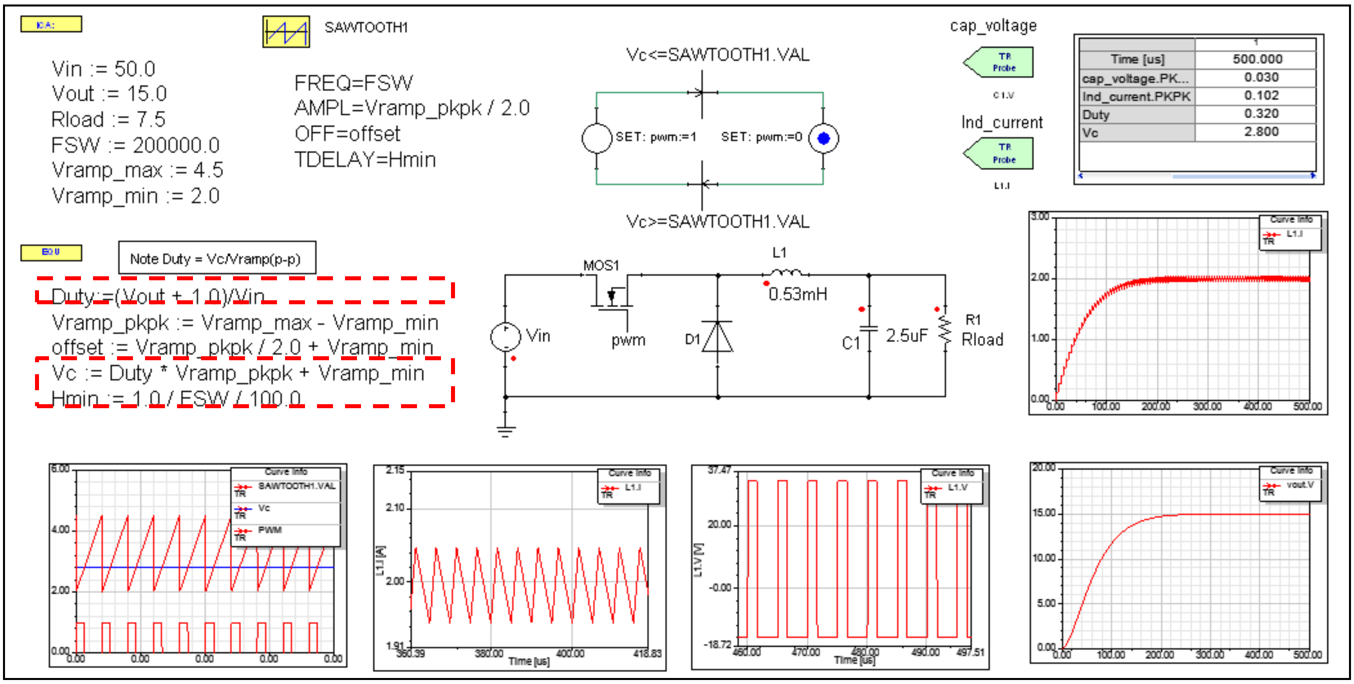
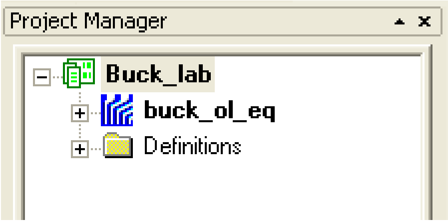
Section One

- Note see addendum for initial software set up
- Several sections of a DC/DC power converter design process will be presented in this lab, each showing a different aspect of Simplorer's capability.
- In the first section, the following Buck power stage will be implemented

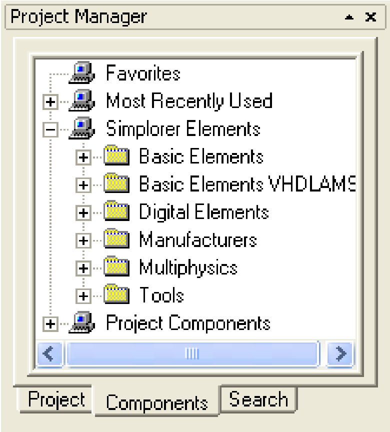


- This example was created to show a variety of Simplorer's capabilities.
  - ICA Initial Condition block to input the specifications of the power stage
  - EQU Equation block to calculate variables that determine the operation
  - Sawtooth source for the desired "ramp" wave
  - State flow diagrams used to calculate the pwm signal for the switch
  - Measurement functions to evaluate design
  - Use of system level MOSFET switch and diode
  - Displays for waveforms and numerical values

- Invoke simplorer and rename the project to be “Buck_lab”, then rename the simplorer design to be “buck_ol_eq”



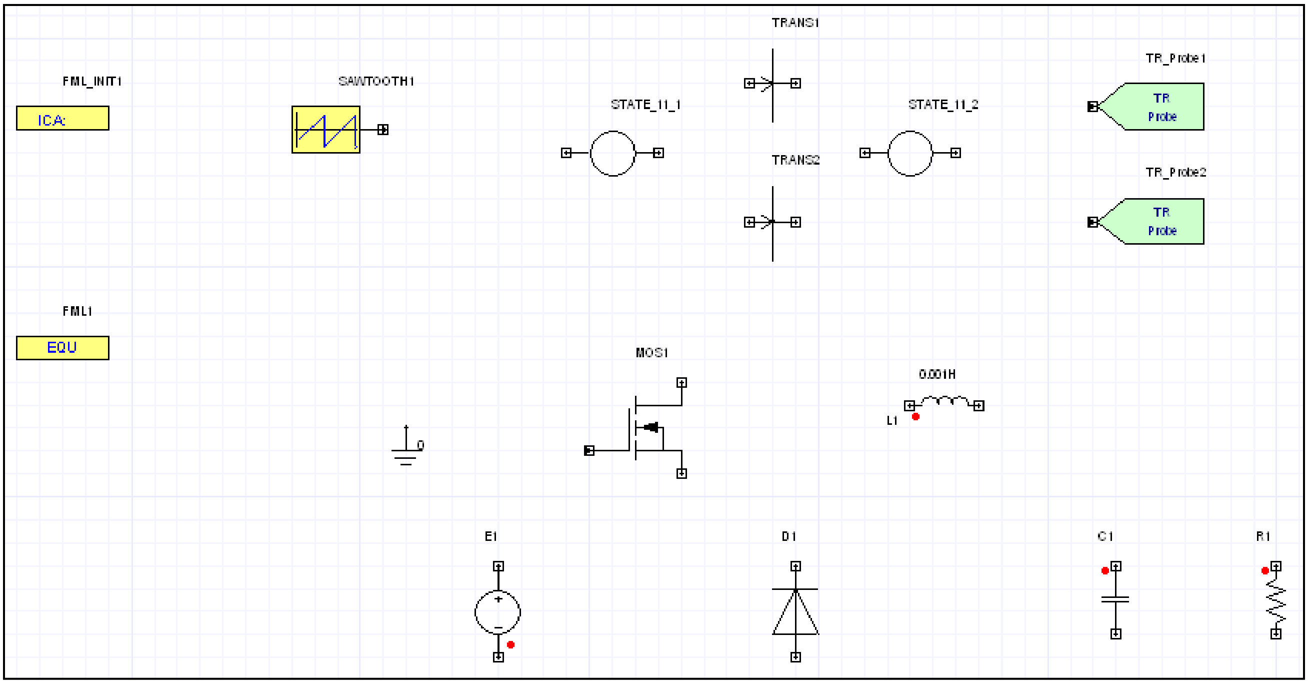
- Open the “buck_ol_eq” design and insert the different components based on the following library locations under the “components” tab. (see next page)



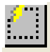
- The ICA initial condition block is found in:
  - Basic Elements/Tools/Equations/ “FML_INIT:Initial Values”
- The EQU Equation block is found in:
  - Basic Elements/Tools/Equations/ “FML:Equation”
- The sawtooth function block is found in:
  - Basic Elements/Tools/Time Functions/”SAWTOOTH:Sawtooth”
- The State logic input/output State is found in:
  - Basic Elements/States/”STATE_11:State 11”
- The State logic transistion is found in:
  - Basic Elements/States/”TRANS:Transition”
- The measurement probes used for the capacitor voltage and inductor current pk-pk measurements are found in:
  - Basic Elements/Signal Char/ “TR_Probe: TR Probe”

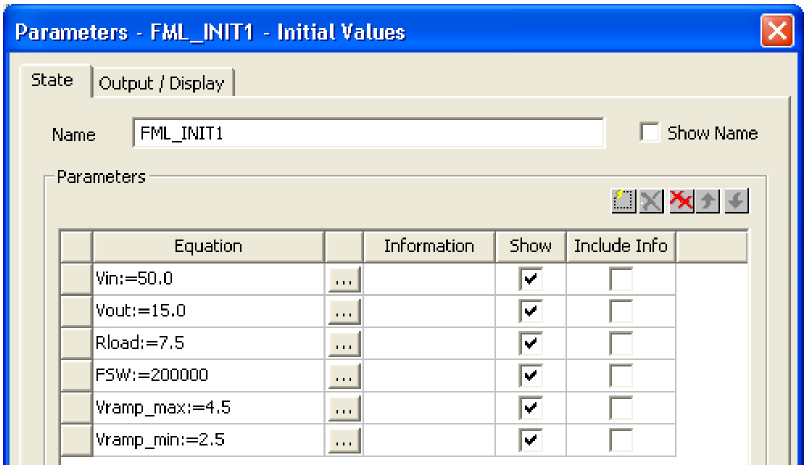
- The Input Voltage uses the “E” voltage source found in:
  - Basic Elements/Circuit/Sources/”E:Voltage Source”
- The system level MOSFET and Diode are found in:
  - Basic Elements /Circuits/Semiconductors System/
- The inductor, capacitor and load resistor are found in:
  - Basic Elements /Circuits/Passive Elements/
- The easiest way to import a ground symbol is to use “ctrl + g” or via the pull down menu “Draw -> Ground”

- Placing the components on the schematic in the approximate positions as seen on the preview schematic should yield the following so far.



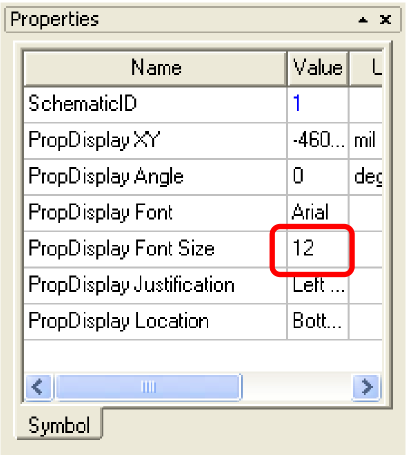
- The next step will be to fill out the properties of each component , rotate as needed, and set up the display format.

