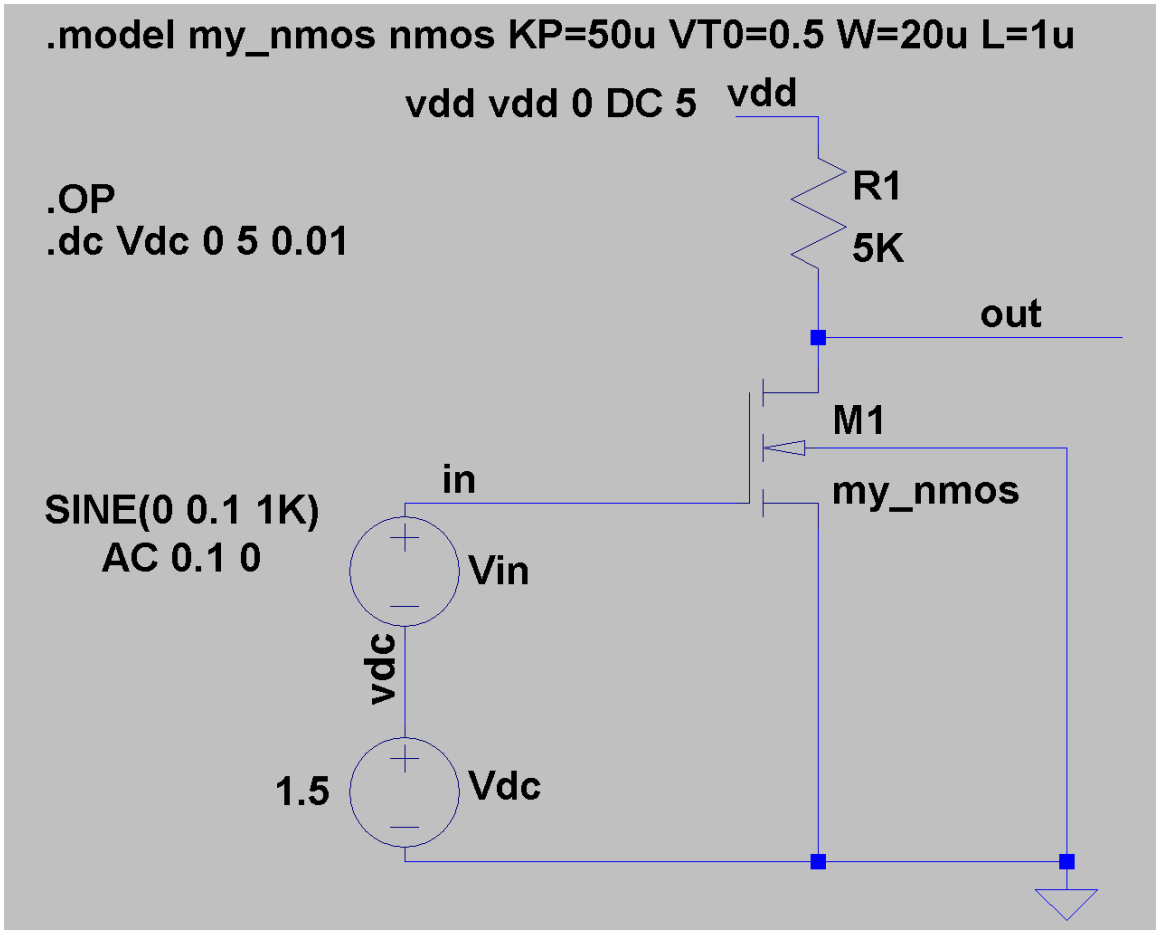
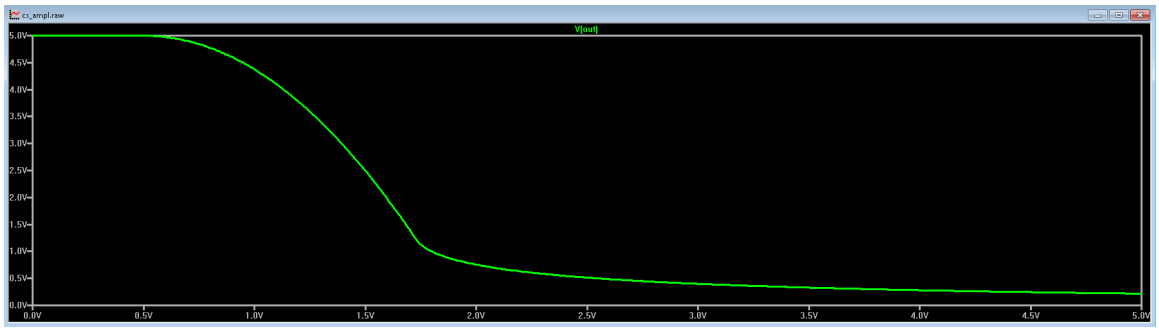
Example

