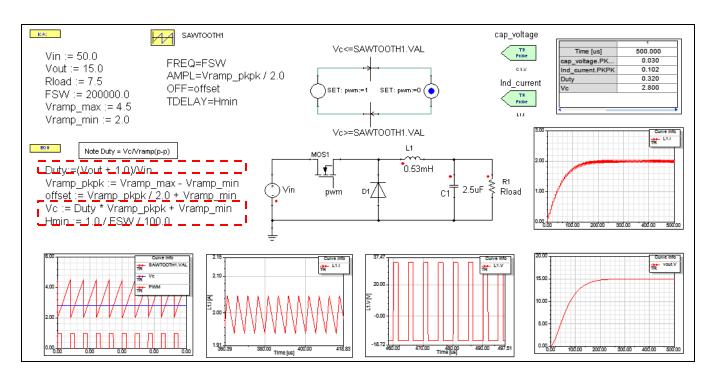


Section One

- 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
 - Most Use of system level MOSFET switch and diode
 - A Displays for waveforms and numerical values



P()1

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)



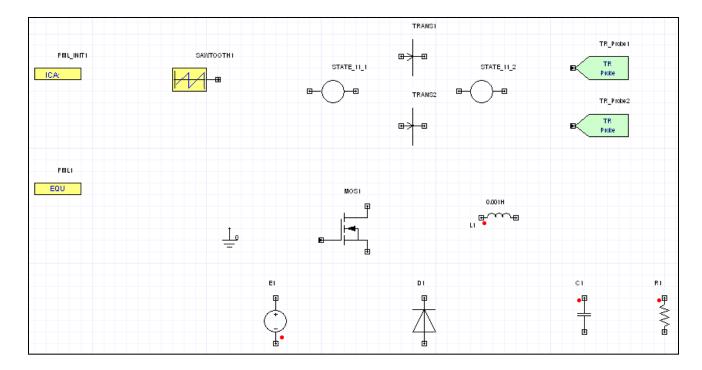
- The ICA initial condition block is found in:
 - Basic Elements/Tools/Equations/ "FML_INIT:Initial Values"
- M The EQU Equation block is found in:
 - Basic Elements/Tools/Equations/ "FML:Equation"
- The sawtooth function block is found in:
 - Basic Elements/Tools/Time Functions/"SAWTOOTH:Sawtooth"
- M 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 pkpk measurements are found in:
 - Basic Elements/Signal Char/ "TR_Probe: TR Probe"

ANSYS[®]

Simplorer v10

P01

- M The Input Voltage uses the "E" voltage source found in:
 - Basic Elements/Circuit/Sources/"E:Voltage Source"
- M The system level MOSFET and Diode are found in:
 - Masic Elements /Circuits/Semiconductors System/
- Mathematical The inductor, capacitor and load resistor are found in:
 - Masic 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.



P01

- 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:

Para	ar	me	ters - FML_INIT1 - Initia	l Va	lues			X
Sta	ate	e	Output / Display					
1	N	ame	FML_INIT1					ihow Name
	P	ara	meters					¥≠€
	Γ		Equation		Information	Show	Include Info	
	ľ		Vin:=50.0			~		
	ľ		Vout:=15.0			~		
	ľ		Rload:=7.5			~		
	ľ		FSW:=200000			~		
	ľ		Vramp_max:=4.5			~		
			Vramp_min:=2.5			~		

- 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.

Vin:=50.0 Vout:=15.0 Rload:=7.5 FSW:=200000 Vramp_max:=4.5 Vramp_min:=2.0

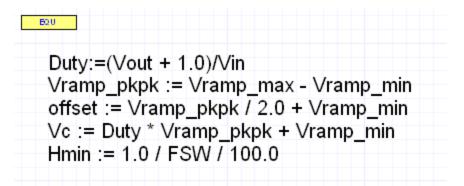
Properties		* X
Name	Value	ι
SchematicID	1	
PropDisplay XY	-460	mil
PropDisplay Angle	0	de <u>c</u>
PropDisplay Font	Arial	
PropDisplay Font Size	12	
PropDisplay Justification	Left	
PropDisplay Location	Bott	
Symbol		>

ANSYS[®]

Simplorer v10

Power Electronics

- Mote 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:



- 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.

