**IWR1843BOOST**

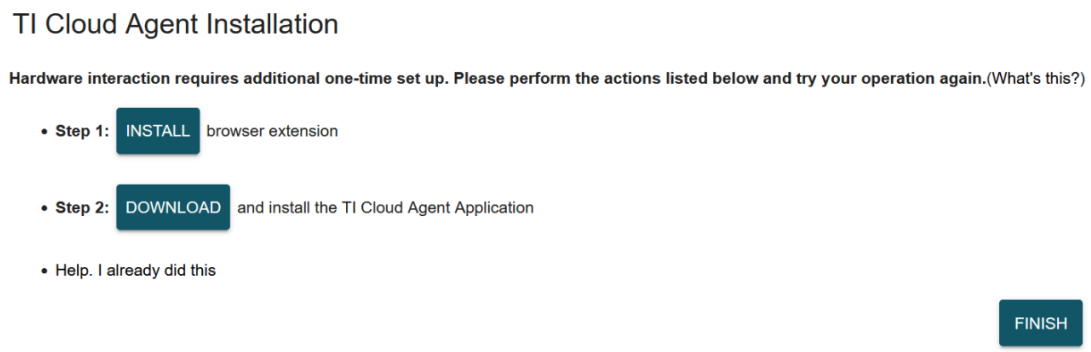
Download XDS Emulation Software Package (Port recognition):
http://processors.wiki.ti.com/index.php/XDS_Emulation_Software_Package

Download mmWave SDK (Development kit):
http://www.ti.com/tool/mmwave-sdk

Download UniFlash (Flash memory):
https://www.ti.com/tool/UNIFLASH
Alternatively, access using online dev tools (Requires installing TI Cloud Agent Application and browser extension):
https://dev.ti.com/

mmWave Demo Visualiser (Version compatible with IWR1843):
https://dev.ti.com/gallery/view/mmwave/mmWave_Demo_Visualizer/ver/3.6.0/

TI Cloud Agent browser extension and Application required for Demo Visualiser (prompted):




Flash Programming (can skip this step if already flashed):
Set SOP0, SOP2 to ON, SOP1 to OFF

Using UniFlash, select Serial Connection > IWR1843
Under Settings & Utilities, change the port number corresponding to XDS110 Class Application/User UART as seen in device manager (e.g. COM5)
Under Program, select binary file corresponding to the demo:

Located at
C:\ti\mmwave_sdk_03_06_00_00-LTS\packages\ti\demo\xwr18xx\mm\xwr18xx_mmw_demo.bin by default.

Click “Load Image” and wait for the flash to happen.

Don’t forget to set SOP2 to OFF for functional mode (remember to power cycle).
Click “Send config to mmWave device” on the Demo Visualiser

If everything went well, congratulations! You set it up correctly this time.

**DCA1000EVM**

In order to use the DCA1000EVM with the IWR1843BOOST, some additional steps are required:

Remove all cables before attempting to connect the two boards.

Follow the instructions on the User Guide for the DCA1000EVM in order to connect the two boards (in the shared Drive).

Once the boards have been connected, the switching for mode selection needs to be done.

In hardware configuration mode (when switch 2.5 at pin 5 is OFF):
Switch 1 should be set to [OFF, OFF, ON] (LVDS 16-bit mode).
Switch 2 should be set to [OFF, ON, ON, OFF, ON, ON, OFF, OFF]
(LVDS capture, Ethernet streaming, 2-lane LVDS, Raw data, (Software / Hardware configuration), FPGA ethernet configuration, Future use (None), Future use (None))

However, using hardware configuration has not been tested by myself, so please use software configuration.

Attach all the cables according to the user guide (ethernet, USB for configuring the DCA1000EVM on J1-Radar FTDI, 60-pin HD connector, USB for configuring IWR1843BOOST on the BOOST) - check the user guide if unsure.

Switch 3 should be set to RADAR_5V_IN in order to receive power from the IWR1843BOOST via the 60-pin HD connector cable.

