Example 7 – Plotting LtSpice Simulations in Matlab

In LtSpice, after doing a simulation you can save the data which was used to generate the simulation and import it into MATLAB for further manipulation. Matlab has a `loadfile` command that allows such files (with minor modification) to be read into Matlab and operated on using Matlab's functions.

There are two advantages to reading LtSpice files into Matlab. First Matlab creates "prettier" reports and graphs that are easier to integrate into Word. Second Matlab can do many functions that can't be done easily in LtSpice and this gives you an opportunity to enrich you simulation.

In this example, we will do a simulation in LtSpice, save the file, modify it for reading into Matlab, and finally read it into Matlab to make a plot.

We begin by constructing the circuit in Figure E7-1 using LtSpice. The simulation looks at Vin and Vout

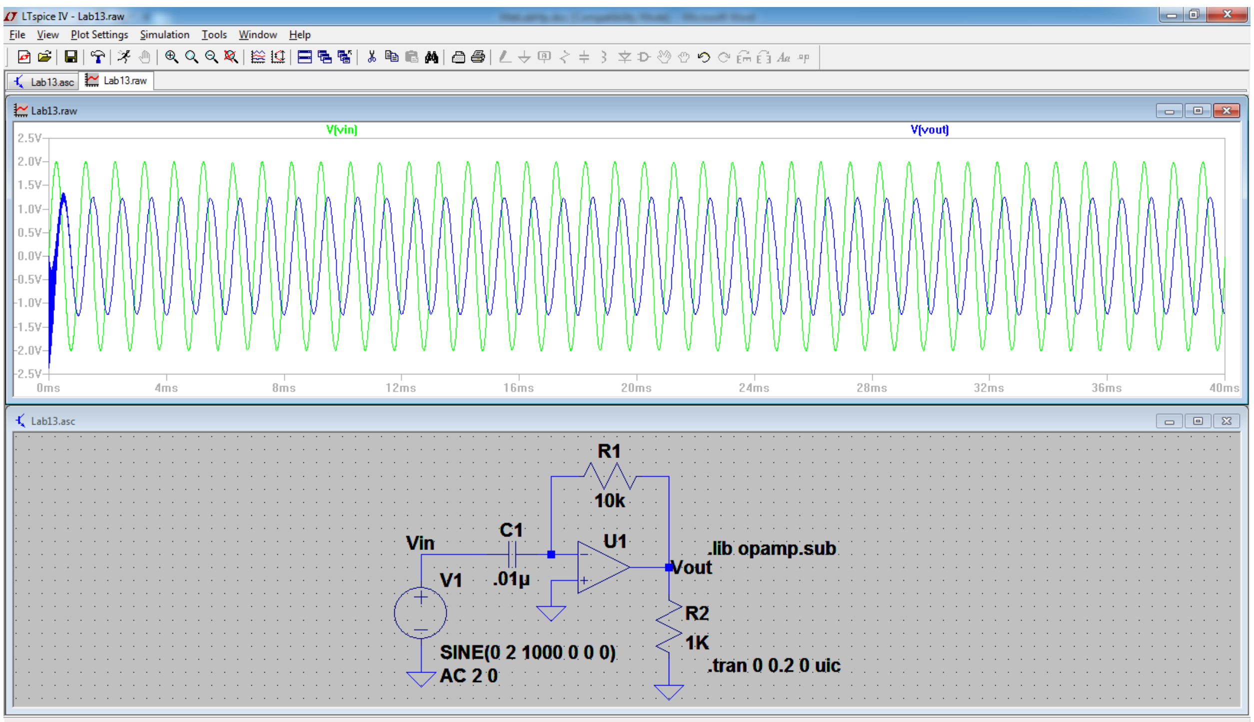

Figure E7-1

Circuit for simulation in Workbench.

After the simulation runs click once on the plot and select File → Export. This will bring up a screen shown in Figure E7-2. Select a path and file name and click OK to save the data as a text file. You can exit LtSpice

