Instructions on Admittance Method Simulations:

==========

1. Create a voxel model (.model format)

2. Create the *.in file (One example is provided below)

- Example: This example has the filename 'modelB.in'

material	1	1e7	0	1e7	0	1e7	0	Air
material	2	0.714	0	0.714	0	0.714	0	GallBladderBile
material	3	0.666	0	0.666	0	0.666	0	BodyFluid
material	11	23.584	0	23.584	0	23.584	0	Nerve
material	17	2.934	0	2.934	0	2.934	0	Muscle
material	254	1e-4	0	1e-4	0	1e-4	0	Metal

 # Meshing control
unitvoxelsize 0.001
maximumsize 2

 # Charge injection
spice i 1a [node 400 70 1470] [node 199 315 1164]
nodename 199 315 1164 0
nodename 400 70 1470 040000701470

 # Other
matrix modelB.model
meshfile modelB.mrm
networkfile modelB.net
nodevoltagefile modelB.vof
ispreadfile modelB.spread

world 586 340 1878

- Explanation of lines in *.in file:

  - Lines with 'material' at the beginning are the material properties.
    - They are in the following form.
    - material 'material #' 'x resistivity' 'x permitivity' 'y res.' 'y perm.' 'z res.' 'z permitivity'

  - Heading '# Meshing control':
    - unitvoxelsize gives the size of the voxel (in this case 1 mm)
    - maximumsize gives the maximum size of the meshes that will be calculated with the mesher program.

(maximumsize == 1 will produce a network that has no meshing, with a value for each voxel)

- Heading '# Charge injection':
  - The first line is the current source stimulation.
  - It is in the following form:
    spice i 'filename of *.cur file without extension' [node '+xcoord' '+ycoord' '+zcoord'] [node '-xcoord' '-ycoord' '-zcoord']

    The *.cur file is discussed in the next section. The two nodes specified on that line are what the src is connected to (+ and -)

  - The lines with 'nodename' at the beginning define the nodes that any sources are connected to.
  - They are in the form:
    Nodename 'xcoord' 'ycoord' 'zcoord'
    'nodename' (0 for ground node, the coordinates without spaces for any other nodes)

  - If you change the current source locations, make sure to change those node definitions appropriately.
  - You can add as many current sources as desired.

- Heading '# Other':

  - This section defines the filenames for all of the files that will be used in the simulation.
  - There is no need to change anything except for the filenames (keeping the same extensions). Just make sure that they match the files that you want to use for the simulation.
    (It's easiest to make them all the same name for consistency and ease of debugging when a sim. is not working)
  - 'world' defines the size of the model in the following form:
    world 'xsize' 'ysize' 'zsize'

****** The .in used in this paper is provided in the folder (P120_waveform0.in) *****

3. Current source files

- File extension: .cur
- Make sure that the filename has the same name as defined in the *.in file (discussed in previous section)
- The first line of the file includes the sampling rate. It is in the following form:
% 'sampling rate' 0
- The next lines are the magnitude of the stimulation in Amps

- Each line will have one value for the stimulation for one step. i.e. the first line is for the '0' step, the second line is for the '1' step, etc.

- The number of steps that you run the simulation for is defined when it is run (discussed in section 4)

- Example:

```
% 1e-7 0
0
100e-6
100e-6
0
0
```

****** The .cur used in this paper is provided in the folder (RetinaCur0.cur) *****

4. Run Simulation:

- mesher:
  - Compile the “mesher” code provided in “AM” folder
  - command line: mesher -f 'filename'
  - This produces the *.mrm file that is defined in the *.in file. It is a meshed version of the *.model file

- netgen:
  - Compile the “netgen” code provided in “AM” folder
  - command line: netgen -f 'filename'
  - This produces the *.net file, which is the electrical network containing resistors and capacitors for the edges of every voxel.

- material properties:
  - Before running the AM code, you are required to also generate the material_properties.txt file.

****** The material_properties.txt used in this paper is provided in the folder *****

- ftdss:
  - Type ftdss on the server to get the get the menu of all of the options. It allows you to specify the tolerance of the solver, the duration, step size, etc.
  - The output is a *.vof file for every timestep (describing the voltages in the model at that specific timestep). This gives you the voltage at each node of the voxel. Then, you need to use interpolation function to compute the voltage at the center of each voxel.