

# EEEE2055: Modelling Methods and Tools - Coursework1 (Fourier Transforms) 22-23

P Evans

November 14, 2022

## 1 Overview

We have covered the theory of Fast Fourier Transforms in lectures but unlike the Fourier Transform, we have not looked at the calculations in detail. This is because in reality, the Fast Fourier Transform is a tool that is built in to many engineering software packages – most of you are unlikely to ever need to program the calculations behind this tool but you will need to be able to use it. The purpose of this coursework is to learn how to use the Fast Fourier Transform tool in the LTSPICE circuit simulation tool that you have already used in other modules. You'll compare the output with analytical methods (Fourier Transform) and try to explain any differences.

## 2 Background

The following section contains some information about how to use the FFT tool in LTSpice and an explanation of how you need to display and analyse the waveforms that you produce. You might already be familiar with much of this through the electronics project but you can use this as a reference if needed. **You don't need to include any of the following exercises in the report, they are given for guidance only**

### 2.1 LTSpice

#### 2.1.1 Adding a voltage/current source

You will need to create voltage sources to generate the signals for analysis. In LTSpice you do this by adding a new component using the toolbar at the top of the window (icon looks like an AND gate), then selecting the *voltage* type. Double click on the voltage source that you have created and configure it to generate a signal of the type required (*DC*, *sine*, *pulsed*, etc), you'll need to select the advanced option for sources other than *DC*. We will be dealing with pulsed sources and so an explanation of the pulse source parameters is given below.

- **Vinitial** - the initial voltage of the source, typically the *low* voltage.
- **Von** - the voltage of the source during a pulse, typically the *high* voltage.
- **Tdelay** - an initial delay before the pulses start being produced, the source is at Vinitial during this time. Set this too zero for a periodic waveform.
- **Trise** - the rise time of the pulses. Note that for an ideal pulsed source this would be zero, but it is impossible for an analogue circuit simulator such as LTSpice to create perfect square pulses. To approximate an ideal pulse you set this to a small time *relative to the pulse width*, e.g. 0.1% of the pulse width. Even if you set this value to zero, it will not actually be zero - the software will automatically choose a sensible value so it is best to specify the value yourself, that way you know what it is!
- **Tfall** - the fall time of the pulses, see comments for Trise.
- **Tpulse** - the pulse width.
- **Tperiod** - the period.