Parameters	s - SAWTOOTH1 - Saw-	Tooth		
Parameters	Output / Display			
Name	SAWTOOTH1			Show Name
Paramete Ramp fun				
Amplitude		Phase	0	deg 💌
Frequenc	Y FSW	▼ Offset	offset	
Period	Tend+1	-		
Periodical	No			
Delay	Hmin	•		

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"

Pa	ram	eters - SA	WTOOTH1 - Saw-Tooth					
P	aram	neters Outp	ut / Display					
		Name	Description	Direction	Show Pin	Sweep	SDB	Visibility
	-	Instance	Description	In	Show Pin	Sweep	506	Visionicy
	-			In			-	None
	<u> </u>	Type	<u></u>				-	
		CompDlg	Options	In -			_	None
		Simulator		In				None
		FREQ	Frequency	In				Both 💌
		TPERIO	Period	In				None Name
		AMPL	Amplitude	In				Value
		PHASE	Phase	In				Both Evaluated Value
		PERIO	Periodic	In				Evaluated Both
		OFF	Offset	In				None
		TDELAY	Initial Delay	In				None
		VAL	Value	Out	~		~	None
		dVAL	Derivative of Value	Out				None
			1					·
	<							
							ОК	Cancel



P01

- 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

	IDELAT	Initial Delay	111		J		DOCH	DOCCOM
	VAL	Value	Out	~		~	None	Bottom
	dvoi	Dariustica of Valua	Out				None	Bottom

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

	SAWTOOTH1	TRANSI			
ICA:				TR_Probe 1	
		0 > 0 STATE_11_1	STATE_11_2	E TR Probe	
Vin:=50.0		'	- 0 -	Probe	
Vout:=15.0	TDELAY=Hmin	TRANS2		TR_Probe2	
Rload:=7.5	OFF=offset	• •		TR	
FSW:=200000	AMPL=Vramp_pkpk/2.0	<u>u</u> - u		Probe	
Vramp_max:=4.5	FREQ=FSW				
Vramp_min:=2.0					
		Ŧ	0.001H		
60 U		Most	u eme		
Duty:=(Vout + 1.0)/Vin					
Vramp_pkpk := Vramp	_max - Vramp_min				
offset := Vramp_pkpk	2.0 + Vramp min	El	D1	C1	R1
Vc := Duty * Vramp_pl			*	• <u> </u>	•
Hmin := 1.0 / FSW / 10		\bigcirc \angle	1		-
			Œ		

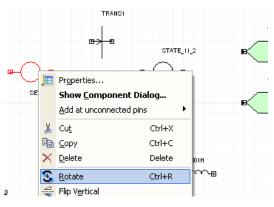


P01

- 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
- M 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

Parameters - STATE_11_	1 - State 11		
State Output / Display			
Name STATE_11_1			🔲 Show Name
Actions	(<mark>×(</mark> *)	Valid for TR C DC	✓ Activate State
Action type Name	e Value	Information	Show
SET pwm	1	Calculation once at the moment of activation	

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





P01

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.

Paramet	ers - STAT	E_11_2 -	State 11			×
State	Output / Disp	blay				
Name		_11_2			Show Name	e
State Actio	Parameters ns		(∱) <u>₹</u>	Valid for	C Activate Stat	:е
	Action type	Name	Value	Information	Show	1
	SET	pwm	0	 Calculation once at the moment of activation	>	

- A 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"

Parameter	rs - TRANS1 - Transition	×
Transition	Output / Display	
Name	TRANS1 Show Name	
Parame	sters	
Priority	5 1 = highest	
Conditi	ion for transition 🔽 Show Condition	
Vc<=	SAWTOOTH1.VAL Edit	
The us	Expression, Value e of equality in logical expressions (<= , == , >=) instead of simple relational operators (<,>), forces the simulator to slow and approach the transition point with maximum accuracy.	

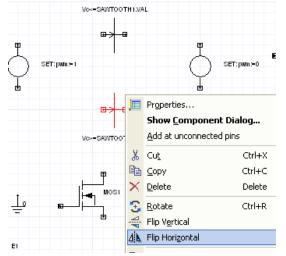


P01

- 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"

Paramet	ters - TRANS2 - Transition	×
Transitio	Output / Display	
Name	e TRANS2 Show Name	
Para	meters	
Prior	ity 5 1 = highest	
Cond	dition for transition 🔽 Show Condition	
Vc>	>=SAWTOOTH1.VAL Edit	
Logic	cal Expression, Value	
	use of equality in logical expressions (<= , == , >=) instead of simple relational operators (<,>), forces the simulator to slow n and approach the transition point with maximum accuracy.	