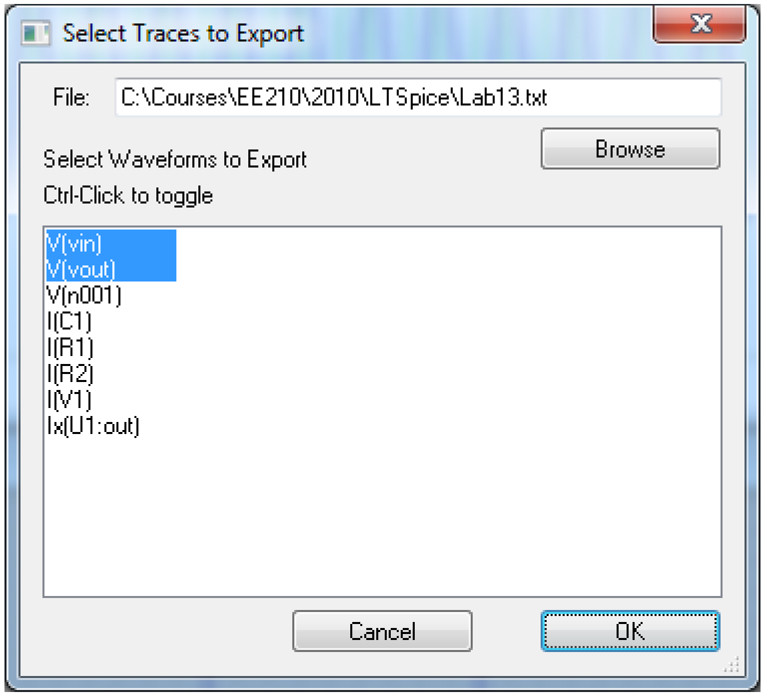

Figure E7-2

The File→Export menu from LtSpice

The saved file is a simple text file that you can open and examine using Notepad or Word. The file consists of some header information followed by three columns of data giving the time base and the corresponding voltage levels for the two nodes. Figure E7-3 shows the beginning and end of this file.

time	V(n002)	V(n003)
0.000000000000000e+000	0.000000e+000	0.000000e+000
1.896585271020485e-009	-1.779854e-006	2.383319e-005
3.793170542040970e-009	-7.007942e-006	4.766639e-005
5.689755813061455e-009	-1.568426e-005	7.149958e-005
7.586341084081940e-009	-2.780881e-005	9.533277e-005
9.482926355102426e-009	-4.338160e-005	1.191660e-004
1.137951162612291e-008	-6.240262e-005	1.429992e-004
1.327609689714339e-008	-8.487187e-005	1.668324e-004
2.556419391731497e-008	-3.034966e-004	3.212491e-004
4.418355913658504e-008	-8.520387e-004	5.552270e-004
	…	
1.998464199406124e-001	-7.143645e-001	-1.640545e+000
1.998609112380335e-001	-8.052113e-001	-1.529819e+000
1.998787348420469e-001	-9.070134e-001	-1.375973e+000
1.998980380558890e-001	-1.004364e+000	-1.190083e+000
1.999218366406368e-001	-1.103188e+000	-9.370503e-001
1.999502564861356e-001	-1.186708e+000	-6.086052e-001

1.9999031893230866e-001	-1.250910e+000	-1.206329e-001
2.0000000000000000e-001	-1.249292e+000	1.571833e-014

Figure E7-3

The exported data file opened using Notepad.

Matlab can read text files using the load 'filename' command. However, Matlab expects these files to contain only the data and not any header information. If we try to read the exported text file with Matlab's load command we will generate an error. To correct this we modify the file by deleting all of the header information from the top of the file down to the first line of data. The first line of the file should read
0.0000000000000000e+000 0.000000e+000 0.000000e+000

Save this amended file with a new name such as Ex7.txt. Ex7.txt can now be read into Matlab and stored as a vector having one row for each row in the file and three columns. Figure E7-4 shows an m-file that reads Ex7.txt into a variable called "scope". We then use the colon operator to separate the scope vector into two new vectors called ChA and ChB. ChA contains the time base in column 1 and the channel A scope data in column 2. ChB contains the time base in column 1 and the channel B scope data in column 2. Both can then be plotted. Figure E7-4 shows the results of the plot.

```
%Example7.m
scope = load('Ex7.txt');
ChA = [scope(:,1),scope(:,2)];
ChB = [scope(:,1),scope(:,3)];
figure(1);clf;
subplot(2,1,1);
plot(ChA(:,1),ChA(:,2));
axis([0 .02 -1.2 1.2]);
subplot(2,1,2);
plot(ChB(:,1),ChB(:,2));
axis([0 .02 -2.2 2.2]);
```

Figure E7-3

This m-file loads a text file named 'Ex7.txt' into a variable called scope. The variable is then separated into two variables called ChA and ChB and the data for each channel is plotted. Figure E7-4 shows the results of the plot.

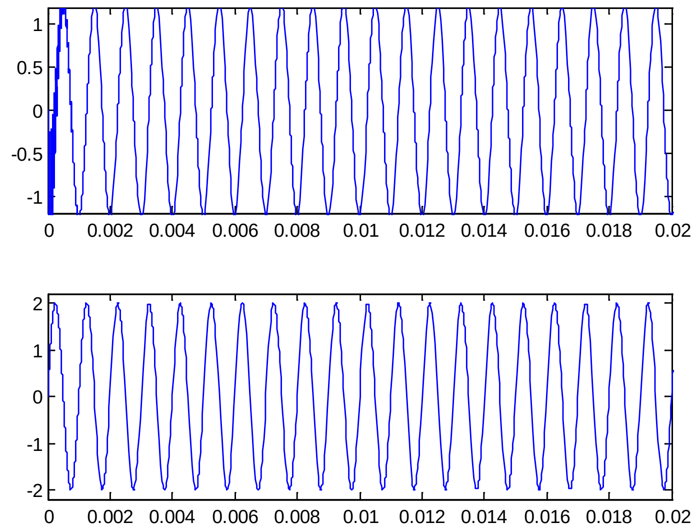

Figure E7-4

The Matlab plotted results from the LtSpice simulation.