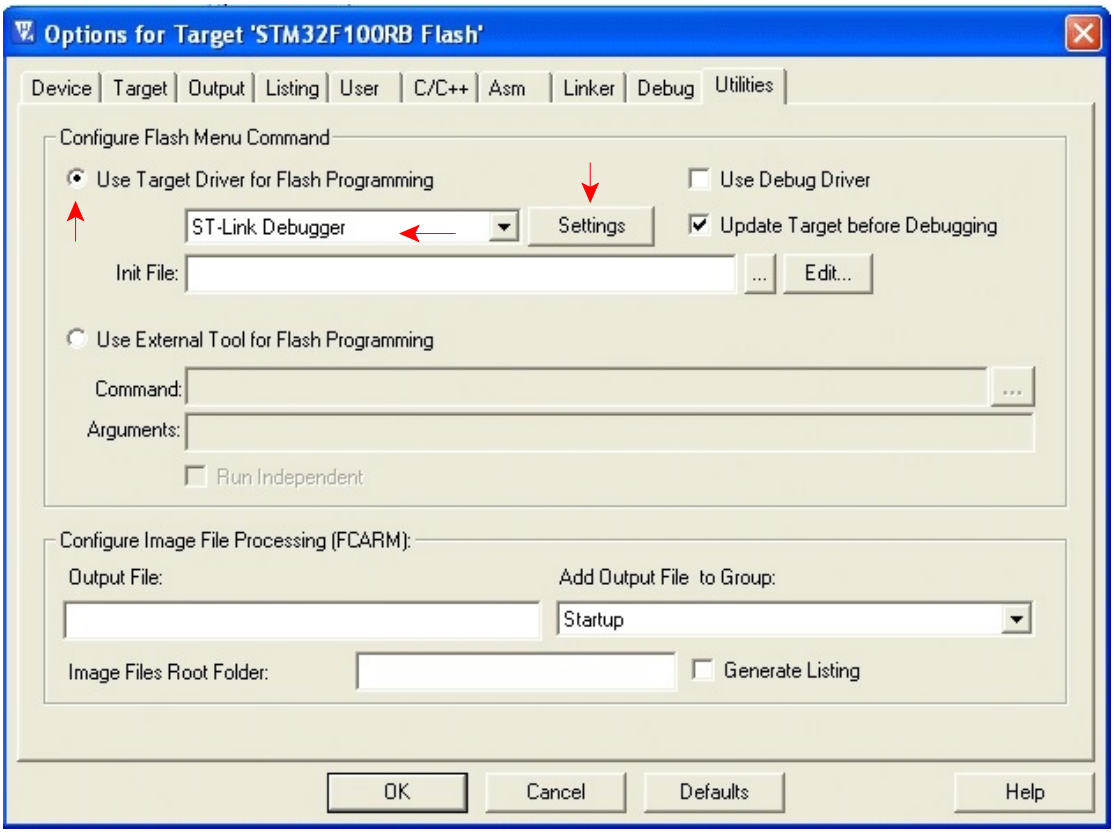
And the Linker tab (default is ok):



