TI mmWave Labs

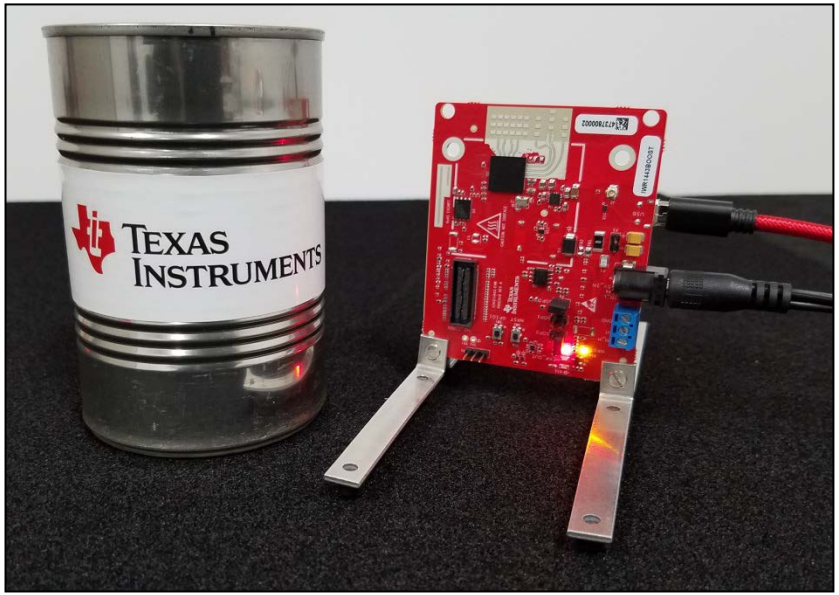
High Accuracy Range Measurement – 14xx

Contents

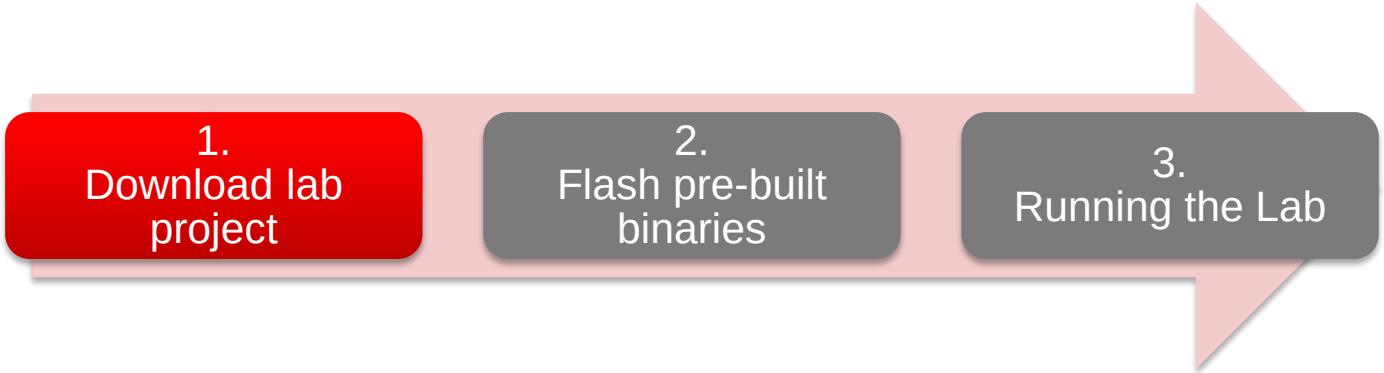
- Lab Overview
- Quick Start with Pre-built Binaries
  - Download the lab project
  - Flash pre-built binaries
  - Running the lab GUI
- Algorithm Description
  - Zoom FFT Algorithm and Code Implementation
  - Trade-offs and design considerations
  - Sub-mm accuracy simulation and test result
- Building Project from Source Code
  - Requirements
  - Software setup
    - Pre-requisites
    - Building the project
  - Hardware setup
    - Preparing the EVM
    - Connecting the EVM
  - Running the Lab from debug mode

Lab Overview

- This lab demonstrates the implementation of zoom FFT to detect object within sub-mm accuracy.
- The mmWave sensor xwr1443 EVM is used for lab demonstration
- The lab is designed to detect the single strongest object only
- 16K FFT is achieved for the peak detection
- Accuracy <0.5mm can be achieved with detection SNR ~55dB
- GUI visualizer tool for lab demonstration



Quick Start Steps



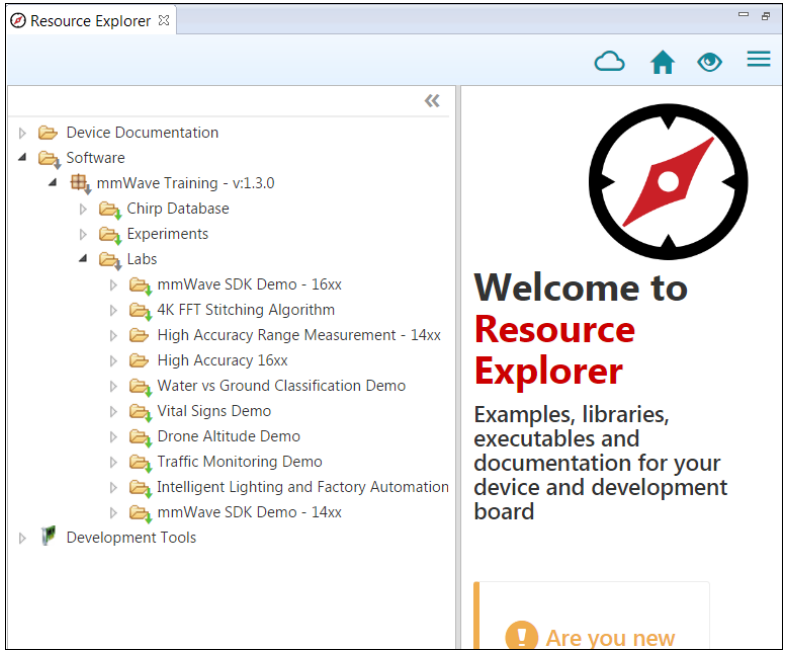
- Required Hardware
  - xWR1443 EVM
  - Micro USB cable (included in the EVM package)
  - 5V/2.5A Power Supply
    - Purchase from Digikey

QUICK START 1. Download the lab

- The mmWave Lab projects are available under **mmWave Industrial Toolbox** or **mmWave Training** in CCS Resource Explorer.

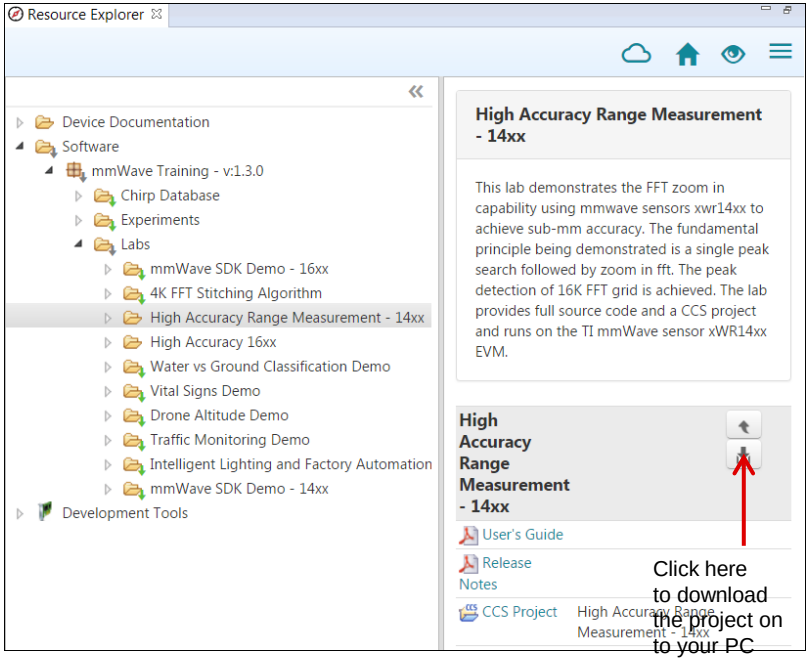
- To download the High Accuracy Range Measurement – 14xx Lab, start CCS v7.1 (or later) and select **View ► Resource Explorer** to open the Resource Explorer.

- In the Resource Explorer Window, select **Software ► mmWave Training ► Labs**.

