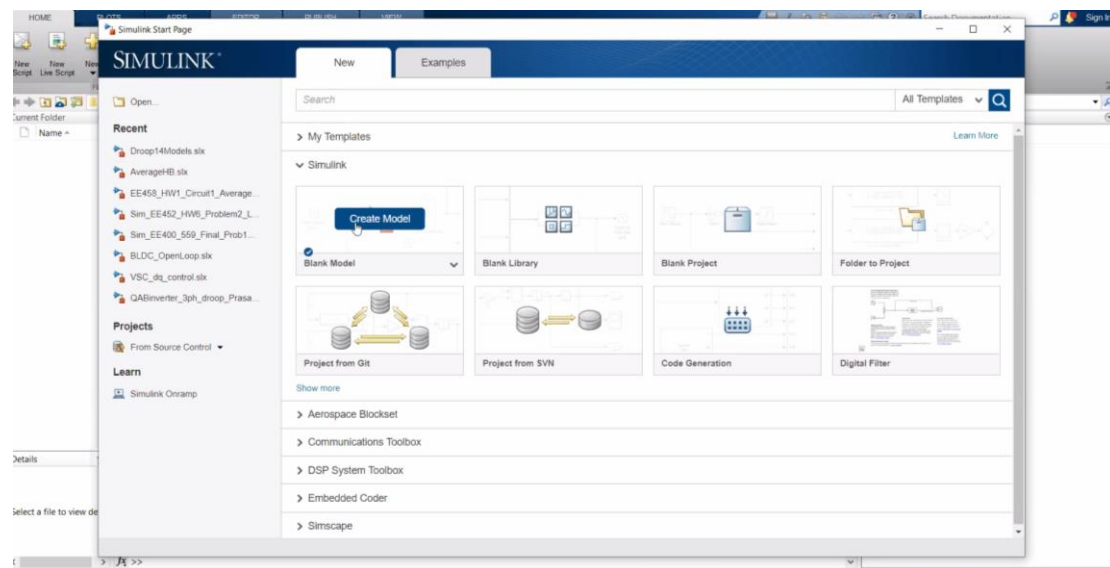


STEPS TO GENERATE PLOTS IN MATLAB ONCE YOUR PLECS MODELS ARE UP AND RUNNING

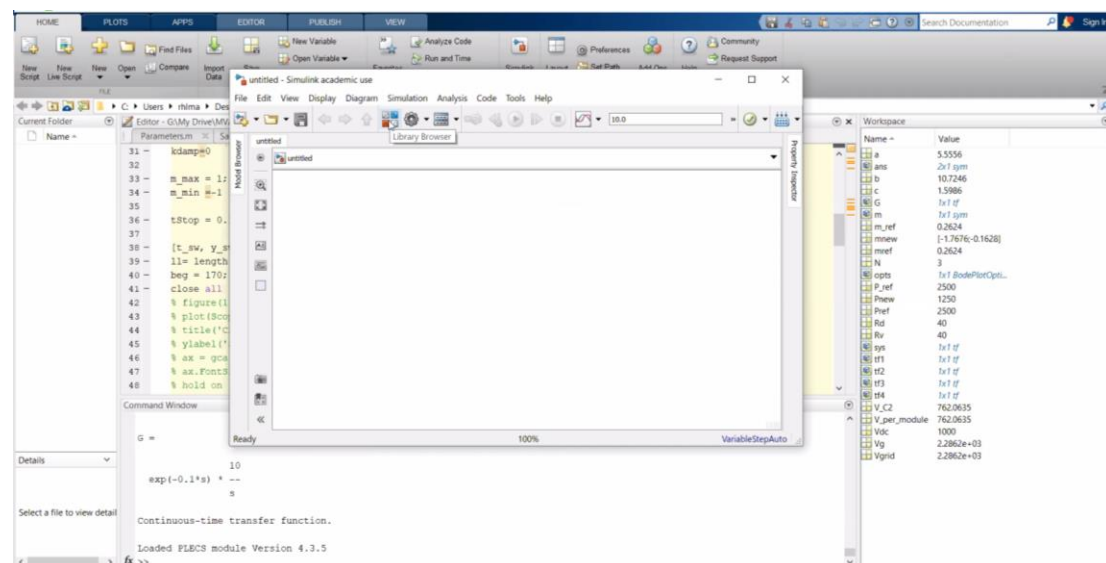
By: Kevin Hung, Isaac Taguchi, Rahul Mallik

Step 0: Use MATLAB/Simulink only to get pretty plots. Make sure you have a simulation running in PLECS without any errors. Once you are done with simulations and only want to make the plots follow these steps. Make sure you have a blockset license for this.

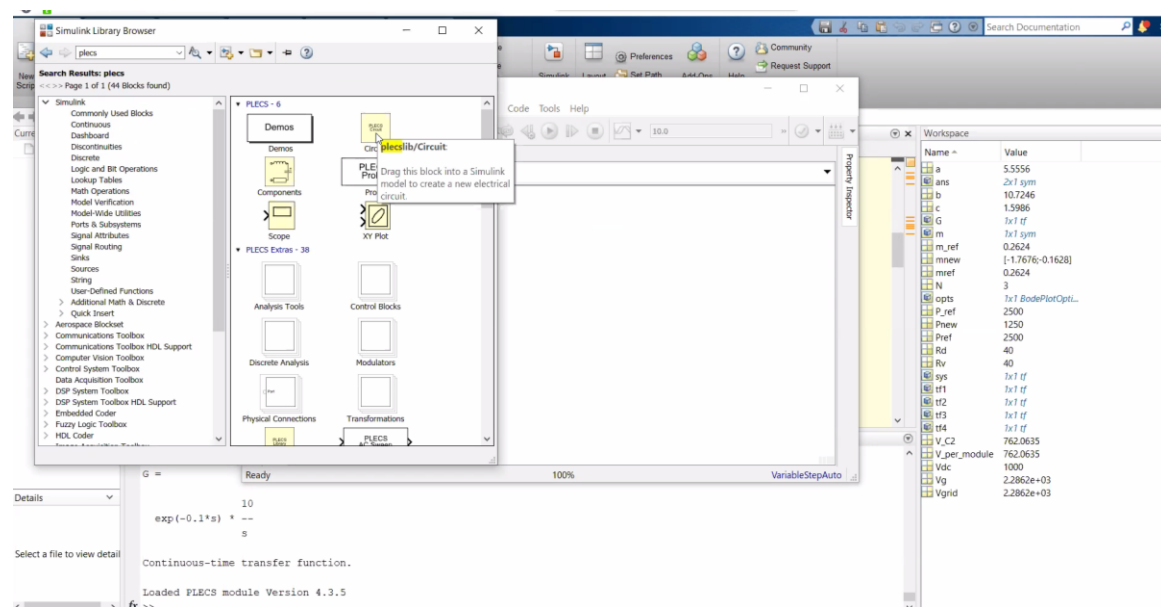
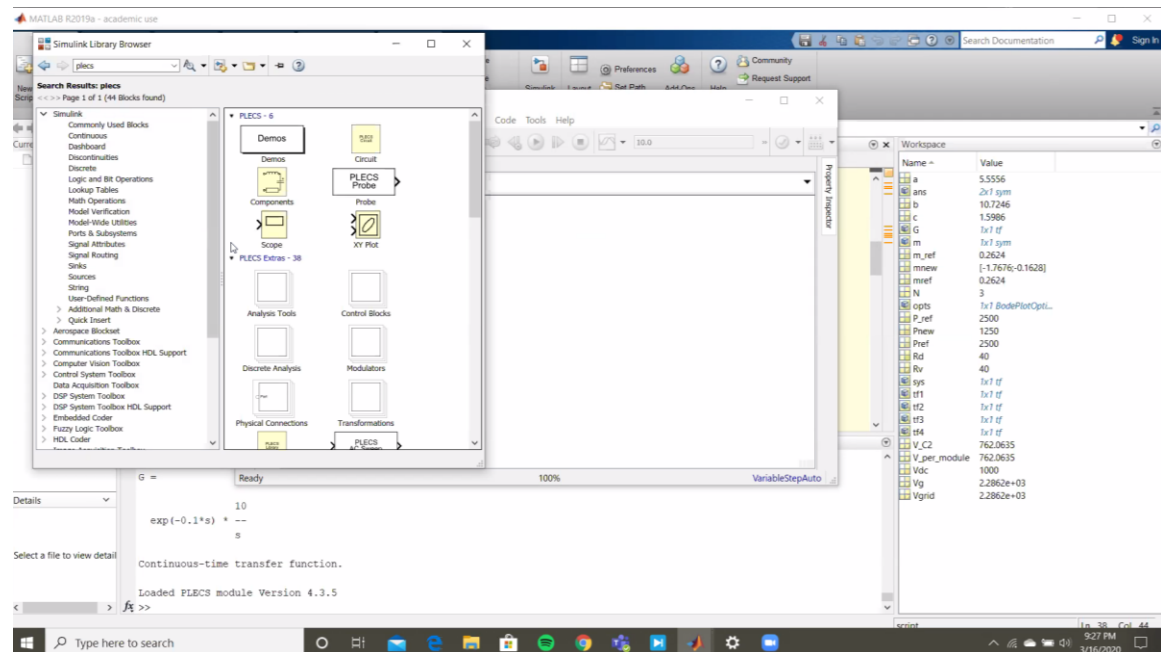
Step 1: Open a new Simulink model

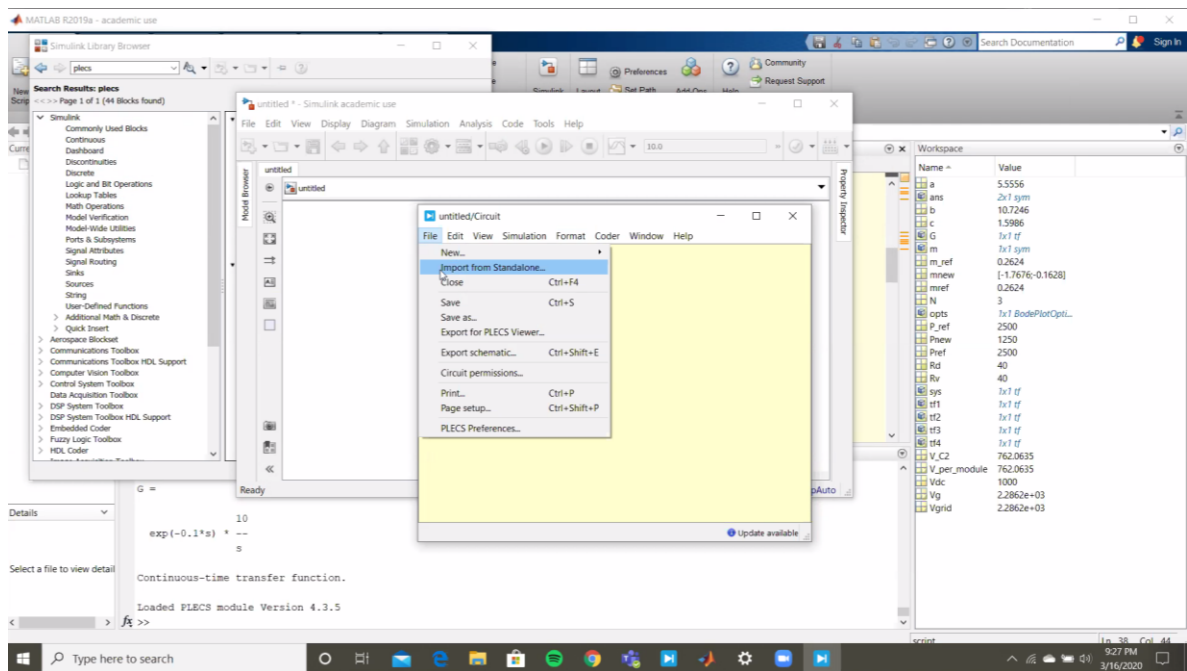
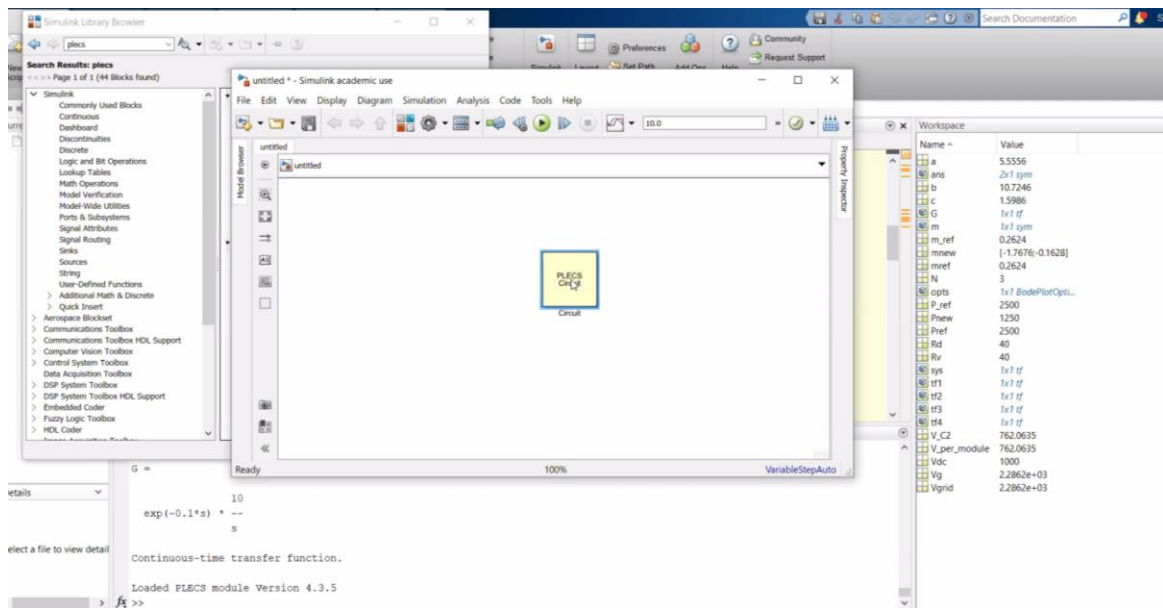


Go to Simulink in matlab

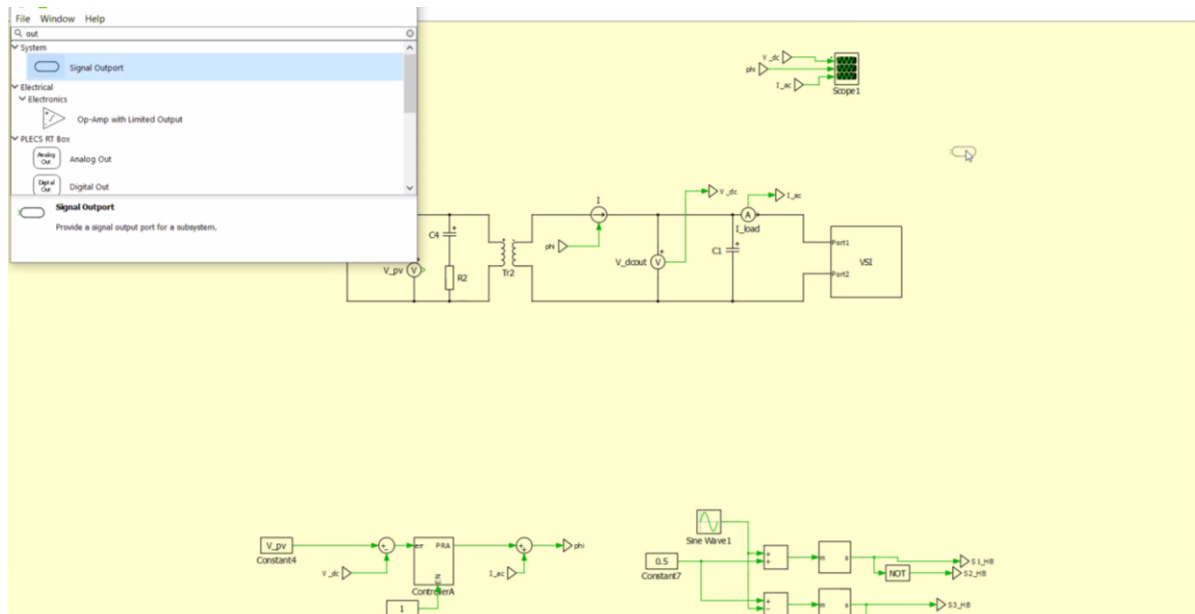


Step 2: Import the already built plecs model into this

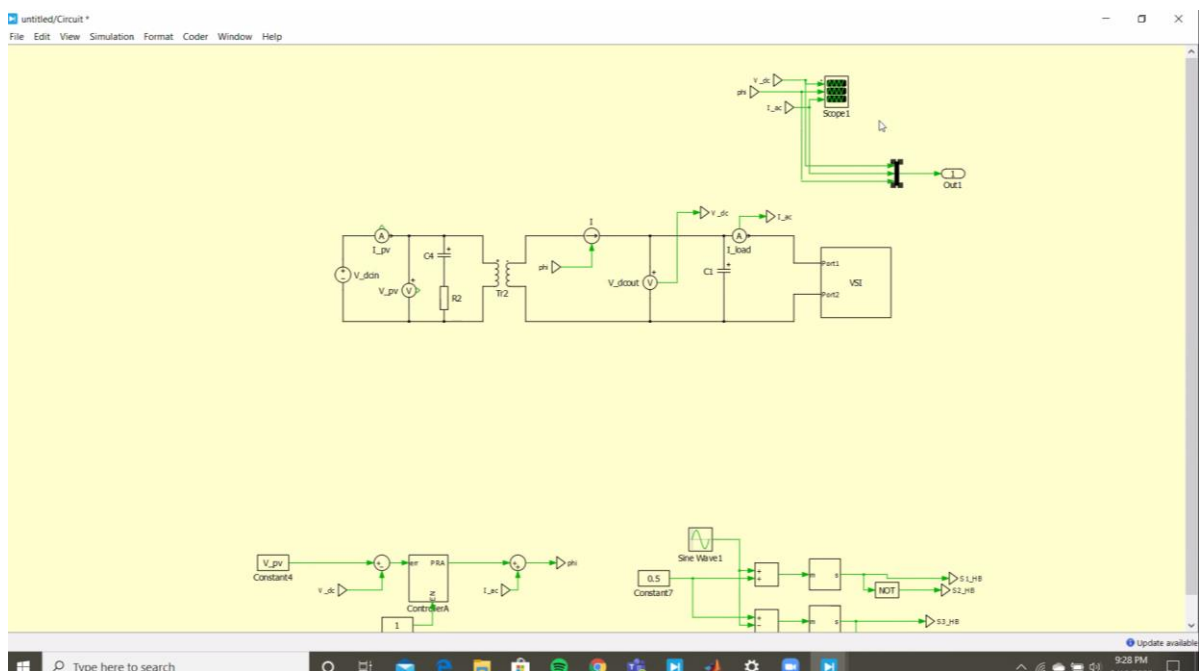




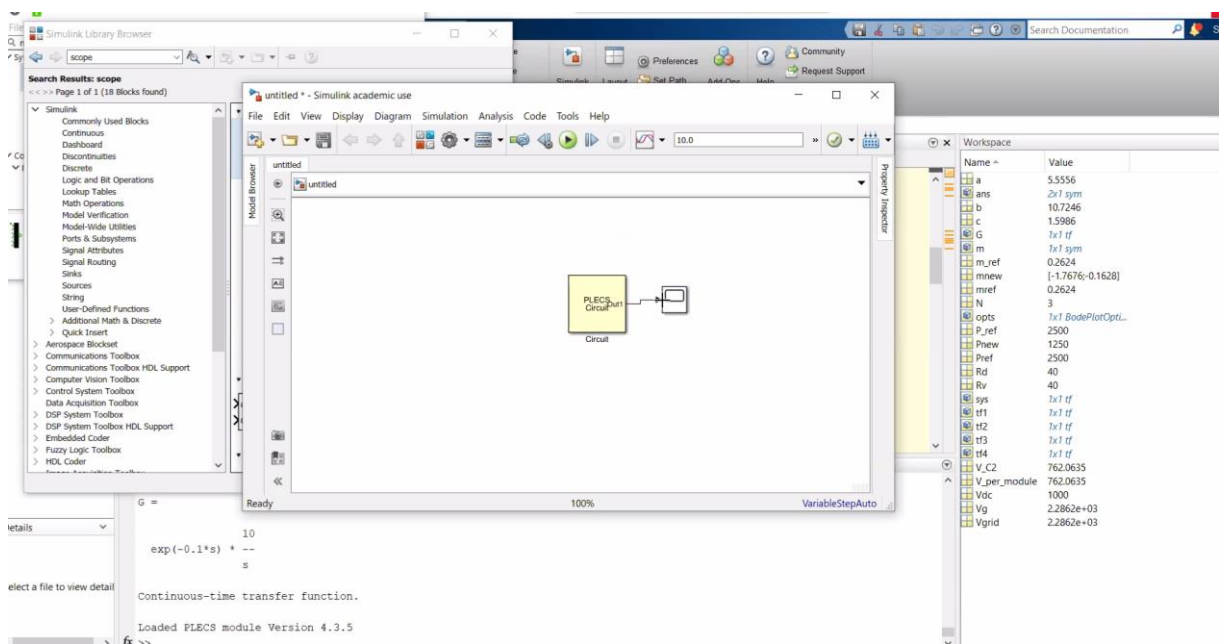
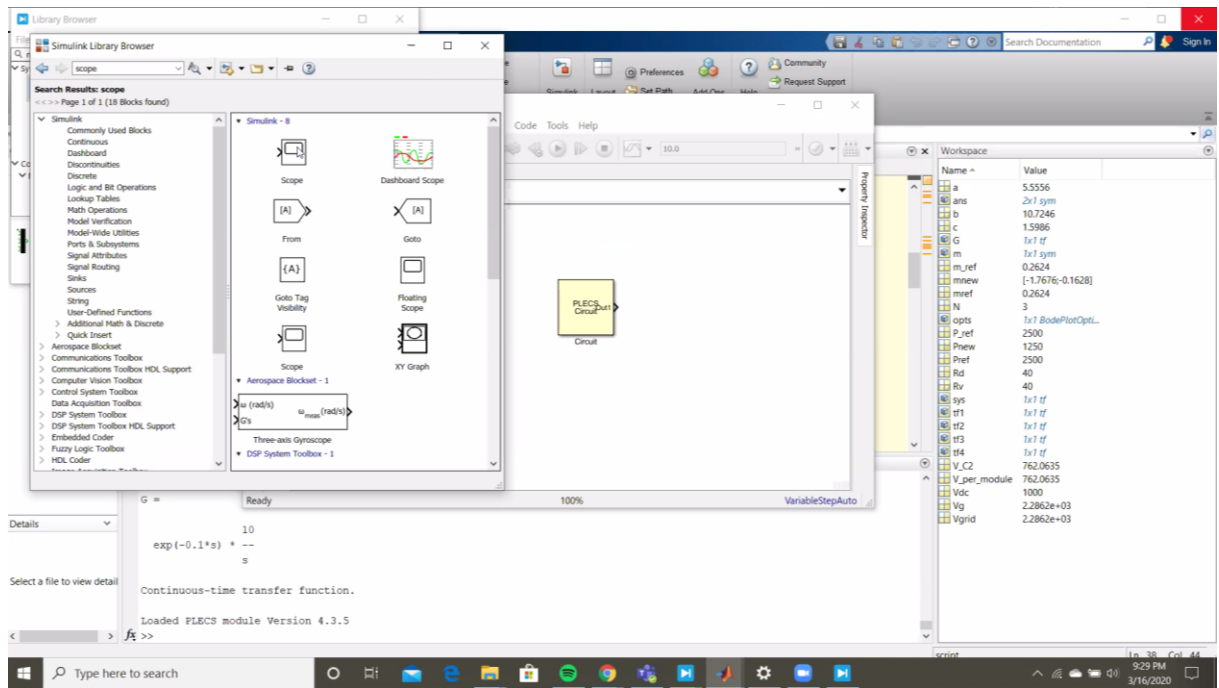
You can import the simulation which is compiled and running without any errors now into this block.



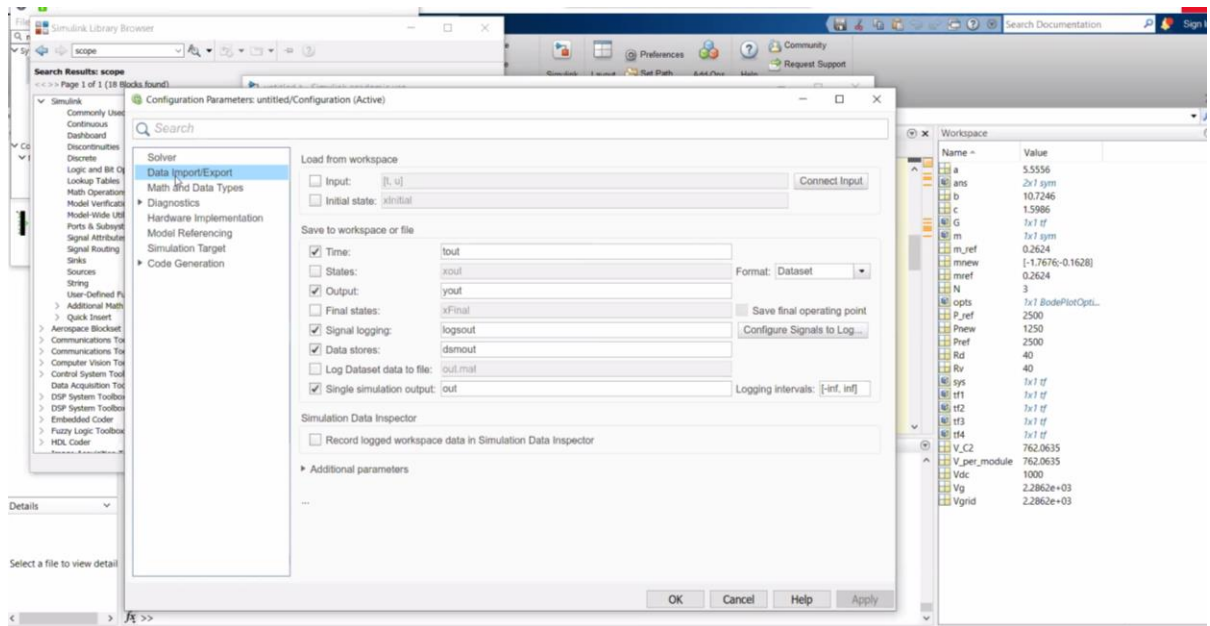
Use the signal import block, add a multiplexer to plot all the signals that you need to plot, number of multiplexer channel = number of things you want to plot



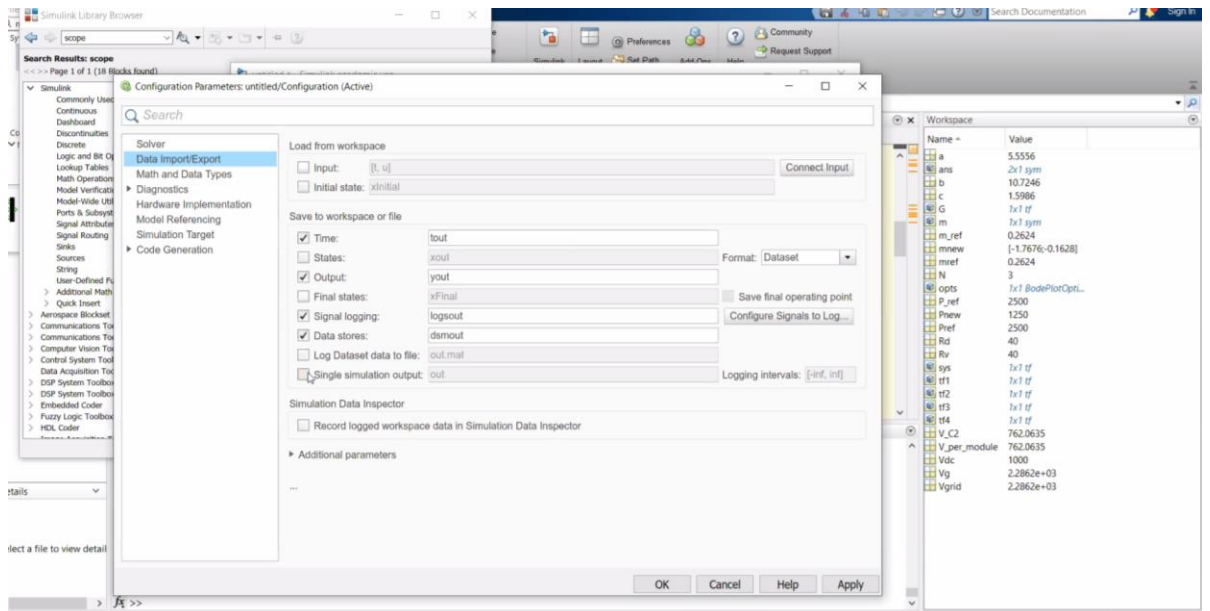
Step 3: Configure MATLAB now, use a scope and



Press CTRL+E to get this screen and untick the “single simulation output”

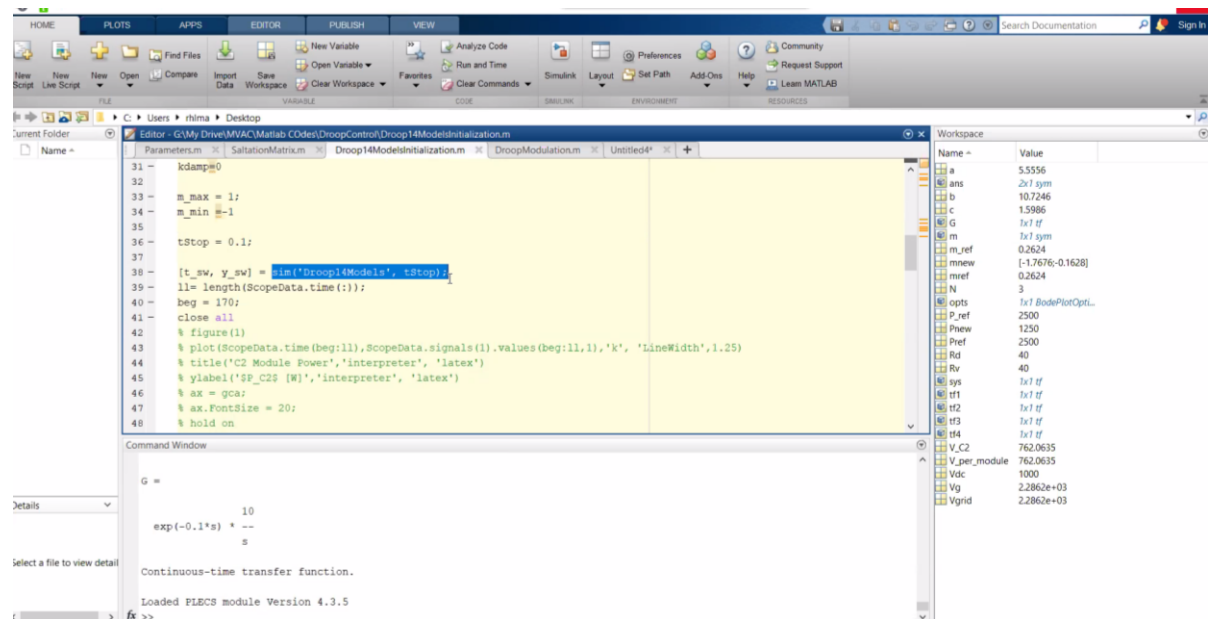


unclick single simulation output



Step 4: Write up the starter file for PLECS. You might have defined inductance

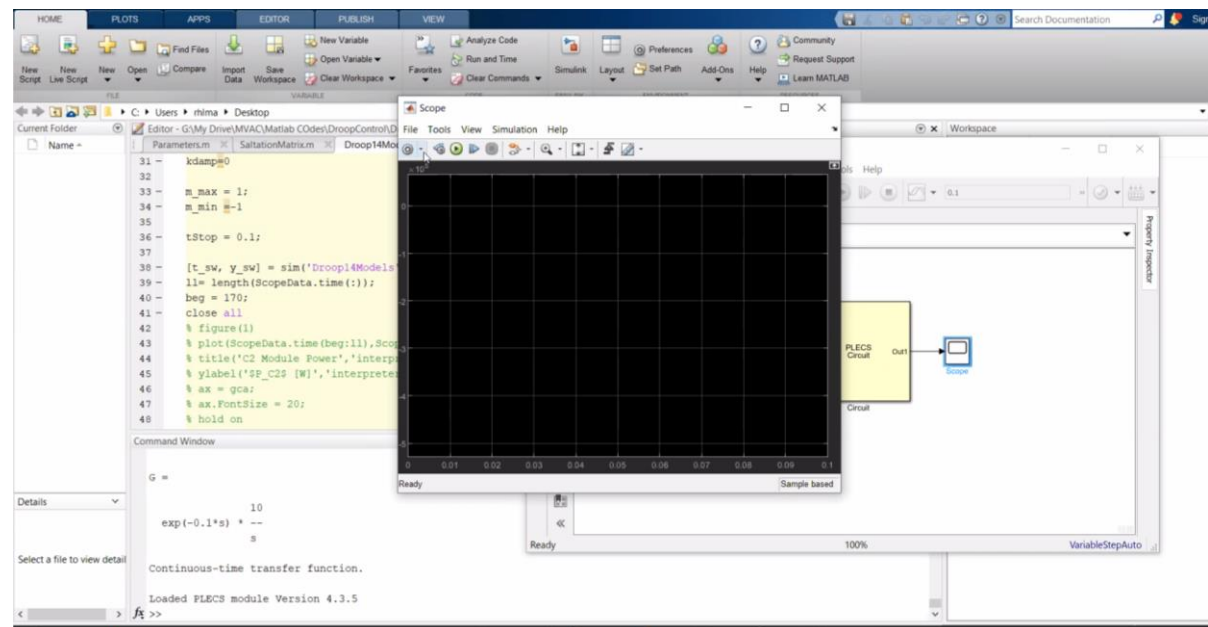
values and other parameters in the **“initialization”** tab of the PLECS. Copy that into a MATLAB script and name it as **starterfile.m** . Make sure this file and your simulation is in the same folder.

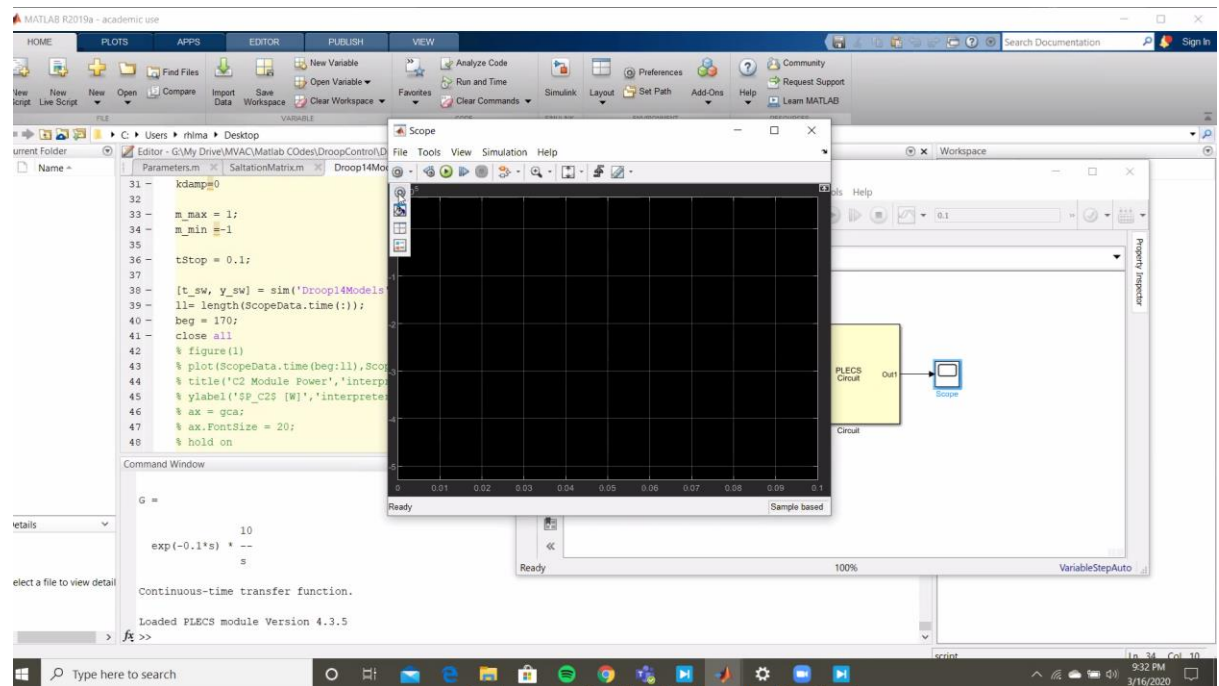


Include the code highlight with the model file name (simulink)

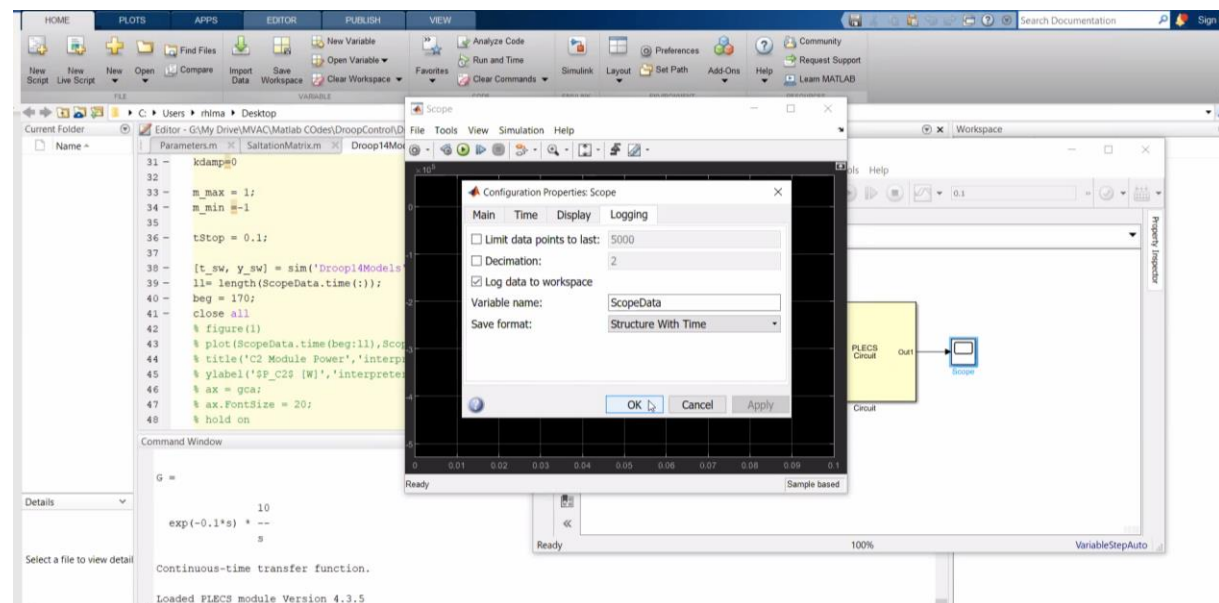
Step 5 : Configure the scope to save the channel data that PLECS is sending out.

Open the scope, click on the SETTING wheel.





Click that



Save the format as struct with time

Run simulation in matlab

Step 6: Make pretty plots

Once you have got till this, time to make pretty plots.

figure(1)

```

plot(ScopeData.time, ScopeData.signals(1).values(:,1),
     'k', 'LineWidth', 1.25)

```



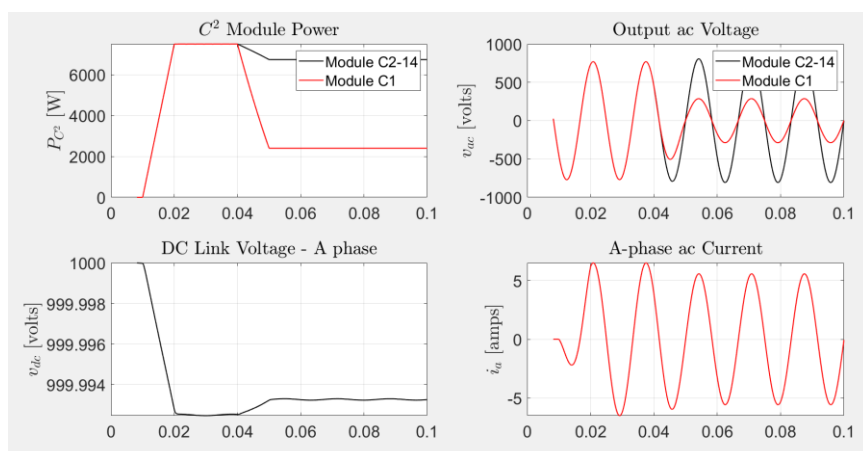
```

title('$C^{2}$ Module Voltage','interpreter',
'latex')
ylabel('$V_{dc}$ [volts]','interpreter', 'latex')
ax = gca;
ax.FontSize = 20;
hold on
grid on
plot(ScopeData.time,ScopeData.signals(1).values(:,2),
'r', 'LineWidth',1.25)
legend('Module C2-14', 'Module C1')

```

More on plotting

1. We usually plot(t,y). The time information is stored in ScopeData.time variable whereas the first signal, lets call it the voltage is stored in the variable ScopeData.signals(1).values(:,1).
2. The structure is simple, ScopeData.signals(1).values(:,i) stores the ith variable in the PLECS module that goes into the signal import module.
3. If you want to get multiple plots like this ,



In this the first few miliseconds of simulation generated bad results. I wanted to crop it.

```
ll= length(ScopeData.time(:));% I find out what is the total length of time recording
```

```
beg = 170; % I just chop off the first 170 data points
```

```
figure(1)
```

```
subplot(2,2,1)
```

```
plot(ScopeData.time(beg:ll),ScopeData.signals(1).values(beg:ll,1),'k', 'LineWidth',1.25)
```

```
title('$C^{2}$ Module Power','interpreter', 'latex')
```

```
ylabel('$P_{C^2}$ [W]','interpreter', 'latex')
```

```
ax = gca;
```

```

ax.FontSize = 20;

hold on

grid on

plot(ScopeData.time(beg:ll),ScopeData.signals(1).values(beg:ll,2),'r', 'LineWidth',1.25)

legend('Module C2-14', 'Module C1')

```

```

subplot(2,2,2)

plot(ScopeData.time(beg:ll),ScopeData.signals(1).values(beg:ll,4),'k', 'LineWidth',1.25)

title('Output ac Voltage','interpreter', 'latex')

ylabel('$v_{ac}$ [volts]','interpreter', 'latex')

ax = gca;

ax.FontSize = 20;

hold on

grid on

plot(ScopeData.time(beg:ll),ScopeData.signals(1).values(beg:ll,6),'r', 'LineWidth',1.25)

legend('Module C2-14', 'Module C1')

```

```

subplot(2,2,3)

plot(ScopeData.time(beg:ll),ScopeData.signals(1).values(beg:ll,3),'k', 'LineWidth',1.25)

title('DC Link Voltage - A phase','interpreter', 'latex')

ylabel('$v_{dc}$ [volts]','interpreter', 'latex')

ax = gca;

ax.FontSize = 20;

hold on

grid on

subplot(2,2,4)

plot(ScopeData.time(beg:ll),ScopeData.signals(1).values(beg:ll,5),'r', 'LineWidth',1.25)

title('A-phase ac Current','interpreter', 'latex')

ylabel('$i_a$ [amps]','interpreter', 'latex')

ax = gca;

ax.FontSize = 20;

hold on

grid on

```