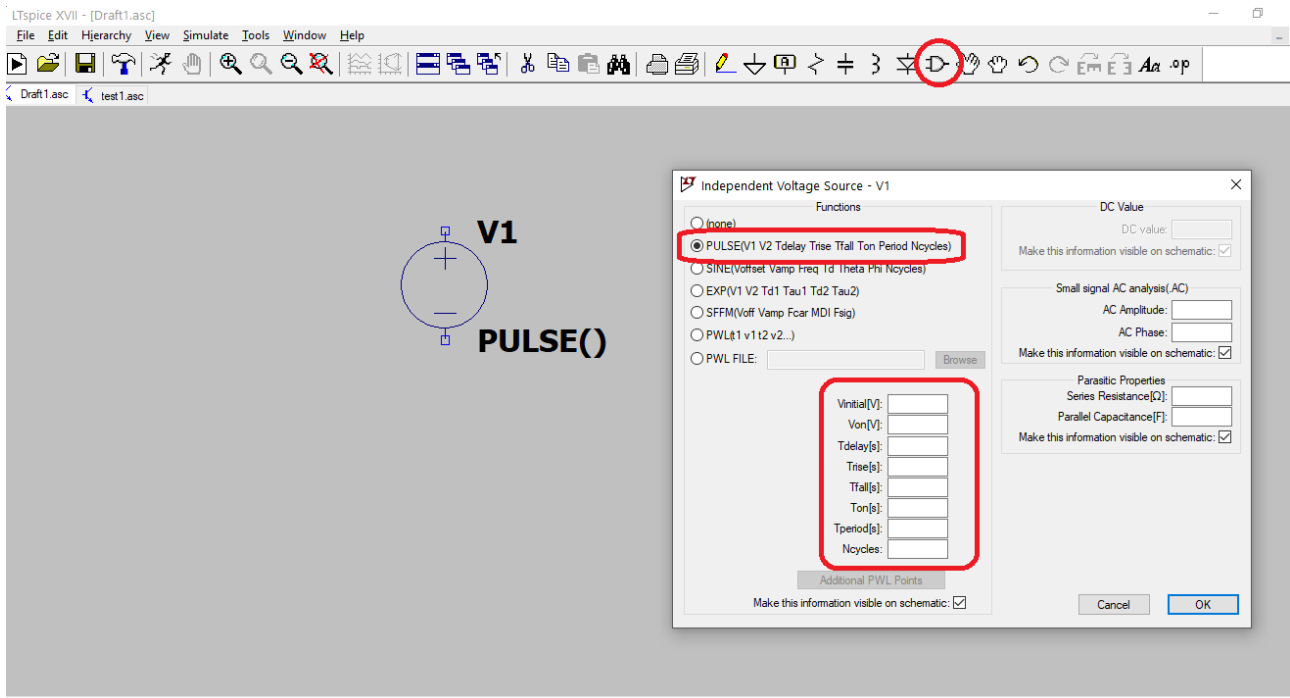


Figure 1: Creating a pulsed voltage source in LTSpice

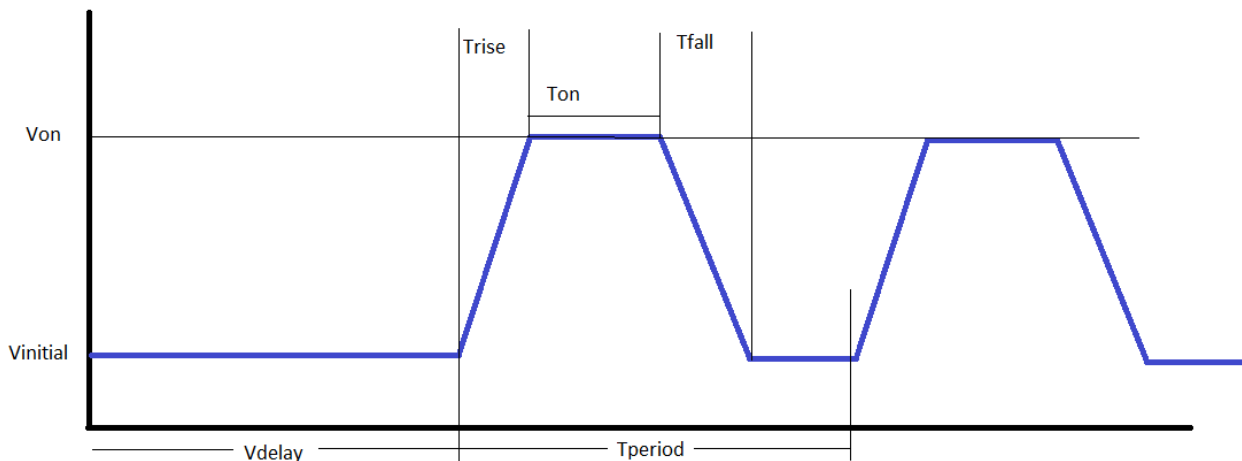


Figure 2: Configuring a pulsed voltage source in LTSpice

- **Ncycles** - the number of cycles. Leave this box empty for a periodic waveform, set to 1 for a single pulse, etc.

As an initial test, setup a 1Hz, -1V - 1V 50% duty cycle periodic square wave with appropriate rise and fall times.

### 2.1.2 Completing the simulation model

Its a good idea (although not essential) to add a resistor across the voltage source. If you include the resistor, make sure you click on the resistor and give it a value otherwise you will get an error.

You will also need to add a ground reference to the model to allow it to run. Without a ground reference the equations that are generated for the circuit model will be *under determined* - there is no unique solution because the voltage source only specifies the voltage difference between two wires and not the absolute voltage at either.

When the voltage source has a value of 1V, the circuit voltages could be 0V and 1V, 1V and 2V, 1.325V and 2.325V, etc. A ground reference defines one of the wires as having a potential of 0V and makes the equations solvable. Note that the PLECS software (that you will use in the energy project) automatically does this for you.

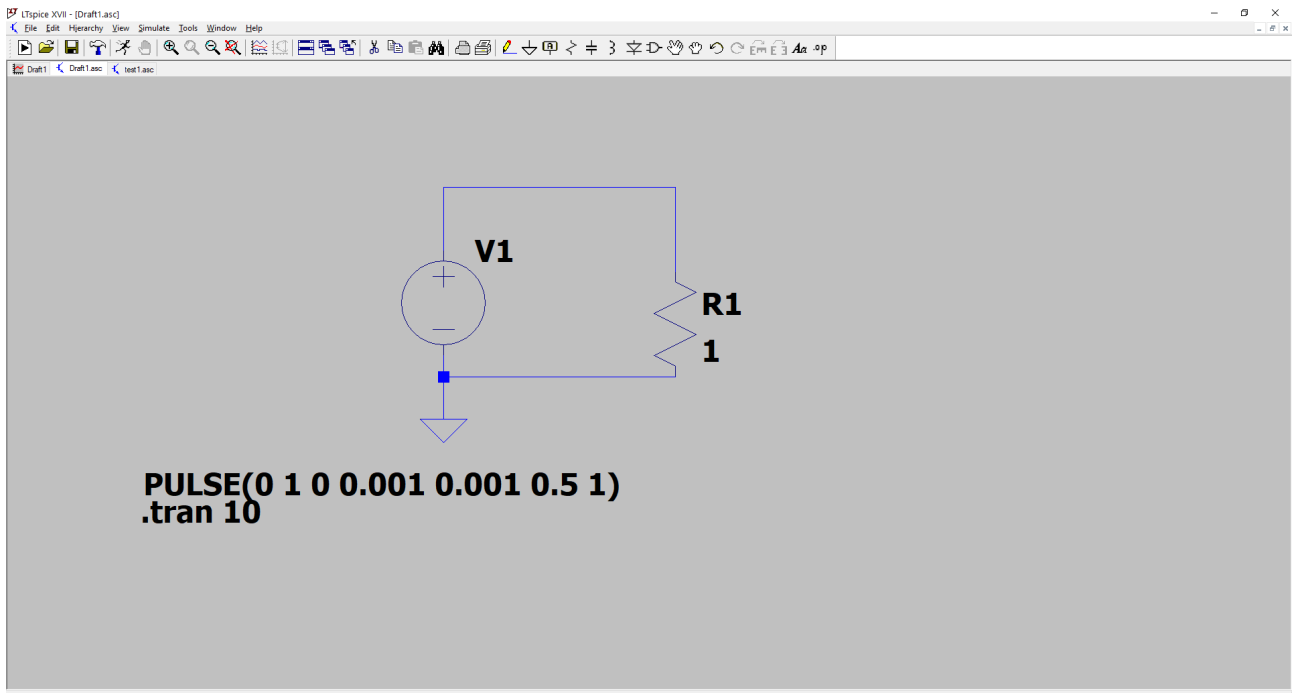


Figure 3: Completed simulation model

### 2.1.3 Running the simulation

To run the simulation, click on the run icon in the toolbar. The first time you do this you will get a dialog asking you to configure the simulation. The only parameter you need to enter for now is the *Stop time* parameter. Note that by default you are running a transient simulation (i.e. a *time-domain* simulation).

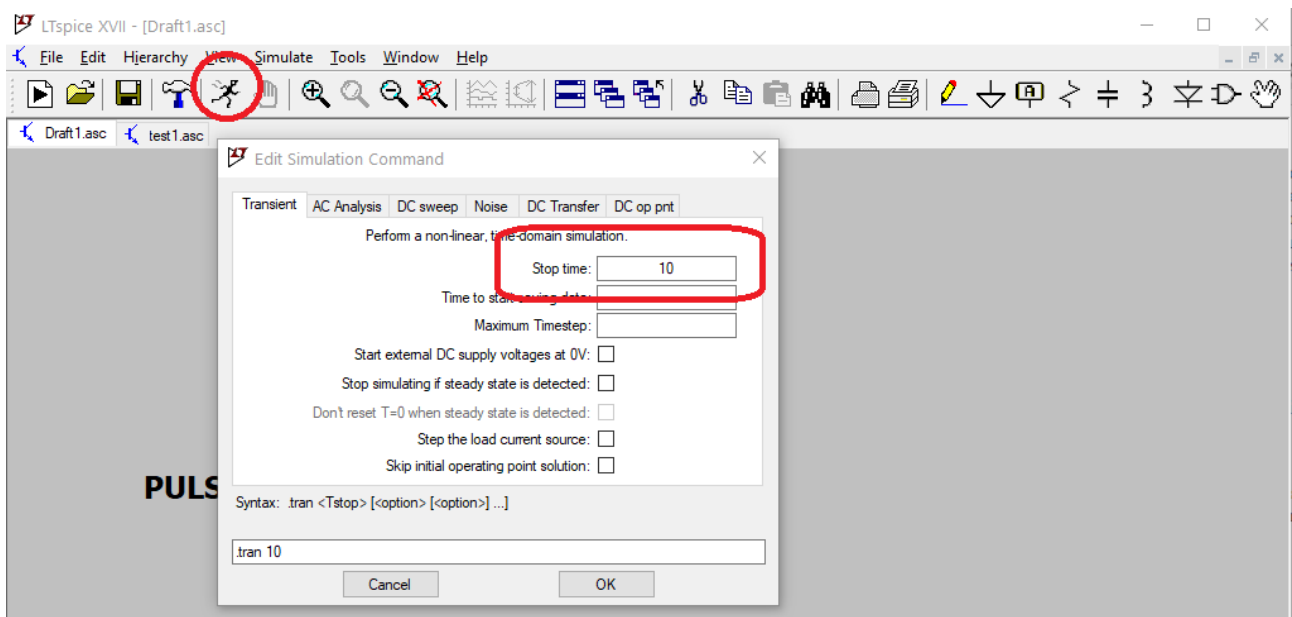


Figure 4: Configuring and running the simulation in LTSpice

### 2.1.4 Displaying the time-domain results

Once the simulation has completed you will have two display tabs to choose from, the circuit model (*XXXX.asc*) and the waveform viewer. If you hover the mouse pointer over the circuit model you should see two different probe types appear - a voltage probe if you hover over wires and a current probe if you hover over components. Click on a the wire at the top of the voltage source to show the voltage waveform in the waveform viewer. Check that the waveform is as expected.