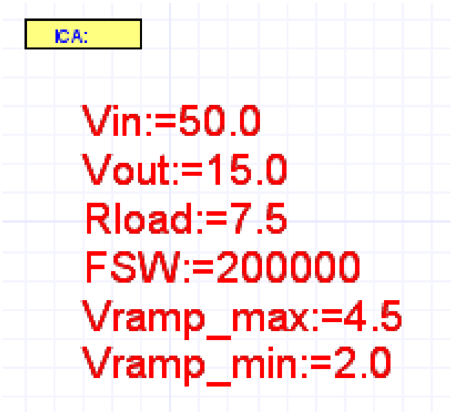
- Setting up the ICA block:
  - Double click on the ICA block, add in variables via the  standard “input” Icon for each variable to be added. Make sure to select the “Show” box to display the variable on the schematic. De-select the “Show Name” for the “Name” of the ICA block. It should look like the following:



  - Select “OK”
  - To resize the text on the schematic, select all the variables by holding the left mouse button down while drawing a box around all the variables, they will turn red to indicate the selection. Note in the “Properties” window on the left side of the Simplorer window has a selection called “PropDisplay Font Size”, by default it is set to 5. change it to 12, select the “Enter” key.

  - To reset the spacing, change the “PropDisplay Location” from Bottom To Center, then back to Bottom. It should Now look as follows.





- Note the values in the ICA block represents the following:
  - Vin = input voltage
  - Vout = output voltage
  - FSW = switching frequency
  - Vramp_max and Vramp_min represents the typical ramp voltage range found in PWM IC Controllers. this is used to compare to the “control voltage” Vc that comes from the typical error amplifier found in a normal feedback loop.

- Repeat the same procedure to fill out the EQU equation block, it should end up looking like the following:

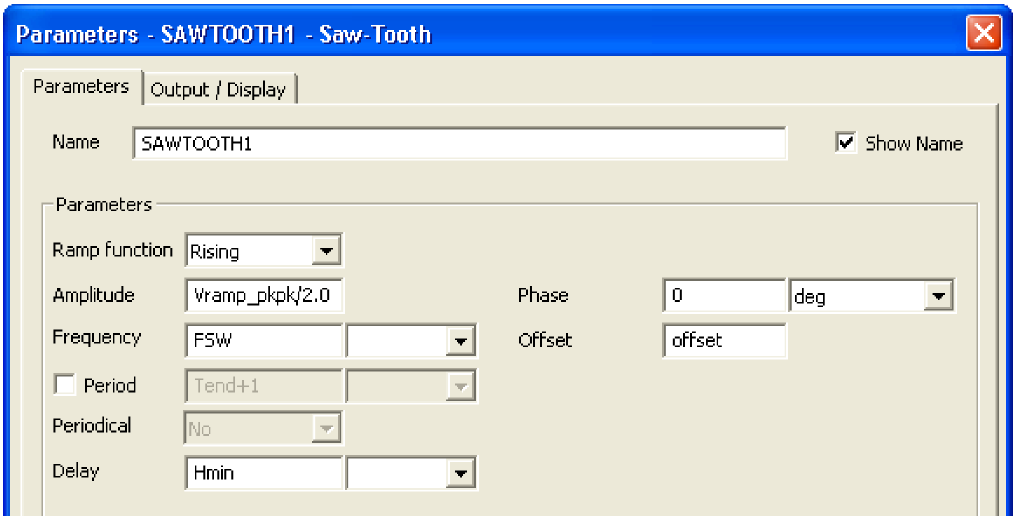
```
EQU

Duty:=(Vout + 1.0)/Vin
Vramp_pkpk := Vramp_max - Vramp_min
offset := Vramp_pkpk / 2.0 + Vramp_min
Vc := Duty * Vramp_pkpk + Vramp_min
Hmin := 1.0 / FSW / 100.0
```

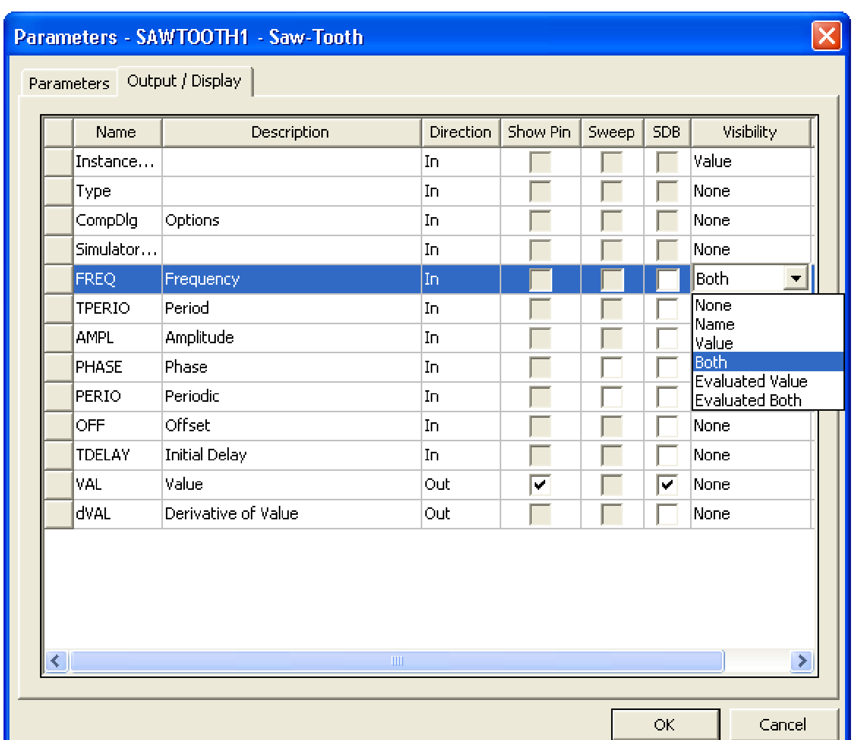
- Note the above equations represent basic voltage mode control relationships used to calculate the control voltage “Vc” as if it had feedback. This control voltage value will then be “compared” with the ramp voltage to generate the pwm signal which controls the switching of the MOSFET, to yield the desired output voltage.

- NOTE variables that control the simulation (ie Hmin, which is the minimum time step used by the simulator) can also be set, and changed during the simulation.

- Double click the mouse over the SAWTOOTH block to set up the ramp voltage signal. Fill out the parameters as shown below.

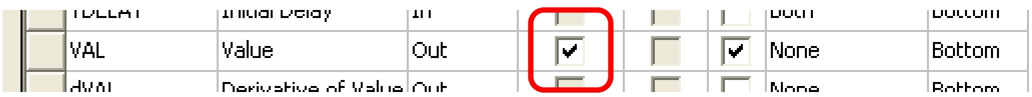


- Before selecting "OK", set up the display that will show up on the schematic by selecting the "Output/Display" tab. Choose to display both the parameter and its value by using the left mouse button (LMB) when the curser is over the desired property's Visibility field (see below). Do this for "FREQ", "AMPL", "TDELAY", and "Offset". Select "OK"

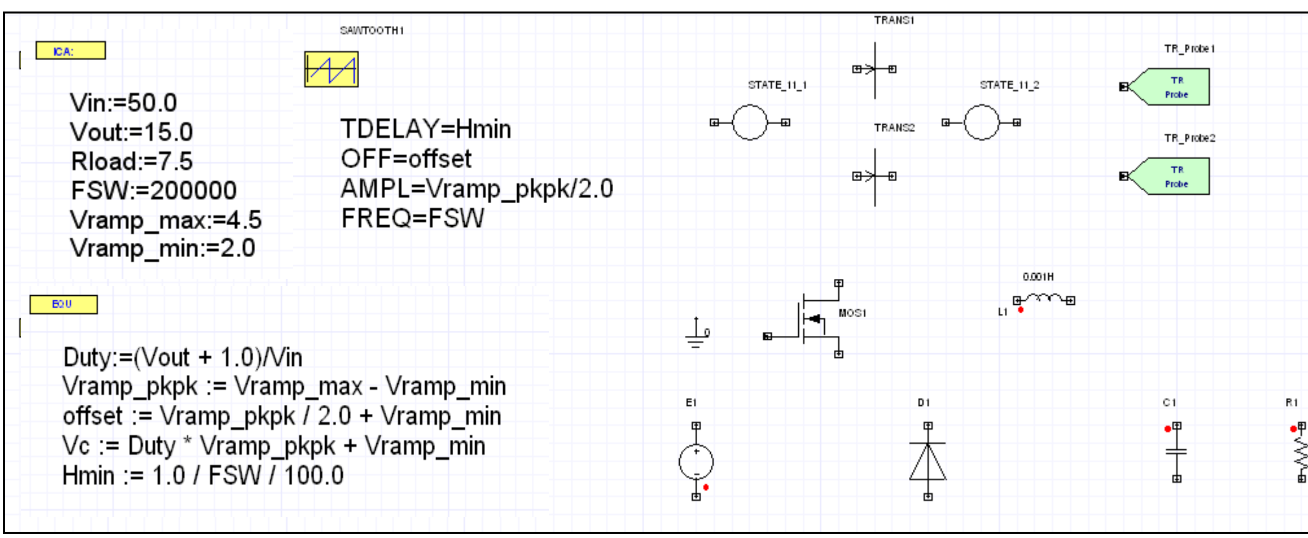


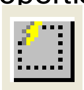
- Now change the SAWTOOTH's properties font size that is shown on the schematic. Use the same process that was done for the ICA and EQU blocks. Hold down the left mouse button and draw a box around the text shown on the schematic, this causes the text to turn red to indicate it is selected. In the Properties window on the left, change the "PropDisplay Font Size" from 5 to 12, then select "Enter". Reset the spacing of the text by changing the "PropDisplay Location" from Bottom, to Center, back to Bottom.

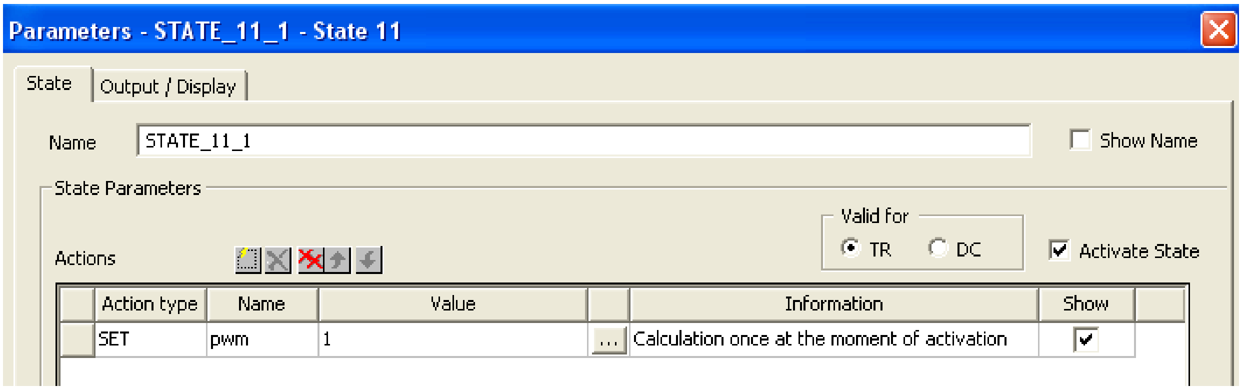
- Note the SAWTOOTH block has a pin for the output. This pin will not be needed in this example and can be hidden by double clicking on the block, then selecting the "Output/Display" tab, then De-selecting the box under the "show pin" column



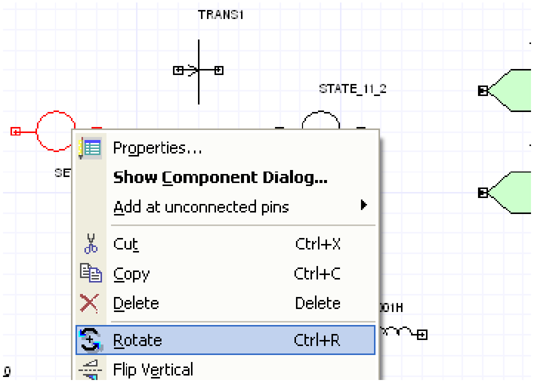
- Note the schematic should now look like the following so far:



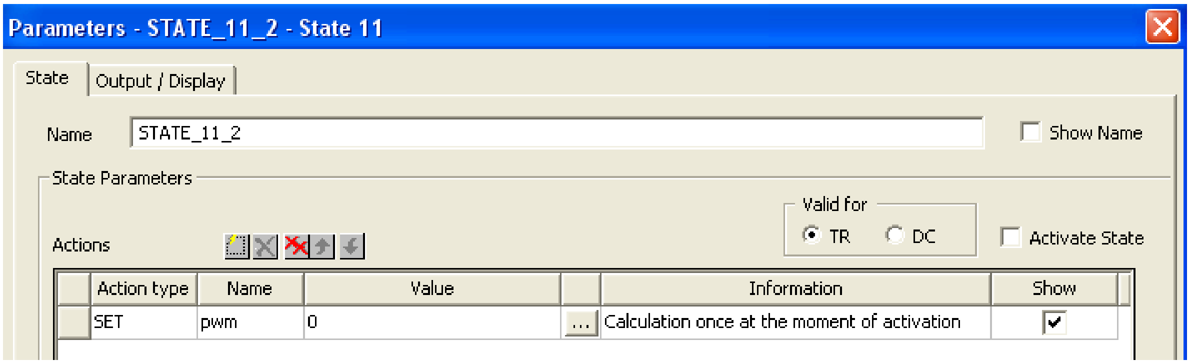
- The next step is to complete the state diagram section that creates the pwm signal used to switch the MOSFET on and off.
- Double click the mouse over the “STATE_11_1” symbol to bring up the properties window. Select the standard input icon to create an equation that will be evaluated when the simulation enters into this state. 
- Set the following (Action type:SET, Name:pwm, Value:1, see below) this indicates that when the simulator enters into this state, it will set the variable “pwm” to be equal to “1”. This will turn on the MOSFET.
- Select the “Show” box next to the Action to view it on the schematic
- De-select the “Show Name” box for the Name
- IMPORTANT: select the “Activate State” box to initialize on this state. This will place a blue dot on the state to indicate it will be used initially.
- See below, then Select “OK



- On the schematic, rotate the State by selecting it, then RMB (Right Mouse Button) to bring up the menu, then select “rotate”

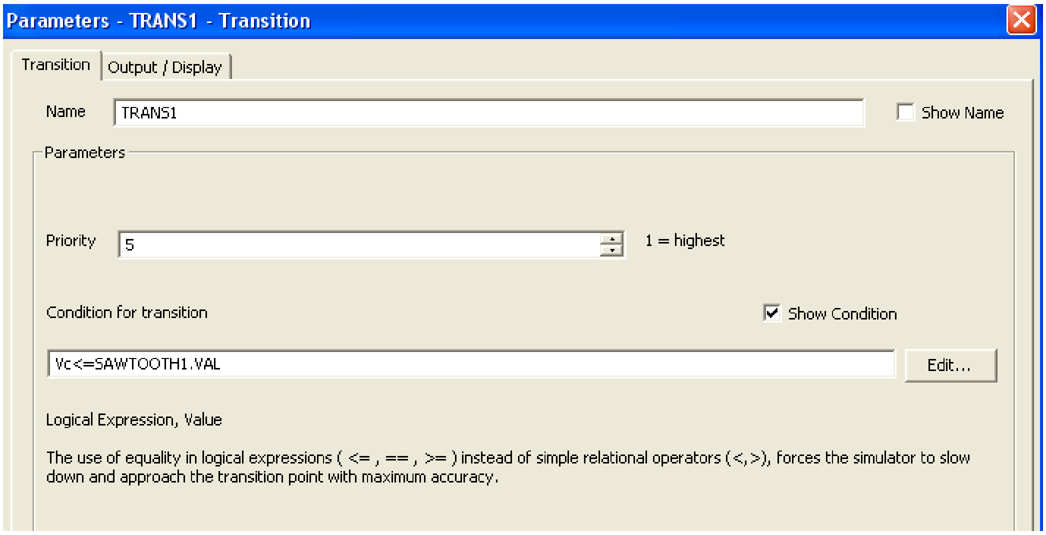


- Repeat the previous process for "STATE_11_2" however now set "pwm" to zero "0" (see below - NOTE do NOT select "Activate State" on this one). Note when the simulator enters this state, it will now set the variable "pwm" to be equal to zero, which will turn the MOSFET switch off.

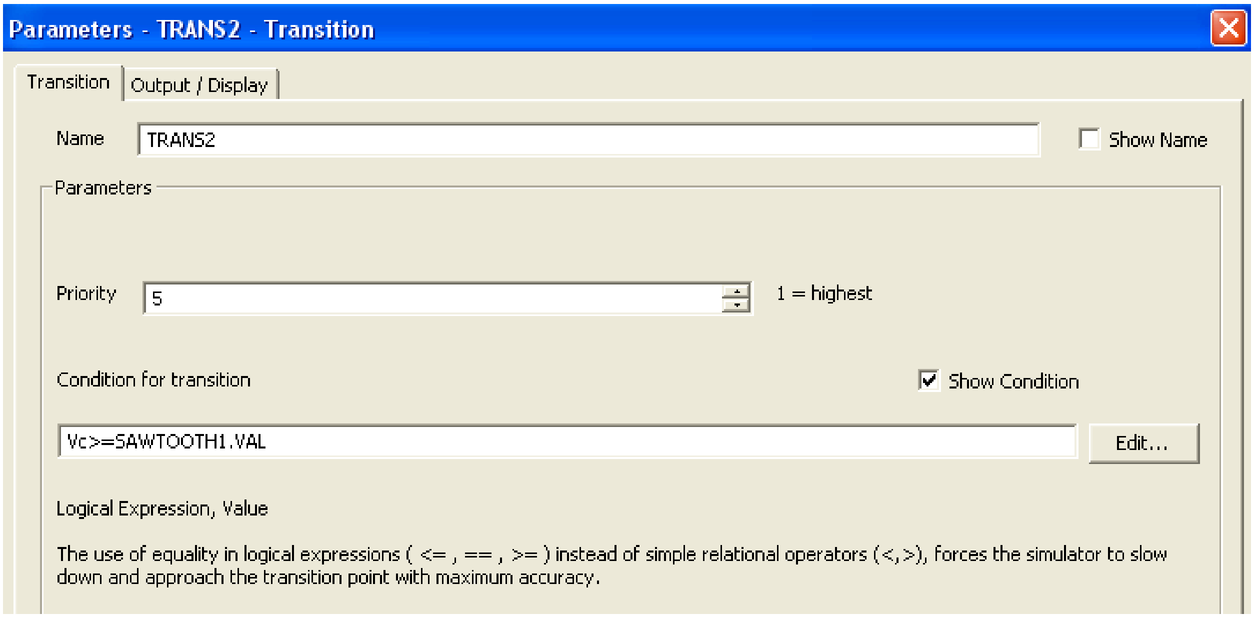


- Rotate it on the schematic as before.

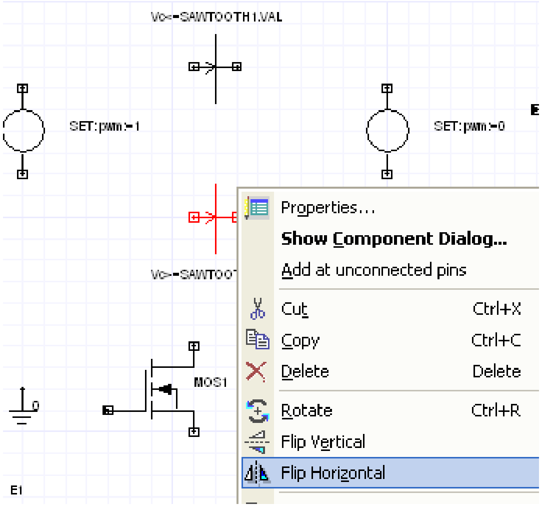
- Next double click the upper "TRANS1" transition symbol to set up conditions that must be met for transition from state (where pwm =1) to state (where pwm = 0).
- See below, note when Vc <= SAWTOOTH1.VAL , pwm will be set to 0, and the MOSFET will be turned off.
- Select the box "Show Condition" so it will appear on the schematic
- De-select the "Show Name" box for the Name, Select "OK"



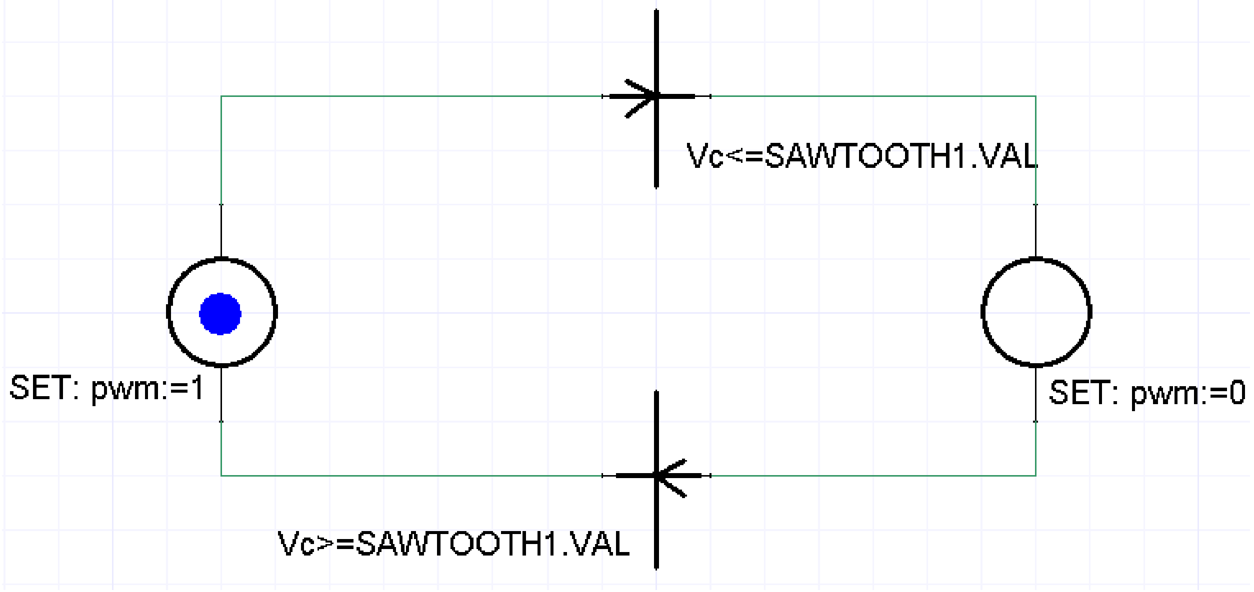
- Double click on the lower “TRANS2” symbol to set up the transition condition where the simulation will go from the state (where pwm =0) to the state (where pwm =1) see below. Note when Vc >= SAWTOOTH1.VAL, pwm will be set to “1” and the MOSFET will be turned on.
- Select the “Show Condition” box and de-select the “Show Name” box as shown, Select “OK”



- IMPORTANT, Flip this symbol horizontally on the schematic

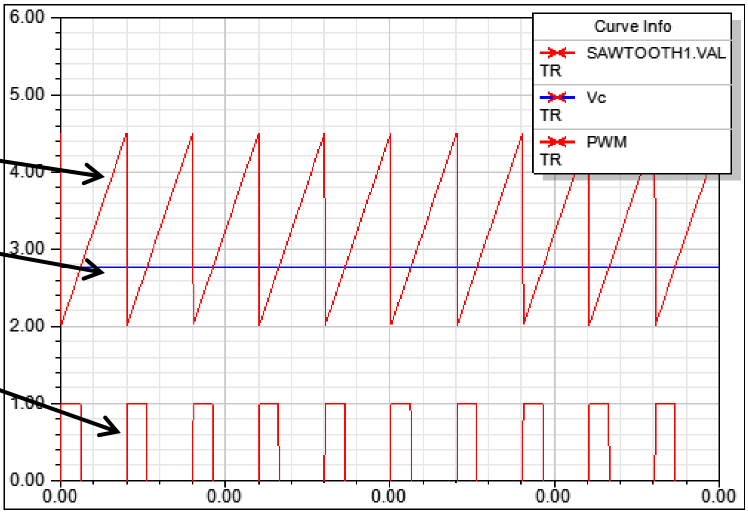


- Connect the state diagram section as shown below, Save the design so far

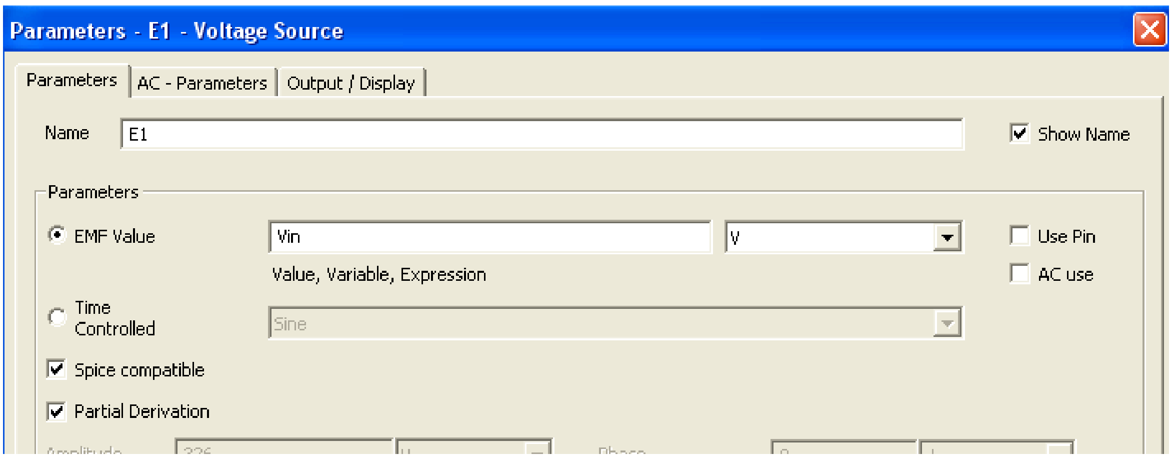


- Note the logic of the state diagram section can be shown below, when Vc (control voltage) is less than or equal to the ramp waveform, the value of "pwm" goes to zero "0" and the MOSFET is turned off, when Vc is greater than or equal to the ramp waveform, the value of "pwm" goes to "1' and the MOSFET turns on.

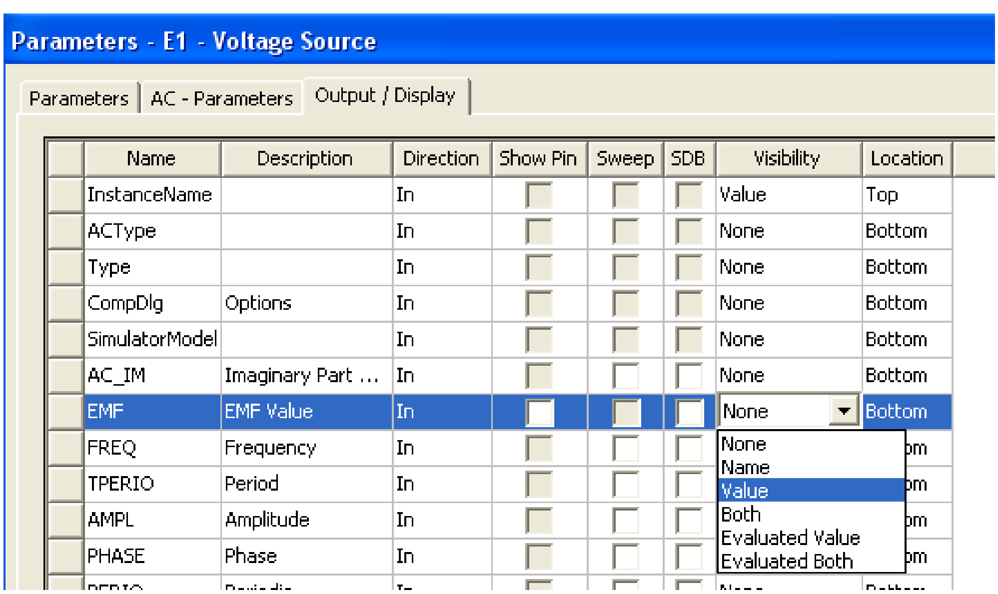
- Ramp waveform
- Vc (control voltage)
- Pwm signal



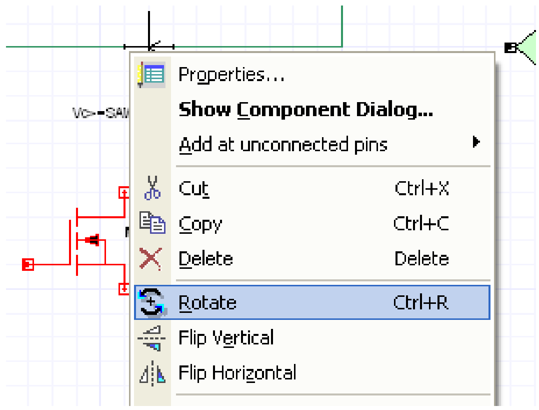
- Next step is to configure the power stage electronics
- Double click on the voltage source and assign the variable “Vin” as seen below. Remember this value of “Vin” comes from the ICA initial condition block. Select the “Spice compatible” box (for spice compatible sign convention)



- Select the “Output/Display” tab and using the left mouse button, make the EMF Value visible as shown below, Select “OK”



- Select the system level MOSFET symbol and rotate it once

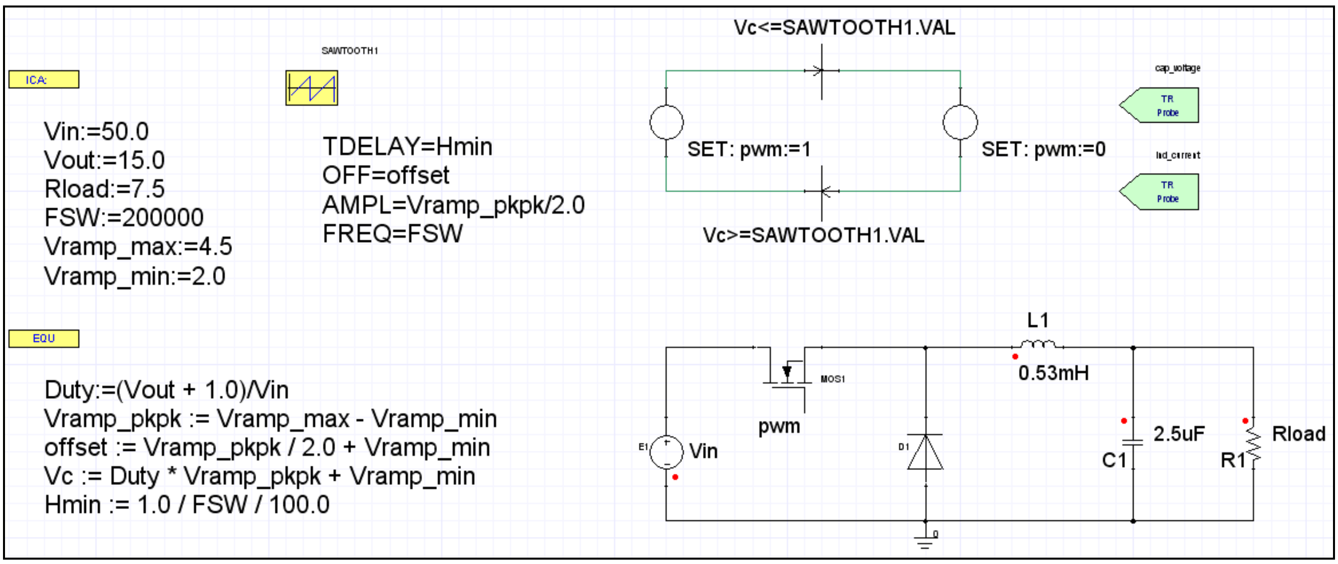


- Double click the mouse on the MOSFET symbol. Note since this is a "system" level MOSFET, it has 1st order behavior like a MOSFET, however does not require the actual high side gate drive circuitry associated with a buck converter switch. This model uses a simple logic control to turn on and off the MOSFET. Here we set the "Control Signal" to be the variable named "pwm" (see below - note must uncheck the "use pin"), if this variable becomes a "1" , the MOSFET turns on, if the variable becomes a "0", the MOSFET is turned off. Remember we had created the state diagram section to control the value of this variable "pwm" based on the control voltage "Vc" and the ramp waveform.

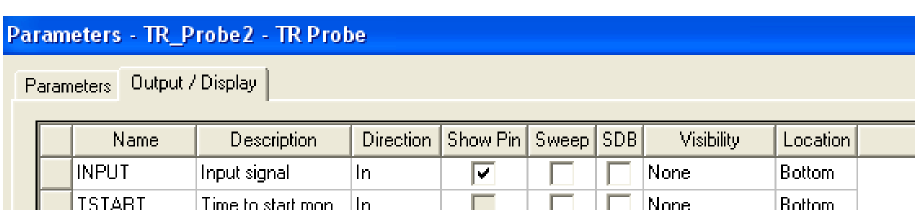
- NOTE Simplorer has several different levels of switch models based on the need

- Note the system level Diode model will not be changed (use default values)

- Next select each of the passive components and assign the following values
- L1 = 0.53m, C1 = 2.5u, R1 = Rload. Set up to display the values, change the font size to 10, rotate as needed. Note “Rload” is a variable that is set in the ICA initial condition block.
- Connect the power stage, when finished, the schematic should look like the following so far: (Save the Design so far)

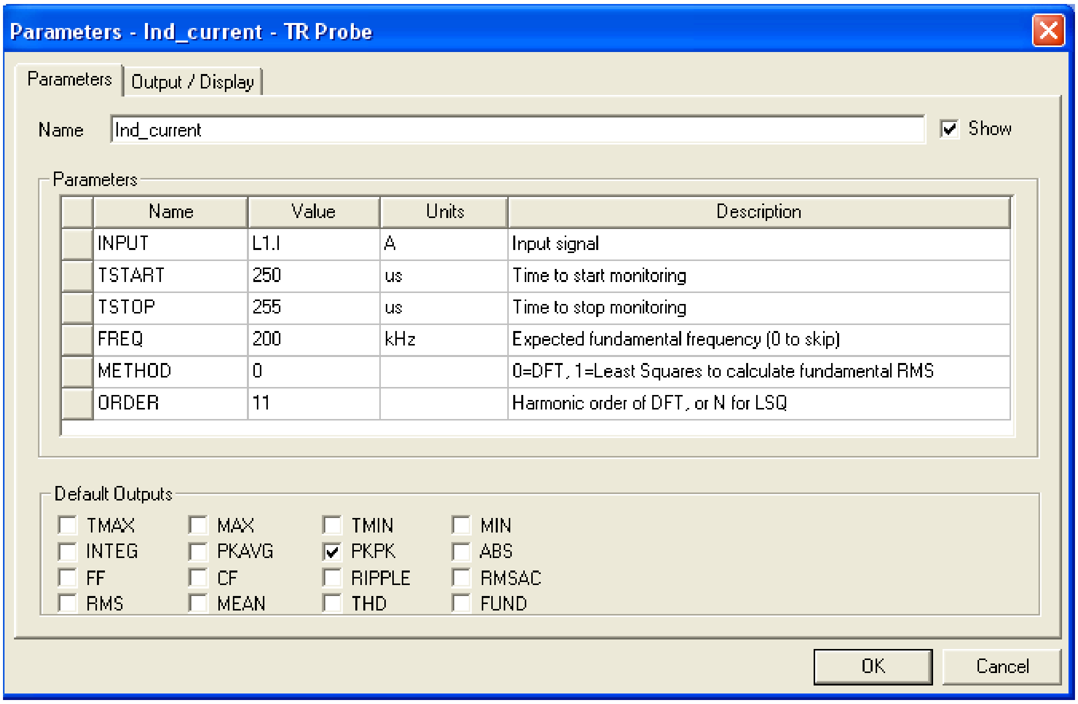


- The next step will be to Set up the measurement “TR_Probes” for the inductor current and the capacitor voltage. This will allow an instant measurement to evaluate the pkpk ripple current thru the inductor and pkpk ripple voltage across the capacitor. The results can then be displayed in a numerical view plot.
- Note to be able to add The “INPUT” as a value, Must first remove the “INPUT” as a pin; double click the mouse on the TR_Probes, select the “Output/Display” tab, then de-select the “Show Pin” for the “INPUT”

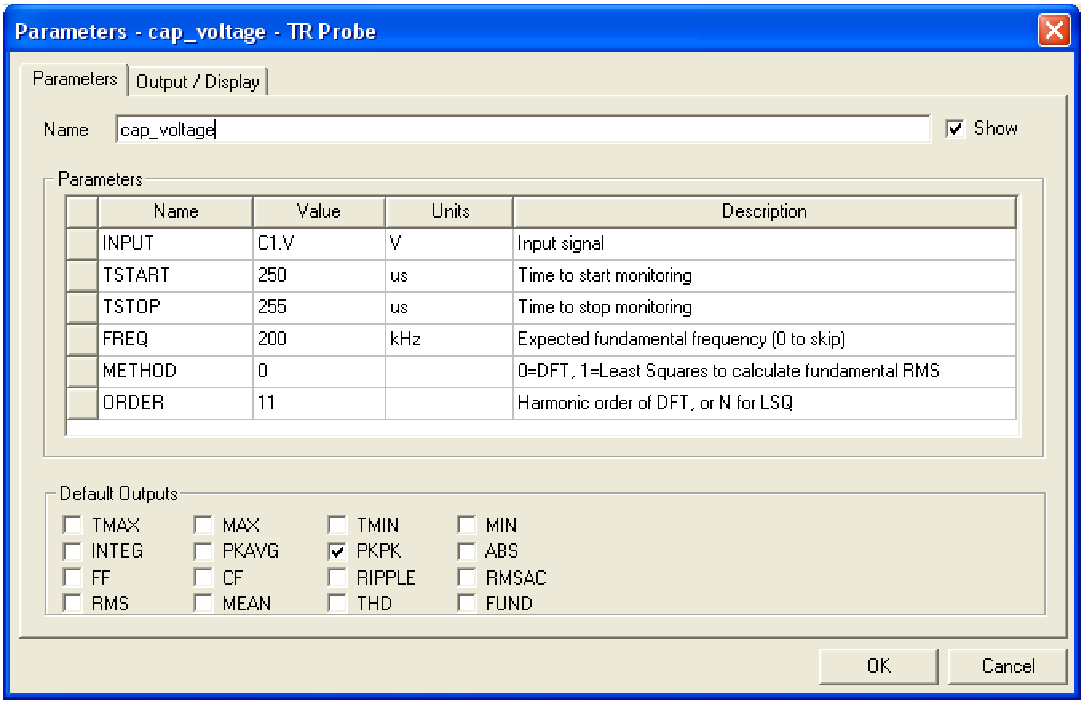


- Note the TSTART and TSTOP are set up to Capture the results After the system has Reached steady state. Note also **only** the “pkpk” Output is selected to save Disc space. Fill out the forms as shown on the next page (INPUT -> L1.I, C1.V, TSTART -> 250us, TSTOP -> 255us, FREQ -> 200kHz)

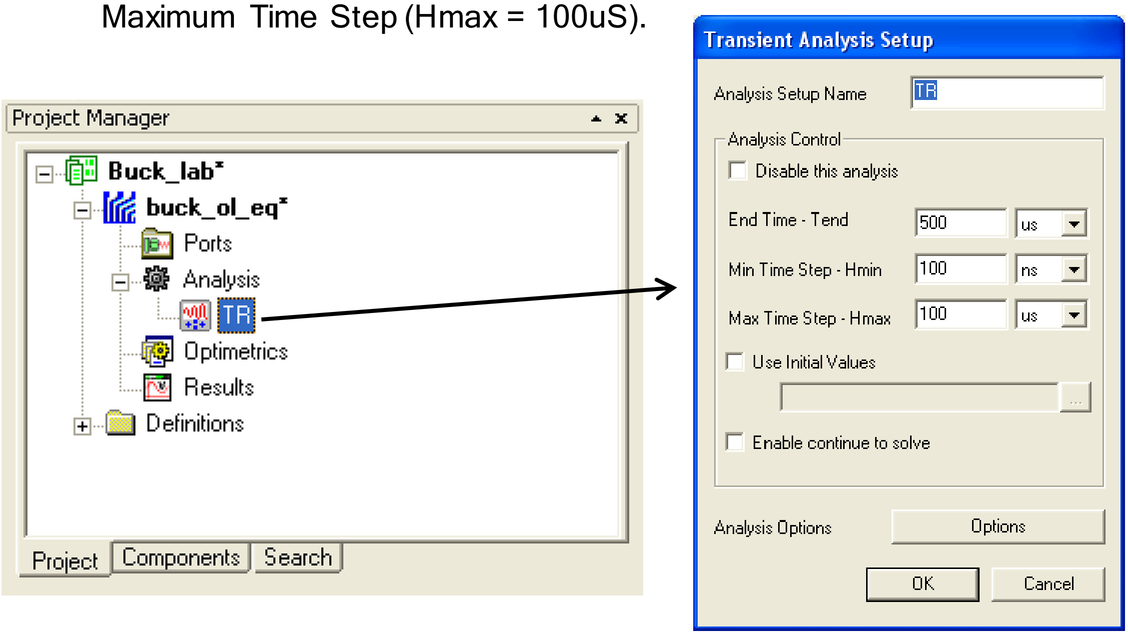
- The following is for the inductor current measurement probe



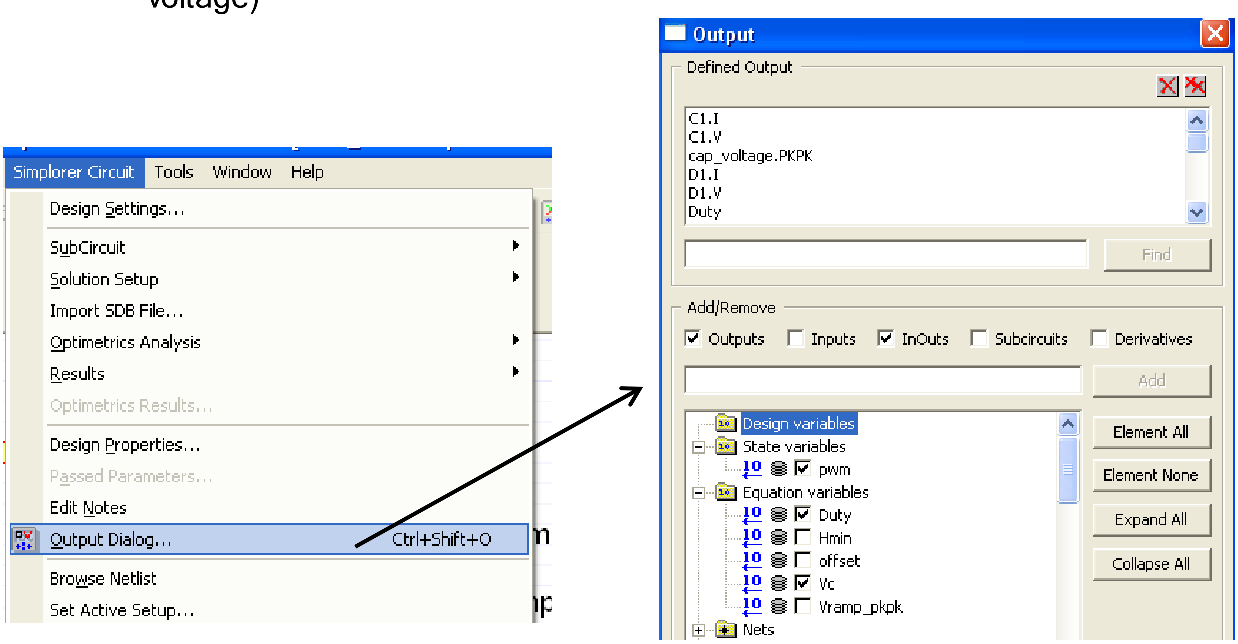
- The following is for the capacitor voltage measurement probe