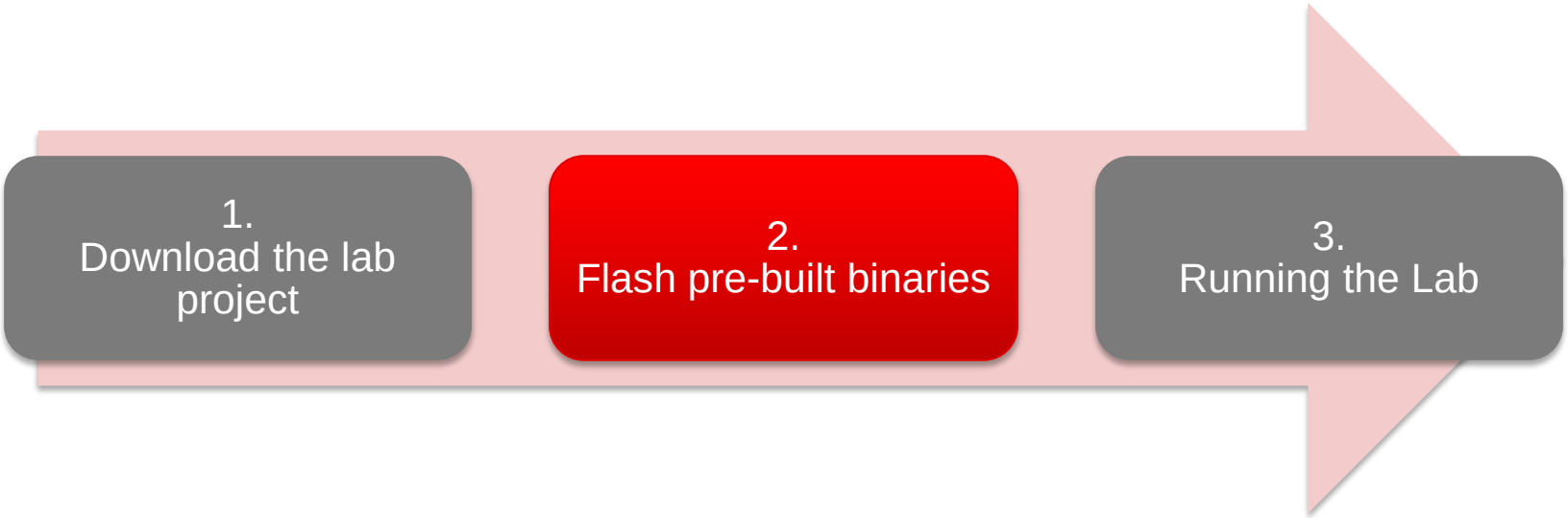


QUICK START 1. Download

- Select the High Accuracy Range Measurement – 14xx in the left view.
- The right view shows the contents of the Lab which contains the CCS Project
- Click on the **Download and Install** button in the top right corner as shown.
- Select the **Make Available Offline** option from the drop down to start downloading the Lab.

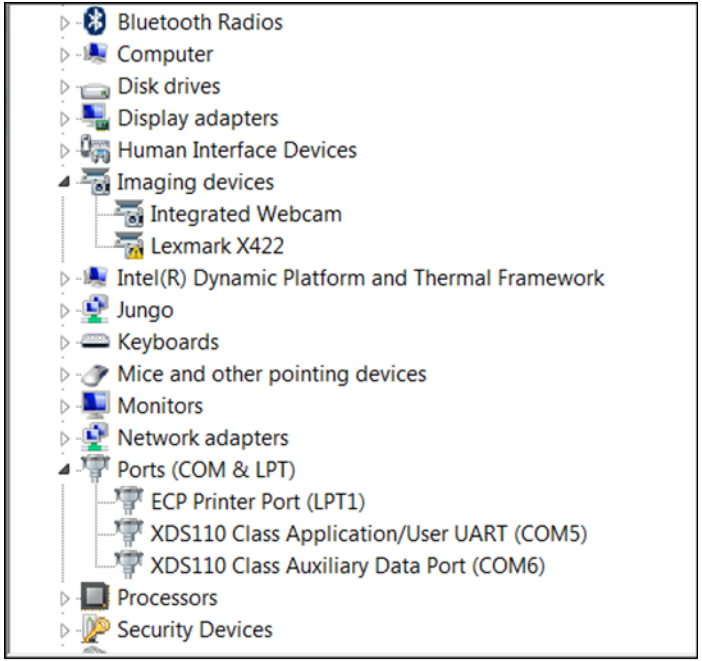


Quick Start Steps



Quick Start 2. Flash the binaries

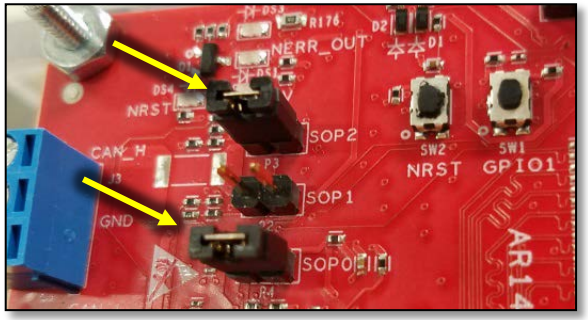
- Power on the EVM using a 5V/2.5A power supply.
- Connect the EVM to your PC and check the COM ports in Windows Device Manager
- The EVM exports two virtual COM ports as shown below:
  - XDS110 Class Application/User UART (COM_{UART}):
    - Used for passing configuration data and firmware to the EVM
  - XDS110 Class Auxiliary Data Port (COM_{AUX})
    - Used to send processed radar data output
- Note the COM_{UART} and COM_{AUX} port numbers, as they will be used later for flashing and running the Lab.

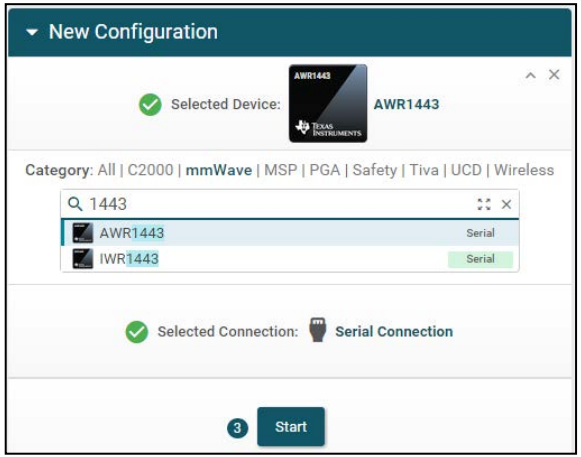


COM_{UART}: COM5 **COM_{AUX}**: COM6 The actual port numbers on your machine may be different

Quick Start 2. Flash – continued

1. Put the EVM in flashing mode by connecting jumpers on SOP0 and SOP2 as shown in the image.
2. Open the **UniFlash** tool
   - Download from TI.com/tool/uniflash
3. In the **New Configuration** section, locate and select the appropriate device (AWR1443 or IWR1443)
4. Click **Start** to proceed





Quick Start 2. Flash – continued

1. In the **Program** tab, browse and locate the Radar SS and MSS images shown below:

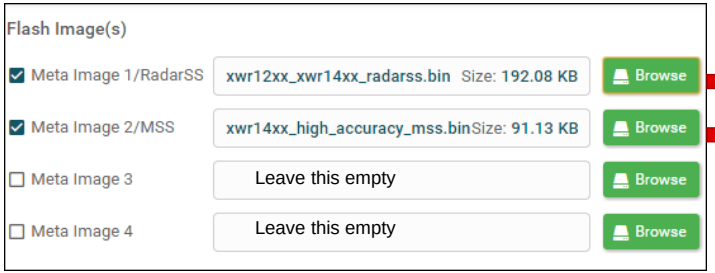
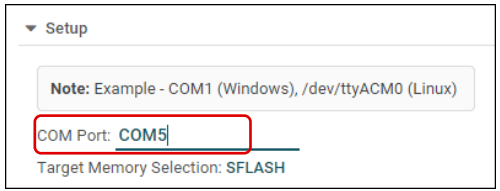


Image	Location
Meta Image 1/RadarSS	C:\ti\mmwave_industrial_toolbox_<ver>\labs\lab0004-high-accuracy-14xx\lab0004_high_accuracy_14xx_pjt\prebuilt_binaries**xwr12xx_xwr14xx_radarss.bin**
Meta Image 2/MSS	C:\ti\mmwave_industrial_toolbox_<ver>\labs\lab0004-high-accuracy-14xx\lab0004_high_accuracy_14xx_pjt\prebuilt_binaries**xwr14xx_high_accuracy_mss.bin**

