This is the folder including Abaqus scripts for the Eurohaptics 2014 paper entitled "Computational Modeling Reinforces that Proprioceptive Cues May Augment Compliance Discrimination When Elasticity Is Decoupled From Radius of Curvature".

All \*.py files are Python 2.6 scripts and modules to perform the analysis. To use this files, you would need to do the following set-ups:

1. Install Abaqus 6.12-1. This code have not been tested on other versions, but should work with later versions since Abaqus is backward compatible.
2. Install SciPy 0.13 onto the python folder that shipped with Abaqus. If you are using Windows, install python 2.6 standalone first, and then install scipy on the standalone python. Copy the scipy folder under site-pakcages to the corresponding folder in the Abaqus python.

To be able to use this code, you will need to have background knowledge to know scripting layer API of Abaqus python. For a complete documentation, please refer to [this link](http://docs.abaqus.com:8000/aoss-hdocs/v6.12/books/cmd/default.htm).

To repeat the result in this paper, the following steps are recommended:

# Get 2D model geometry from the 3D model

1. Open Abaqus and set working folder at where the readme.docx is located.
2. Import the module extractContourCoord.py from getgeometry package.
3. Run the extractNodeCoordList function in that module. This should convert the raw input file ‘rawmodel.inp’ to two csv files named ‘frontCoordArray.csv’ and ‘sideCoordArray.csv’, which contains the info for 2D model building.
4. Check sideCoordArray.csv to make sure the result is reasonable. It should look like a side projection of a human finger contour.

# Fit the material properties to skin deflection

1. Setup the models for fitting by using createmodel.setfit
2. Exhaust all parameters by using fitmodel.defelction.exhaust. Two tips needs to be run, namely ‘pe\_rigid\_d317\_tip’, meaning the PE (plane strain) model, rigid tip, diameter 3.17, and another ‘pe\_rigid\_d952\_tip’.
3. Open the output csv file from the exhaust, and then use Excel manually to take average material properties with fitting above 0.80. Check the ./fitmodel/csv/r2/r2table\_pe\_rigid\_tip\_averaged.xlsx for my result – you value should be similar to mine (5.71, 4.33).
4. Then, run del.magnitude.autoOptimize() (after imported). This should automatically find the right magnitude of materials to fit the force-displacement relationship. Check the same excel file in step 2.3 – that is what I used to average material properties from different subjects.
5. The material property is now complete, in the excel file. It should be comparable to what Wu and Maeno got – I got 1211643.334, 50698.6528, 2372.830354 Pa for epidermis, dermis and hypodermis, and yours should be reasonable close to this value.

# Run simulations

1. The simulation driver code are all in the package of ‘sim’. It is intentionally named differently from ‘simulation’ because Abaqus itself has a package called simulation. Import the whole package.
2. Use sim.sphereas.init to initiate the classes for the simulation. The parameters are listed in the same module.
3. After the classes are initiated, run sim.sphereAs.run() to compute the FE analysis.
4. After all model runs have been completed, use sim.sphereAs.analyze() to extract the odb data to python pickle files.

# Data visualization

1. All code necessary to generate the plots in the paper are in the folder ./fig/. Not all figures generated are placed in the paper, so feel free to shop around.
2. Note that, the python code under this folder need to be run using a standalone python, with matplotlib installed. The matplotlib is not compatible with Abaqus because of the Tkinter backend was modified in Abaqus.
3. Most of the file names of the python script under fig folder is self explanatory – if you do not know what this file is used for, just check the output figure directly.