



Getting Started with X2C[®]

***X2C[®] v6.6.4504
Free Edition***

August 28, 2026

© Linz Center of Mechatronics GmbH

Contents

I	Installation	2
1	Software versions	2
2	Setup with <i>Scilab</i> & <i>Xcos</i> support	2
2.1	Installation	2
2.2	Uninstallation	2
II	How-To	4
3	<i>X2C</i>[®] code generation with <i>Scilab</i>	4
4	Loading and building the demo application Blinky in <i>Code Composer Studio</i>	6
5	Loading and building the demo application Blinky in <i>MPLAB</i>[®] <i>X</i>	7
6	Loading and building the demo application Blinky in <i>STM32CubeIDE</i>	9
7	Loading and building the demo application Blinky in <i>DAVE IDE</i>	12

Part I

Installation

1 Software versions

Following software versions were tested for full X2C[®] functionality:

Software	Version
<i>Required:</i>	
Scilab (www.scilab.org)	6.1.1
<i>Optional (for standalone operation):</i>	
Java Runtime Environment	Java SE 8 / ojdkbuild 13 JRE
<i>Optional (for documentation):</i>	
MiKTeX (www.miktex.org)	2.9
Doxygen (www.doxygen.org)	1.9.0
Graphviz (www.graphviz.org)	2.38
<i>Optional (for programming):</i>	
TI Code Composer Studio	11.x
TI Code Generation Tools	c2000_16.9.5.LTS / arm_16.9.4.LTS
TI TivaWare	TivaWare_C_Series-2.x.x.xxx
ST STM32CubeIDE	1.15.x
Infineon DAVE IDE	4.x.x
Microchip MPLAB [®] X	6.xx
Microchip XC16	1.xx

Different versions of these programs may work but without warranty.

2 Setup with Scilab & Xcos support

2.1 Installation

1. Open *Scilab* and with the *File Browser* navigate to <X2C_ROOT>\System\Scilab\Scripts. Right click on **setup.sce** and click *Execute in Scilab*.
2. Restart *Scilab*
3. The setup command creates a X2C configuration file which will automatically load X2C libraries and palettes at startup of *Scilab*.

2.2 Uninstallation

1. Open *Scilab* and execute the command `initX2C(%f)` in the *Scilab* console.
2. Restart *Scilab*
3. Once above command was executed, the X2C configuration file is deleted and *Scilab* will not load any X2C libraries or palettes anymore.

For the unlikely event that Scilab freezes at startup and remains in a deadlock state, the deinstallation can be done manually by deleting the file **scilab.ini** located in the Scilab home directory (for Windows typically C:\Users\<your user name>\AppData\Roaming\Scilab\scilab-6.x.x).

Part II

How-To

3 X2C[®] code generation with *Scilab*

The following section describes X2C code generation of a *Xcos* model based on the *Blinky* demo application.

1. Open *Scilab* and in the file browser navigate to your project directory (e.g. <X2C_ROOT>\DemoApplication\Blinky_TI_TMS320F28069_controlSTICK\X2CCode).
2. Double click on **DemoApplication.zcos**. The example project contains a few blocks used to demonstrate the basic function of X2C (see Figure 1). The *Inport* and *Outport* blocks define the interface between the generated X2C code and the peripheral functions (e.g. ADC or GPIO Pins) on the target. For details about each block function read *X2Copen.Doc.pdf* in the documentation folder of the X2C directory.

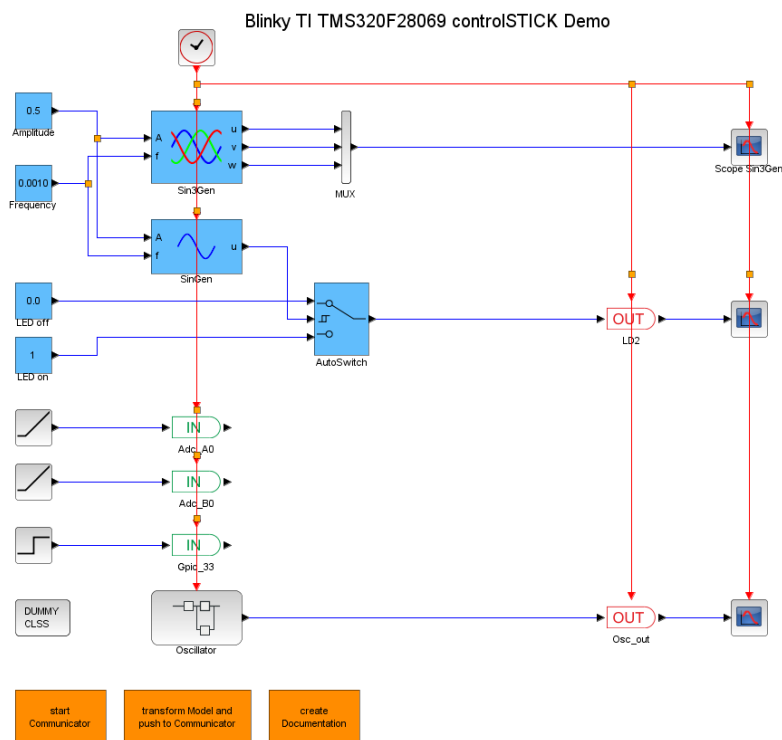


Figure 1: *Blinky* demo application in *Scilab*

3. Double click on **start Communicator**. Some details of the current actions of the *Communicator* are shown in the *Log* area of the *Communicator* window and the *Scilab* command line:

```
Starting Communicator
done
Successfully connected to Communicator
```

4. Double click on **Transform model and push to Communicator** and check the pop-up window for the end of the transformation process.
5. Click **Create Code** in the *Communicator*. Now the files *X2C.h* and *X2C.c* are generated in the <PROJECT_ROOT>\X2CCode directory and the Log screen should contain the lines:

```
[...]  
Model updated  
Model XML file write: OK  
Create code successful.
```

6. The *C* code for the *X2C* application has been created. Depending on the used target start the programming tool (e.g. *Code Composer Studio* , *STM32CubeIDE* or *MPLAB X*) and import the *Blinky* demo application project as described in Section [4](#), or [5](#) respectively. Follow the instructions on how to configure and download the application to the target.

4 Loading and building the demo application Blinky in Code Composer Studio

The demo application *Blinky* is intended to be used with a *TI F28069 Piccolo controlSTICK*.

1. Connect the *TI F28069 Piccolo controlSTICK* to the computer.
2. Open *Code Composer Studio* (choose workspace directory as you like). Now click **Project** → **Import Existing CCS Eclipse Project**. Browse to the location of the *Blinky* project (<X2C_ROOT>\DemoApplication\Blinky_TI_TMS320F28069_controlSTICK). Click **Finish** to import the project.
3. In the *Code Composer Studio* file structure of the *Blinky* demo project there are two virtual folders *Blocks* and *Core*, which should be linked directly to the *X2C* directory. To ensure this go to **Project** → **Properties** drop down **Resource** and click **Linked Resources**. Double click on folder **X2C_ROOT** and set the correct link to your *X2C* installation directory (<X2C_ROOT>). After hitting **OK** two times there should not be any warning signs (like shown in Figure 2) at the icons for the linked files in the *Blocks* and *Core* folders.

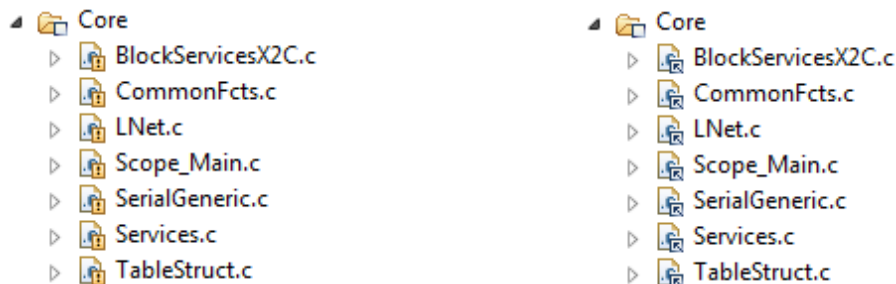


Figure 2: *Code Composer Studio* invalid (left) and valid (right) *X2C* root directory

4. The generated code from *X2C* is located in the folder <X2C_ROOT>\DemoApplication\Blinky_TI_TMS320F28069_controlSTICK\X2CCode. To check if code generation went fine go to the *X2CCode* folder and open *X2C.c*. Make sure time and date of code generation is plausible.
5. Build the project in *Code Composer Studio* by clicking **Project** → **Build all** or by clicking on the **Hammer** symbol as seen in Figure 3 at the top of the screen. Check for errors while building in the console at the bottom of the screen.



Figure 3: *Code Composer Studio* build and debug buttons

6. If your target is connected to the computer click **Run** → **Debug** or click on the *Bug* symbol as seen in Figure 3 at the top. The program is now transferred to the target and can be started with the **green arrow** button at the top.
7. After starting the program the on-board LED of the *TI F28069 Piccolo controlSTICK* should be blinking!

5 Loading and building the demo application Blinky in MPLAB® X

The demo application *Blinky* is build for the combination of the *Microstick II* with the *dsPIC33FJ128MC802* processor and the *MicrostickPlus* developer board (for details see www.microstick.com).

Info: To download a new application only the *Microstick II* needs to be connected with the computer.

1. Connect the *Microstick II* with the computer.
2. Open *MPLAB X* and click **File** → **Open Project**. Browse to the location of the *Blinky* demo application in the *X2C* directory <X2C_ROOT>\DemoApplication\... \Blinky_Microchip_dsPIC33Fxxxx_MicrostickPlus. Click **Open Project**.
3. In the case the demo application is copied/moved to a different location, the include paths have to be adapted. To ensure the compiler uses the correct path variables right click on the **Projectname** → **Properties** → **XC16 Global Options** → **xc16-gcc**. In the drop down menu **Option categories** choose **Preprocessing and messages**. Click on the dots beside *C include dirs*. There are relative paths to the needed include files listed as seen in Figure 4. Correct the links by double clicking on the path variables.
Info: Only the links to the *Library* and *Controller* path need to be updated.

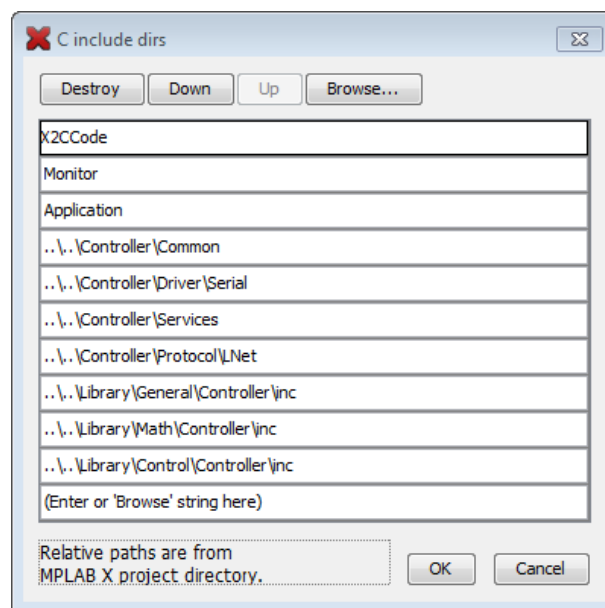


Figure 4: Default path variables for the include files

4. Go to **Run** → **Clean and Build Main Project** or click the *hammer with brush* button as seen in Figure 5. After building there should be a message BUILD SUCCESSFUL in the message area at the bottom of the screen.



Figure 5: MPLAB X Clean and Build Main Project button

5. If the build process was successful go to **Run** → **Run Main Project** or click the *Green*

Arrow button as seen in Figure 5. If there is a message similar to *MICROSTICK not Found* try to select the *Starter Kits (PKOB)* item which represents your board.

6. After starting the program the LED (RB12) on the *MicrostickPlus Board* should be blinking!

6 Loading and building the demo application Blinky in *STM32CubeIDE*

The demo application *Blinky* is intended to be used with the *ST STM32F072RB Nucleo* or the *ST STM32G474RE Nucleo* kit.

1. Connect the ST development kit with the computer. You may have to install the ST-Link USB driver (available on www.stm.com) to get the board recognized by your operating system.
2. Open *STM32CubeIDE* and click **File** → **Import...** Select **General/Existing Projects into Workspace** and click **Next**. Select the root directory of the *Blinky* project (either <X2C_ROOT>\DemoApplication\Blinky_ST_STM32F072RB_Nucleo or <X2C_ROOT>\DemoApplication\Blinky_ST_STM32G474RE_Nucleo). Click **Finish** to import the project.
3. In the *STM32CubeIDE* file structure of the *Blinky* demo project are two virtual folders *X2C-Blocks* and *X2C-Core*, which are linked relatively to the *X2C* directory. If the *Blinky* demo project is copied/moved to a different location, the resource property *X2C_ROOT* as seen in Figure 6 and the build variable *X2CDirPath* as seen in Figure 7 have to be adapted.

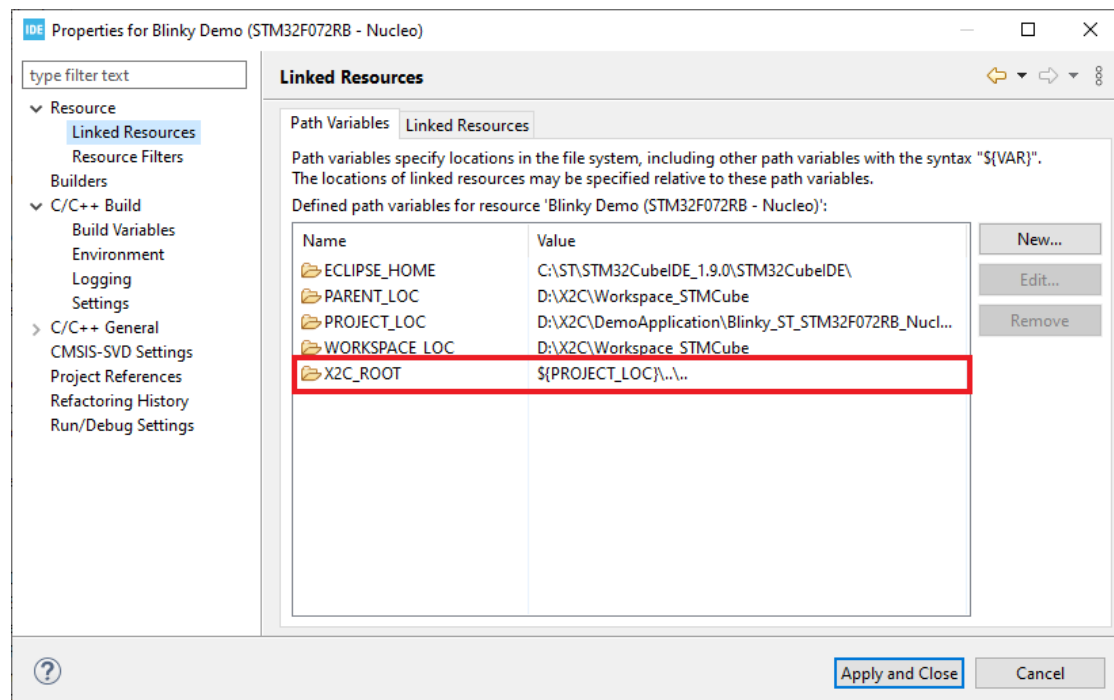


Figure 6: *STM32CubeIDE* resources settings

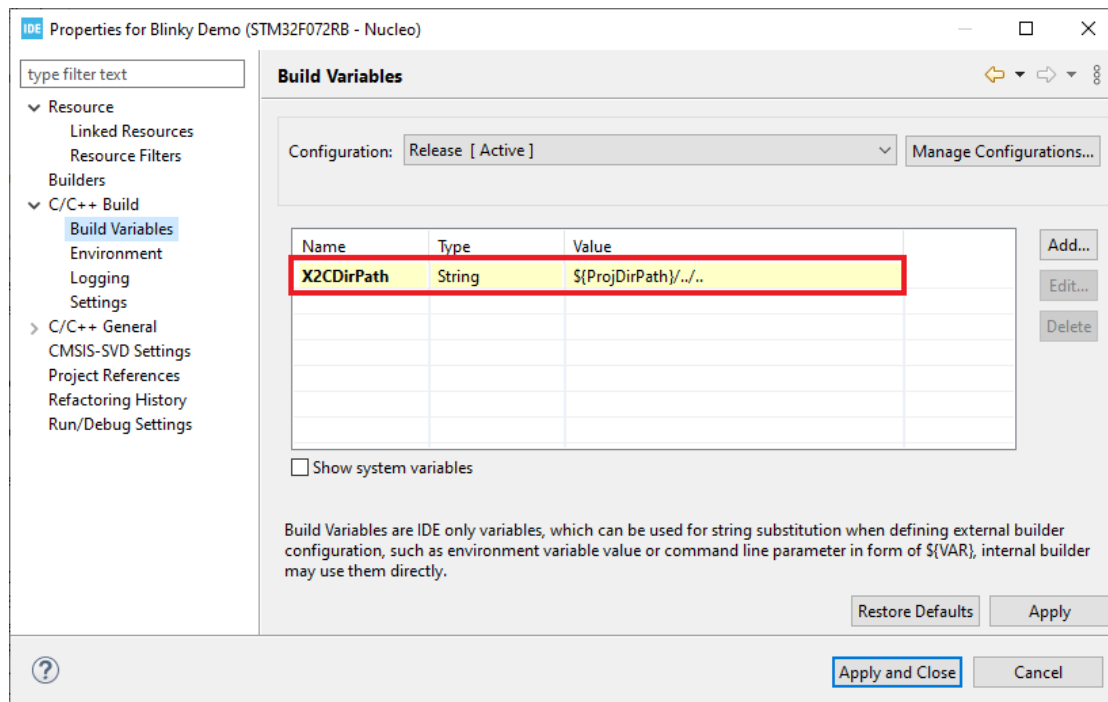


Figure 7: STM32CubeIDE build variables setting

To open shown windows go to **Project** → **Properties**.

- The generated code from X2C is located in the X2CCode folder (eg. <X2C_ROOT>\DemoApplication\Blinky_ST_STM32F072RB_Nucleo\X2CCode). To check if code generation went fine go to the X2CCode folder and open X2C.c. Make sure time and date of code generation are plausible.
- Before building the project the first time some hardware specific code has to be generated. Open the STM32CubeMX configuration window with a double-click on the *.ioc file in STM32CubeIDE's project explorer. Then the code can be generated with **Project** → **Generate Code** or by clicking on the *Code Generation* icon as seen in Figure 8.

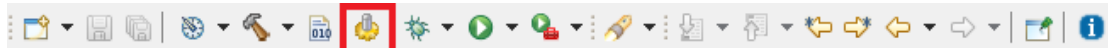


Figure 8: STM32CubeIDE menu icons - Code Generation

- Build the project in STM32CubeIDE by clicking **Project** → **Build Project** or by clicking on the *Build* icon as seen in Figure 9. Check for errors while building in the console at the bottom of the screen.



Figure 9: STM32CubeIDE menu icons - Build

- If your target is connected to the computer click **Run** → **Run** or click on the *Run* icon as seen in Figure 10. The program is now transferred to the target and is automatically started.

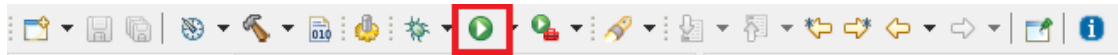


Figure 10: *STM32CubeIDE* menu icons - Run

8. After starting the program the green on-board LED of the ST development kit should be blinking!

7 Loading and building the demo application Blinky in *DAVE IDE*

The demo application *Blinky* is intended to be used with the *Infineon XMC4700 Relax Kit*.

1. Connect the Infineon development kit with the computer. Use a Micro-USB cable to connect the USB port of the computer with the USB port labeled X101 on the kit.
2. Open *DAVE IDE* and click **File** → **Import....** Select **General/Existing Projects into Workspace** and click **Next**. Select the root directory of the *Blinky* project (<X2C_ROOT>\DemoApplication\Blinky_Infineon_XMC4700_RelaxKit). Click **Finish** to import the project.
3. In the *DAVE IDE* file structure of the *Blinky* demo project are two virtual folders *X2C-Blocks* and *X2C-Core*, which are linked relatively to the *X2C* directory. If the *Blinky* demo project is copied/moved to a different location, the resource property *X2C_ROOT* as seen in Figure 11 and the build variable *X2CDirPath* as seen in Figure 12 have to be adapted.