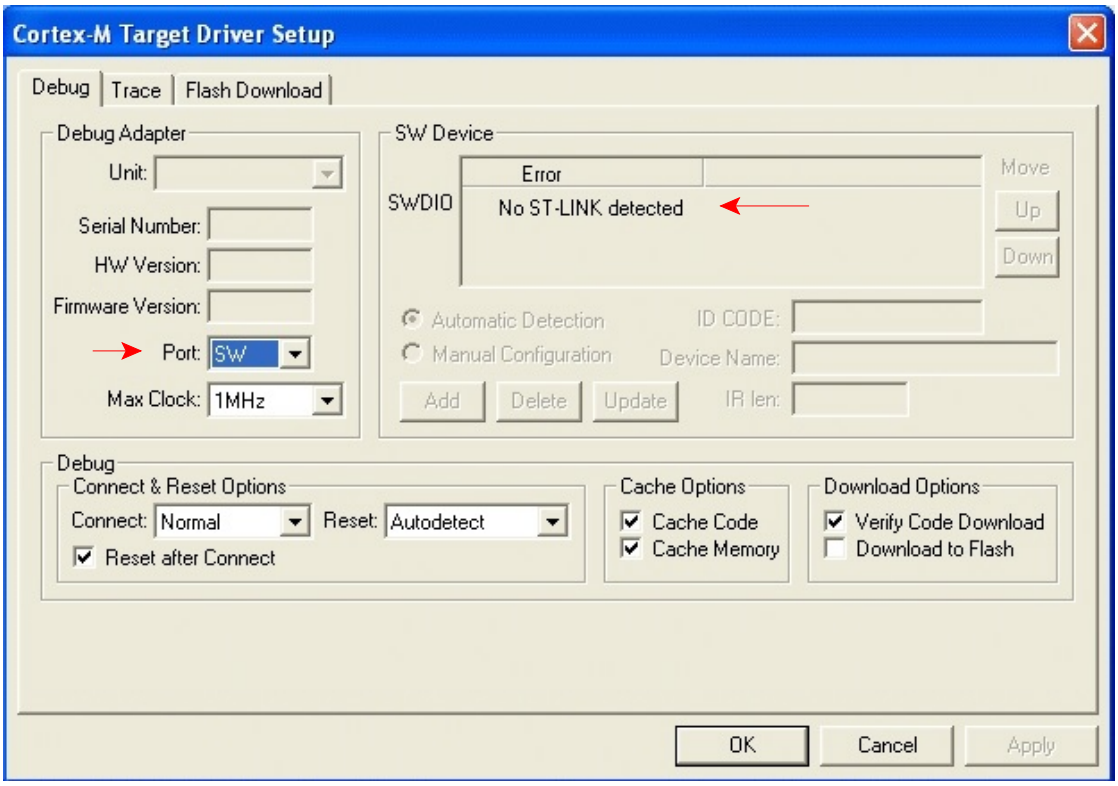
The Debug tab opens this window. Note that the ST-Link Debugger must be chosen:



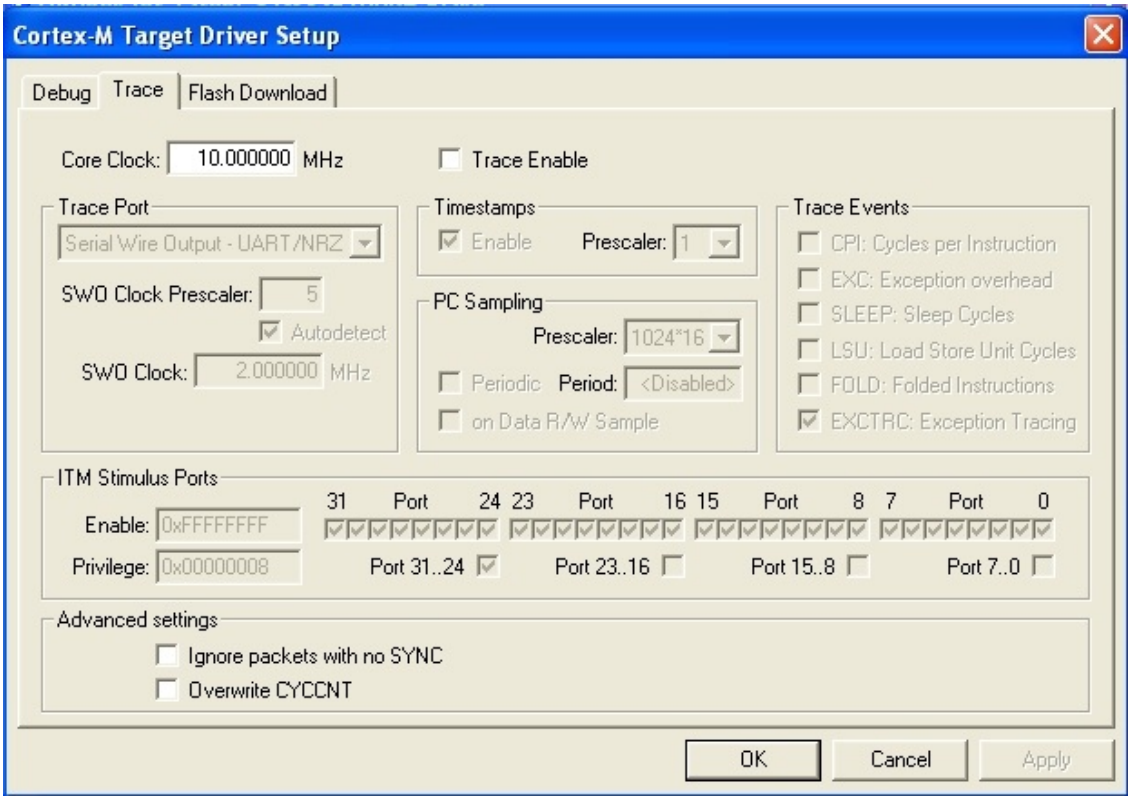
And finally, the Utilities window. The ST-Link Debugger needs to be selected. Note the Settings button. Click Settings to open additional windows as shown on the next page.