Downloading mmWave Studio is required for interfacing the DCA1000EVM with a PC:
https://www.ti.com/tool/MMWAVE-STUDIO#downloads
Downloading this software requires that you gain approval first. Fill out the required fields and you should be approved to download the software.

Before using the DCA1000EVM, first set the power source to DC (effective off) and check that the IWR1843BOOST is working in the mmWave Demo Visualiser first. If this is working then you can continue by setting the power source to RADAR_5V_IN. After setting the power source, make sure to wait until all the error LEDs on the DCA1000EVM have gone off.

On the PC that the ethernet cable is connected to (either your laptop or a University computer), open a command prompt and run the command netstat -e twice (see https://kevincurran.org/com320/labs/netstat.html for more). If data is being sent via ethernet, then the number under “Bytes Received” should be increasing. To reset this count, toggle the power source switch on the DCA1000EVM.

On the PC connected to ethernet, open the control panel, then under the network and sharing centre, click on change adapter settings. You should set the IP to a static IP: 192.168.33.30 and the subnet mask to 255.255.255.0. This requires administrator access.

*For windows 11, follow these steps:*
*https://pureinfotech.com/set-static-ip-address-windows-11/*

To use mmWave Studio, you will need to install the MATLAB runtime engine version 8.5.1 32-bit from https://au.mathworks.com/products/compiler/matlab-runtime.html.
If you attempt to run mmWave Studio without this, it will throw you an error. If it has thrown you an error and you installed MATLAB runtime engine, restart mmWave Studio.

If mmWave Studio has launched correctly, it should detect the runtime engine, and a new tab called RadarAPI will open.
Using mmWave Studio:

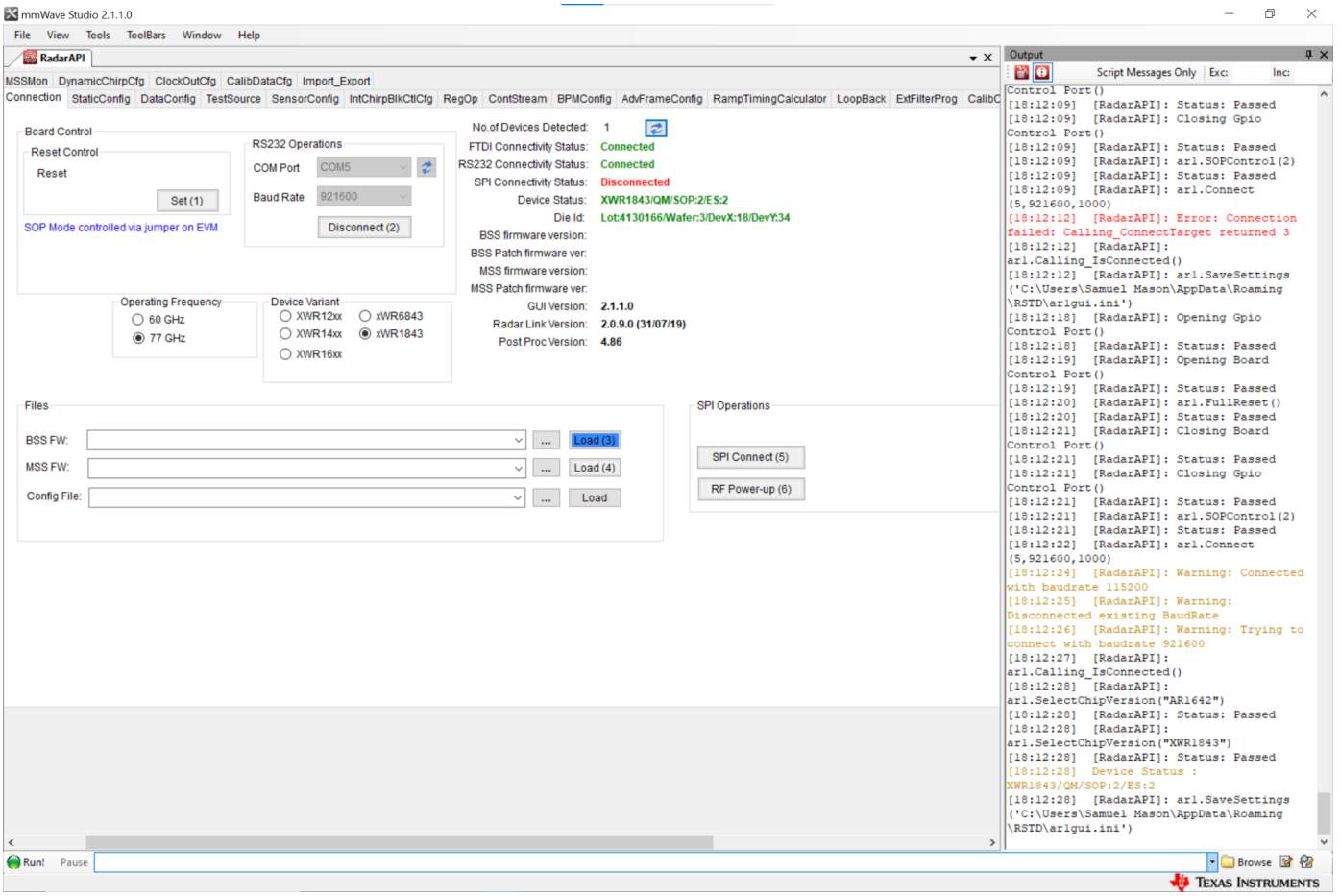
In device manager, the ports should now appear as follows:


Under connected devices, FTDI connectivity status should be connected, with RS232 and SPI disconnected.

To connect to RS232, first make sure that you are NOT connected to the mmWave Demo Visualiser using the port, otherwise access will be denied. Then, make sure you are in debug mode by setting the SOP switch to 011. Then press Set (1) followed by Connect (2) in mmWave Studio.

IMPORTANT: When connecting power to the IWR1843BOOST, make sure that the SOP mode is set to functional (001). Only once power has been provided should the mode be set to debug mode (011). For some reason this will cause an error otherwise, it is unclear why.

At this point, the mmWave Studio window should look like this:

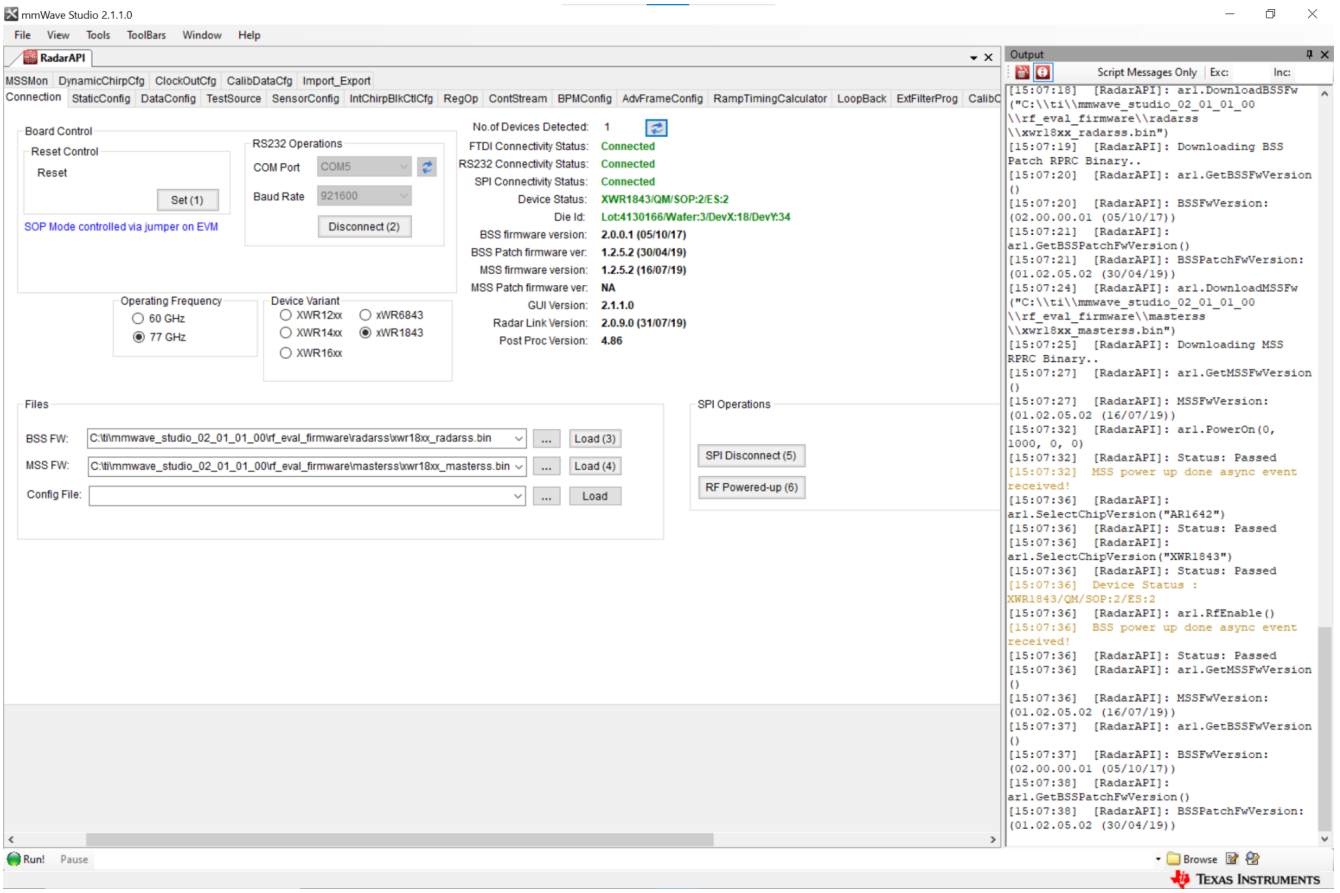



Set the BSS to C:\ti\mmwave_studio_02_01_01_00\rf_eval_firmware\radarss\xwr18xx_radarss.bin
Set the MSS to C:\ti\mmwave_studio_02_01_01_00\rf_eval_firmware\masterss\xwr18xx_masterss.bin

Click Load (3) followed by Load (4).

Make sure the SPI/CAN switch on the BOOST is set to SPI, then click SPI connect (5) followed by RF Power-up (6).

