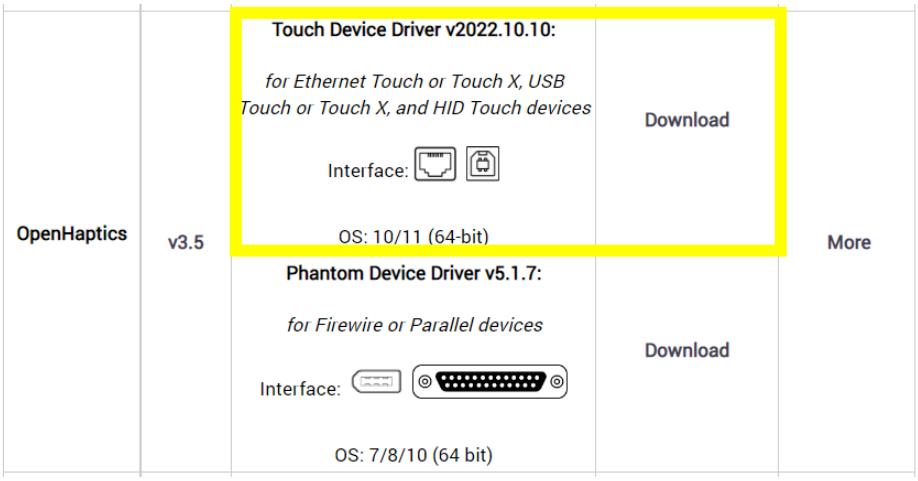
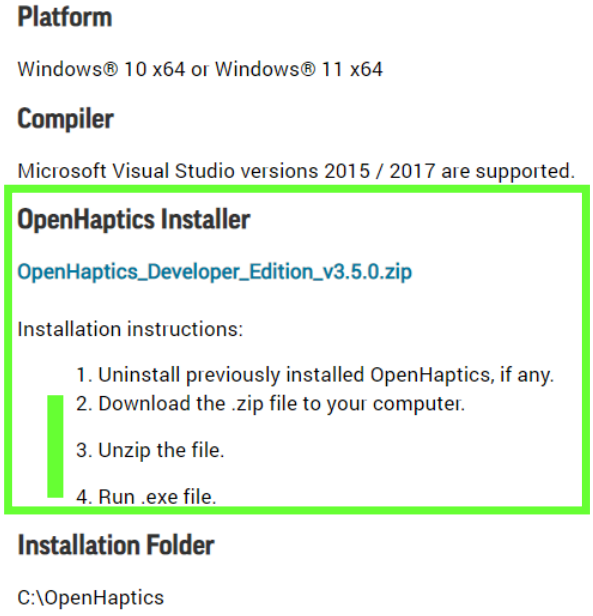
How to install the PhanTorque Simulink block for Matlab 2022b

You should have Matlab2022b with Simulink already installed on your PC.

Follow this link https://support.3dsystems.com/s/article/Haptic-Device-Drivers?language=en_US and look for OpenHaptics – v3.5. Download **Touch Device Driver v2022.10.10** and install it to get the Touch Smart Setup app. We will use this app to communicate with the Phantom robot and calibrate it.

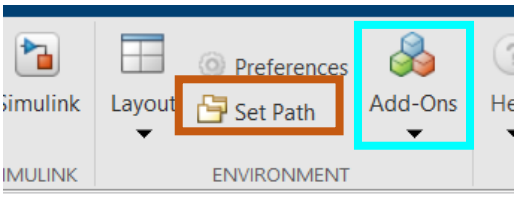


Then, from here https://support.3dsystems.com/s/article/OpenHaptics-for-Windows-Developer-Edition-v35?language=en_US download the zip folder OpenHaptics_Developer_Edition_v3.5.0.zip and follow the instruction in the screenshot to install:



Download from Teams the .zip folder phantom3Dof.zip into the MATLAB folder (usually it's in C:\Users*yourUserName*\Documents\MATLAB) and unzip it.

Open Matlab, on the upper tab choose Home and click on the Set Path icon. Add the folder I gave you to the Matlab path, with its subfolders.



Let's get some Matlab add-ons! Look for MinGW (**MATLAB Support for MinGW-w64 C/C++ Compiler**) and install it. Do the same with the **Simulink 3D-Animation toolbox**, you will need it for the assignment to use the VR sink block.

Now it's time to install the Simulink block for the Phantom robot. In the Matlab command window copy, paste – all on the same line - and run this command (*be careful to adapt the path!* It depends on where you have unzipped the phantom3Dof folder):

mex -IC:\Users**yourUserName\Documents\MATLAB**phantom3Dof\ -LC:\Users**yourUserName\Documents\MATLAB**phantom3Dof\library PhanTorque_3Dof.c -lhdu -lhd

In my case, it looks like this:

You should get this message

Building with 'MinGW64 Compiler (C)'.

MEX completed successfully.

You can check that everything is working by opening the .slx files for the 2 exercises: the PhanTorque block should now be found!

What to do in lab

You can plug the Phantom USB cable to your PC. Run the Touch Smart Setup app, check the robot name (usually it's Default Device) and click on save configuration. Open Matlab and your Simulink files: you are good to go! Just check that the robot name in the PhanTorque Simulink block is the same as the Setup app: change it to 'Default Device'