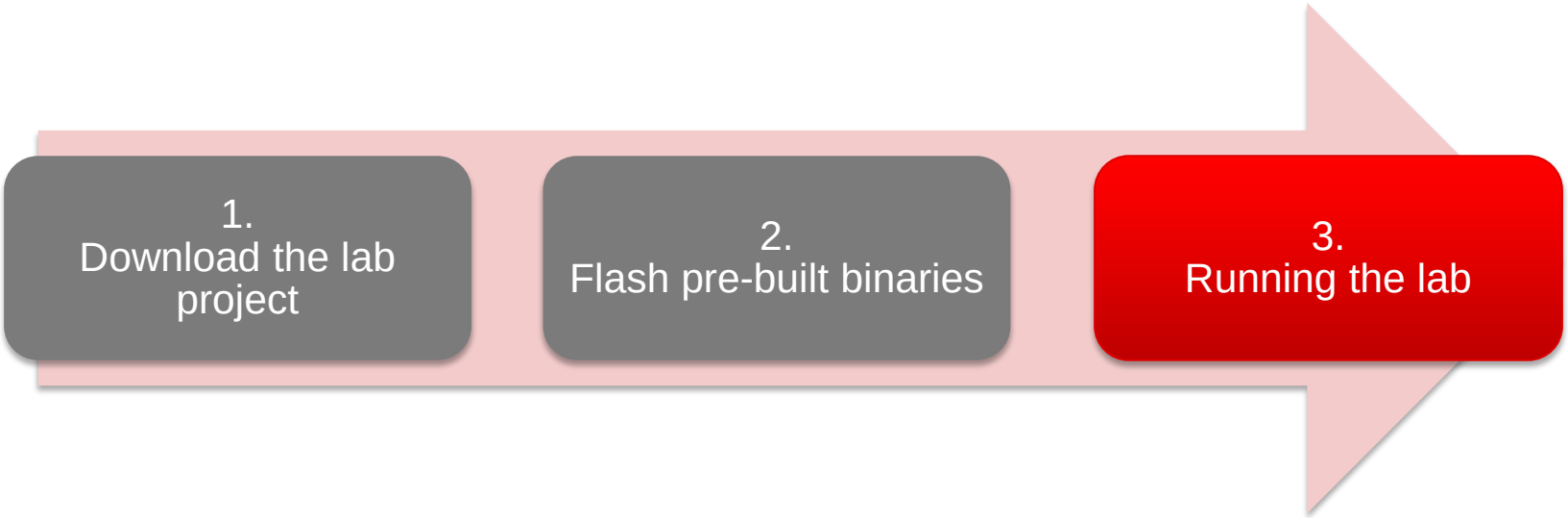
2. In the **Settings & Utilities** tab, fill the **COM Port** text box with the Application/User UART COM port number (**COM_{UART}**) noted earlier



3. Return to the **Program** tab, power cycle the device and click on **Load Images**
4. When the flash procedure completes, UniFlash's console should indicate: [SUCCESS] Program Load completed successfully
8. Power off the board and remove the jumper from only header **SOP2** (this puts the board back in functional mode)

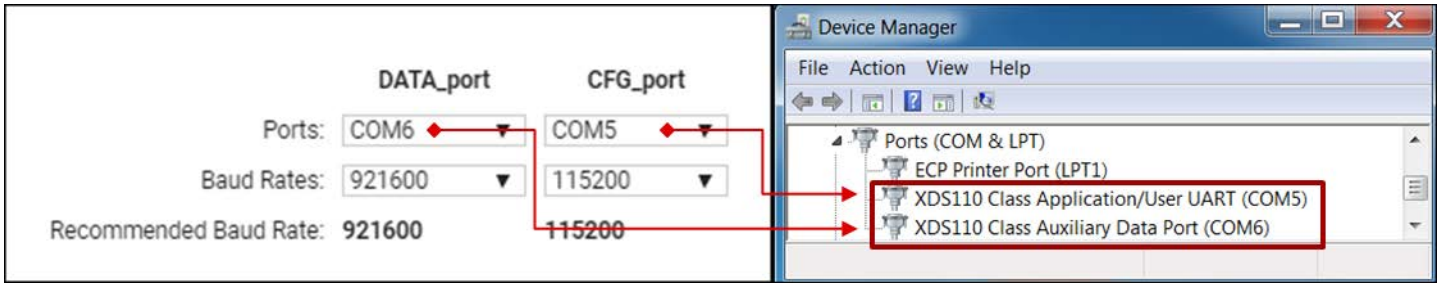
Texas Instruments

Quick Start Steps



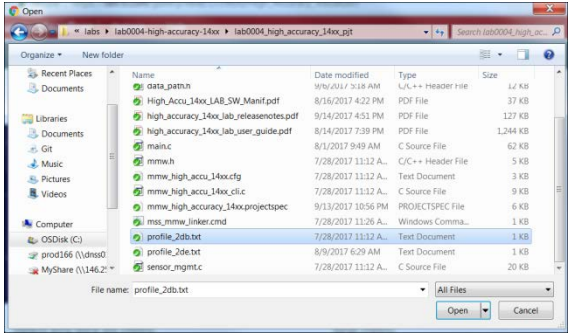
Quick Start 3. Running the Lab GUI

1. Using Google Chrome, navigate to the following URL: https://dev.ti.com/gallery/view/1359600/High_Accuracy_Visualizer/
   – Alternatively, go to https://dev.ti.com/gallery and search for “High Accuracy Visualizer”
2. If prompted, follow the on-screen instructions for installing TI Cloud Agent
3. Once the demo is loaded, go to **Options → Serial Port**
4. In the serial port window, enter the appropriate port in each of the drop down menus based on your port numbers from Step 2



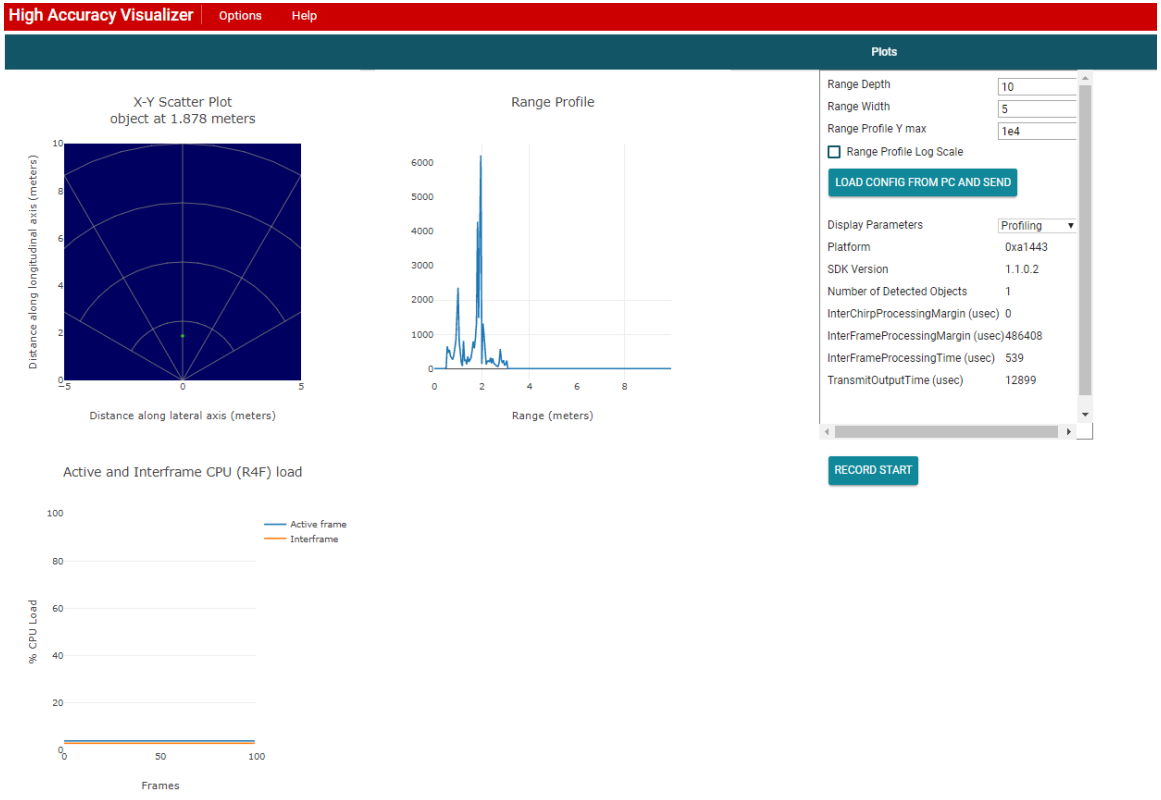
Quick Start 3. GUI - continued

5. Click on **Configure** and the demo will automatically connect to the EVM
   - Not connected: Connected:
   - If the connection fails, try clicking on the connection icon in the bottom left corner
6. Press “LOAD CONFIG FROM PC AND SEND” button and select the profile_2db.txt or profile_2de.txt* under C:\ti\mmwave_industrial_toolbox_<ver>\labs\lab0004-high-accuracy-14xx\lab0004-high-accuracy-14xx_pjt
7. Move a highly reflective object in front of the EVM and see how the demo responds



* Note: the two optimized chirp configurations provided in the high accuracy 14xx lab are:
  - Profile_2db.txt: sample size 512
  - Profile_2de.txt: sample size 1024, (coarse resolution is 4cm, best accuracy expected)

Quick Start 3. GUI - continued



Texas Instruments

Quick Start 3. Limit the range

8. Use CLI command to disable or enable the range limited function.
9. User can change the parameter to suit their own application.
10. Command is RangeLimitCfg <dis/enable><min range limited><max range limited>
    - <dis/enable>: disable 0, enable 1
    - <min range limited> : in unit of meter
    - <max range limited>: in unit of meter
11. Example: RangeLimitCfg 1 0.5 3.0
    - Enable the range limited function and the min range is 0.5m, max range is 3.0m

Texas Instruments

Quick Start 3. Record the result

12. While using GUI to shown the detected result, user can use record function to record it to a CSV file.
13. Press “LOAD CONFIG FROM PC AND SEND” button and select the profile_2db.txt or profile_2de.txt under C:\ti\mmwave_industrial_toolbox_<ver>\labs\lab0004-high-accuracy-14xx\lab0004-high-accuracy-14xx_pjt
14. When the application is running, press “Record Start” button and select the save file name in your disk. Record is running.
15. When stop recording, please press “Record Stop” button to stop it.
16. The .csv file format is shown on the right.

Result Template.csv

Frame	object at(m)
0	1.7998
1	1.8584
2	1.8584
3	1.8576
4	1.8576
5	1.7998
6	1.7998
7	1.8573
8	1.8573
9	1.8552
10	1.8552
11	1.7996
12	1.7996
13	1.7996
14	1.7996
15	1.8004

Zoom FFT Algorithm

- Configuration is designed to have 1 TX and 1 RX, with infinite loops of frames (one chirp per frame). Each frame only one iteration of data is processed
- Two step FFT:
  - 1D coarse FFT for peak search – 1D FFT size N1
  - Zoom in for fine peak detection – Peak interpolation ration N2
  - Zoom FFT size N = N1 * N2, N is fixed at 16K (16386)
- 1D FFT to obtain coarse peak frequency:
  - Coarse FFT (1D FFT) size N1 <=1024
  - Statistics block of Hardware accelerator (HWA) is enabled – the maximum value and its index of 1D FFT are logged in registers
  - The index of maximum value (coarse peak frequency) will be used for zoom FFT
- Zoom in for fine peak detection:
  - HWA in slow DFT mode (refer to HWA user guide for details)
  - Number of bins near peak used for zoom FFT is 4
  - Interpolation on R4F to get finer resolution
  - Total number of points in peak after interpolation is 4*N2

17

Zoom FFT Code Implementation

- HWA parameter sets for both coarse1D FFT, peak search and zoom in FFT with slow DFT are configured under function MmwDemo_dataPathConfigCommon (data_path.c)
  - Coarse1D FFT output data is in saved in M2 local memory
  - If calibDcRangeSig is enabled, DC range bins will be accumulated. When chirp accumulation is finished, 1D FFT output data in M2 will have DC range bins cancelled
  - Peak search step utilizes HWA statistics block to generate the max peak index
  - Zoom in FFT with slow DFT will generate zoon-in bins data to M3 local memory
- The maximum peak index is loaded into register MmwDemo_DataPathObj->rangeProcStats.maxIndex
- With maximum peak index from 1D FFT, MmwDemo_configInterp_HWA function is used for setting starting frequency of slow DFT(data_path.c)
- After zoom in DFT, interpolation codes are implemented in MmwDemo_processInterpolation function(data_path.c)

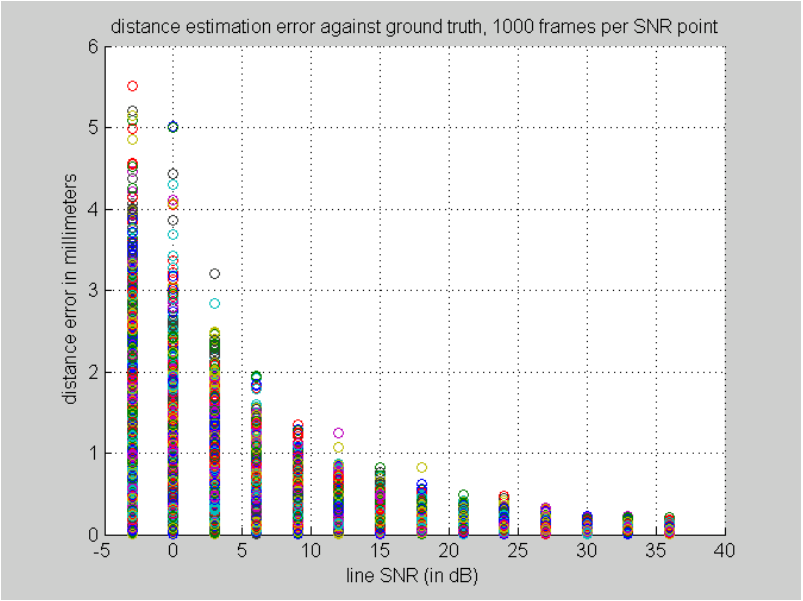
Trade-offs and Considerations

- Comparison of the high accuracy lab 16xx vs. 14xx:
  - Both 16xx and 14xx has the flexibility to select the range where highest peak is for zoom FFT;
  - 14xx: the max FFT size is 16K due to HWA limitation but for 16xx with DSP, resolution can go higher without limitation
  - 14xx will have much lower processing latency by using HWA, in addition to lower power consumption
- Chirp configurations:
  - The two optimized chirp configurations provided in the high accuracy 14xx lab are:
    - Profile_2db.txt: sample size 512, max range 20m
    - Profile_2de.txt: sample size 1024, with resolution of 4cm for best accuracy
  - The high accuracy lab only supports 1TX and 1RX with sample size <=1024
  - Frame configuration: infinite loops of frames with one chirp per frame

Simulation Result

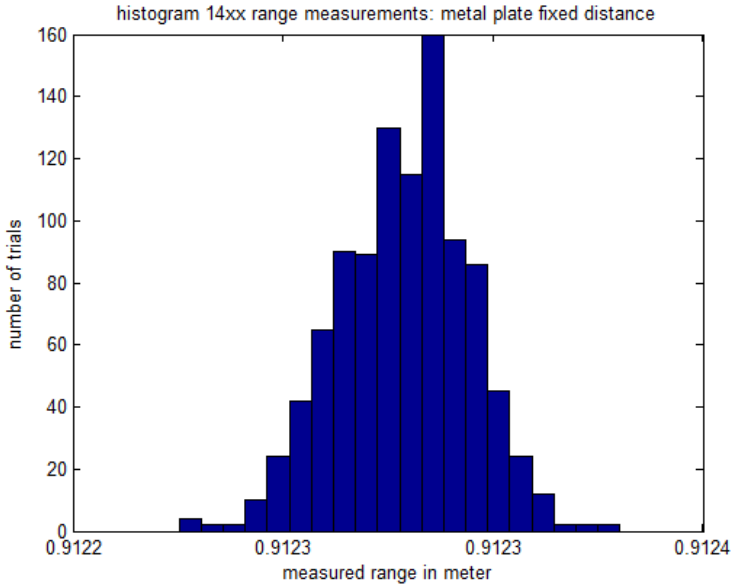
- Error vs time domain SNR scaler plot
  - Can read of the max error for a given time domain SNR.
  - Processing gain to convert to Detection SNR is approximately 30 dB (10log10(1024))
- Conclusions
  - We read the following results from the plot:

Target Accuracy	Required Detection SNR
< 0.5 mm	~ 55 dB
< 1 mm	~ 34 dB



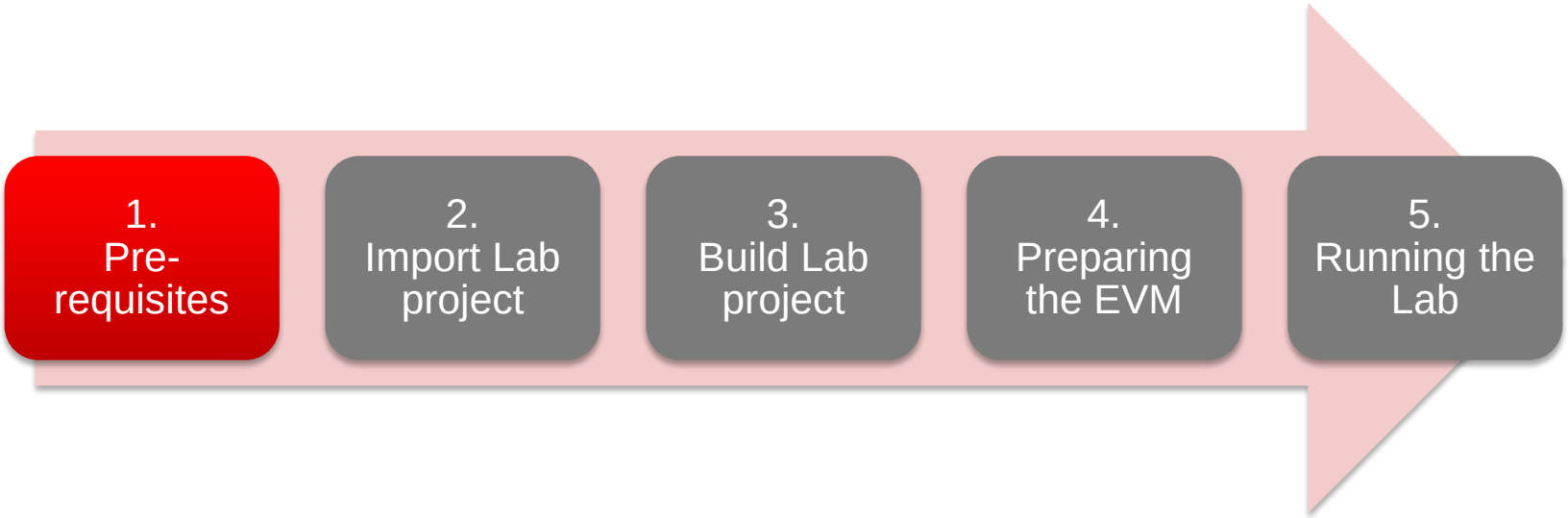
Line SNR or Time domain SNR translates to detection SNR by adding 30dB.

Lab test with metal plate



- For a stationary target, the measurement is done 1000 times

Steps for Building from the Source Code and Run

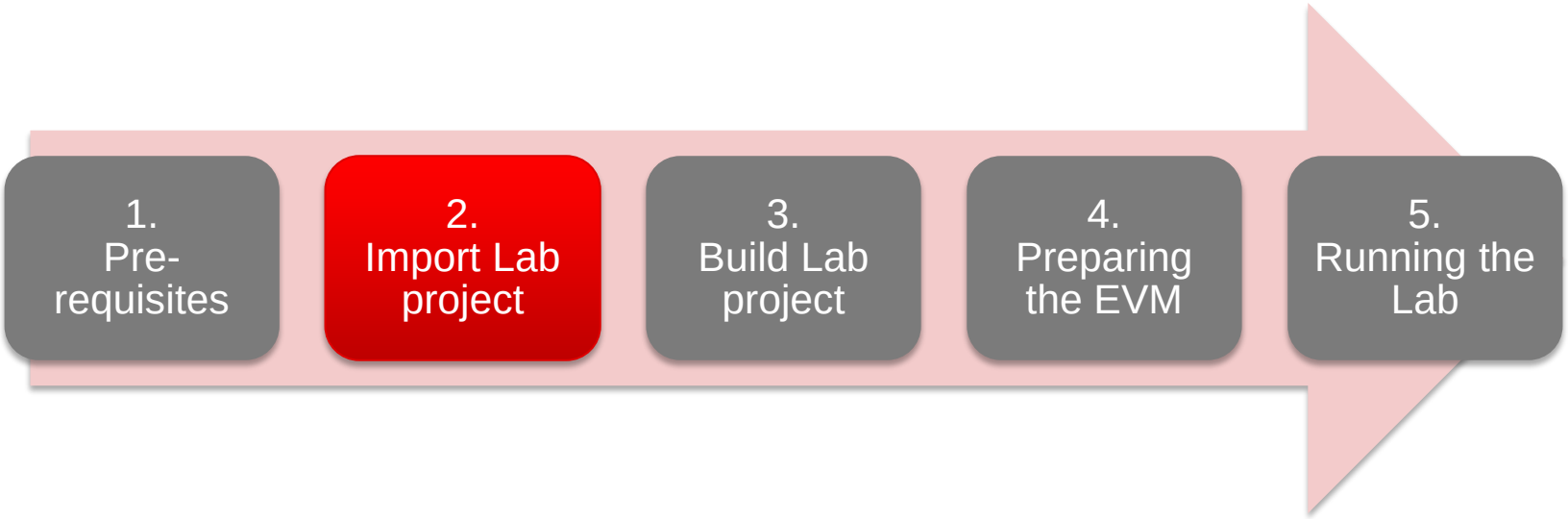


1. Pre-requisites

- It is assumed that you have the latest TI mmWave SDK and all the related tools installed as mentioned in the mmWave SDK release notes.
  - The mmWave SDK release notes include the links for downloading the required versions of the above tools.
- If you have already installed the mmWave SDK and all the required tools, you can move on to the next step i.e. downloading the lab on to your machine.

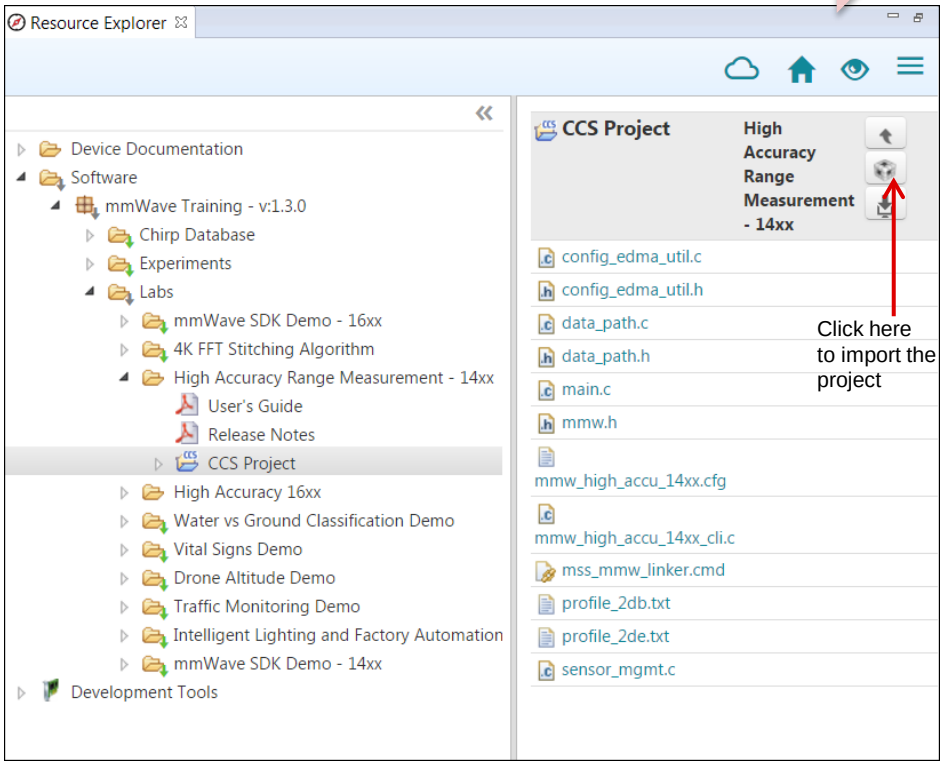
Tool	Version	Download link
CCS	7.4 or later	download link Please note that CCS v7.1 or later is mandatory. CCSv6.x cannot be used
TI SYS/BIOS	6.53.02.00	Included in mmwave sdk installer
TI ARM compiler	16.9.6.LTS	Included in mmwave sdk installer
TI CGT compiler	8.1.3	Included in mmwave sdk installer
XDC	3.50.04.43	Included in mmwave sdk installer
C64x+ DSPLIB	3.4.0.0	Included in mmwave sdk installer
C674x DSPLIB	3.4.0.0	Included in mmwave sdk installer
C674x MATHLIB (little-endian, elf/coff format)	3.1.2.1	Included in mmwave sdk installer
Mono JIT compiler	4.2.1	Only for Linux builds
mmwave device support packages	1.5.5 or later	Upgrade to the latest using CCS update process (see SDK user guide for more details)
TI Emulators package	6.0.0576.0 or later	Upgrade to the latest using CCS update process (see SDK user guide for more details)

Steps


1.
Pre-
requisites
2.
Import Lab
project
3.
Build Lab
project
4.
Preparing
the EVM
5.
Running the
Lab

2. Import Lab Project

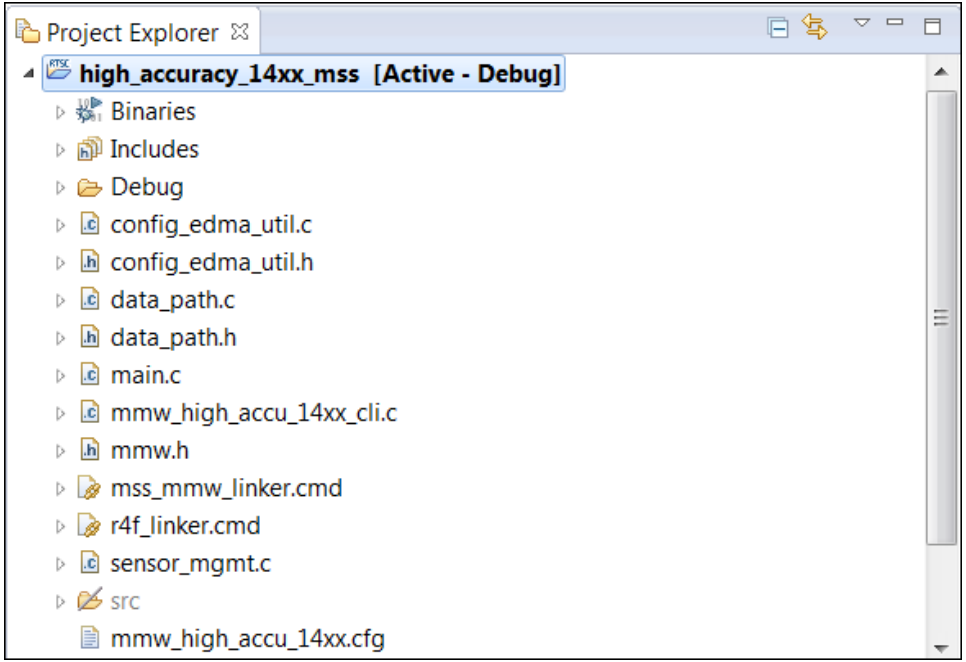
- As shown in quick start section, the lab projects are downloaded and should be available from C:\ti\mmwave_industrial_toolbox_<ver>
- Select the CCS project in the left view as shown.
- Click on the **Import to IDE** button which should be visible in the right side view after a successful download.
- This copies the project in the user's workspace and imports it into the CCS project explorer.
  - It is important to note that the copy created in the workspace is the one that gets imported in CCS. The original project downloaded in mmwave_training is not touched.



Texas Instruments

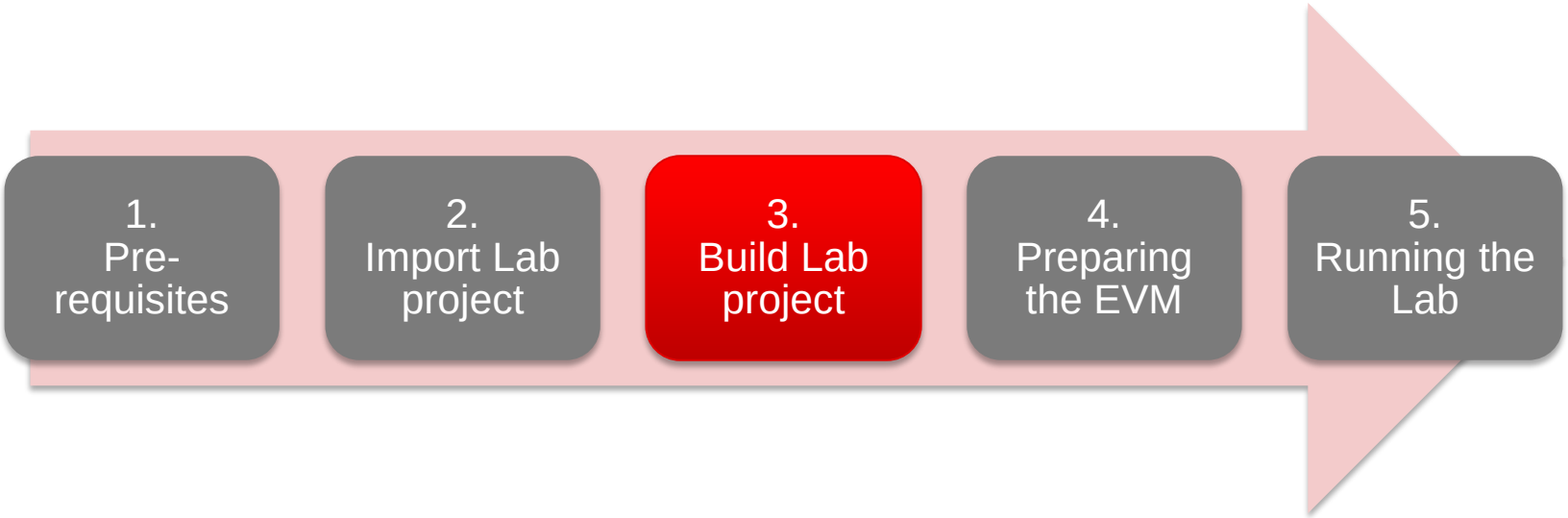
2. Import - continued

- After successfully completing the **Import to IDE** operation, the project should be visible in CCS Project Explorer as shown here.
- At this point, we have successfully downloaded the high accuracy 14xx mss and imported it in CCS.
- We are ready to move on to the next step i.e. Building the project.



Texas Instruments

Steps

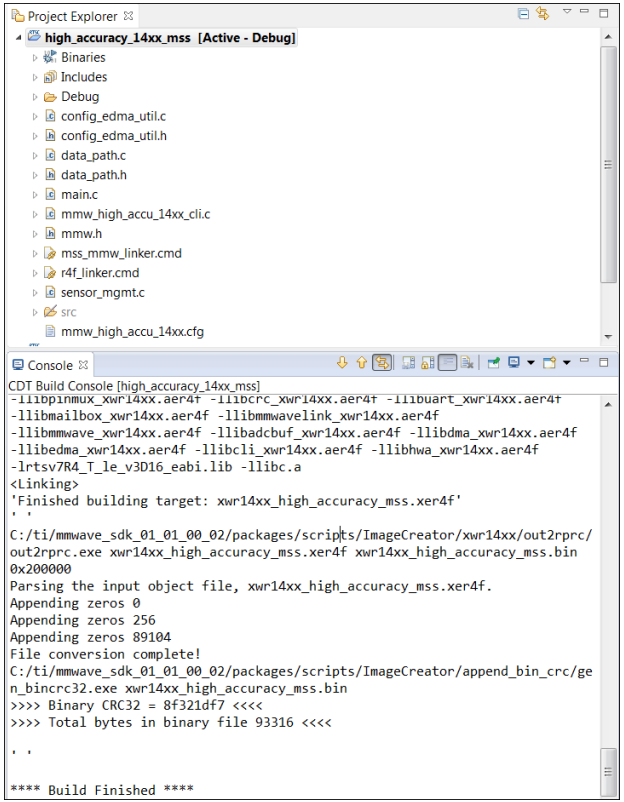

1.
Pre-
requisites
2.
Import Lab
project
3.
Build Lab
project
4.
Preparing
the EVM
5.
Running the
Lab

3. Build the Lab

- With the high_accuracy_14xx_mss project selected in Project Explorer, right click on the project and select **Rebuild Project**.
  - Selecting **Rebuild** instead of **Build** ensures that the project is always re-compiled. This is especially important in case the previous build failed with errors.

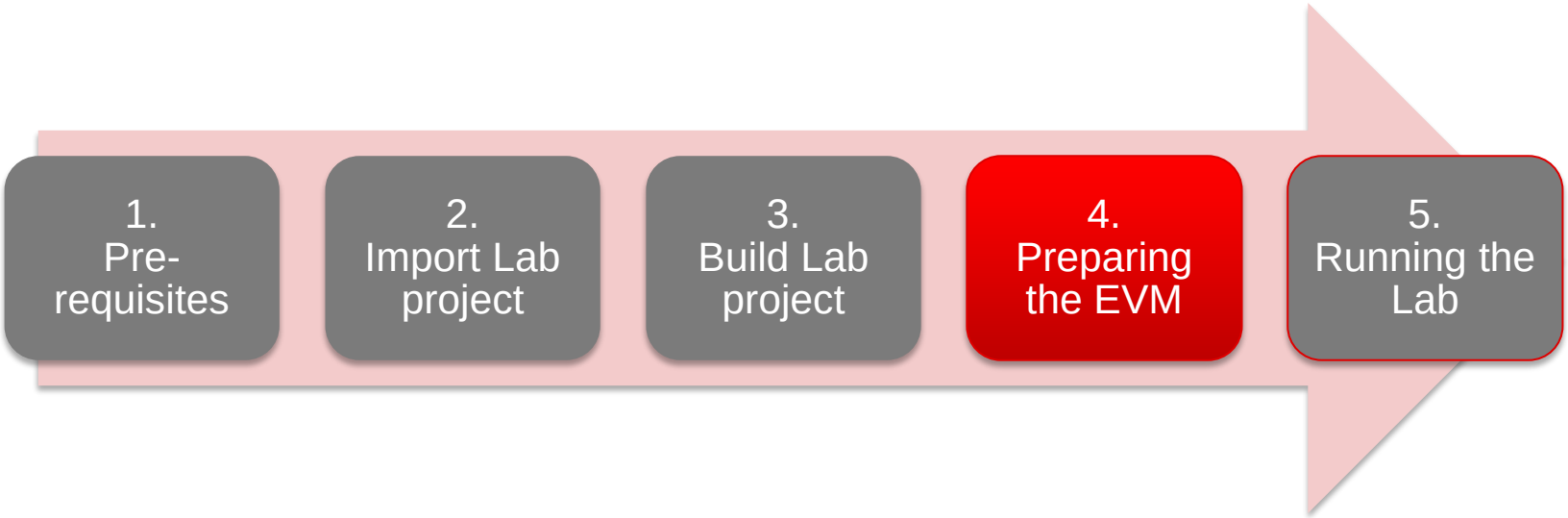
- On successful completion of the build, you should see the output in CCS console as shown here and the following two files should be produced in the project debug directory
  - xwr14xx_high_accuracy_mss.xer4f
  - xwr14xx_high_accuracy_mss.bin

- If the build fails with errors, please ensure that all the pre-requisites are installed as mentioned in the mmWave SDK release notes.



