

# A How-To Guide for Performing Parameter Sweeps in FEKO

William Miyahira

July 22, 2021

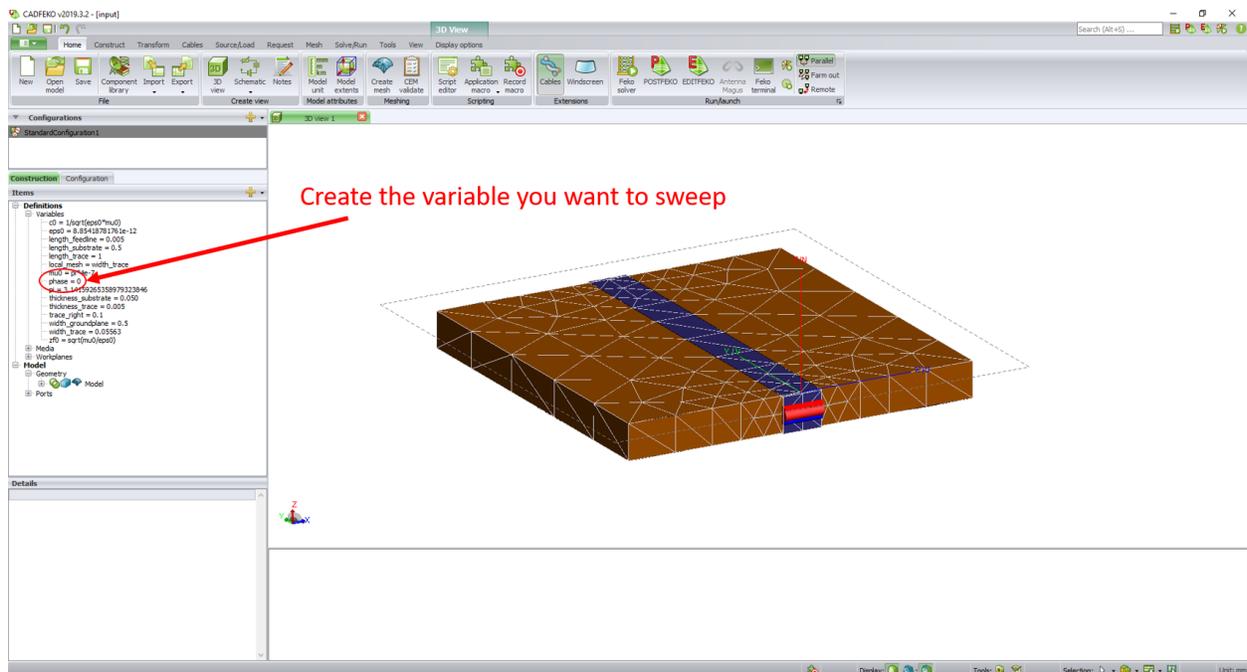
This document outlines how to perform a sweep of an arbitrary parameter in FEKO, both locally and on the W&M HPC. It should be noted that at the time of writing this I have only looked into sweeping a single parameter at a time (including frequency). Though there is the option to select multiple parameters to sweep, doing will increase the overall run time of the job.

At the end of the document, I also outline how to change the location of the FEKO temporary directory. This is primarily useful when running out-of-core jobs, as you can change it to a hard disk with more space for larger models.