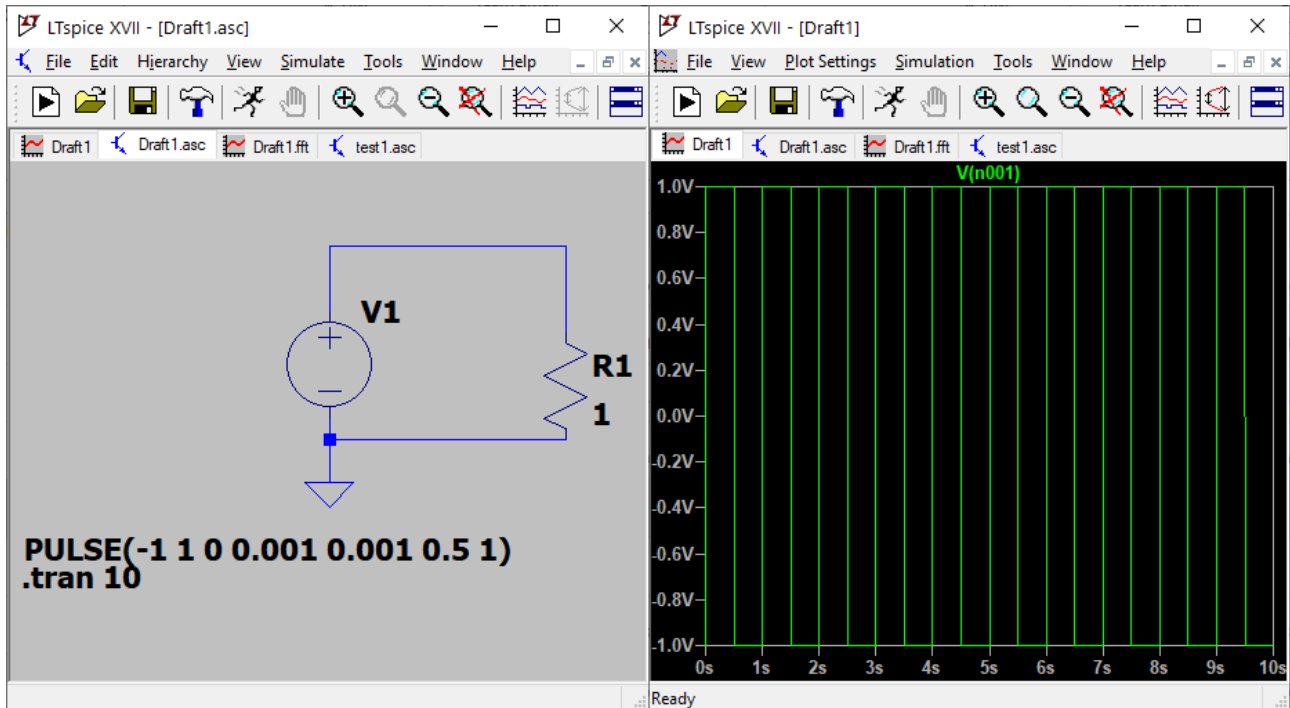


Figure 5: Displaying time-domain results in LTSpice

You can export the waveforms a text data (e.g. for importing into excel to compare with other waveforms) by selecting *File*→*Export data as text*.

### 2.1.5 Performing a Fast Fourier Transform

When viewing the waveform viewer tab you can create an FFT by selecting *View*→*FFT*. You don't need to change any of the options for now, just select the waveform that you want to create an FFT for.

By default, LTSpice will use a log-log scale for the FFT. This is usually the most appropriate way of viewing a real FFT but be aware that we used linear scales in lectures for Fourier Transforms as it makes it easier to understand the relationship between the analytical expression we calculated and the plot. You can change the scales to linear by right clicking on them and un-checking the *Logarithmic* option. You'll need to do this for both horizontal and vertical scales separately. You will also probably need to change the axis range for one/both of the horizontal and vertical axes to zoom in on the region of interest, the reason for using log-log scales is that it allows a much larger range of values to be displayed on a single plot when compared with linear scales. **Note for report, Log-Log scales are usually the most appropriate way of displaying Fourier Transforms. Linear scales tend to obscure some details and make analysis more difficult**

You should notice that the linear-linear scale FFT of this pulsed waveform now looks similar to what you would expect for a periodic square wave (Fourier Series lecture material). We will check the result in the following section.

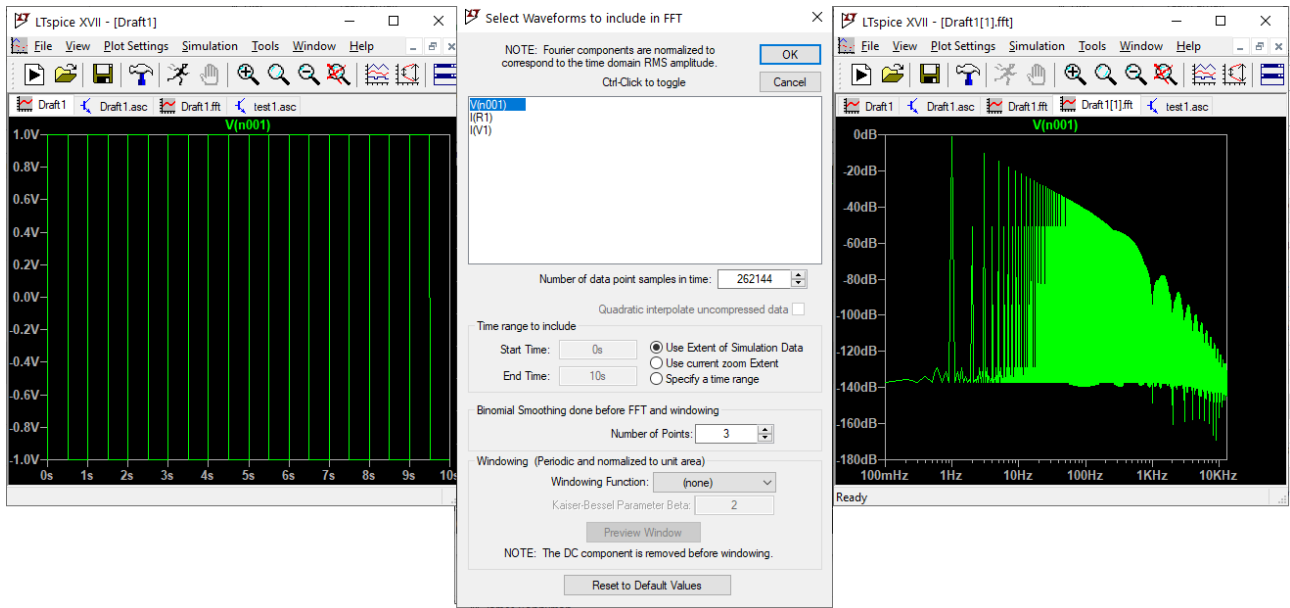


Figure 6: Creating an FFT plot in LTSpice

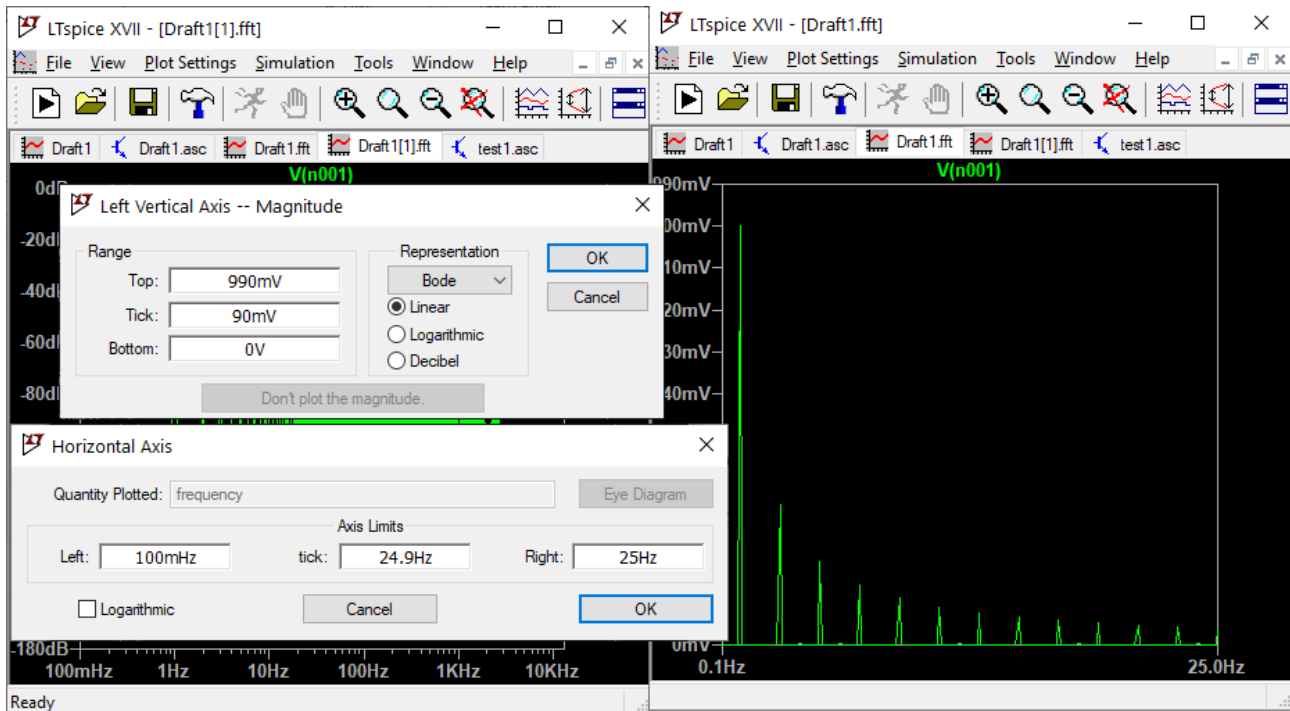


Figure 7: Creating a linear FFT plot in LTSpice

## 2.2 Using Microsoft Excel to produce Fourier Series / Fourier Transform Plots

### 2.2.1 Computing and plotting the Fourier Series representation of a pulsed waveform

We will now compute the Fourier Series representation of the square wave from the previous section. You should recognise that it is an odd waveform and so only the  $B_n$  coefficients need to be computed. We need to integrate over one period:

$$\begin{aligned}
B_n &= \frac{2}{T} \int_0^T S(t) \sin(2\pi n f t) dt \\
&= \frac{2}{T} \int_0^{0.5} \sin(2\pi n f t) dt - \frac{2}{T} \int_{0.5}^1 \sin(2\pi n f t) dt \\
&= \frac{2}{T} \left[ \frac{\cos(2\pi n f t)}{2\pi n f} \right]_{0.5}^1 - \frac{2}{T} \left[ \frac{\cos(2\pi n f t)}{2\pi n f} \right]_0^{0.5} \\
&= \frac{2}{T} \frac{\cos(2\pi n f) - 2\cos(\pi n f) + 1}{2\pi n f}
\end{aligned} \tag{1}$$

We know that  $f = 1$  for our 1Hz square wave and also that  $\cos(2\pi n f) = 1$  for all  $n$  and so we can simplify this:

$$B_n = 2 \frac{1 - \cos(\pi n)}{\pi n} \tag{2}$$

This gives us an equation for computing any  $B_n$  coefficient and we know that all  $A_n$  coefficients are zero for this odd waveform. We can use this information to create a table of coefficients vs frequency in Excel. We want columns for  $n$ ,  $f = n f_0$ ,  $A_n$  and  $B_n$ . For the general case, where both  $A_n$  and  $B_n$  might be non-zero, it is easier to plot the combined magnitude of the coefficients at each frequency  $|C_n|$ , rather than the individual  $A_n$  and  $B_n$  components. Remember from the notes on complex Fourier Series coefficients that  $|C_n| = \sqrt{A_n^2 + B_n^2}$  so we'll also create a column for this. LTSpice will also plot the magnitude of the Fourier Transform and so this allows for easy comparison.

	A	B	C	D	E	F	G
4							
5	n	f	An	Bn	Magnitude	Phase	
6	0	0	0	0	0		
7	1	1	0	=2*(1-COS(PI()*A7))/(PI()*A7)	1.27324	1.27323954	
8	2	2	0	0	0		
9	3	3	0	0.424413	0.42441318		
10	4	4	0	0	0		
11	5	5	0	0.254648	0.25464791		
12	6	6	0	0	0		
13	7	7	0	0.181891	0.18189136		

	A	B	C	D	E	F	G
4							
5	n	f	An	Bn	Magnitude	Phase	
6	0	0	0	0	0		
7	1	1	0	1.27324	1.27323954		
8	2	2	0	0	0		
9	3	3	0	0.424413	=SQRT(C9^2+D9^2)		
10	4	4	0	0	SQRT(number)		
11	5	5	0	0.254648	0.25464791		
12	6	6	0	0	0		
13	7	7	0	0.181891	0.18189136		

Figure 8: Creating a table of Fourier Series Coefficients in Excel (also see Excel Spreadsheet provided)

Finally create a plot of the coefficients. Because we only want to plot the magnitude at discrete points ( $n f$ ) for Fourier Series, it is best to use something like a bar chart with discrete columns rather than a continuous curve. We now need to compare the plots from theory and from LTSpice. You can hover the mouse pointer over the LTSpice chart to get a measurement of the location and amplitude of each spike (or click on the waveform label to enable the cursors) and compare these to Excel. LTSpice gives spikes of approximately: 0.895V@1Hz, 0.301V@3Hz, 0.180V@5Hz, ... The Excel spreadsheet gives values of: 1.273@1Hz, 0.424@3Hz, 0.254@5Hz, ... At first glance these appear to be different results but if you look at the ratios of the amplitudes:  $1.273/0.895 = 1.42$ ,  $0.424/0.301 = 1.41$ ,  $0.254/0.180 = 1.41$ . This factor of approximately 1.41 is in fact  $\sqrt{2}$  - LTSpice is displaying the RMS amplitude rather than the peak. With any frequency domain simulation results you need to be careful when looking at results in V or I units as it isn't always obvious if RMS or peak values are being used.

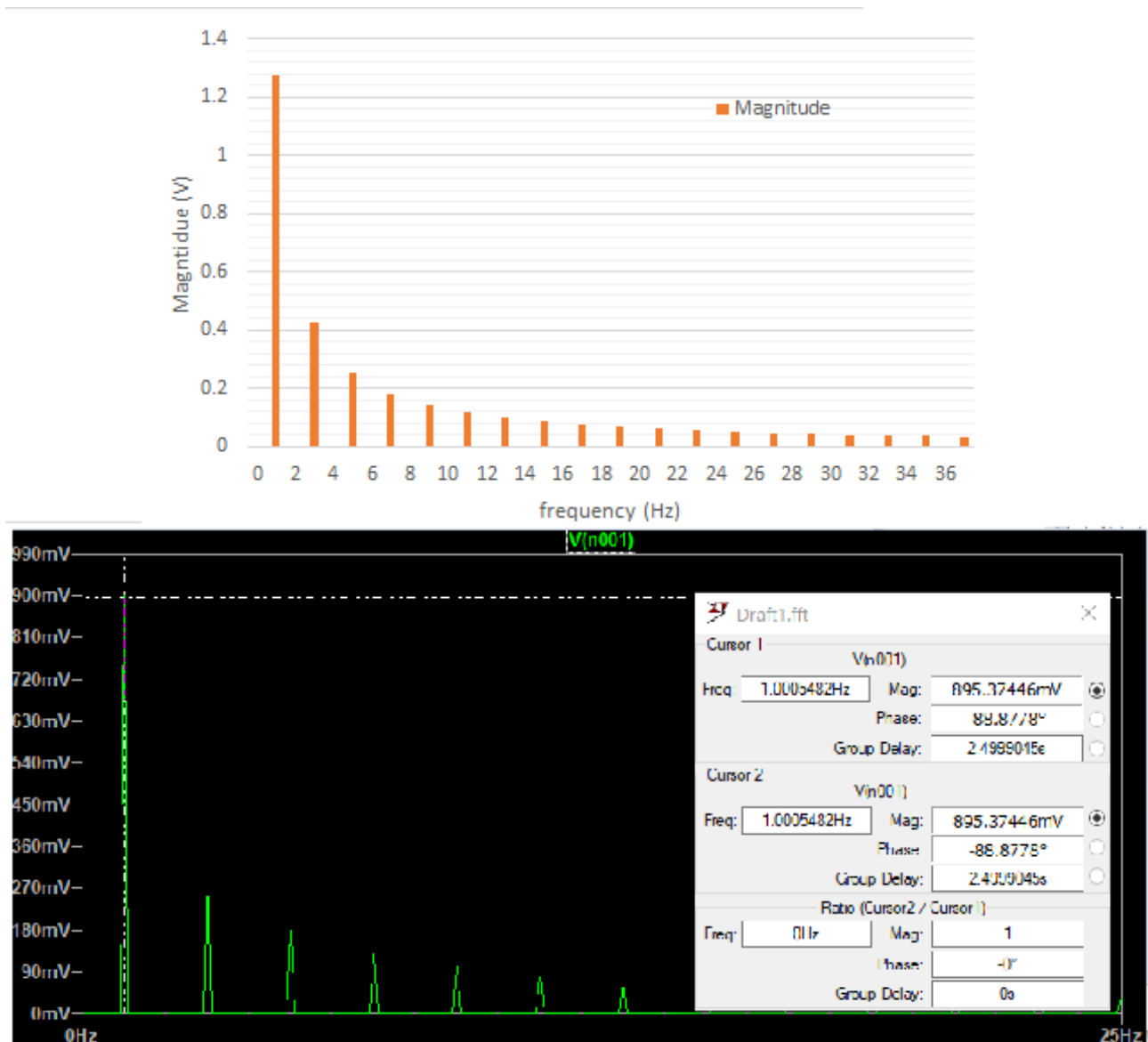


Figure 9: Creating a plot of Fourier Series Coefficients in Excel (also see Excel Spreadsheet provided)

### 2.2.2 Computing and plotting the Fourier Transform of a pulsed waveform

A second example is included in the template spreadsheet that illustrates how to plot a Fourier Transform.