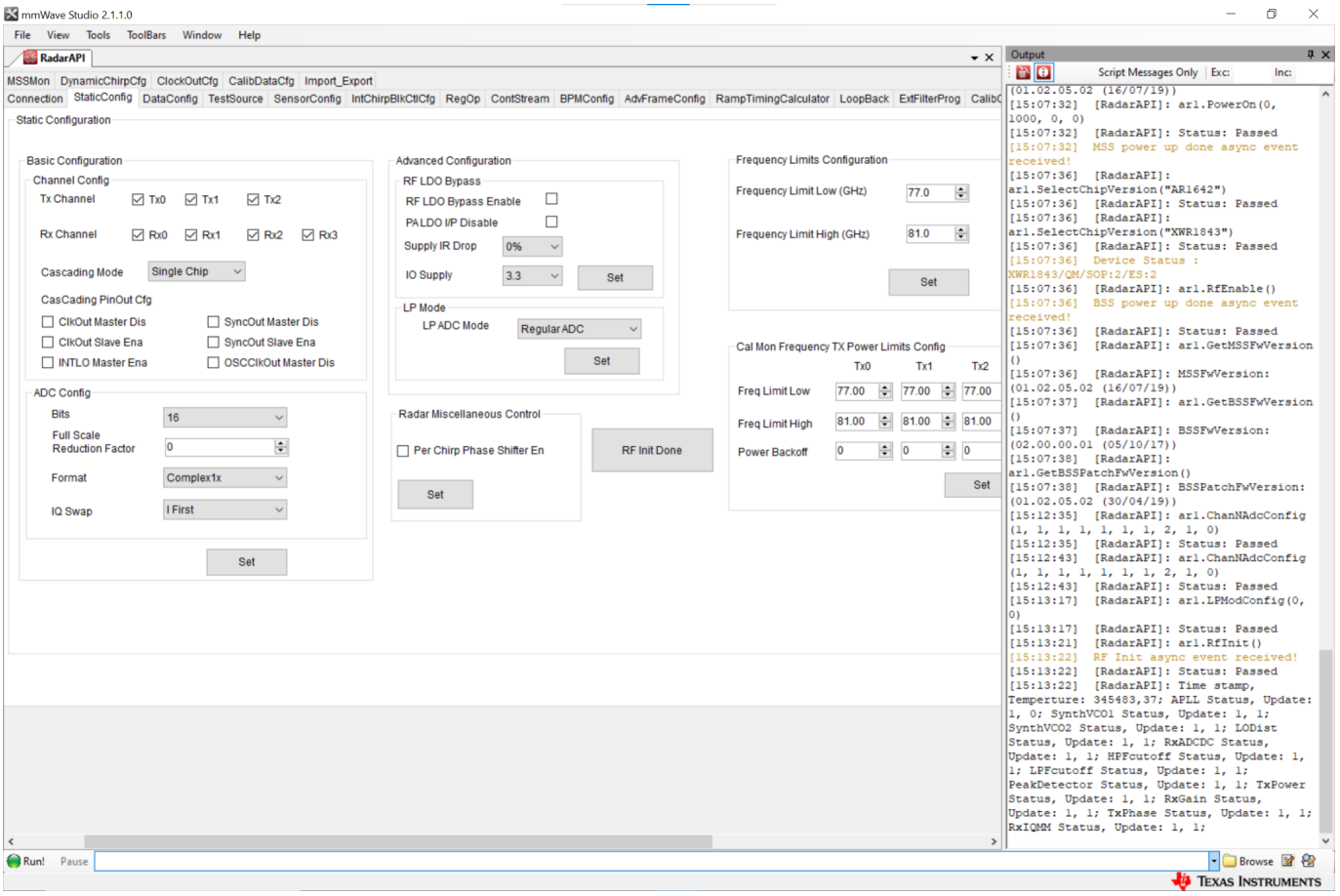
The mmWave Studio window should look like this after all these steps have been completed:



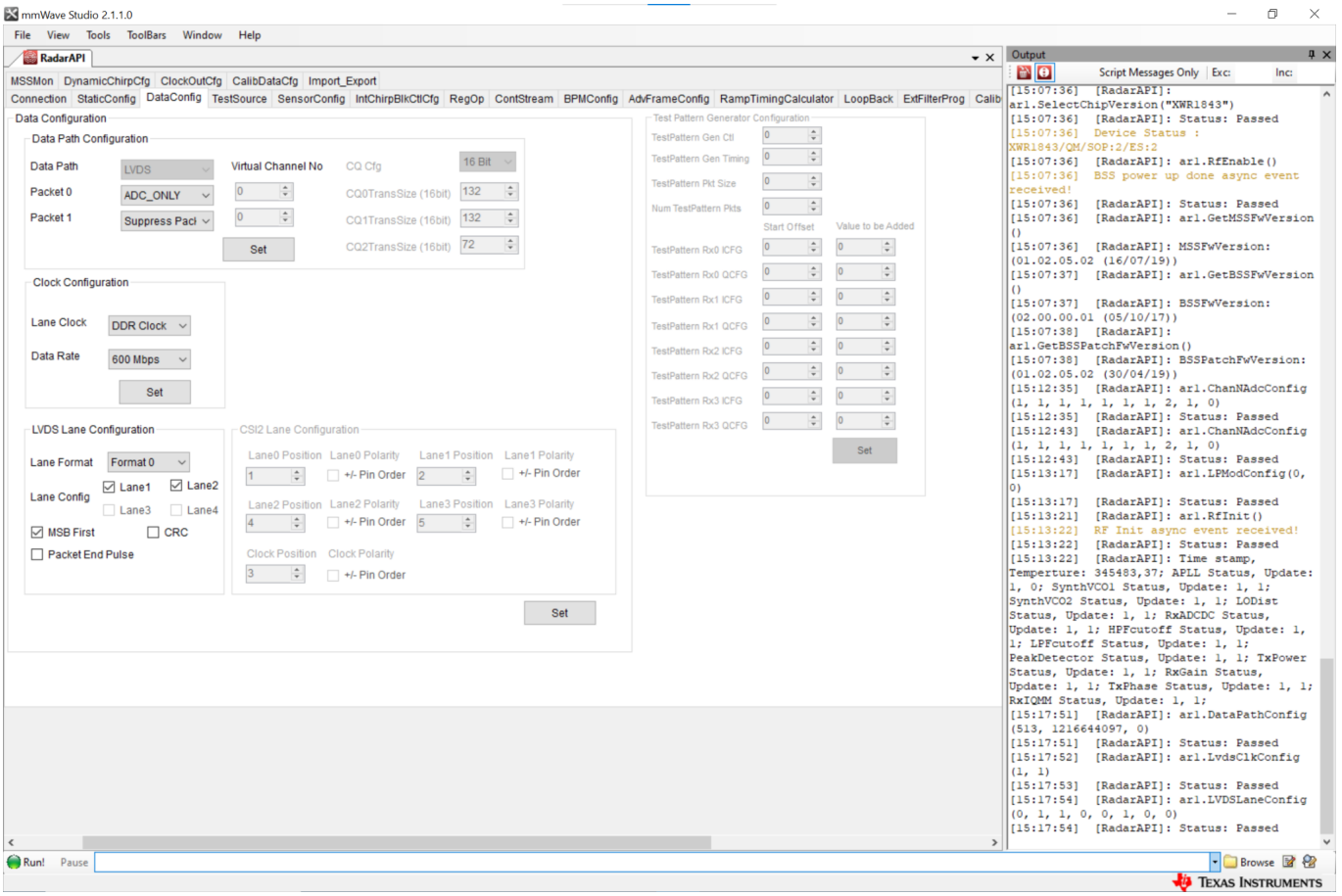

At this point, everything is set up correctly. You should refer to this video: https://www.ti.com/video/5827389052001 @ 7:45 onwards for more steps.

Settings for each of the tabs are as follows (these can be considered as defaults). Make sure to press set after each section (they will highlight blue, ONLY press the blue highlights unless you are certain of what you are doing).