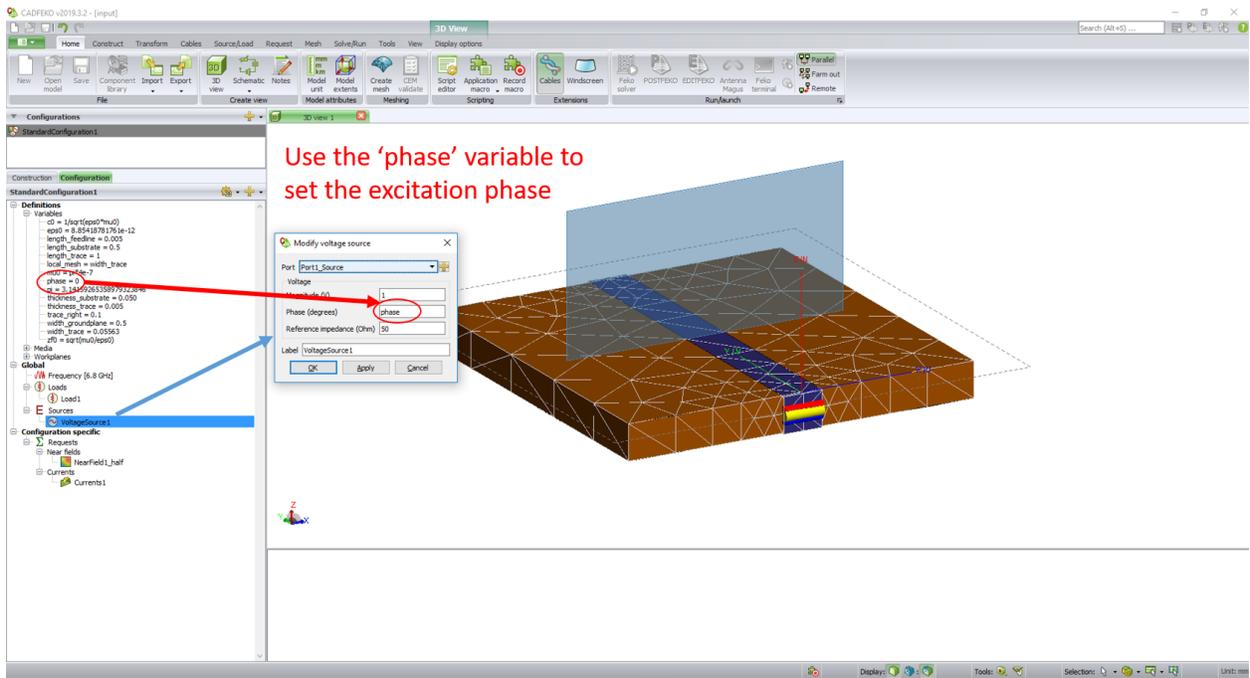
## Local Parameter Sweeps

First let's look at how to do a parameter sweep locally on the lab workstations. In this example I sweep the phase of a voltage source for a single microstrip, but the technique can be applied to any arbitrary parameter(s).

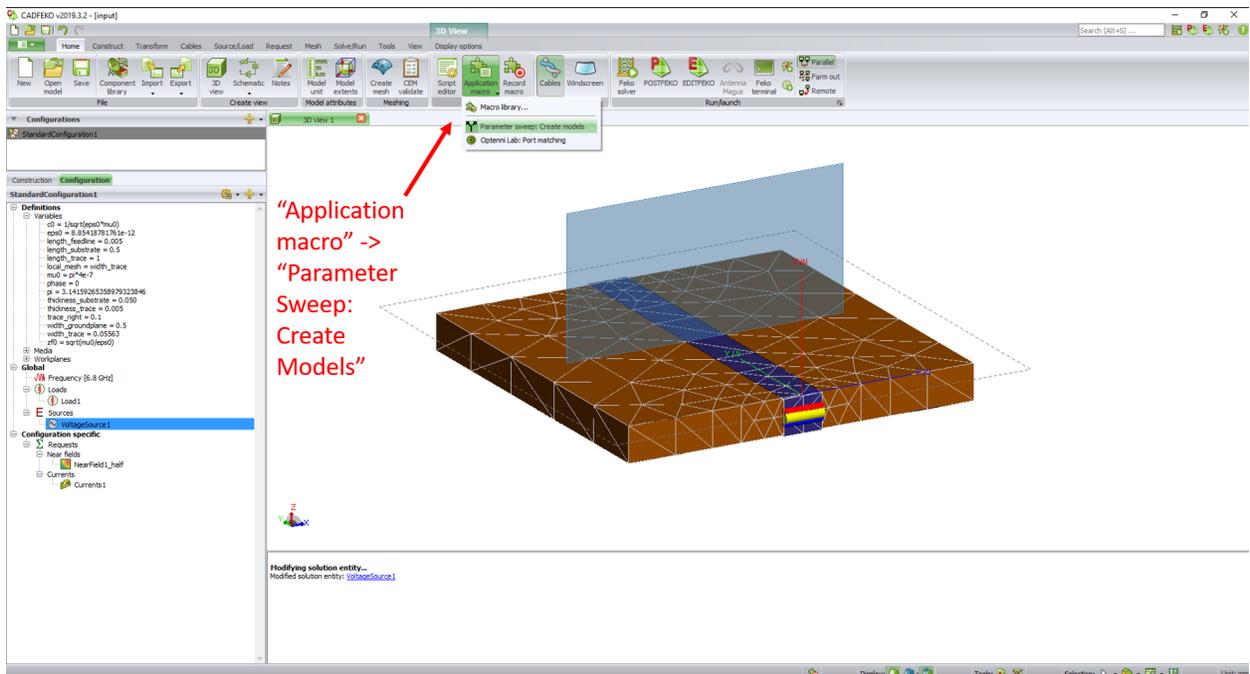
The idea in this phase sweep example is that we want to set the excitation phase of the voltage source equal to a variable rather than a fixed value. So to start let's define a variable called *phase*. Here I set  $phase = 0$ , but the value doesn't really matter since we choose what values it takes later on.