IMPORTANT, Flip this symbol horizontally on the schematic

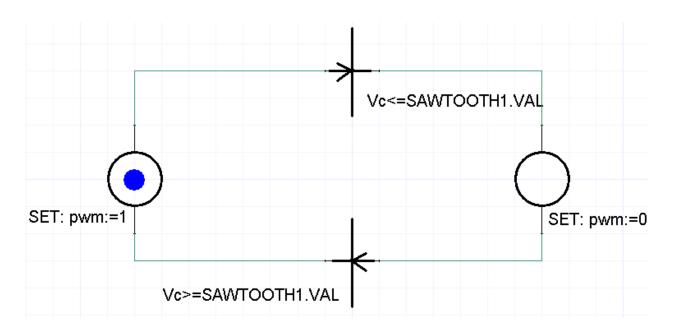




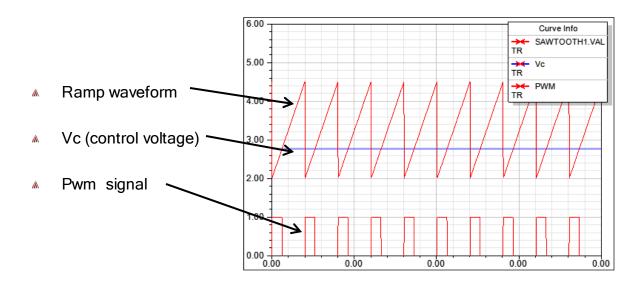
Power Electronics

P01

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.





- Next step is to configure the power stage electronics ٨
- Double click on the voltage source and assign the variable "Vin" as seen below. ٨ Remember this value of "Vin" comes from the ICA initial condition block. Select the "Spice compatible" box (for spice compatible sign convention)

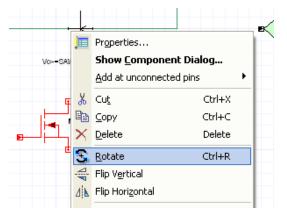
Parameters - E1 - Voltage	e Source		×
Parameters AC - Parameters	s Output / Display		
Name E1			🔽 Show Name
Parameters			
EMF Value	Vin V	•	🗌 Use Pin
	Value, Variable, Expression		C use
C Time Controlled	Sine	-	
Spice compatible			
🔽 Partial Derivation			
Amplitudo 224	U Desce	1	

Select the "Output/Display" tab and using the left mouse button, make the EMF ٨ Value visible as shown below, Select "OK"

an	neters AC - Par	rameters Output /	Display					
	Name	Description	Direction	Show Pin	Sweep	SDB	Visibility	Location
	InstanceName		In				Value	Тор
	АСТуре		In				None	Bottom
	Туре		In				None	Bottom
	CompDlg	Options	In				None	Bottom
	SimulatorModel		In				None	Bottom
	AC_IM	Imaginary Part	In				None	Bottom
	EMF	EMF Value	In		Г		None 💌	Bottom
	FREQ	Frequency	In				None	pm
	TPERIO	Period	In	Г	Γ		Name Value	pm
	AMPL	Amplitude	In				Both	pm
_	PHASE	Phase	In		Γ		Evaluated Value Evaluated Both	pm
-		Deviedie	T-				Nees	Dathan



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.

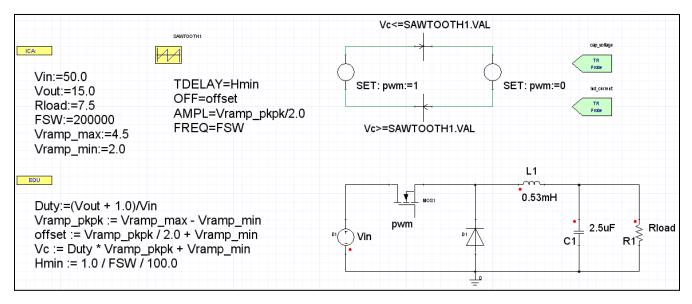
pwm	Use Pin
Value, Variable, Expression	
	12

- 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)



P01

- 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"

Pa	ram	eters - TR_P	robe2 - TR Prot)e						
P	aram	neters Output /	Display							
		Name	Description	Direction	Show Pin	Sweep	SDB	Visibility	Location	
		INPUT	Input signal	In	~		\Box	None	Bottom	
		TSTART	Time to start mon	In				None	Bottom	

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)