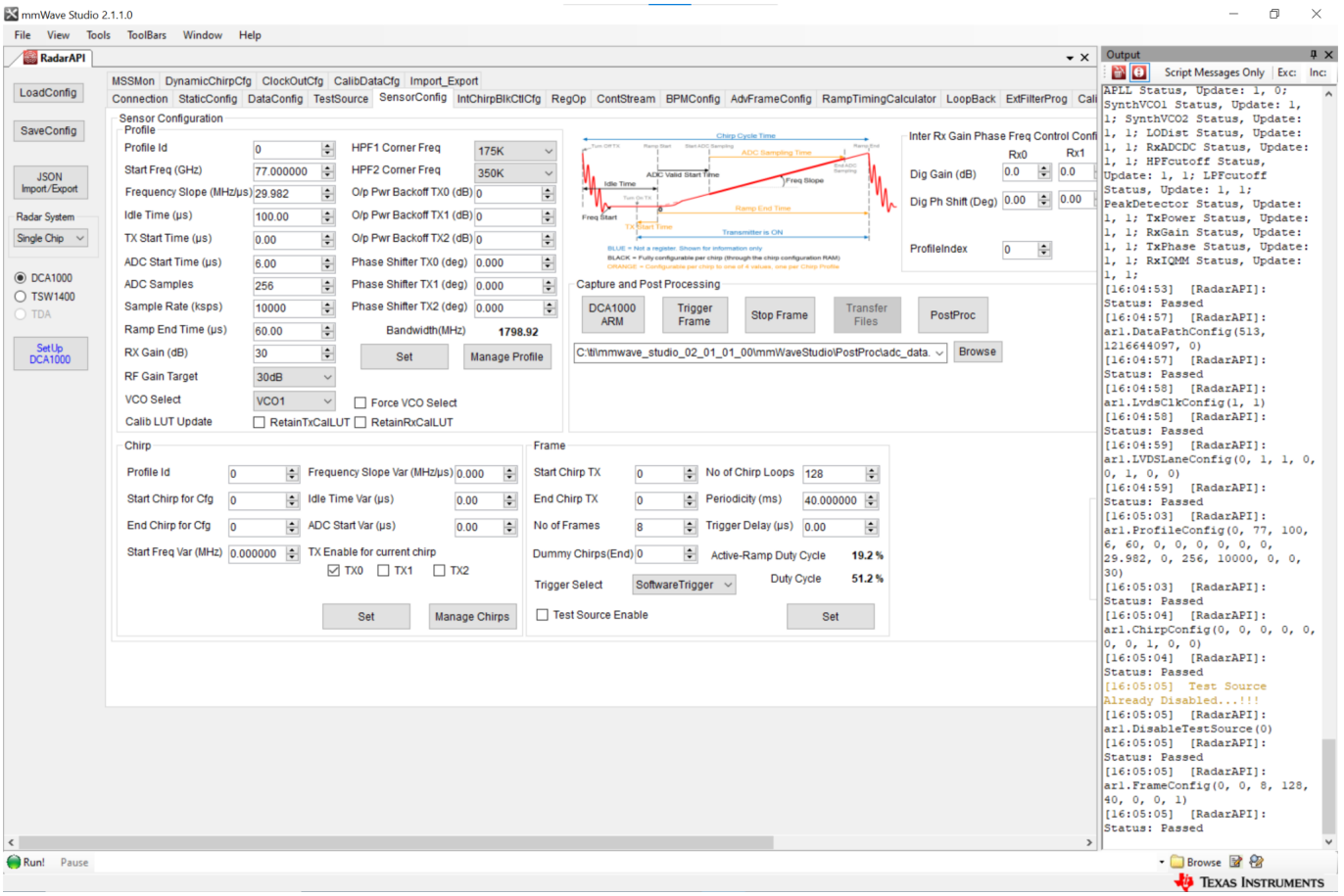
In the static configuration tab:




In the data configuration tab:

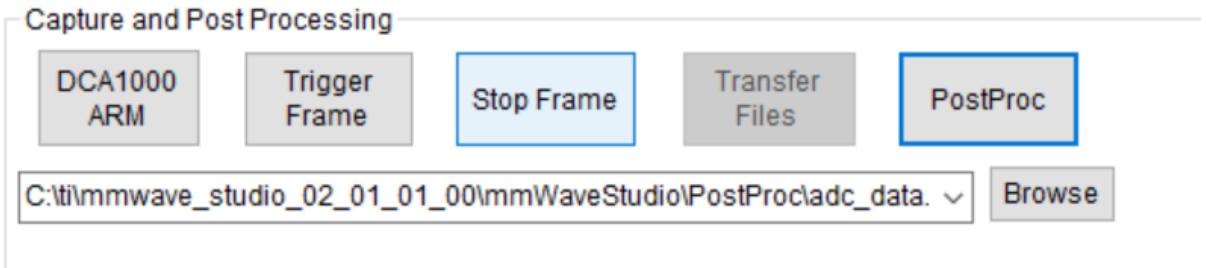



In the sensor configuration tab:



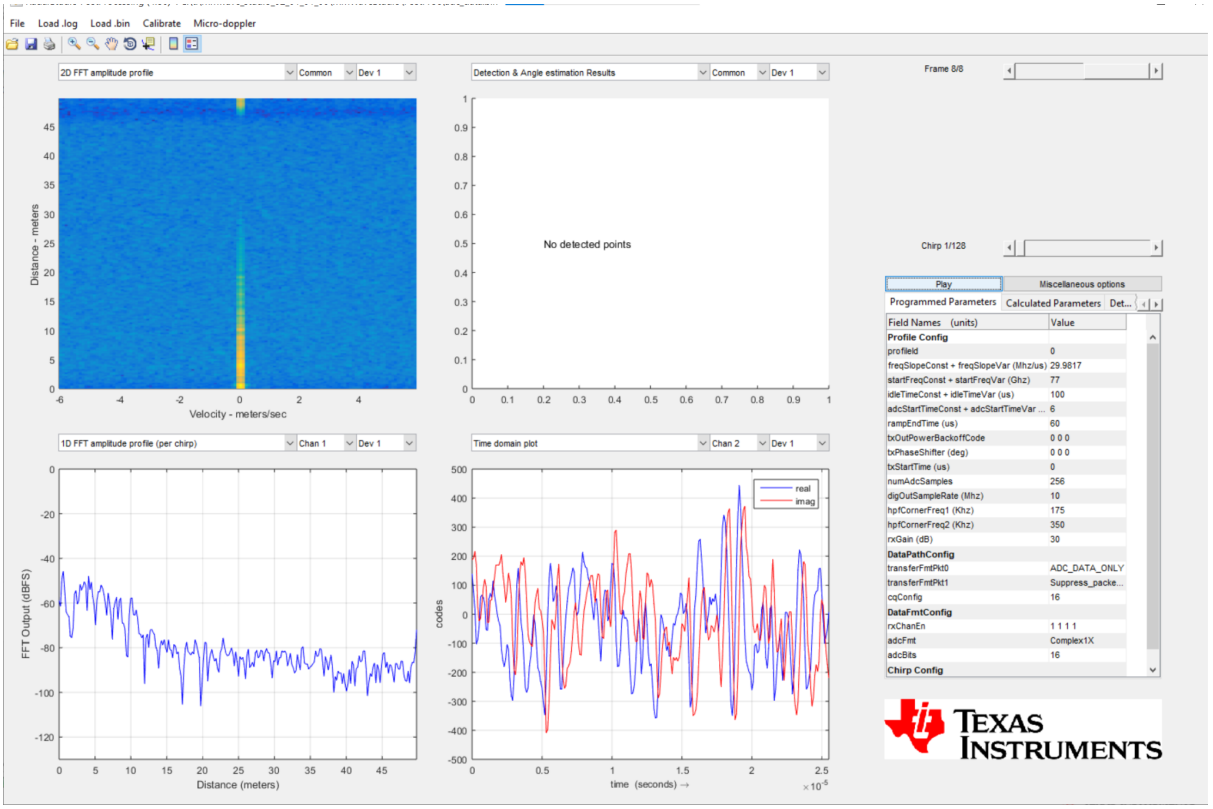

Once everything has been set, you want to save your configuration (if applicable) then click on Set Up DCA1000 (you may need to scroll to the left at the bottom).

IMPORTANT: Make sure that mmWave Studio and the DCA1000EVM CLI Control have been allowed in windows firewall settings for both public and private networks, else you will be unable to connect.




The above are used to capture data. Press DCA1000 ARM to start capturing, then Trigger Frame to stop capturing. Saved data is stored in the location specified (default).

Using the PostProc tool, you can view information about the captured data.
Press the play button to view changes over the frames captured.




See the video from before @ 13:25 for information about the data capture flow and data format.

You can also use the PostProc tool to view data that was previously captured. First you should leave the path blank, then click on PostProc. This will bring up an empty window. You can then select the binary and log files in order to do post processing on the collected data. The binaries for each capture should be loaded separately, but the log file is common to all captures within a single session.

See mmWave Studio Cascade User Guide section 8.2.2 in the google drive for information on this.