Texas Instruments

Steps

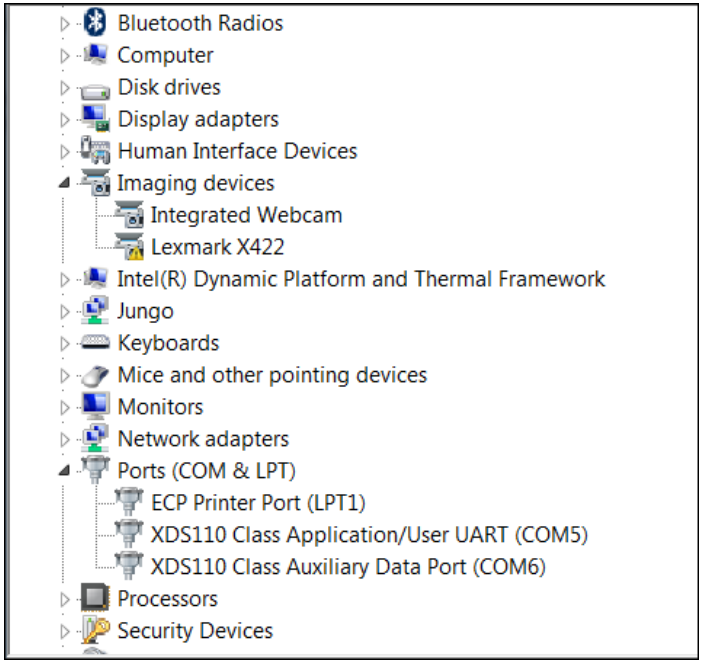

1.
Pre-
requisites
2.
Import Lab
project
3.
Build Lab
project
4.
Preparing
the EVM
5.
Running the
Lab

4. Preparing the EVM

- There are two ways to execute the compiled code on the EVM:
  - Deployment mode : Flashing the binary (.bin image) on to the EVM serial flash
    - In this mode, the EVM boots autonomously from flash and starts running the bin image.
  - Debug mode: Downloading and running the executable (.xer4f image) from CCS.
    - You will need to flash a small CCS debug firmware on the EVM (one time) to allow connecting with CCS. This debug firmware image is provided with the mmWave SDK.
  - As a recap, the build process in Step 3 produces both the .bin and .xer4f images.
- The Quick Start section has demonstrated the deployment mode
- The following presentation explains the second method i.e. Debug mode (CCS).
  - To prepare the EVM for debug mode, we start with flashing the CCS debug firmware image.
  - Please note that the same flashing process can be used to flash the Lab binary to run it in deployment mode.

4. Connecting to the EVM

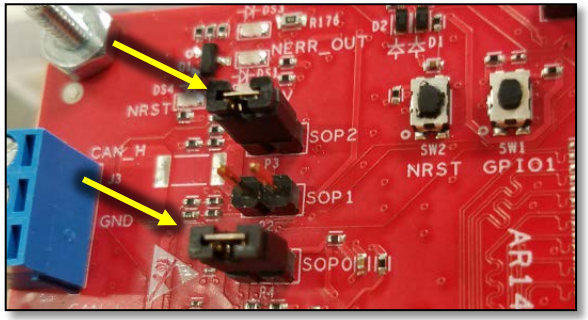
- Power on the EVM using a 5V/2.5A power supply.
- Connect the EVM to your PC and check the COM ports in Windows Device Manager
- The EVM exports two virtual COM ports as shown below:
  - XDS110 Class Application/User UART (COM_{UART}):
    - Used for passing configuration data and firmware to the EVM
  - XDS110 Class Auxiliary Data Port (COM_{AUX})
    - Used to send processed radar data output
- Note the COM_{UART} and COM_{AUX} port numbers, as they will be used later for flashing and running the Lab.

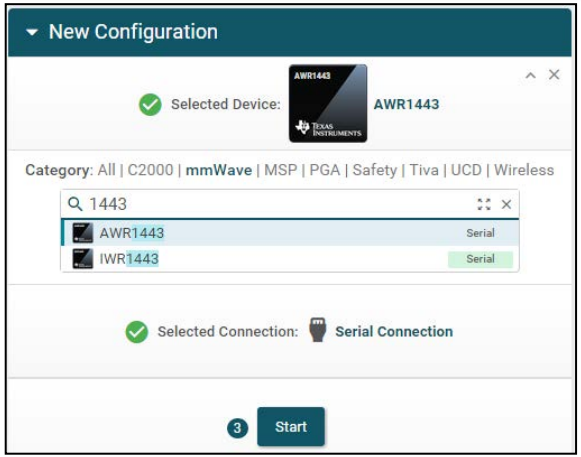


COM_{UART}: COM5 **COM_{AUX}**: COM6 The actual port numbers on your machine may be different

4. Flashing CCS debug firmware

1. Put the EVM in flashing mode by connecting jumpers on SOP0 and SOP2 as shown in the image.
2. Open the **UniFlash** tool
3. In the **New Configuration** section, locate and select the appropriate device (AWR1443 or IWR1443)
4. Click **Start** to proceed





4. Flashing CCS debug firmware

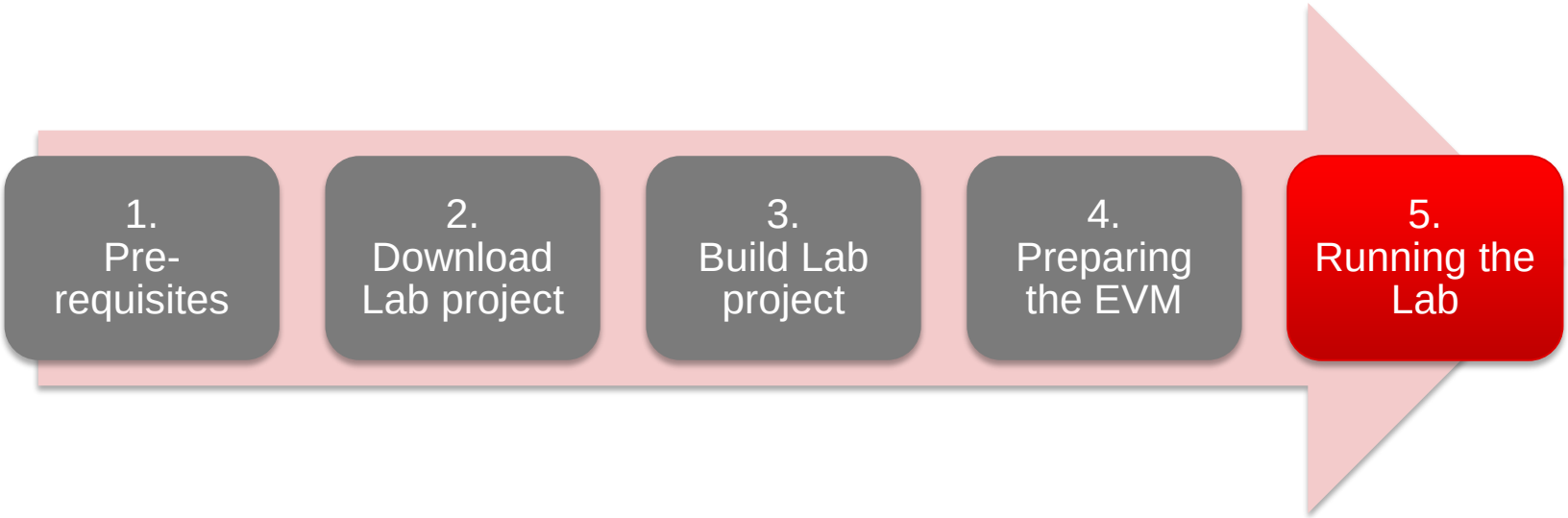
1. In the **Program** tab, browse and locate the Radar SS and MSS images shown below:

Image	Location
Meta Image 1/RadarSS	C:\ti\mmwave_sdk_<ver>\firmware\radarss**xwr12xx_xwr14xx_radarss.bin**
Meta Image 2/MSS	C:\ti\mmwave_sdk_<ver>\packages\ti\utils\ccsdebug**xwr14xx_ccsdebug_mss.bin**

2. In the **Settings & Utilities** tab, fill the **COM Port** text box with the Application/User UART COM port number (**COM_{UART}**) noted earlier

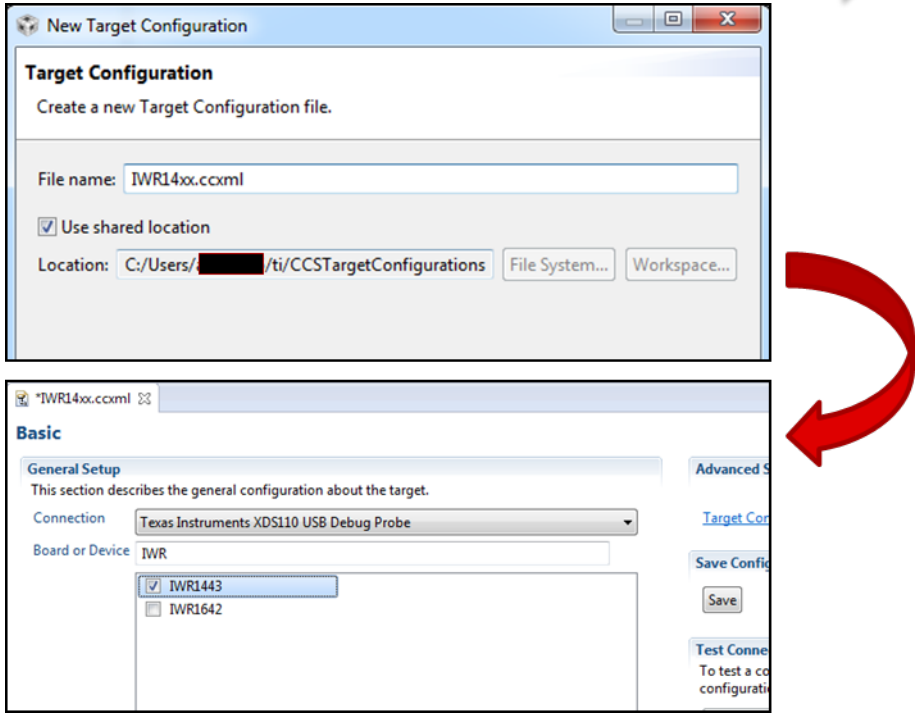
3. Return to the **Program** tab, power cycle the device and click on **Load Images**
4. When the flash procedure completes, UniFlash's console should indicate: [SUCCESS] Program Load completed successfully
8. Power off the board and remove the jumper from only header **SOP2** (this puts the board back in functional mode)

Steps


1.
Pre-
requisites
2.
Download
Lab project
3.
Build Lab
project
4.
Preparing
the EVM
5.
Running the
Lab

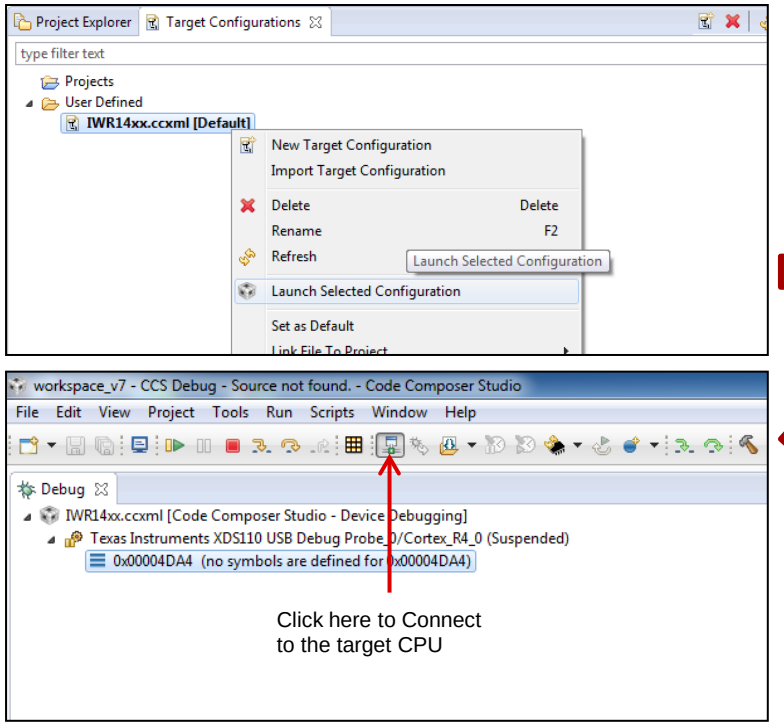
5. Connecting EVM to CCS

- It is assumed that you were able to download and build the Lab in CCS (completed steps 1, 2 and 3)
- To connect the Radar EVM to CCS, we need to create a target configuration
  - Go to File ► New ► New Target Configuration File
  - Name the target configuration accordingly and check the “Use shared location” checkbox. Press Finish
  - In the configuration editor window:
    - Select “Texas Instruments XDS110 USB Debug Probe” for **Connection**
    - Select AWR1443 or IWR1443 device as appropriate in the **Board or Device** text box.
    - Press the **Save** button to save the target configuration.
    - You can press the **Test Connection** button to check the connection with the board.



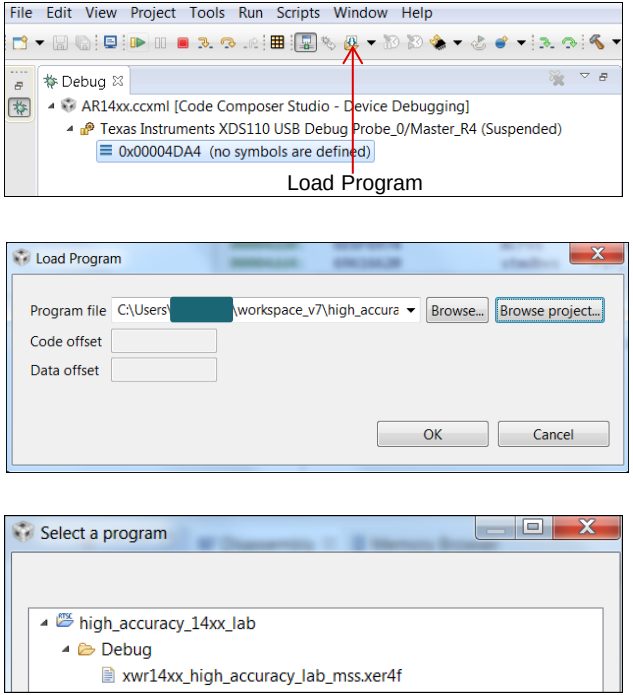
5. Connecting - continued

- Go to **View ► Target Configurations** to open the target configuration window.
- You should see your target configuration under **User Defined** configurations.
- Right click on the target configuration and select **Launch Select Configuration**.
- This will launch the target configuration in the debug window.
- Select the Texas Instruments XDS110 USB Debug probe and press the **Connect Target** button



5. Loading the binary

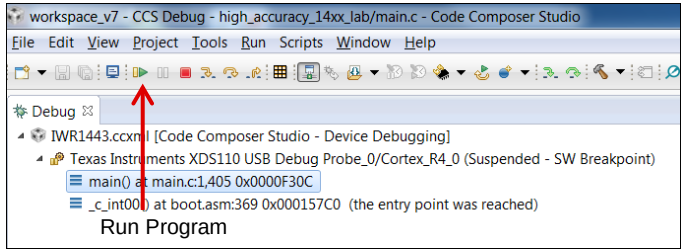
- With the target connected, click on the **Load** button in the toolbar.
- In the **Load Program** dialog, press the **Browse Project** button .
- Select the lab executable (.xer4f) as shown and press OK.
- Press OK again in the **Load Program** dialog.

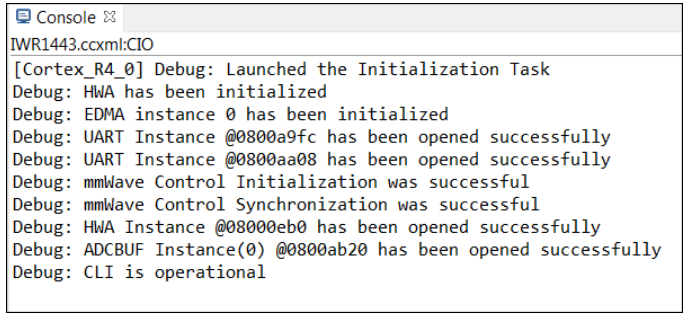


Texas Instruments

5. Running the binary

- With the executable loaded, press the Run/Resume button
- The program should start executing and generate console output as shown.
- If everything goes fine, you should see the “CLI is operational” message which indicates that the program is ready and waiting for the sensor configuration.
- Next use Google Chrome and navigate to the High Accuracy Visualizer for GUI display. Follow Quick Start Section 3 for more details.





Texas Instruments

Learn more about TI mmWave Sensors

- Learn more about xWR1x devices, please visit the product pages
  - IWR1443: http://www.ti.com/product/IWR1443
  - IWR1642: http://www.ti.com/product/IWR1642
  - AWR1443: http://www.ti.com/product/AWR1443
  - AWR1642: http://www.ti.com/product/AWR1642
- Get started evaluating the platform with xWR1x EVMs, purchase EVM at
  - IWR1443 EVM: http://www.ti.com/tool/IWR1443BOOST
  - IWR1642 EVM: http://www.ti.com/tool/IWR1642BOOST
  - AWR1443 EVM: http://www.ti.com/tool/AWR1443BOOST
  - AWR1642 EVM: http://www.ti.com/tool/AWR1642BOOST
- Download mmWave SDK @ http://www.ti.com/tool/MMWAVE-SDK
- Ask question on TI's E2E forum @ http://e2e.ti.com

© Copyright 2017 Texas Instruments Incorporated. All rights reserved.

This material is provided strictly “as-is,” for informational purposes only, and without any warranty.
Use of this material is subject to TI’s **Terms of Use**, viewable at TI.com