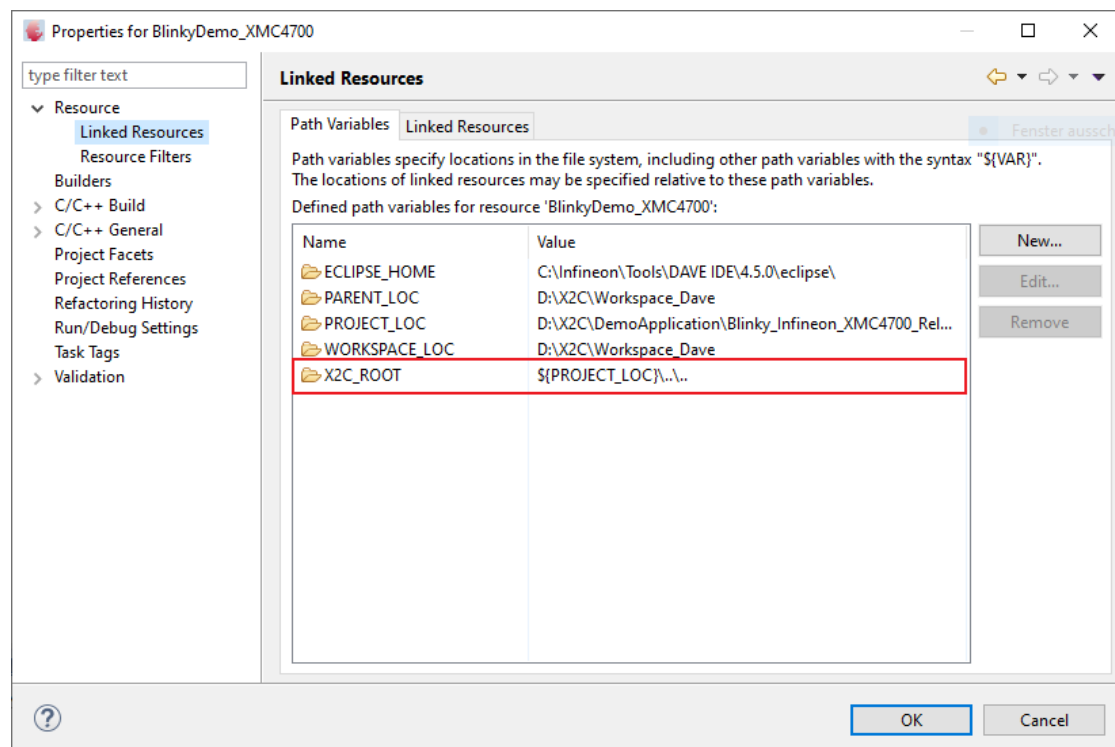


Figure 11: *DAVE IDE* resources settings

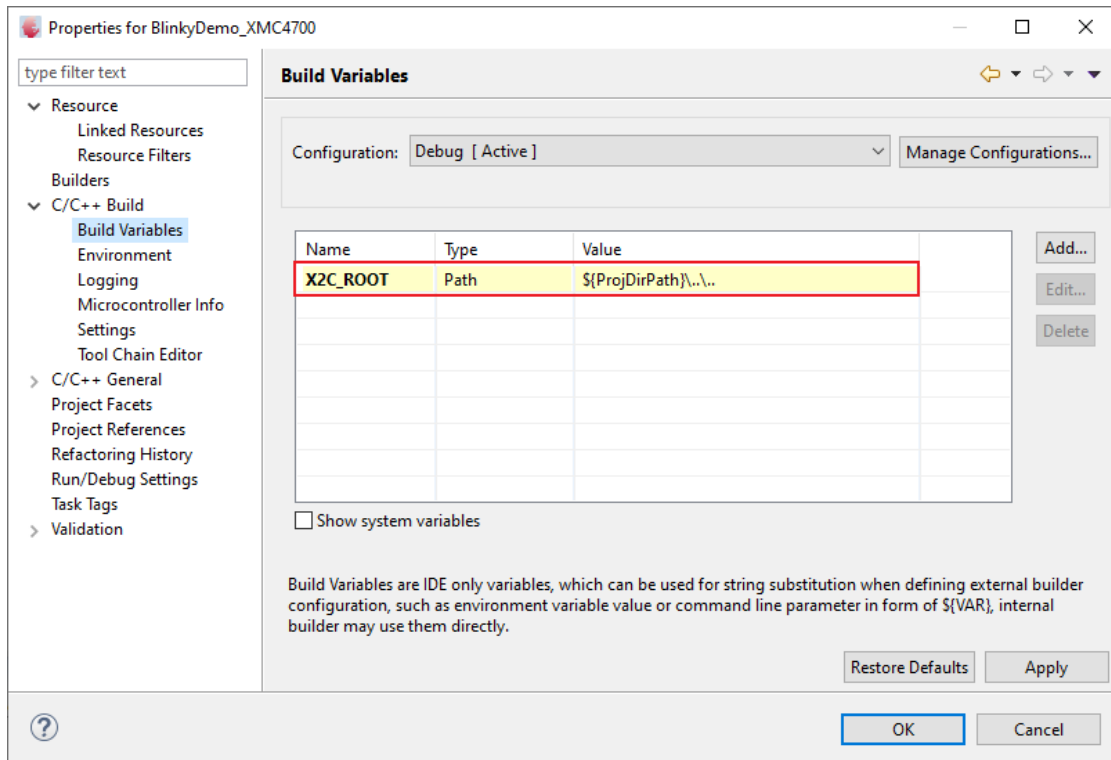


Figure 12: DAVE IDE build variables setting

To open shown windows go to **Project** → **Properties**.

- The generated code from X2C is located in the X2CCode folder (eg. <X2C_ROOT>\DemoApplication\Blinky_Infiniteon_XMC4700_RelaxKit). To check if code generation went fine go to the X2CCode folder and open X2C.c. Make sure time and date of code generation are plausible.
- Before building the project the first time some hardware specific code has to be generated. Right-click on the project name in the project tree to open the context menu and select **Set Active Project**. This activates the menu icons which are needed for code generation and project building.

The hardware specific code can be generated with **DAVE** → **Generate Code** or by clicking on the *Generate Code* icon as seen in Figure 13.



Figure 13: DAVE IDE menu icons - Code Generation

- Build the project in DAVE IDE by clicking **Project** → **Build Active Project** or by clicking on the *Build Active Project* icon as seen in Figure 14. Check for errors while building in the console at the bottom of the screen.



Figure 14: DAVE IDE menu icons - Build

- If your target is connected to the computer click on the *Debug* icon as seen in Figure 15.



Figure 15: *DAVE IDE* menu icons - Debug

When the debug session is started for the first time a *Debug Configurations* window will open. Double-click on the *GDB SEGGER J-Link Debugging* entry on the left hand side. This will create a debug configuration as depicted in Figure 16.