M The following is for the inductor current measurement probe

Paramet	ers - Ind_curre	nt - TR Probe			×
Paramete	ers Output / Displa	y]			
Name	Ind_current			Show	
	, -			Je onow	
Parar	meters				
	Name	Value	Units	Description	
	INPUT	L1.I	А	Input signal	
	TSTART	250	us	Time to start monitoring	
	TSTOP	255	us	Time to stop monitoring	
	FREQ	200	kHz	Expected fundamental frequency (0 to skip)	
	METHOD	0		0=DFT, 1=Least Squares to calculate fundamental RMS	
	ORDER	11		Harmonic order of DFT, or N for LSQ	
					-
Defa	ult Outputs				
	гмах 🗆 мах				
	NTEG 📃 PKA				
	FF 🗌 CF RMS 🗐 ME/		,	ISAC	
	RMS 🗍 MEA	AN 🗆 THI	D 🗆 FUI	ND	-
				OK Canc	el

M The following is for the capacitor voltage measurement probe

Para	met	ers - cap_voltag	ge - TR Probe			×
Par	ameti	ers Output / Displa	y			
Na	ame	cap_voltage			Show	,
_	D	meters				
		Name	Value	Units	Description	r
		INPUT	C1.V	V	Input signal	
		TSTART	250	us	Time to start monitoring	
		TSTOP	255	us	Time to stop monitoring	
		FREQ	200	kHz	Expected fundamental frequency (0 to skip)	
		METHOD	0		0=DFT, 1=Least Squares to calculate fundamental RMS	
		ORDER	11		Harmonic order of DFT, or N for LSQ	
	I					
	D-6-					
Γ		ult Outputs MAX 🗆 MAX	к ⊏тм	IN E MIN	4	
		NTEG PKA				
	E F	F 🗌 CF	🗖 BIF	PLE 🗌 RM	SAC	
	F	RMS 🗆 MEA	AN 🗆 TH	D 🗌 FUI	ND	
					OK Can	cel



Set up the Transient Analysis by double clicking the mouse over the "TR" in the ٨ "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). Transiont Analysis Sotun

	manoione maryon oorap			
Project Manager	Analysis Setup Name Analysis Control Disable this analysis End Time - Tend 500 us Min Time Step - Hmin 100 ns Max Time Step - Hmax 100 us Lise Initial Values Lise Initial Values Lise Initial Values Analysis Options Options			
Project Components Search	Analysis Options Options Oct			

Check to make sure all desired signals are available for plotting from the results ٨ 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) 🔲 Output

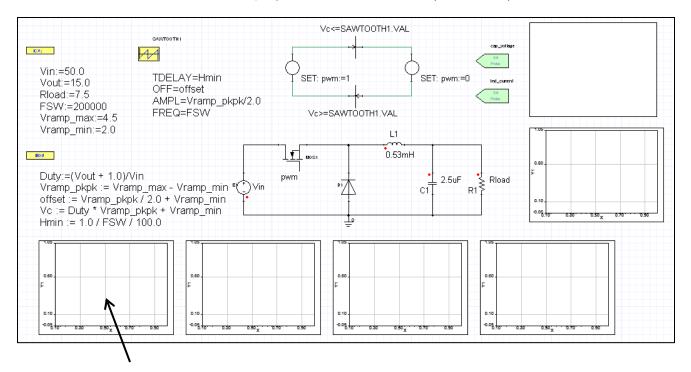
		C Defined Output	
			\times \times
		C1.I	<u>~</u>
	· · · ·	C1.V cap_voltage.PKPK	
Sim	olorer Circuit Tools Window Help	D1.I	
	Design Settings	D1.V Duty	~
	SubCircuit		Find
	Solution Setup		
	Import SDB File	Add/Remove	
	Optimetrics Analysis	🔽 Outputs 🔲 Inputs 🔽 InOuts 🗐 Subcircuits 🗍	Derivatives
	Results		Add
	Optimetrics Results		
	Design Properties	Design variables State variables	Element All
	Passed Parameters	- 10 ⊜ 🔽 pwm □ 🔤 Equation variables	Element None
	Edit Notes	10 € V Duty	Expand All
!!!	Qutput Dialog Ctrl+Shift+O n	20 🛢 🗖 Hmin -	
	Browse Netlist		Collapse All
	Set Active Setup 10		
		😟 😥 Nets	

×



P01

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)

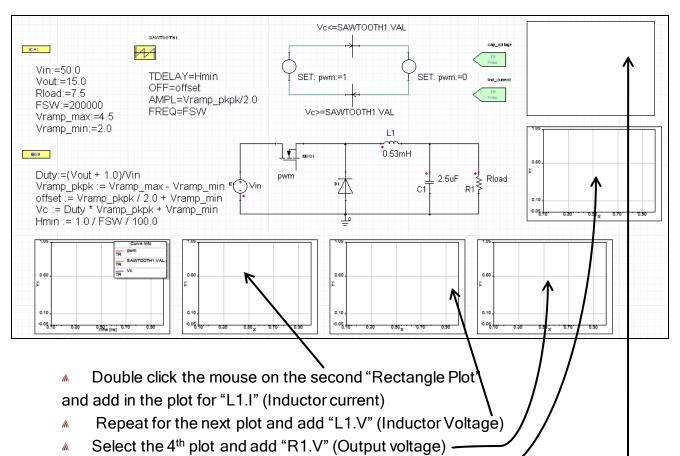


double click on the first rectangular plot and select the trace "SAWTOOTH1.VAL", then hold down the "ctrl" key and select also "pwm" and "Vc", then select "Add trace", then select "close".

🔀 Report: Buck_lab - buck_ol_eq - Re	ctangular Plot0_4 - New Trace(s)
Context	Trace Families Families Display
Solution: TR	Primary Sweep: Time All
Domain: Time 💌	X: 🔽 Default Time
Optimetrics setup: None	Range Range
Select Quantities	, Policion
	Category: Quantity: filter-text Function:
	Variables MO51.V Annu Annu Annu Annu Annu Annu Annu Ann
	All R1.I acos
	Current R1.V acosh
Update Report	Others SAWTOOTH1.VAL ang_deg Percentage STATE_11_1.ST ang_rad
Real time Update	Time Vc value asin V
Output Variables Options	New Report Apply Trace Add Trace Close



Power Electronics

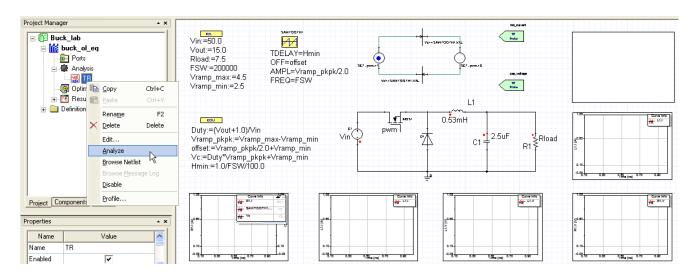


- Select the 5th plot and add "L1.I" again
- Double click on the "Numeric Display" plot (which will display the last value of the chosen waveform in numeric form) and add "Cap_voltage.PKPK", hold the "ctrl" key down and add "Ind_current.PKPK", "Duty", and "Vc"

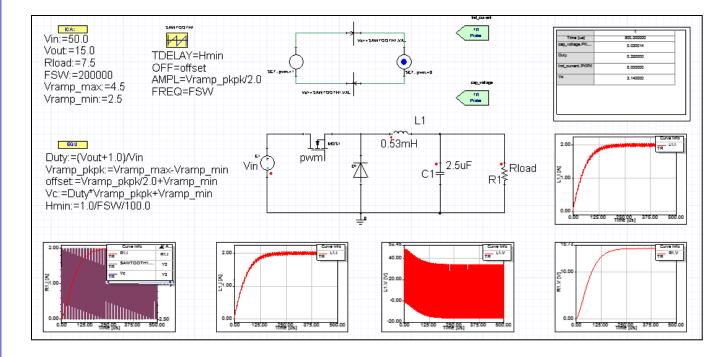
III Report: Buck_lab - buck_ol_eq - Da	itatable1 - New Trace(s)	×
Context Solution: TR Domain: Time Optimetrics setup: None	Trace Families Families Display Primary Sweep: Time All X: Image: Default Time Y: cap_voltage.PKPK; Duty; Ind_current.PKPK; Vc Range Puncti	
Select Quantities	Category: Quantity: filter-text Function: Variables OL.V Output Variables DI.I All DL.V Current Others EI.I Voltage EI.V Voltage EI.V MOS1.V MOS1.V MOS1.V Cos cos acos	
Update Report	SAWTOOTH1.VAL STATE_11_1.ST Vc v	ze



- ▲ 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)



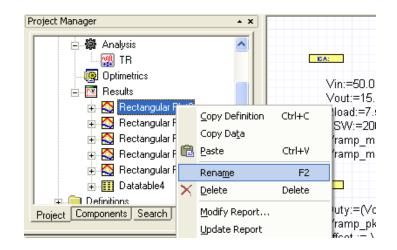
The results should appear as follows:



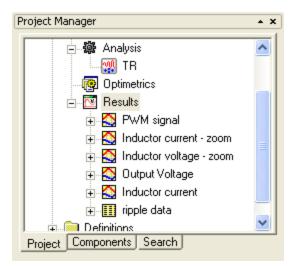


P01

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

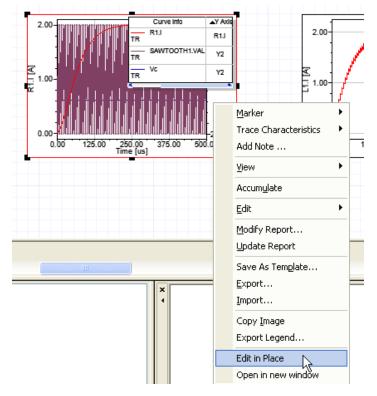


The following new names have been applied:

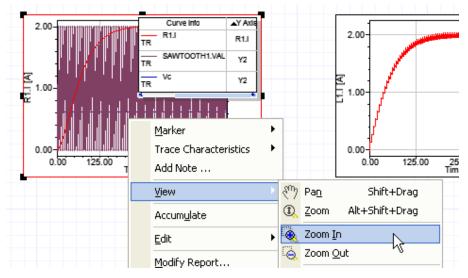




Edit the plots on the schematic to zoom the waveforms for better representation. Select the first plot on the schematic, then use the right mouse button to bring up the menu, then select "Edit in Place". (Note the edit in place allows such things as moving, re-sizing, or re-defining the legend box as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



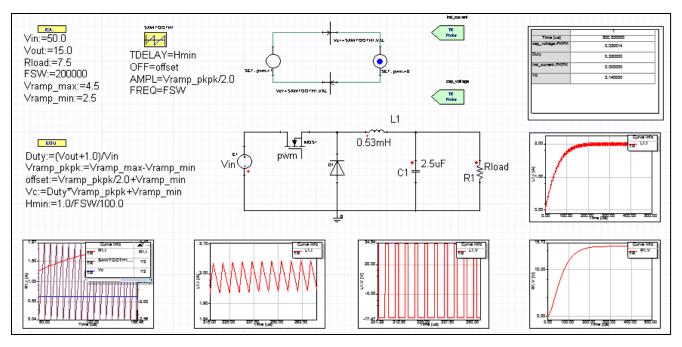
Move the mouse in the plot area, and again use RMB to invoke the menu, select the "View-> Zoom In", then draw a zoom box within the plot.



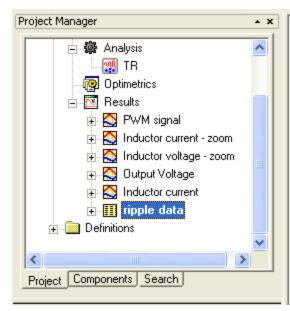


Power Electronics

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.



1
500.000000
0.030014
0.320000
0.102217
3.140000



P01

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:

Name	Value
Number Format	Decimal
Field Width	5
Field Precision	3
Specify Min	
Min	
Specify Max	
Max	
Pare To	No
	1

	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

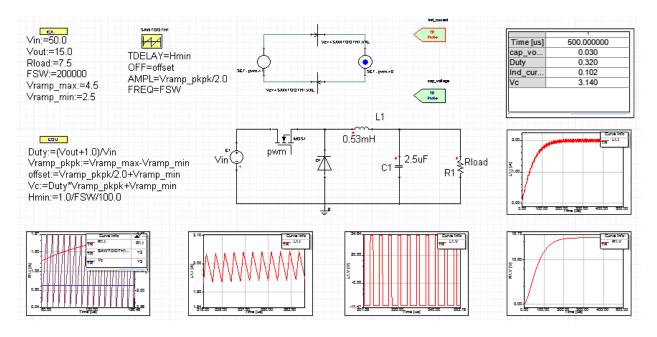
Select the "Data Table" tab as shown below, and change the font to size 16, and de-select the "Show Solution Name". The results are shown below:

Name	Value	1
Show Global Min		
Transpose	~	
Num Data Per Page	2500	
Show Trace Name	~	
Show Solution Name		
Show Variation Key	~	
Font	Font	
Header Row Font	Font	
Header Row Back		
		~

	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)