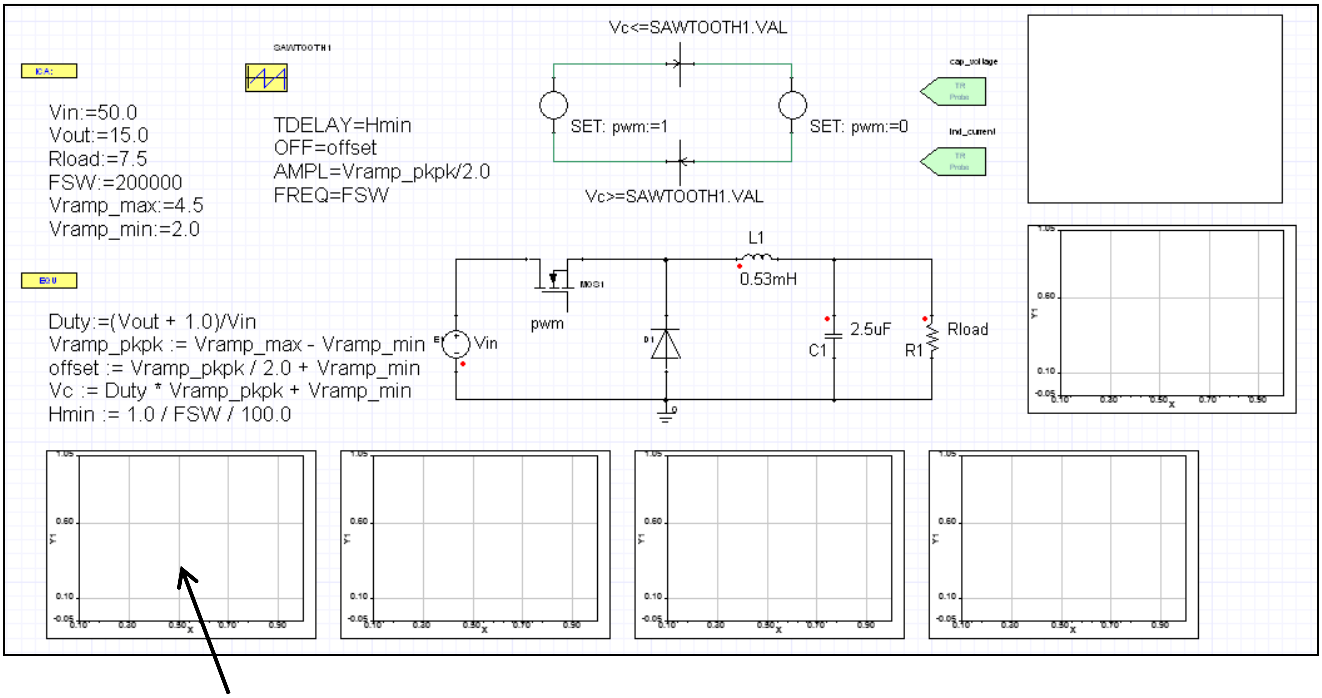
- Set up the Transient Analysis by double clicking the mouse over the "TR" in the "Analysis" section (see below). This will bring up the Setup window and set the End time (Tend = 500uS), the Minimum Time Step (Hmin = 100nS), and the Maximum Time Step (Hmax = 100uS).



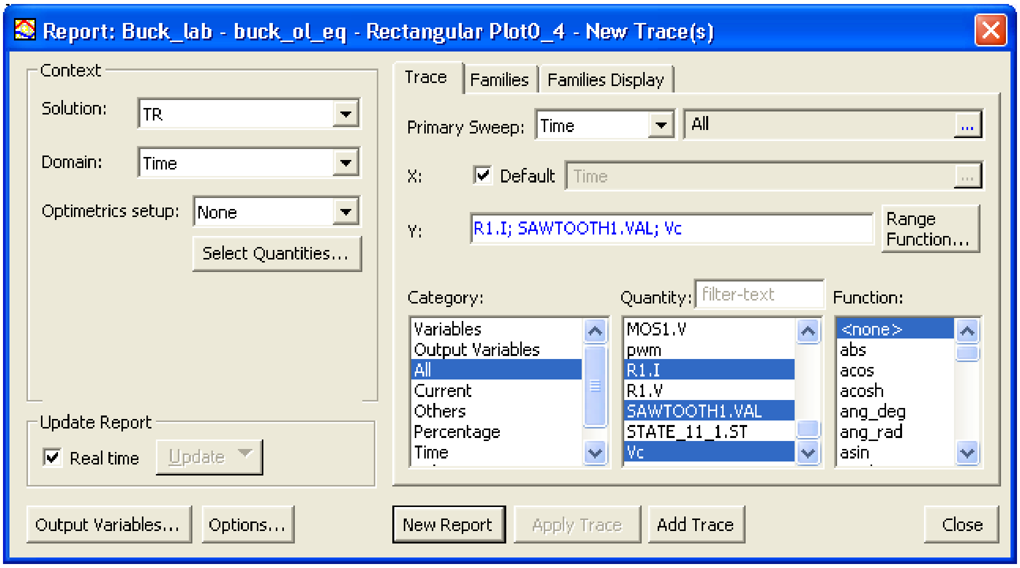
- Check to make sure all desired signals are available for plotting from the results of the transient analysis. Select the menu "Simplorer Circuit -> Output Dialog" . Check the boxes for "pwm" (switch signal), "Duty" (Duty cycle), and "Vc" (control voltage)

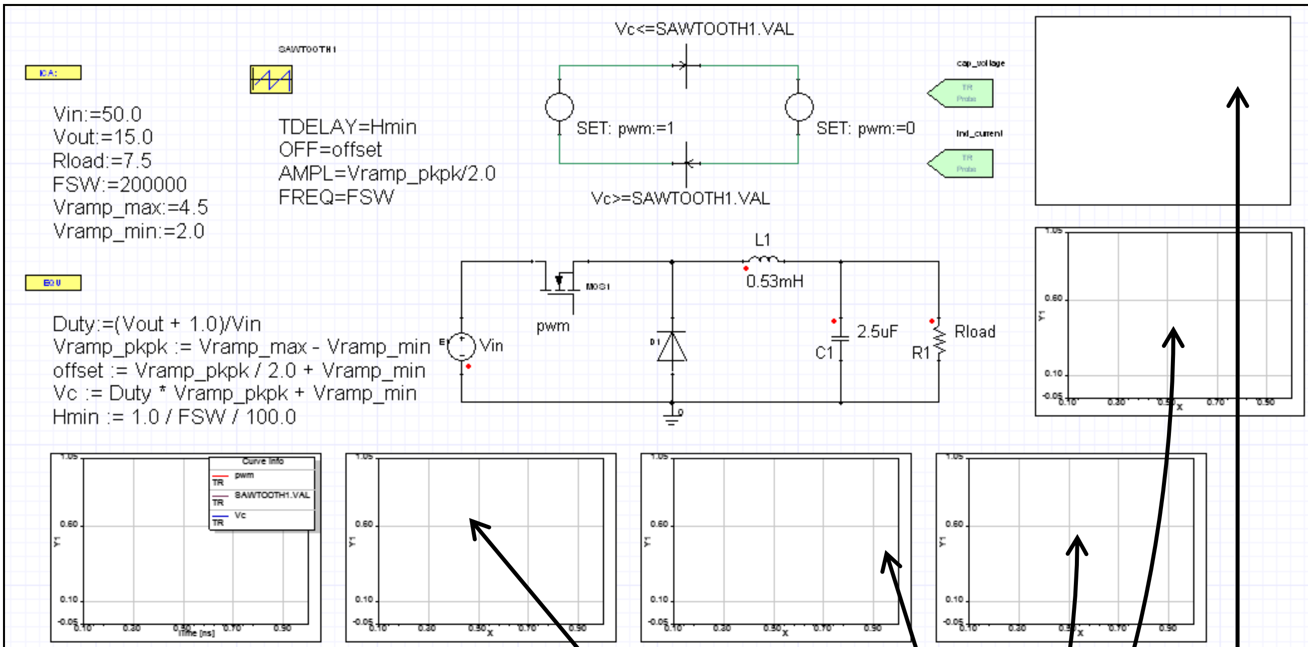


- Zoom out on the schematic and add in a "rectangular plot" via "Draw -> Report", then select it in the schematic, and copy and paste 4 more into the schematic. Add one "Numeric Display" onto the schematic (see below)

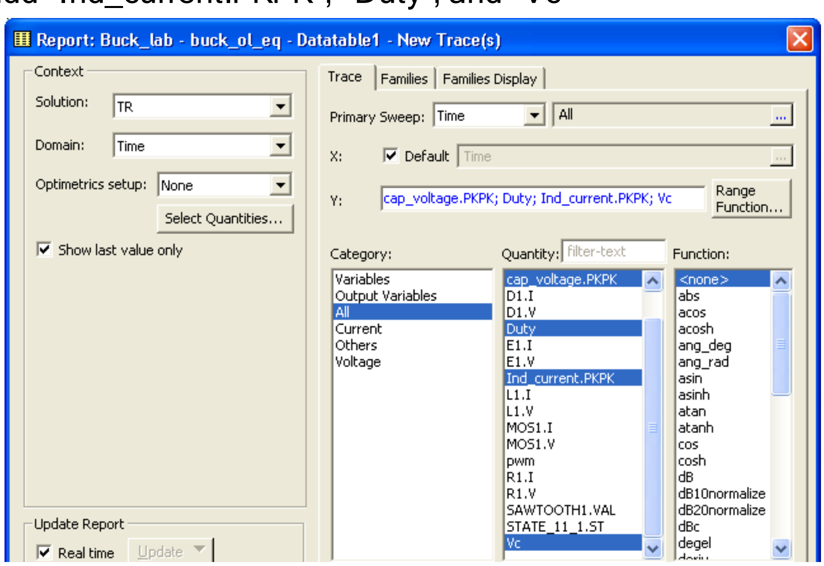


- double click on the first rectangular plot and select the trace "SAWTOOTH1.VAL", then hold down the "ctrl" key and select also "pwm" and "Vc", then select "Add trace", then select "close".