The example in the spreadsheet is a plot of the FT2 example from lectures. The lecture calculations tell us

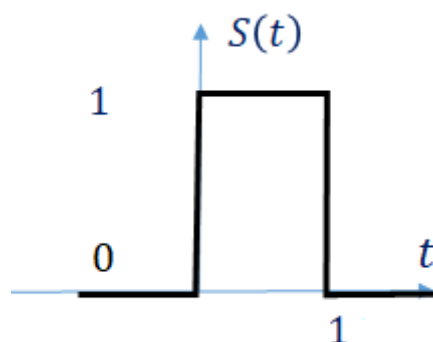


Figure 10: Time domain plot signal for lecture example FT

that the Fourier Transform of a single pulse of width 1, amplitude 1 occurring at  $t = 0$  is:

$$S(\omega) = \frac{\sin(\omega)}{\omega} + j \frac{\cos(\omega) - 1}{\omega} \quad (3)$$

The real part of this equation is equivalent to the  $A_n$  coefficient and the imaginary equivalent to the  $B_n$  coefficient and so creating a magnitude plot is similar. The main difference is the frequency domain plot is now produced for all frequencies, not just  $nf_0$ . You must decide how many points you need to plot and over what range and it is also better to use a *scatter* or *x-y* plot for the continuous function. The steps for plotting calculating and plotting this in Excel are the same as above. You can look at the spreadsheet provided for an example of how this can be plotted.

You can try the following as an additional practice activity if you like. Again, there are no marks for this but you can use the data provided in the Excel spreadsheet to check you can get results from LTSpice that agree with the theory. Also see below for some notes on exporting and interpreting FFT data from LTSpice.

- Create an LTSpice simulation for the FT2 pulse, simulate and extract the FFT.
- Export the results to Excel and compare with the data given in the *FourierTransformEx - Bode* spreadsheet.

### 2.2.3 A few final points on plotting Fourier Transforms produced by LTSpice

In the following tasks, you will need to compare Fourier Transforms that you calculate with Fourier Transforms that you obtain from LTSpice. The following two points will help you do this:

- As with the Fourier Series example, LTSpice can apply unexpected scaling to the Fourier Transform it produces. The *factor of  $\sqrt{2}$*  still applied but LTSpice also normalises the plots using the RMS value of the time-domain waveform. This normalisation has little effect for periodic waveforms but it does affect the scaling of non-periodic waveforms (how much time you add after your pulse affects the RMS value of the overall waveform and therefore the FFT scaling). For this reason, it is best to compare the normalised magnitudes (i.e. divide through by the maximum so your plots have a maximum value of 1). The Fourier Transform example in the spreadsheet also shows normalised plots. If comparing normalised values, make sure the range of frequencies over which the normalisation is performed is the same for both plots (produce plot data from your calculations to match the frequency range produced by LTSpice)!
- A time-domain (or transient) simulation produces time-sampled data (think back to the Discrete Fourier Transform notes - a set of measurements, with a certain sample period). These samples are the input data to the FFT. The transient simulation parameters in the *Edit Simulation Command* menu affect the samples that are generated - *Stop Time* sets the total time range for which data is recorded, and *Maximum Timestep* sets the maximum sample period (the sample period may be less than this, if LTSpice decides the value you have set is too large). There is also an option in the *View FFT* menu that controls how many time samples are actually used for the FFT (*Number of data point samples in time*)
- It is useful to be able to plot the LTSpice Fourier Transform in Excel (so you can directly compare it with your own calculation, for example). LTSpice allows you to export the FFT data (same process as for time-domain results) but it isn't in a helpful format:
  - You can get too many data points to easily deal with in Excel - you may need many time-steps (small sample period, long simulation time) to get an accurate FFT. This will produce many time samples, and therefore many frequency samples. 10,000-100,000 samples is not uncommon.
  - The data file format isn't the easiest to import into Excel.

There is a program on Moodle, *ltfftconvert.exe*, that you can use to convert LTSpice's default output to something that is easier to plot in Excel. To export data in LTSpice, go to *File*→*Export data as text* and select the (one) FFT waveform you want to export. Export using the default *Polar* format option. This will produce a *.txt* file, if you drag this file onto *ltfftconvert.exe* it will produce a corresponding *\_out.txt* file containing data that can be pasted into Excel with a sensible number of data points. This is just provided for convenience, you can use the LTSpice data however you like.