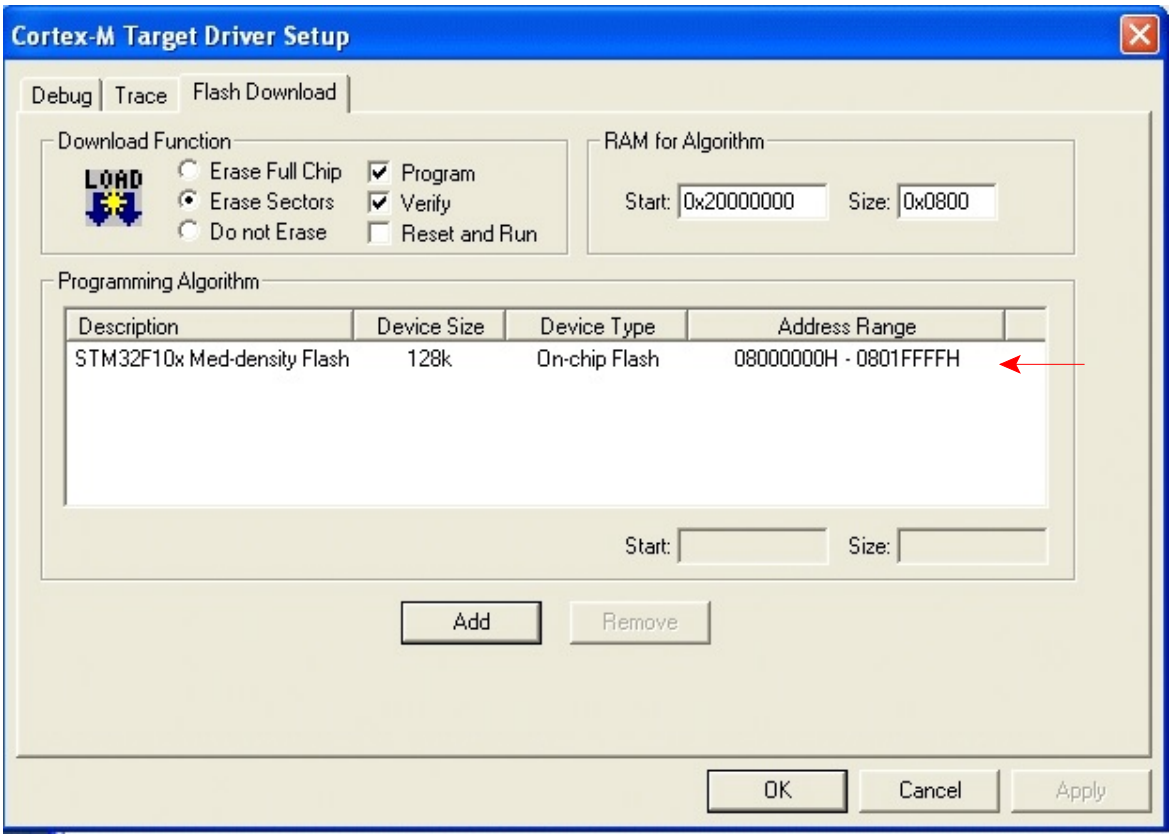
Clicking Settings in the Utilities window and its Debug tab gives this. Note that Port must be set to SW (single wire). No embedded board was connected when this screen shot was taken and thus the No ST-LINK message. That should change when a board is connected.



The trace tab. Default values should work.



On the Flash Download tab, if there isn't an entry showing in the Programming Algorithm box, click on the Add button which opens another window, scroll down to find STM32F10x Med-density Flash, click on it to select, and then click Add..



10) On the Project tab, select Build Target (or use the button on a tool bar, or press F7). Your project should build unless there are errors in your source file.

11) Connect hardware, download, and test your program.