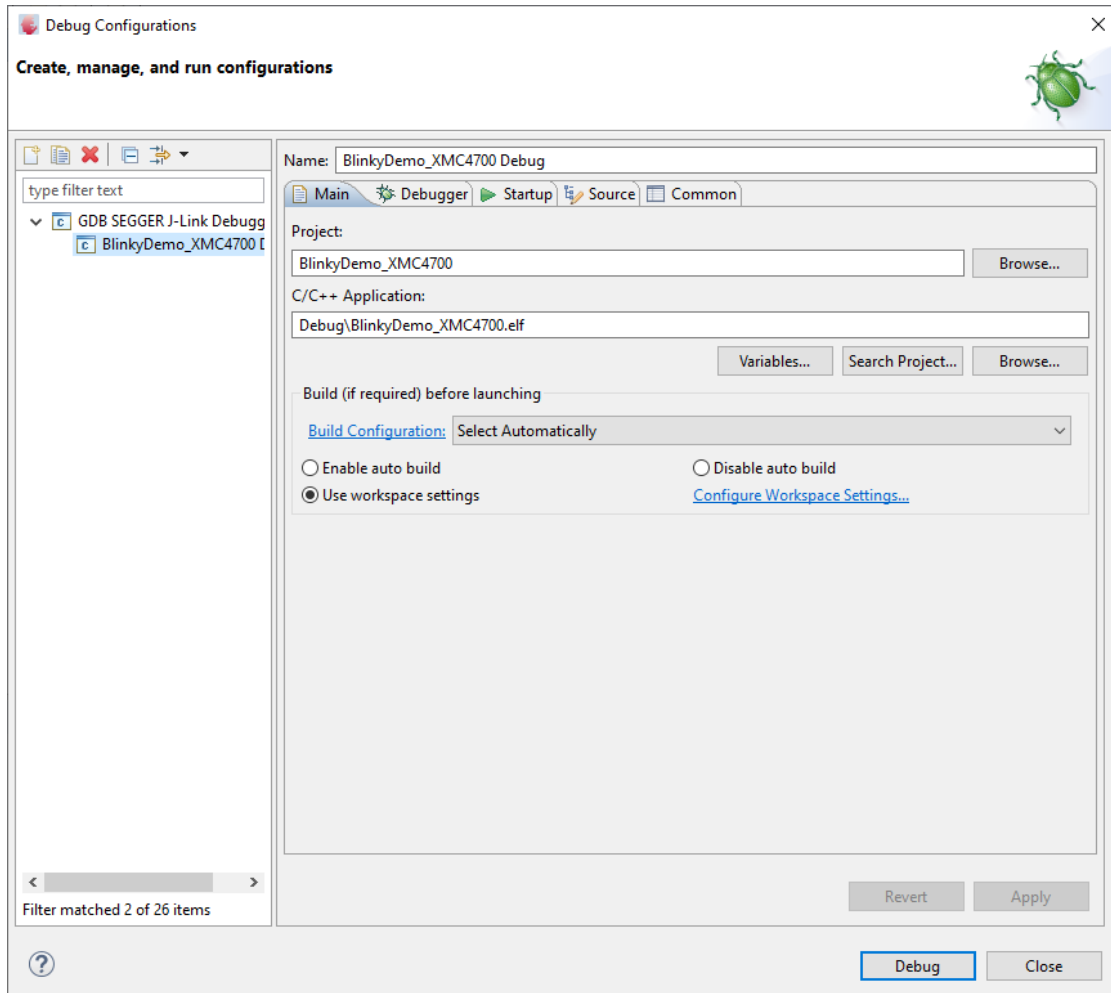


Figure 16: *DAVE IDE* Debug configuration

A click on **Debug** will start the debug session. This means the application will be transferred to the target and *DAVE IDE* will switch to the debug perspective.

8. To start the execution of the application press **F8** or click on the *Resume* icon as seen in Figure 17.

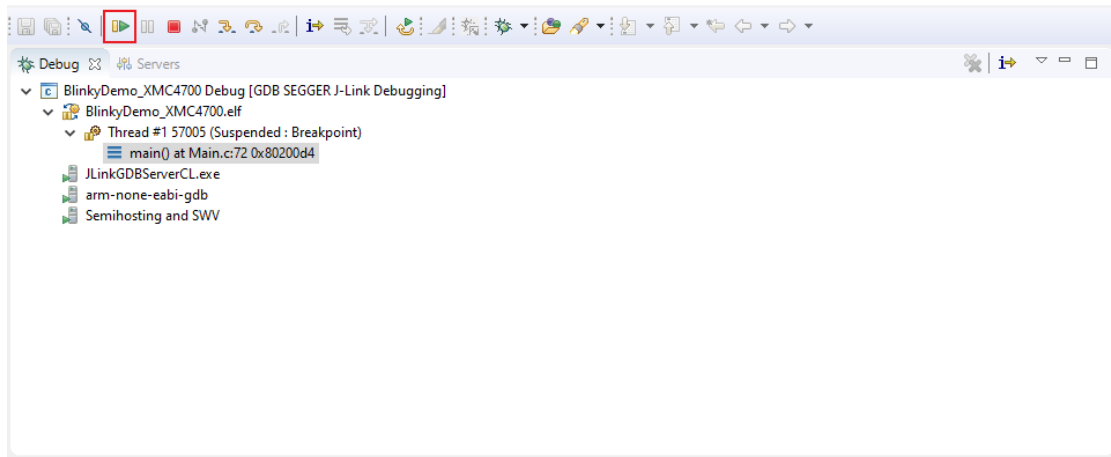


Figure 17: *DAVE IDE* Debug view

9. After starting the program the red on-board LED1 of the Infineon Relax Kit should be blinking!