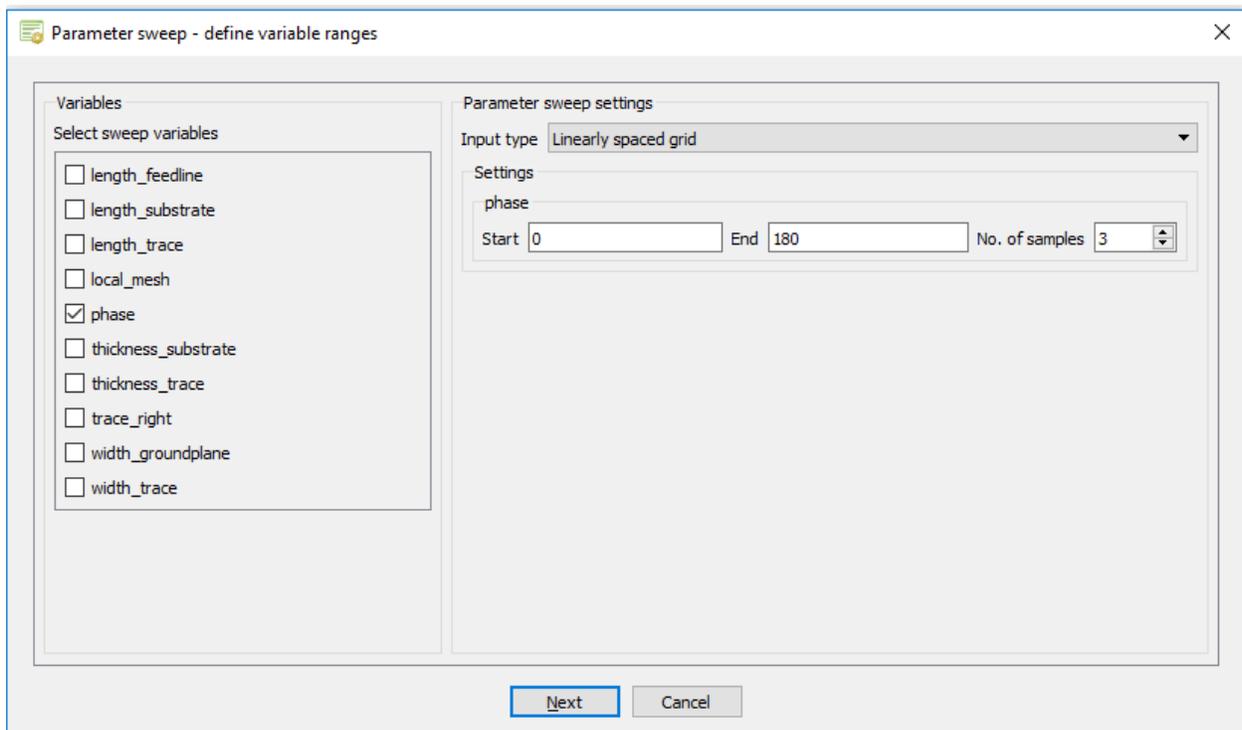
Next, define the phase of the voltage source to be the variable *phase*.



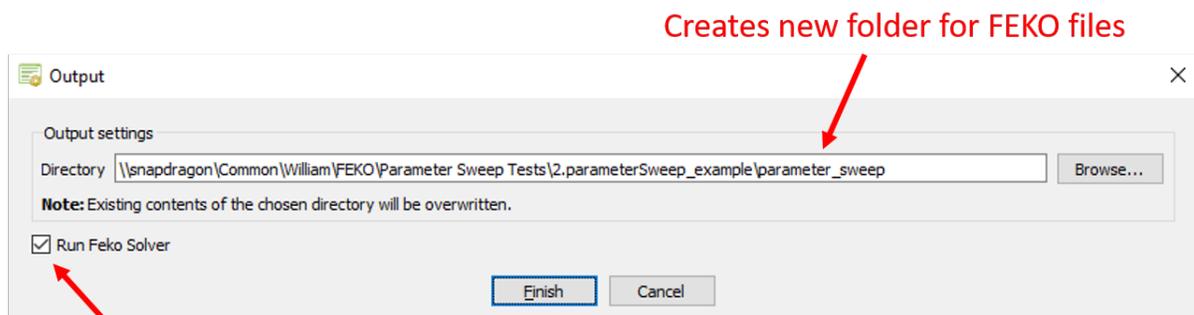
When you are ready to run the sweep, save your work. Then, on the *Home* tab of the top menu go to *Application macro* and select *Parameter Sweep: Create models*.



When selected, a menu will pop up that allows you to select which parameter(s) you want to sweep and which values that parameter will take during the sweep. Here I choose a linearly spaced grid going from  $0 \rightarrow 180$  with three samples. Thus, the variable *phase* will take on values 0, 90, and 180 when ran.



When you are done selecting the sweep variables and their ranges, select *Next*. Another menu will pop up that asks where you would like FEKO to put the files generated for the sweep, as well as whether you would like to run the models when *Finish* is selected. FEKO will automatically add the *parameter\_sweep* folder to your current directory to put the sweep files, but you can choose whichever directory you want by selecting *Browse...*



**"Run Feko Solver" selected**

Once *Finish* is selected, FEKO will generate  $n$  models, with  $n$  being the number of sweep samples (in this example,  $n = 3$ ) and put them all in the chosen directory.

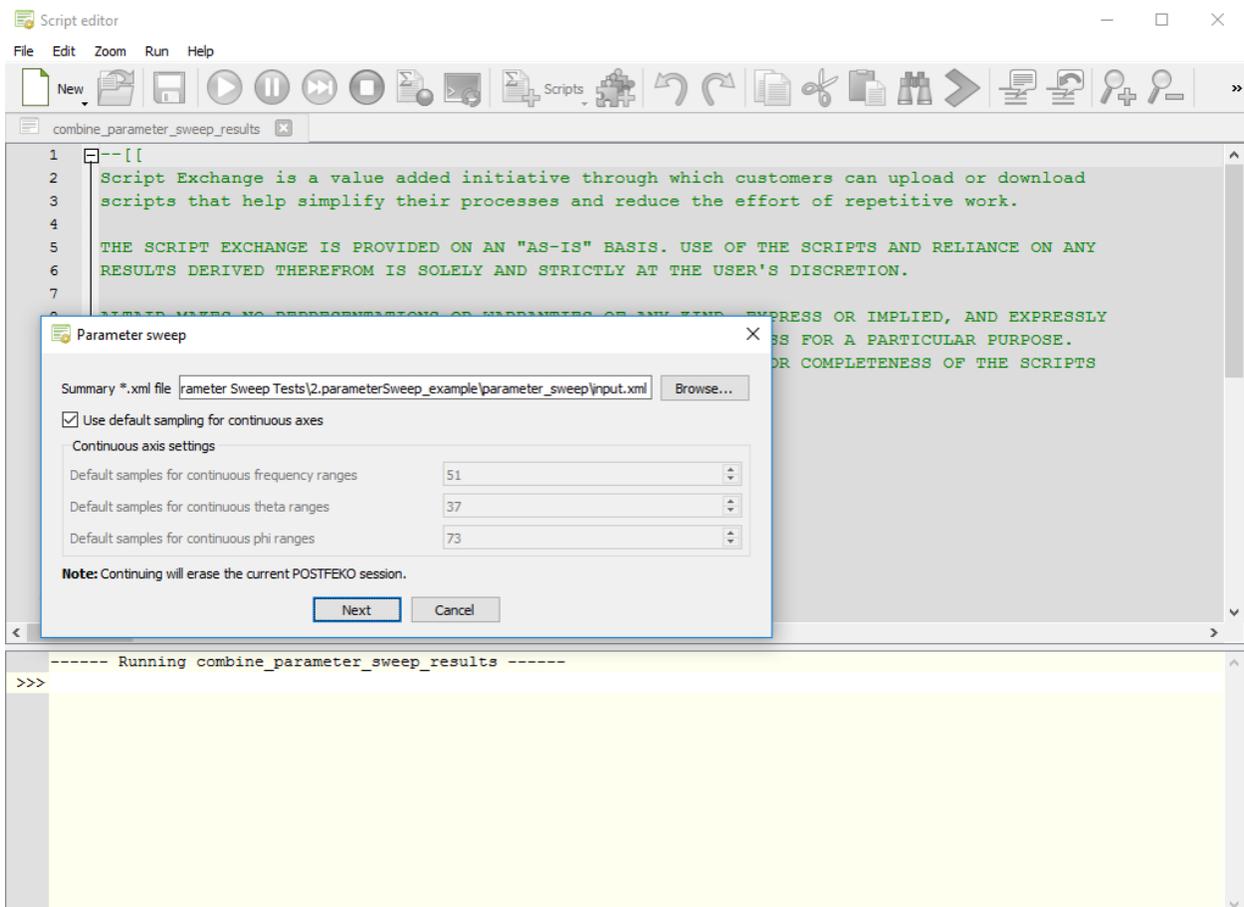
A \*.bat file will then open and run the generated FEKO files sequentially. When all the models have finished running, text will appear that asks if you'd like to merge the results in POSTFEKO. Type *y* or *Y* on the keyboard to choose this option.

```

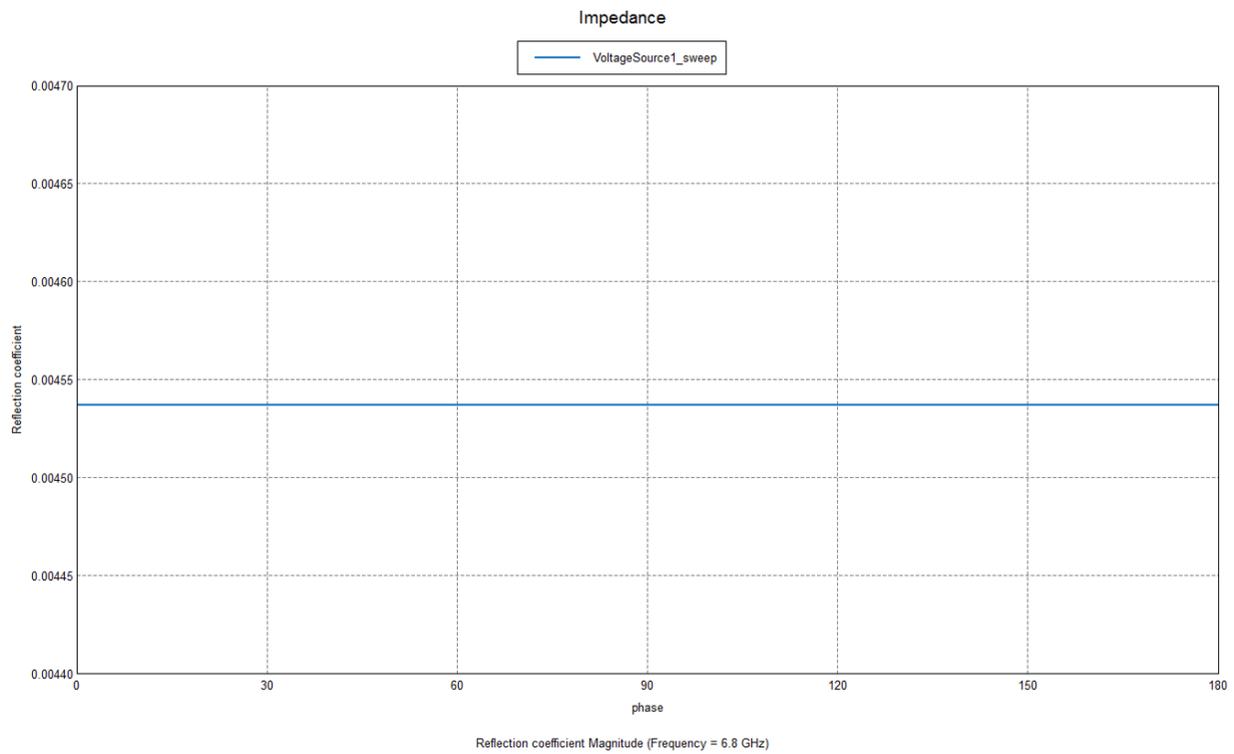
input.bat
See also message in the output file input_03.out
The following message from the master process (MYID= 0):
WARNING 1765: Wrong value of a singular integral, possibly observation point on a triangular surface
See also message in the output file input_03.out
The following message from the server process (MYID= 2):
WARNING 1765: Wrong value of a singular integral, possibly observation point on a triangular surface
See also message in the output file input_03.out
The following message from the master process (MYID= 0):
WARNING 1765: Wrong value of a singular integral, possibly observation point on a triangular surface
See also message in the output file input_03.out
The following message from the master process (MYID= 0):
WARNING 1765: Wrong value of a singular integral, possibly observation point on a triangular surface
See also message in the output file input_03.out
The following message from the master process (MYID= 0):
WARNING 1765: Wrong value of a singular integral, possibly observation point on a triangular surface
See also message in the output file input_03.out
The following message from the master process (MYID= 0):
WARNING 1765: Wrong value of a singular integral, possibly observation point on a triangular surface
See also message in the output file input_03.out
100%
Calculate the magnetic near field
(NearField1_half)
100%
Finished
      DONE: Parameter Sweep Solve
*****
* Launch POSTFEKO and merge the results.      *
*****
Do you want to run POSTFEKO now (y/n):

```

Another menu will appear asking which \*.xml file you want to use to merge the results of the sweep. This file is automatically generated by FEKO along with the sweep models and basically just tells POSTFEKO which files to merge. Luckily it's already chosen, so we just have to select *Next*.



And that's it! The results for the sweep are merged and you can view the results. Below I show the reflection coefficient as a function of phase.



## Parameter Sweeps on the HPC

Running a parameter sweep on the W&M HPC is very similar to running it locally. Just follow all the steps as before up to the *Output* pop-up menu, but now make sure *Run Feko Solver* is **NOT** selected. This will still create all of the sweep files and put them in your desired directory, but it won't automatically run the \*.bat file to solve the models.

## Submitting a Job

Next, transfer the sweep \*.cfx files (labeled by *feko\_model\_name\_{01,02,...}.cfx*) onto your W&M HPC account. To see how to use the HPC, see ShuangLi Du's guide on the lab website (bottom of the *Docs* page). You will also need the *runfeko\_sweep* script (and *starttime* and *endtime* files if you want to know when your job starts/ends). This script can be found in my Snapdragon folder: `\\Snapdragon\Common\William\HPC\runfeko files`. The *runfeko\_sweep* script is shown below:

```

1  #!/bin/tcsh
2  #PBS -N fekoName
3  #PBS -l nodes=1:vortex:ppn=12
4  #PBS -l walltime=15:00:00
5  #PBS -j oe
6
7  module load hyperworks/19
8  cd $PBS_O_WORKDIR
9  cat $PBS_NODEFILE > hostfile
10
11 setenv FEKO_TMPDIR /sciclone/home20/whmiyahira/scr20/temp
12
13 bash starttime
14
15 set totalSweeps=03
16
17 foreach numSweep (`seq -f "%02g" 1 $totalSweeps`)
18
19  cadfeko_batch input_${numSweep}.cfx >& LOG_${numSweep}
20  prefeko input_${numSweep}.pre >& LOG_${numSweep}
21  runfeko input_${numSweep}.cfx -np all --machines-file hostfile >>& LOG_${numSweep}
22
23 end
24
25 bash endtime

```

Annotations in the image:

- Red arrow pointing to line 3: **Name, # nodes, walltime**
- Red arrow pointing to line 11: **FEKO directory**
- Red circle around the path in line 11: **FEKO directory**
- Red arrow pointing to line 15: **# sweeps**

Here's what you will have to change:

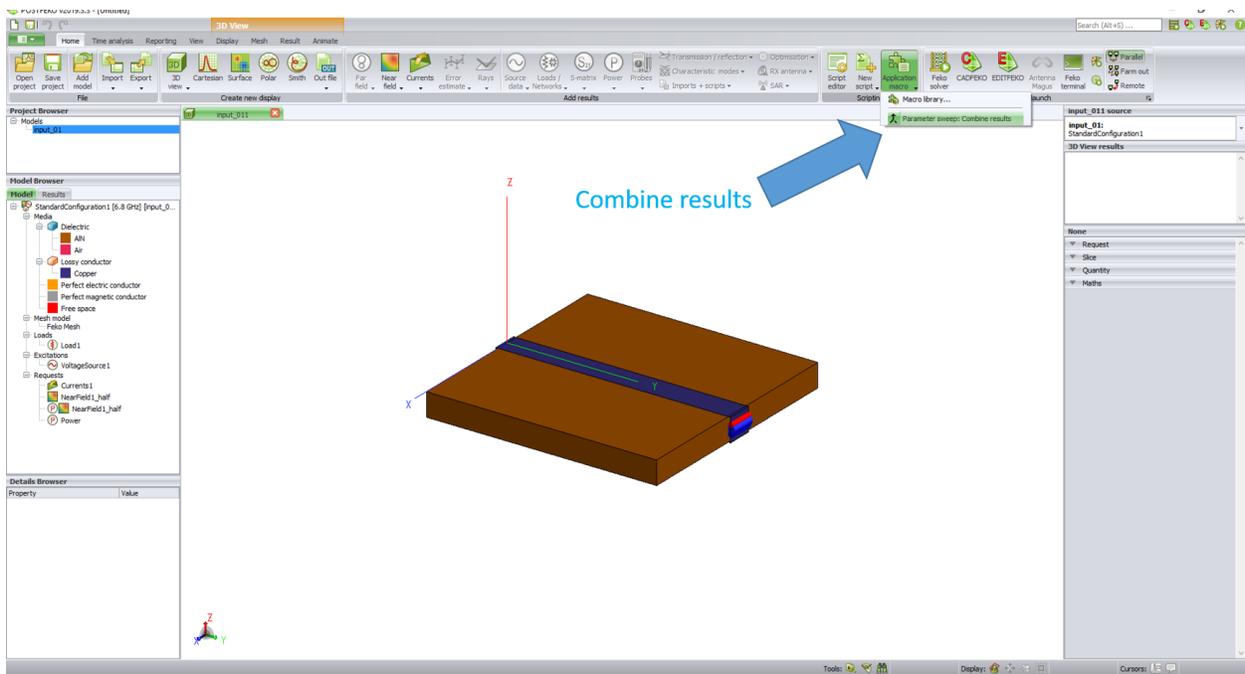
- The name (not necessary, but makes it easier to keep track of multiple jobs), number of nodes, and walltime of your job
- The FEKO\_TMPDIR. Change this to your hpc username and wherever you want to put the Temp folder. Here I made a folder called *temp* in my *scr20* directory.
- The number of sweeps. This should be equal to the number of sweep models you have. For the phase sweep example I set this number to 03.
- Additionally, don't forget to have the name of your file match the name in the script (i.e. replace *input* in lines 19-21 with your file name).

If you are also interested in how long it takes to run your job, you can include the *starttime* and *endtime* files found in the same Snapdragon folder. When your job has finished running it will output a text file showing the start and end times of your job. Note that you can also include these files for any other job you submit, just make sure you let the HPC know by including lines 13 and 25 in the above script. Also of note is that even if you have these lines in your script, there is no penalty for not including the corresponding *starttime* and *endtime* files in your HPC directory, you just won't get the run time information.

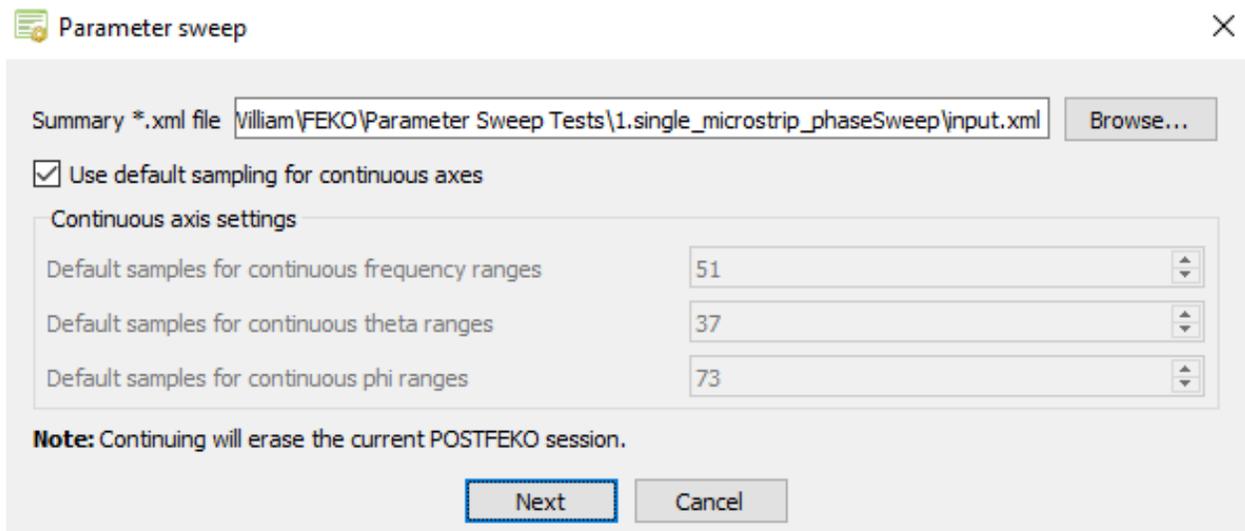
Now just submit this job to the HPC as you normally would (*qsub runfeko\_sweeps*) and transfer the files from your HPC account to your desired directory in Snapdragon (or wherever else).

## Merging Results in POSTFEKO

To merge results in POSTFEKO, open up one of the sweep files \*.fek files to open up POSTFEKO.



A pop-up menu will then ask you which \*.xml file you want to use to combine the results. Although it's already selected, for some reason it has trouble merging the files as is. So you just have to select *Browse...* and choose the \*.xml file manually. It will be in the same folder as the generated sweep files (should automatically take you to that folder).



And that's it! It will open up the merged POSTFEKO file and you can view your results as a function of the sweep parameter(s).

## Nodes and Walltime

Here I make a couple of notes about running parameter sweep jobs on the HPC.

Firstly, since the *runfeko\_sweep* file runs one FEKO file at a time, you only need to request the number of nodes needed for the **largest** sweep file. It's important to keep this in mind if you are doing a sweep of a variable that changes your model's geometry (such as the width of the trace in the example presented). This will change the number of mesh elements you have, which directly affects the memory required to run the job (memory  $\propto$  ( $\#$  mesh elements)<sup>2</sup>).

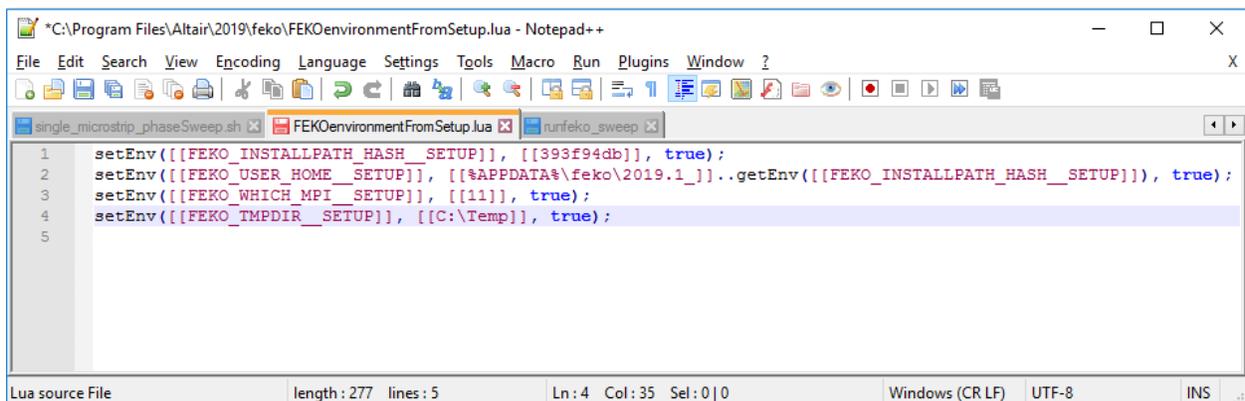
Secondly, keep in mind the maximum walltime for the HPC clusters, which can be found on the W&M HPC website. It may be good to run a single sweep file to get an idea for how

long each one will take to run if you are worried about hitting that maximum.

## Changing the FEKO Temporary Directory

In this section I go through the steps of changing the FEKO temporary directory (FEKO\_TMPDIR) on your computer. According to the 2019.3.3 FEKO user manual, “this variable specifies the directory where Feko will write paging files, when using the out-of-core solution.” When performing an out-of-core solution, FEKO will use the hard disk (specified by the location of the temporary directory) to store the matrices while the RAM solves chunks of the matrix at a time. Thus, one could accommodate a larger out-of-core solution by having the FEKO\_TMPDIR associated with a disk with more space. Here’s how to change it:

1. First, create a folder called *Temp* in the disk you want to use for the FEKO\_TMPDIR.
2. Next, locate the file *FEKOenvironmentFromSetup.lua*. This will be on the hard disk used to install FEKO, and should be located in the folder: C:\Program Files\Altair\2019\feko.
3. Open the file using a text editor such as *Notepad++* that allows you to edit the code.



```
*C:\Program Files\Altair\2019\feko\FEKOenvironmentFromSetup.lua - Notepad++
File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?
single_microstrip_phaseSweep.sh FEKOenvironmentFromSetup.lua runfeko_sweep
1 setEnv([[FEKO_INSTALLPATH_HASH_SETUP]], [[393f94db]], true);
2 setEnv([[FEKO_USER_HOME_SETUP]], [[%APPDATA%\feko\2019.1_]].getEnv([[FEKO_INSTALLPATH_HASH_SETUP]], true);
3 setEnv([[FEKO_WHICH_MPI_SETUP]], [[11]], true);
4 setEnv([[FEKO_TMPDIR_SETUP]], [[C:\Temp]], true);
5
Lua source File length: 277 lines: 5 Ln: 4 Col: 35 Sel: 0|0 Windows (CR LF) UTF-8 INS
```

4. On the last line you see where we choose to set the FEKO\_TMPDIR. All you need to do here is change the current disk (C in this case) to the new disk.
5. With the change made, save the file. It will as if you want to run *Notepad++* in administrator mode. Select “Yes” to save.
6. You might have to close and reopen FEKO to solidify the change. You can see if it’s been made by going into the Feko terminal and running “set”. Here I’ve changed it from C→E, as verified in the Feko terminal.