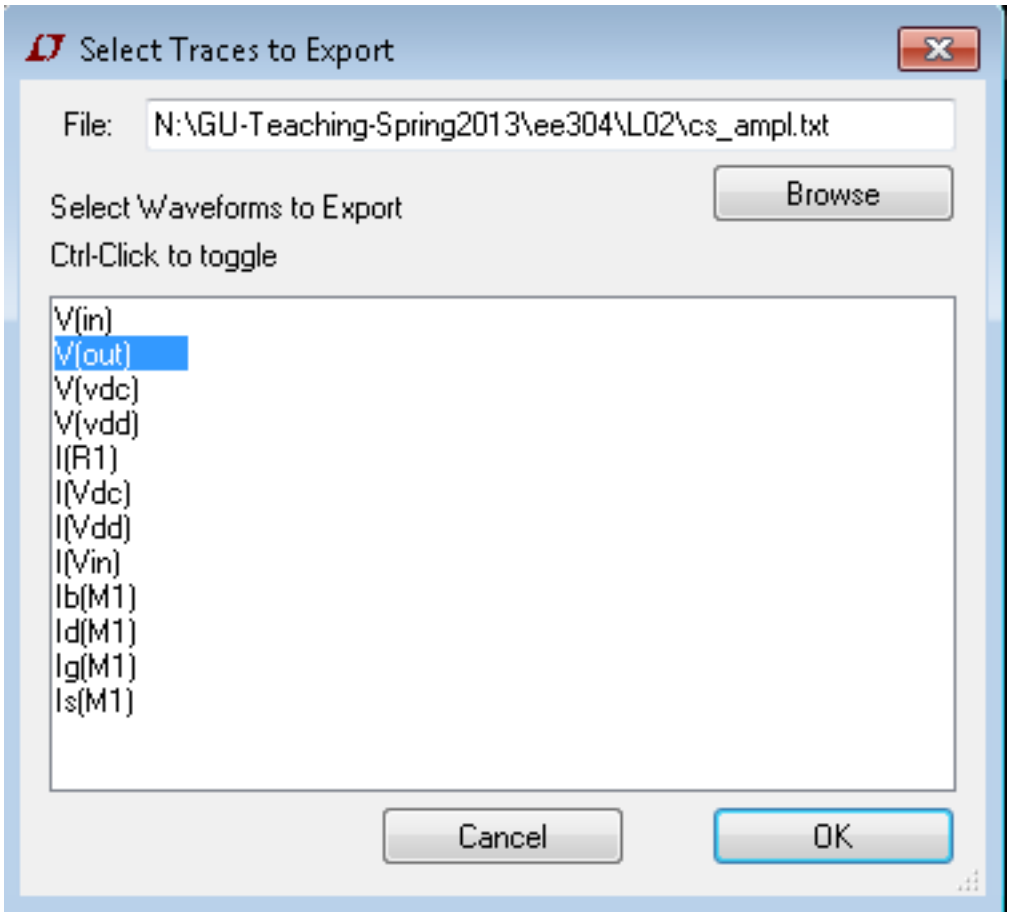
This example shows how to import the simulation results of a LTspice DC sweep in Matlab.



After running the simulation click once on the plot and select File → Export.