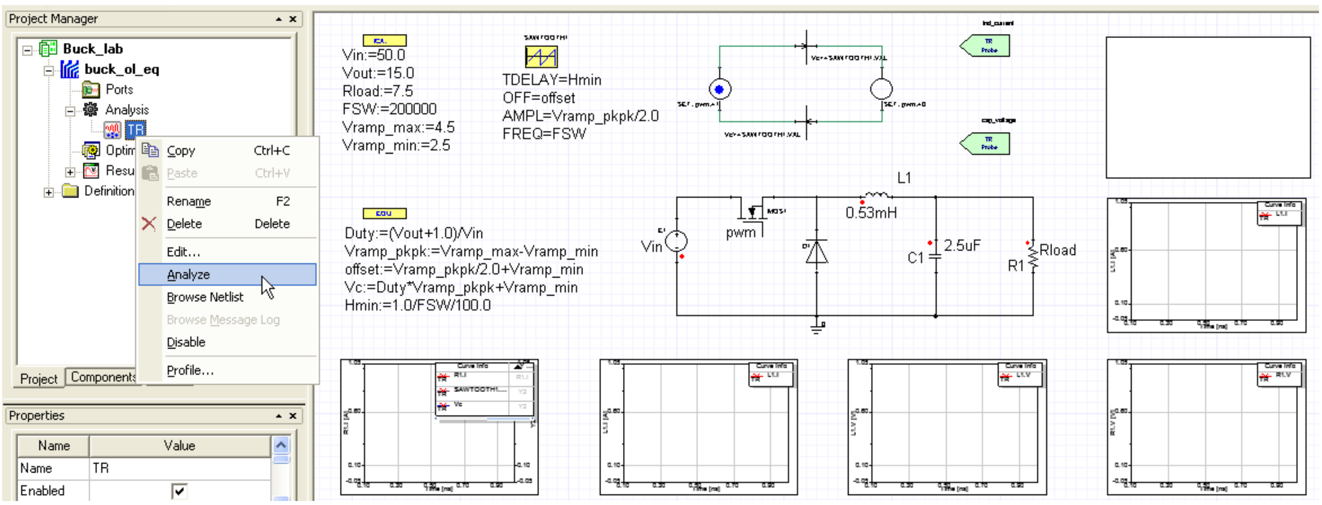




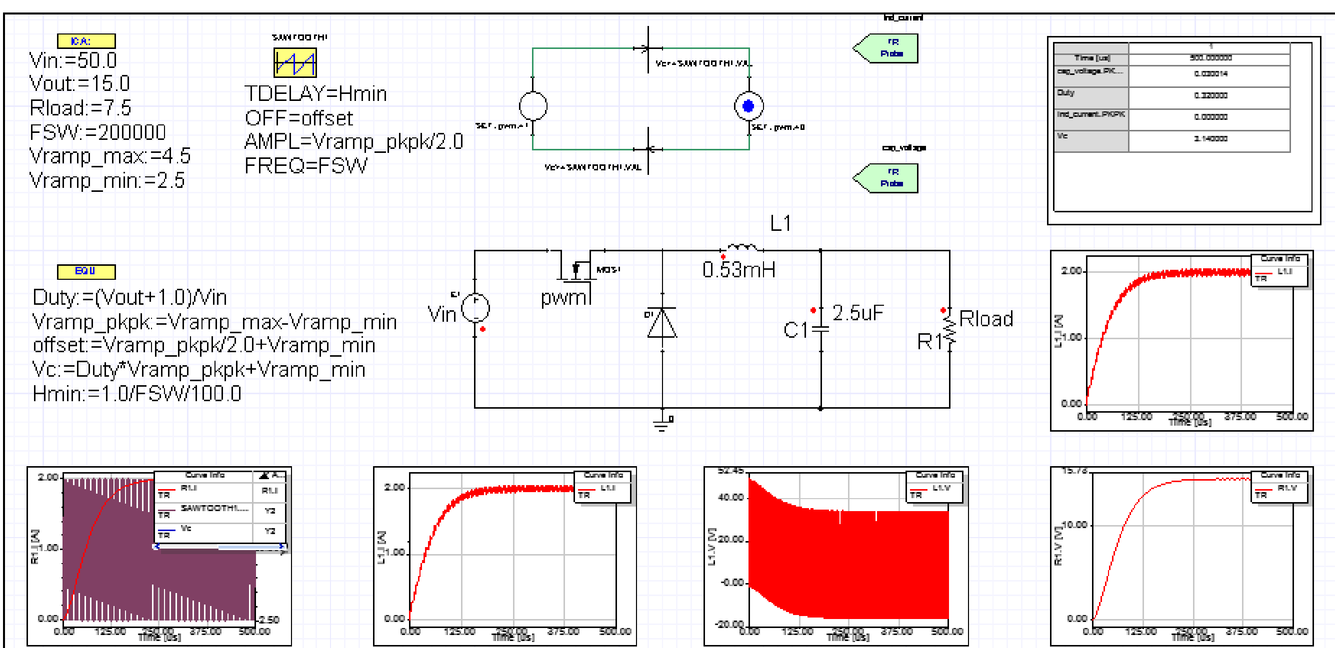
- Double click the mouse on the second "Rectangle Plot" and add in the plot for "L1.I" (Inductor current)
- Repeat for the next plot and add "L1.V" (Inductor Voltage)
- Select the 4th plot and add "R1.V" (Output voltage)
- Select the 5th plot and add "L1.I" again
- Double click on the "Numeric Display" plot (which will display the last value of the chosen waveform in numeric form) and add "Cap_voltage.PKPK", hold the "ctrl" key down and add "Ind_current.PKPK", "Duty", and "Vc"



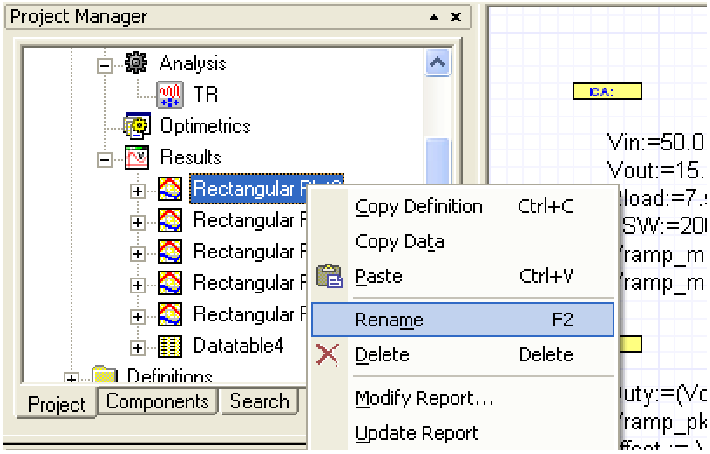
- Save the Design
- Simulate the design by placing the mouse over “TR” (that was defined in the “Analysis” section earlier), then using the right mouse button, bring up the menu then select “Analyze” (see below)



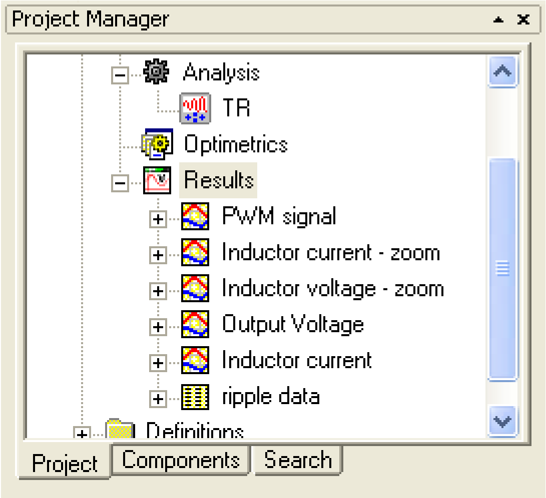
- The results should appear as follows:



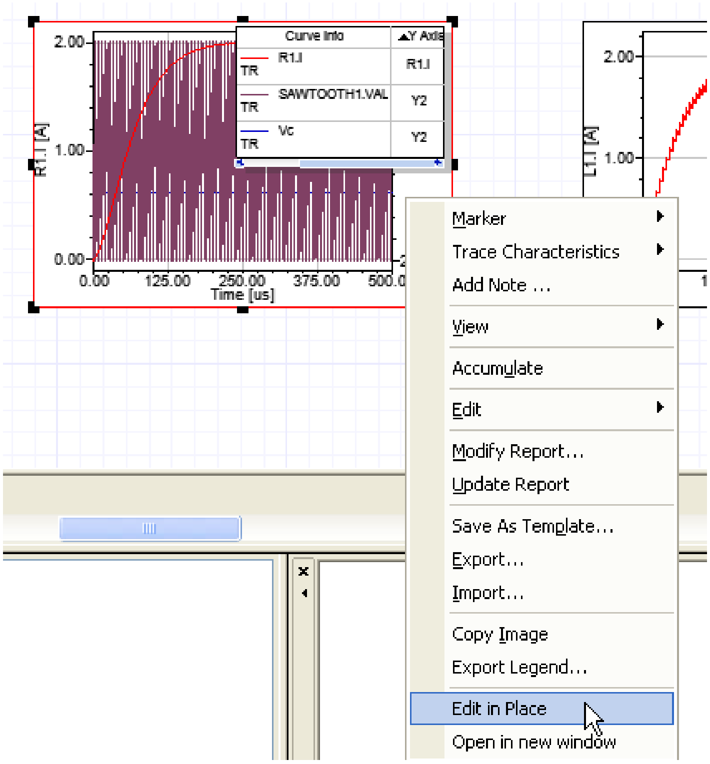
- Rename the plot files in the “Results” section to better reflect their content. Move the mouse over the name, then right mouse button -> Rename



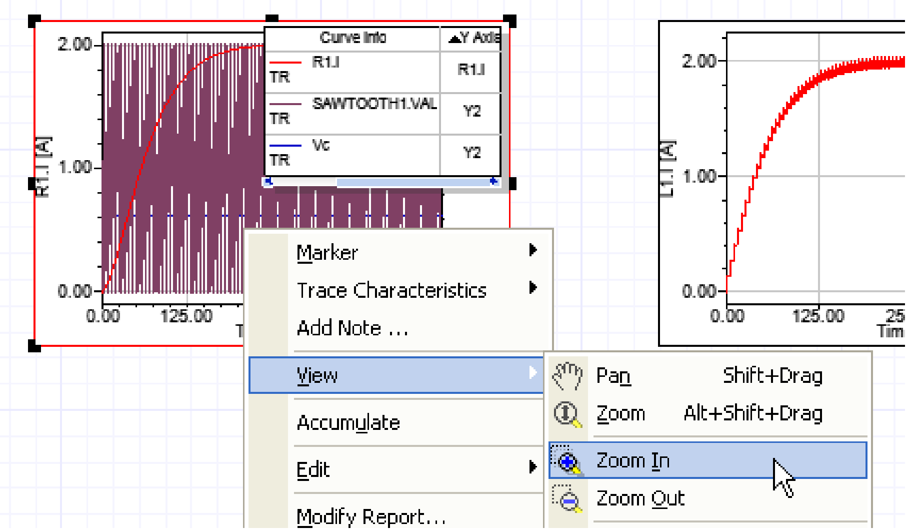
- The following new names have been applied:



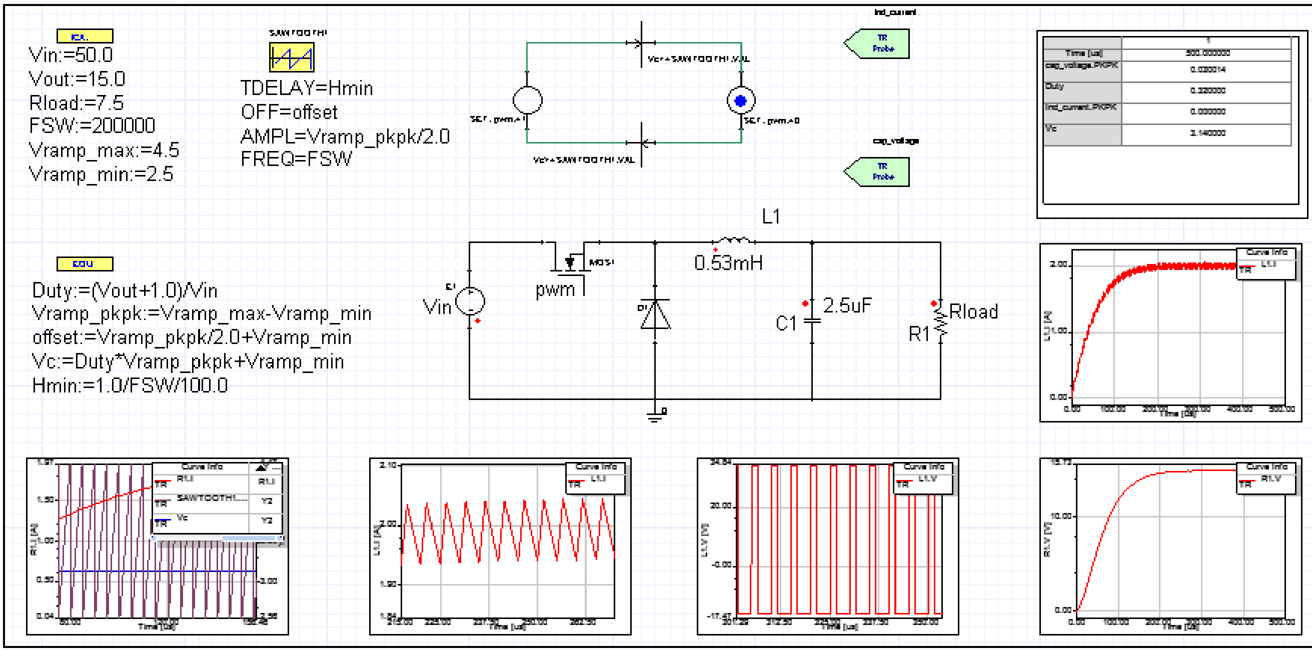
- Edit the plots on the schematic to zoom the waveforms for better representation. Select the first plot on the schematic, then use the right mouse button to bring up the menu, then select "Edit in Place". (Note the edit in place allows such things as moving , re-sizing , or re-defining the legend box as well)



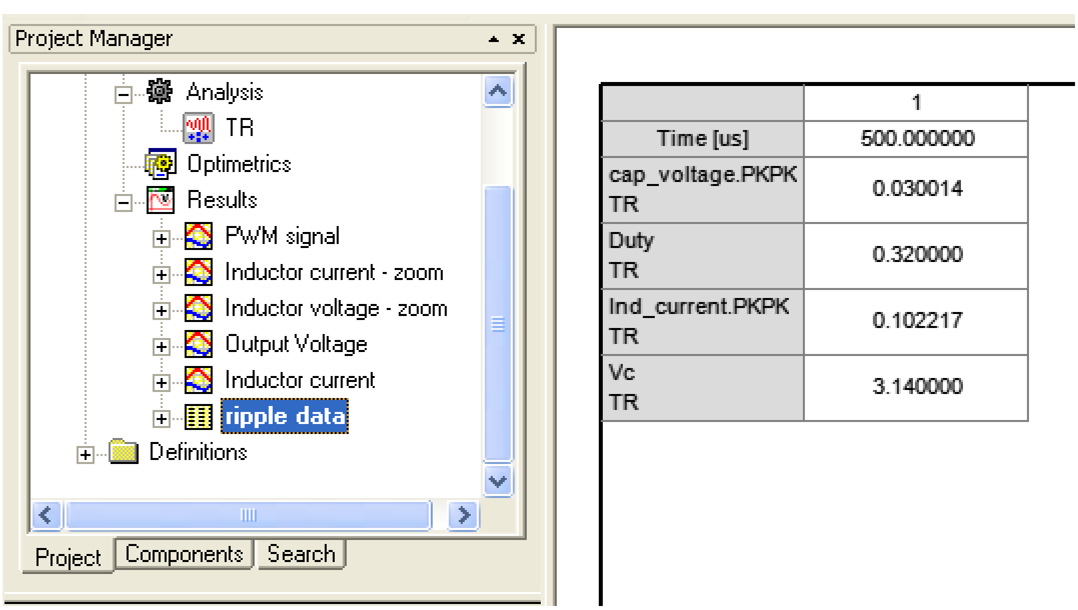
- Move the mouse in the plot area, and again use RMB to invoke the menu, select the "View-> Zoom In", then draw a zoom box within the plot.



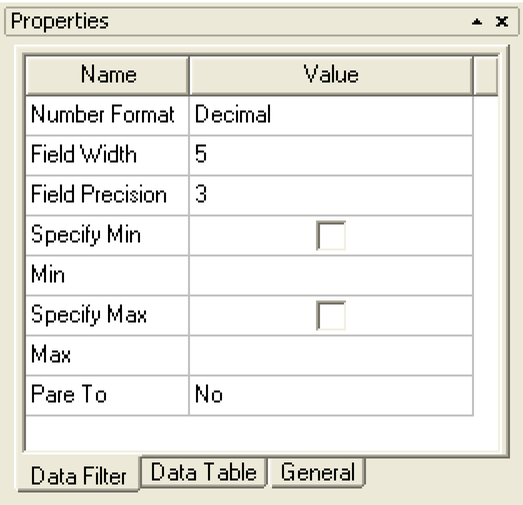
- Note can also zoom in without using the “edit in place” concept. Simply select the plot, then RMB to bring up the menu, then use View -> zoom in. Zoom in on next two plots for the Inductor current and Inductor voltage as shown below:



- Edit the Numeric Display plot (“ripple data”) to resize and reformat the numerical results. In the Project Manager window, double click the mouse over its name “ripple data”. this brings in the report to the larger window.

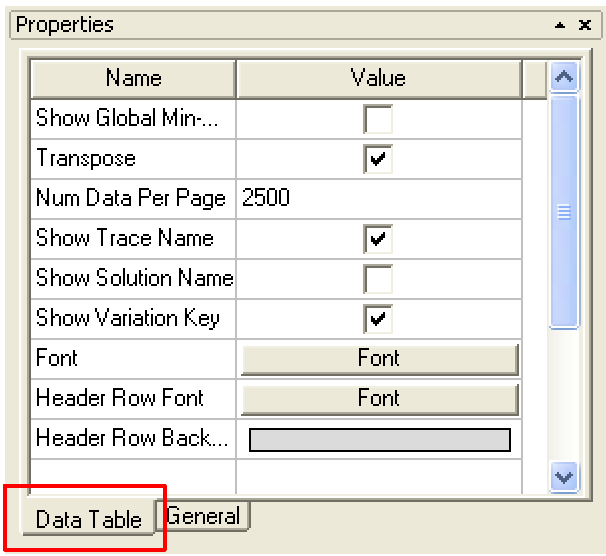


- Use the menu “Edit -> select all” to select all the fields in the DataTable report. Note the “Properties” window on the left side of the simplorer desktop window (see below) . Change the Field Width to 5 and the Field Precision to 3, select “Enter” key. Note this will now show the DataTable as shown below:



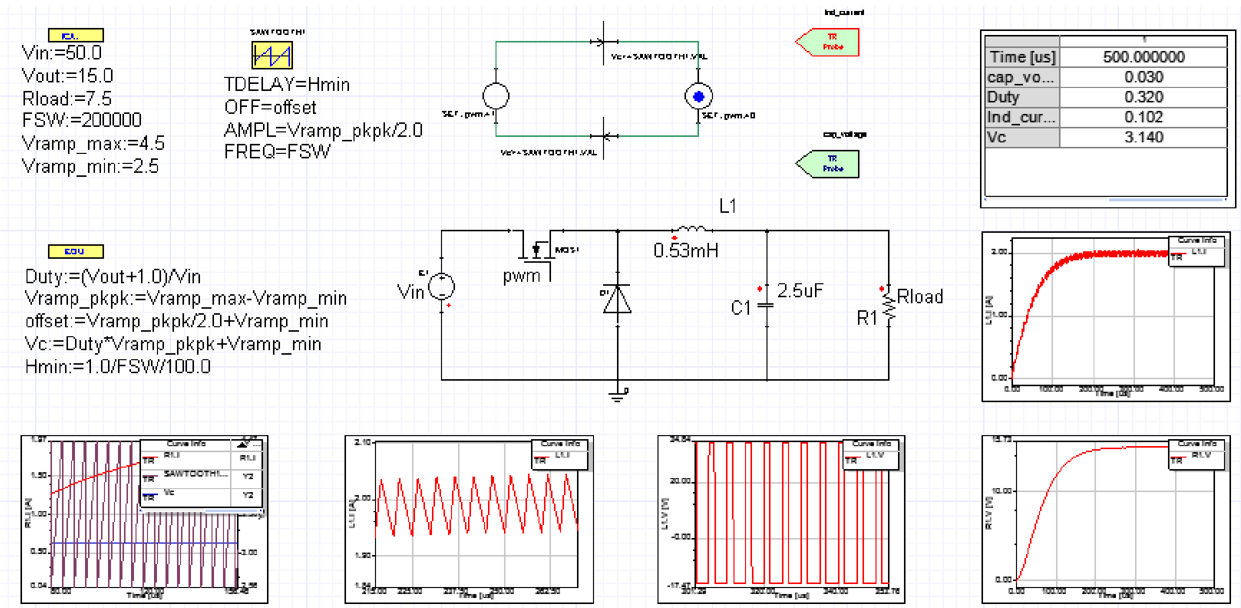
	1
Time [us]	500.000000
cap_voltage.PKPK TR	0.030
Duty TR	0.320
Ind_current.PKPK TR	0.102
Vc TR	3.140

- Select the “Data Table” tab as shown below, and change the font to size 16, and de-select the “Show Solution Name”. The results are shown below:



	1
Time [us]	500.000000
cap_vol...	0.030
Duty	0.320
Ind_curr...	0.102
Vc	3.140

- The final results should look like the following (note this should be the same as the initial view showed at the start of this lab) : (Save the design)



- **NOTE** can change the default property font size that is used in the schematic ahead of time via the menu "Tools -> Options -> Schematic Editor Options"
- **NOTE** another way to permanently change the visual representation of the plots on the schematic is to edit them in the Report window. For example double click the mouse over the name "inductor voltage - zoom" which brings that plot into the larger report window. Select the x-axis and note the properties window to the left, Select the "Scaling" tab to specify desired range (see below). Note the same can be done with the Y axis or other parts of the plot.
- **NOTE** can also change what is displayed on the plot schematic by selecting the plot in schematic, RMB to bring up menu, "View -> Visibility" (see below)