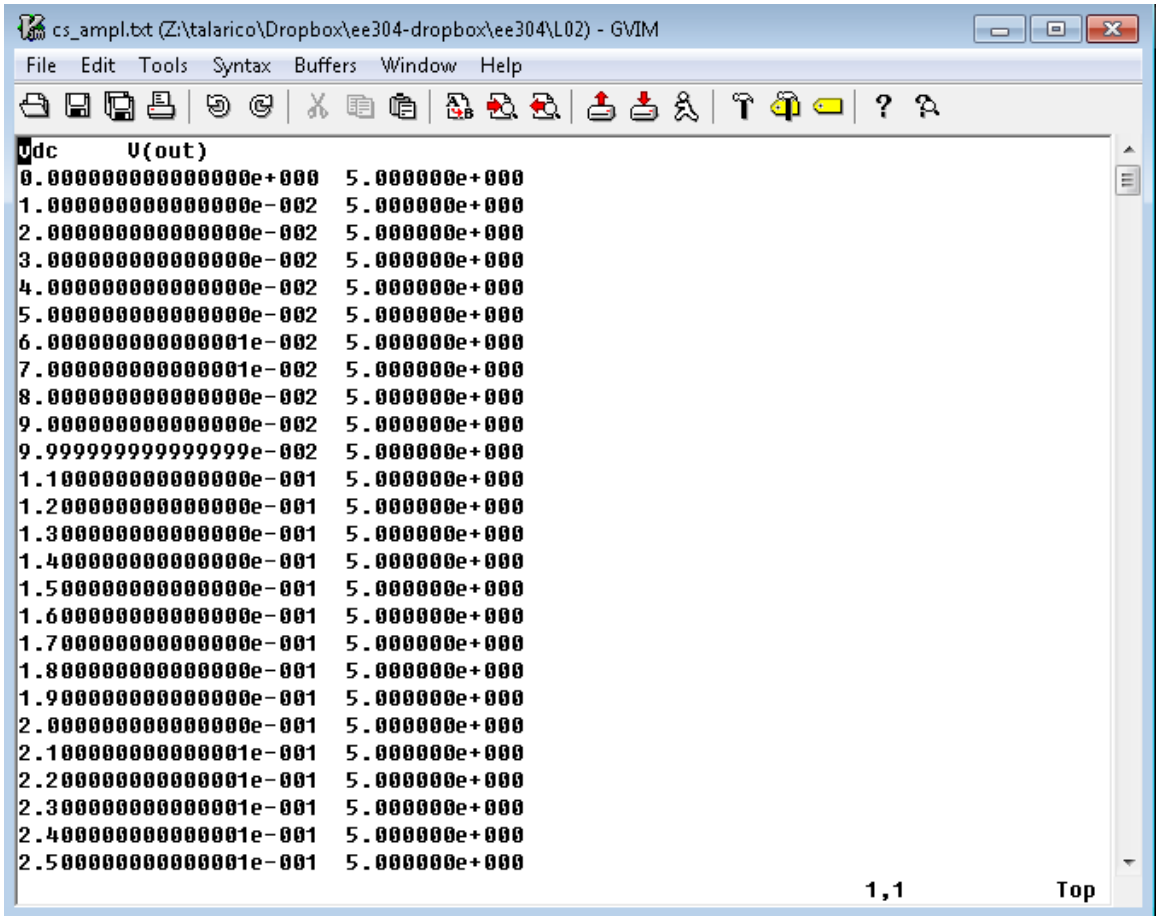


This will bring up the screen shown in the following Figure. Select the waveform(s) you want to export (e.g. V(out))



Select a path and file name and click OK to save the data as a text file.

The saved file (cs_ampl.txt) is a text file that you can open and examine using any text processor of your choice (e.g. vi or notepad). The file consists of some header information followed by columns of data. The following figure shows how the first few lines of the text file cs_ampl.txt looks like.



To read and post process the text file you can write a matlab script similar to the one provided next:

```
% C. Talarico
% cs_ampl_DCTF.m
% post process an LTspice DC sweep simulation with Matlab
clear all; clc; close all;

% dlmread: read ASCII-delimited file of numeric data into matrix
data = dlmread('cs_ampl.txt','',1,0);  % skip one row (headings)
% '' means let matlab figure out the delimiter
% 1,0 means reads data whose upper left corner is at row 1 and col 0

vin  = data(:,1);
vout = data(:,2);

%plot the DC tranfer function
plot(vin,vout,'linewidth',2);
xlabel('vin [V]','fontsize',14);
ylabel('vout [V]','fontsize',14);
title('DC Transfer Function','fontsize',16);
maxvout = max(vout) + 0.1;
axis([0 max(vin) 0 maxvout]);
```

The following figure shows the result:

