INDIAN INSTITUTE OF SCIENCE Department of Electrical Communication Engineering E3-238 Lab Experiments

Lab 0: Introduction to Cadence¹

Contents

1.	Introduction	1
2.	Cadence Tutorial	1
2.1	. Cadence Setup and Launch	1
2.2	. Cadence overview	2
2.3	DC simulation - Resistive divider	8
2.4	. AC and transient simulation - RC low-pass	12

1.Introduction

This lab is a tutorial on Cadence Virtuoso, which is the simulation tool we will use for the rest of the semester. The official program name is Virtuoso, but the common name among users is just Cadence. We will use the name Cadence in this class.

2. Cadence Tutorial

2.1. Cadence Setup and Launch

- > Follow the steps as shared in Class' Teams Group.
- Make sure your system is connected to IISc network, either directly or via <u>IISc VPN</u>.
- > Open Terminal & type ./icl and hit Enter.

¹This document is a modified version of <u>https://inst.eecs.berkeley.edu/~ee105/fa17/labs/Lab0.pdf</u>

2.2. Cadence overview

After opening Cadence, you'll see the main window:

Ē	Virtuoso® 6.1.7-64b - Log: /home/ff/ee105/CDS.log	_ 🗆 ×
File	<u>T</u> ools <u>O</u> ptions <u>H</u> elp	cādence
' <mark>cds</mark> (peE (peE	but was defined in libFile '/share/instsww/cadence/IC.06.17.703/share/cdssetup/dfII/cds.lib' for Lib DefTechLib'. ditProp) no properties found for 'cadence_files' ditProp) viewing directory attributes.	
< 8	IIII	
mou	se L: M:	R:
1 >		

Go to Tools->Library Manager, it should open the following window:

•	Library Manager: WorkArea: /home/ff/ee105/cadence_files	_ = ×
<u>File Edit View D</u> esign Manager <u>H</u> elp		cādence
Show Categories Show Files Library Cadence_files ahdLib analogLib cadence_files cc8bCeTEchLib ee105_components rfExamples rfLib		View A Lock Size
Messages		
Log file is "/home/ff/ee105/cadence_files/ Warning: The directory: Yshare/instswu/c but was defined in IbFle', Warning: The directory: Yshare/instswu/c but was defined in IbFle'	libManager.log'. adence/IC.06.17.703/tools.lnx86/dfli/etc/cdslib/artist/functionaf does not exist /share/instsww/caderce/IC.06.17.703/tools.inx86/dfli/etc/cdsDotLibs/artist/cds.lib' for Lib 'funct adence/IC.06.17.703/tools.lnx86/dfli/etc/cdsDefTechLib' does not exist /share/instsww/cadence/IC.06.17.703/share/cdssetup/dfli/cds.lib' for Lib 'cdsDefTechLib'.	ionaf.
		Lib: cadence_files Free: 1.18T

The hierarchy in Cadence is:

Library (left side) -> Cell (middle) -> View (right).

A library contains multiple cells, and each cell contains multiple views.

The *libraries* that we will use in this tutorial are:

- analogLib the basic analog components (resistors, capacitors, voltage and current sources, etc)
- **umc65ll** the actual components that we will use in the lab (transistors, diodes, opamps, ...). We won't use them in this tutorial.
- **lab0** your designs

The *views* that we will use for each cell are:

- schematic the actual circuit, the components and interconnections
- **spectre** the simulation setup
- symbol the appearance of the cell in another schematic view

Creating a new Library

To create a new library, go to the library manager and click File \rightarrow New \rightarrow Library. A new window will pop up. Go into "**mylibraries**" folder & t\ype "**lab0**" in the name field and press OK.

		New Libra	iry			,
Library Name lab0	I		•		ei	
Computer	Name	ni <u>rityiotaries</u>		Size	Туре	Da
File type Directo	ories					
Use NONE Use No DM						
Compression enab	led	ОК	Apply	Can	icel	Help

At this point, Cadence will prompt you for something called a Technology File. The technology file is collection of information and libraries that define the layers and devices available for a given process technology.



For this class, we will not use any technology file. Therefore, go ahead and choose 'Do not need process information'.

Creating a new schematic

To create a new schematic design:

Click on the lab0 library in the Library Manager.

File -> New -> Cell View

A new window pops up, but it may be at the background:

ile Edit View Design Manager	Help	cādence
<u>N</u> ew	🔟 Library	
Dpen Ctrl+O	<u>C</u> ell View	
Open (Read-Only) Ctrl+R	C <u>a</u> tegory	View
₿ Open Wit <u>h</u>		
Load Defaults	-	
<u>S</u> ave Defaults		
Open Shell Window Ctrl+P		
Exit Ctrl+X		
Messages		
In /snare/instsww/cadence/iC61 Or remove or comment out DEFINE in /home/aa/ugrad/sameetr/ee14	5.1517tools.inx867atii/etc70 analogLib 0_ta/gpdk/90nm/gpdk090	cosDotLips/anist/cos
to suppress this warning message.		100

This is a general tip in Cadence - if you expect a window to open and it's not there, check the taskbar!

	New File ×
File	
Library	cadence_files
Cell	tutorial
View	schematic
Туре	schematic 🔽
Application	
Open with	S chematics L
Always use this	application for this type of file
Library path file	
/home/ff/ee105/	/cadence_files/cds.lib
	OK Cancel Help

We will give it a name "tutorial", the type should be "schematic". Note that you can make cells in any available library by choosing proper one from **'Library'** (if you have permissions to edit). Click OK. The following window will open:

Next License	×
WARNING (icLic-201) Failed to check out license Virtuoso_Schematic_Editor_L ("95100") to run Schematics L bu Would you like to check out license Virtuoso_Schematic_Editor_XL ("95115") instead?	ecause of status code -18.

Launch <u>File</u> Edit	View <u>C</u> reate	e Check	Opti	ons	Windo	w Hel	р															cā	d e n	C
	. I & C		×	0	T/	9	2	12	T	• •	th	- 3	•	Q	Q	0		16	1	1	abc			
0-0-0	Basic					-		2	t's	203	-13	T			a, -Se	arch		-	-					
Navigator	78×										-		•	1 I					-					
Schematic tutorial																								
OBJECTS																								
All	-																							
Instances	-																							
Nets	Þ																							
Pins	E.																							
Nets and Pins	Þ																							
GROUPS																								
Cells																								
Types																								
, ypcs																								
+ -																								
Deserves to Editors	2.5.4																							
Property Editor	100																							
_	_																							
																	12 12							
mouse L: schAddSeled	:tPt()									M:														R
(3)																						Cm	d: Sel	:0

Click "Always" to avoid getting this message later. The schematic window will open:

This is the main window where we'll draw our circuit. Generally, we won't use the menus, but keyboard shortcuts.

Adding components

To add an element, click "i". The following window will appear:

	Add Instance	×
Library	Browse	
Cell		
View	symbol	
Names		
⊻ Add Wi	re Stubs at: all terminals • registered terminals only	
Array	Rows 1 Columns 1	
	🕰 Rotate 🛛 🕼 Sideways 🛛 🚭 Upside Down	
	Hide Cancel Defaults Help	

You can type the library, cell and view names, or click Browse:

Select "analogLib" library, "res" cell and "symbol" view. Another window will open:

ца П	Library Browser - Add Inst	tance _	×	A	dd Instance	×
Show Categories			Library	analogLib	Browse	4
Library	Cell	View	Cell	res		
analogLib	res	symbol	View	symbol	-	
a hdl Lib	psin	View 🗠 Lock Size	Names			
analogLib	psoi	auCdi	19k			_
cdsDefTechLb	otft	auLVS	19k 🗹 Add V	Vire Stubs at:		
ee105_components	pvccs	spectre	194	🔾 all terminals 🤅	registered terminals only	
rfExamples	pvccs2	symbol	19k Array	Dour	1 Columns 1	
rfLib	pvccs3	symbol_xform	13k	Rows		
	pvccsp			A Rotate	🛯 Sideways 🛛 🚭 Upside Down	
	pvcvs					- 18
	pvcvs3		Model na	ame		
	pvcvsp		Resistance	:e	20K Ohms	
	rcwireload		Longth			
	res		Length			
	scasubckt		Width			
	sccvs		Multiplier	r		
	schottky		Carla fast			
	scr		Scale laci	tor		
	simulinkCoupler		Temp ris	e from ambient		
	spitswitch	\sim	Tempera	ture coefficient 1		
	sp3tswitch		Tempera	ture coefficient 2		
0	J.J the date		Resistanc	e Form		
		Lib: a nalogLib Free: 2	5.22G	poice?		
Close	Filters	Display He	P Comparison	- maradi		
			- Capacitar	nce		
			Alias for l	Lin. temp. co.		
					Hide Cancel Defaults	Help

Here you specify the parameters of the component. A resistor has a single parameter (resistance), change it to $20k\Omega$.

In Cadence you don't have to write the units (Ohms, volts, etc.). For the resistance, type 20k and hit Tab. The Ohms will be automatically completed. The useful prefixes in Cadence are single letters: p - pico, n - nano, u - micro, m - milli, k - kilo, M - mega, G - giga.

Click on the schematic window to place the resistor. The useful components in the analogLib library are:

res	Resistor
cap	Capacitor
gnd	Ground
vdc/idc	DC voltage/current source
vsin/isin	Sinusoidal voltage/current source
vpulse/ipulse	Square-wave voltage/current source
iprobe	Current meter

Now add another resistor of $10k\Omega$. Click "Rotate" to make it horizontal and place it on the schematic:



Your schematic should look like this:



Adding wires and labels

To connect the resistors with a wire, click "w". Click on the first terminal to connect, and then on the



To create a wire label, click "l" (lowercase L). Type out and click on the wire. Click Esc. Now you have the following sementic: virtuoso@schematic Editor L Editing: cadence_files tutorial schematic*



Labels can be used to connect nodes. If you want to connect two nodes in your circuit, you can give them the same label, without connecting them with a wire. It is usually useful for large circuits, to reduce the number of wires. Labels are also useful for output expressions, as we will see later.

Other useful shortcuts

- **Components** click on the desired component, then click:
 - **c** copy component
 - \circ **m** move component (preserves the wire connections)
 - Shift+M move component (without the wire connections)
 - o q edit component properties (same window as the add component window)
- **f** fits the circuit to fit the screen
- mouse scroll zoom in and out

- z selects area to zoom
- Shift+X check and save. Check that all nodes are connected properly. If you have errors, you have to fix them to simulate the circuit. You can run simulations if you have warnings. Pay attention to the warnings, usually they indicate a problem in your circuit, like unconnected nodes.

2.3. DC simulation - Resistive divider

Add a DC voltage source and grounds to create the resistive divider circuit shown in Figure 1.



Figure 1: Resistor circuit to build

You should get the following schematic:



Click Shift+X to check and save your schematic.

To open the simulation window, click Launch -> ADE L. You will see the following window:





The ADE window will open:



The Analysis box - specifying the simulation type

In the Analysis box: right click -> Edit.

Here we select the different simulation types for our schematic.

The useful simulations in our class are:

- **dc** DC simulation. Only DC sources are used, and the results are DC voltages and DC currents. This is in general a non-linear analysis (unless we only have linear components, like in our case).
- **ac** AC simulation. This is a linear phasor analysis of the circuit. The simulation result is a phasor (magnitude and phase) of the voltages and the currents in our circuit. We can use it to calculate the transfer function from the input to the desired output. Here we define the frequency range to perform the simulation.
- **tran** transient simulation. This is a non-linear time-domain simulation. The simulation results are time-domain waveform of the voltages and the currents in our circuit.

In this part of the tutorial we will perform a DC simulation. Select dc, and check "Save DC operating point":

-	Choosing	g Analys	es ADE	L(1) ×						
Analysis	🔾 tran	🖲 dc	🔾 ac	noise						
	🔾 xf	🔘 sens	O dcmatch	acmatch						
	🔾 stb	🔾 pz	🔾 sp	envlp						
	🔾 pss	🔾 pac	🔾 pstb	O pnoise						
	🔾 pxf	🔾 psp	🔾 qpss	🔾 qpac						
	🔾 qpnoise	🔾 qpxf	🔾 qpsp	ight hb						
	🔾 hbac	hbnoise	🔾 hbsp							
		DC Analys	is							
Save DC Ope	rating Point	2								
Hysteresis S	weep									
Sweep Vari	able									
Tempe	rature									
Design	Variable									
Compo	nent Paramete	r								
Model	Parameter									
Enabled 💆	Enabled 🕑 Options									
	OK	Cancel	Defaulte	Analu Hala						
		Lancer	Detaults	мрну пер						

Click OK.

The Outputs box - specifying the simulation outputs

After performing the simulation, we should specify the results that we are interested in.

In the Outputs box: right click -> Edit.

In the Name section type: out_dc.

In the Expression section, type: VDC("/out"):

	Setting Outputs ADE	L (1)			×
	Selected Output	Table Of Outputs			
Name (opt.)	out_dc	Name/Signal/Expr	Value	Plot	Save Options
Expression	VDC("/out") From Design				
Calculator	Open GetExpression Close				
Will be	Plotted/Evaluated				
Add	Delete Change Next New Expression				
		ОК	Cancel	A	pply <u>H</u> elp

Click OK.

We created an output expression named "out_dc" for the DC voltage at the node "out".

A very useful tool in Cadence for the output expressions syntax is the calculator. In the main ADE window: Tools -> Calculator. At the bottom, you have a list of the various functions that can be performed on the simulation results. If you are not sure about the command syntax, the Calculator is a very useful place to start.

The syntax for the output expressions is:

VDC/IDC	DC voltage/current (dc analysis)
VF/IF	AC voltage/current (ac analysis)
VT/IT	Transient voltage/current (tran analysis)

For voltage outputs, the syntax is VDC ("**/node_name**"). For current output, the syntax is IDC ("**/component_name/terminal name**").

In our circuit to see the terminal name of the 20k resistor connected to "out", click on it (the red square) and press q.

You will see the following window:



	Selected Output	Table Of Outputs			
	(C. 27)	Name/Signal/Expr	Value	Plot	Save Option:
Name (opt.)	1_R0_dc	1 out_dc		yes	
Expression	IDC("/R0/PLUS") From Design				
Calculator	Onen CetExpression Close				
concurator	Open detexpression close				
Will be	Plotted/Evaluated				
		< [
Add	Delete Change Next New Expression				

So, the component name is R0 and the terminal name is PLUS. For the output DC current through this node add the following output expression: IDC("/R0/PLUS"):

Another option is to click on idc in the Calculator, and then click on the resistor terminal.

To save your simulation setup: Session -> Save State. At the top change to "Cellview":

irectory Options			
ate Save Directory			
	~/.artist_states		Browse
Save As	state1		
Existing States			
eliview Options			
Library	cadence_files		
Cell	tutorial 🔽	Browse	
State	spectre_state1		
escription			
<u>(</u>			
What to Save		Select All Clear A	
Vhat to Save ✓ Analyses	uu ✓ Variables	Select All Clear A	
✓ Analyses	∭ ✓ Variables ✓ Operating Points	Select All Clear A	
Vhat to Save Vhat to Save Subckt Inst Simulation Files	✓ Variables ✓ Operating Points ✓ Environment Options	Select All Clear A Outputs Model Setup Simulator Options	
Vhat to Save ✓ Analyses ✓ Subokt Inst ✓ Simulation Files ✓ Convergence Setup	Variables Variables Operating Points Finitronment Options Waveform Setup	SelectAll Clear A V Outputs V Model Setup V Simulator Options V Graphical Stimuli	
✓ Analyzes ✓ Subokt Inst ✓ Subokt Inst ✓ Simulation Files ✓ Convergence Setup ✓ Conditions Setup	¥ Variables ¥ Operating Points ¥ Environment Options ¥ Waveform Setup ¥ Results Display Setup	SelectAll Clear A V Outputs V Model Setup V Simulator Options V Graphical Stimuli V Device Checking Setup	<u>.</u>

Click OK. It will a view named "spectre_state1" in the "tutorial" cell.

Click the "play" button to perform the simulation. You should see the simulated DC voltage and current at the Value column. Add the screenshot of the ADE window with the simulated result to your lab worksheet.

2.4. AC and transient simulation - RC low-pass

We will build the RC low-pass circuit shown in Figure 2.



Figure 2: RC circuit to build

Note that the output node should have a different name than "out", otherwise it will be shorted to the resistive divider output. The source should be "vsin":

	A	dd Instance
Library	analogLib	Browse
Cell	vsin	· · · · · · · · · · · · · · · · · · ·
View	symbol	
Names		
⊻ Add W	fire Stubs at:	registered terminals only
Array	Rows	1 Columns 1
	A Rotate	🛓 Sideways 🛛 🚭 Upside Down
First frequ	uency name	
Second fr	equency name	
Noise file	name	
Number	of noise/freq pairs	0
DCvoltag	je	
AC magni	tude	1 V
AC phase		
XF magnit	tude	
PAC mag	nitude	
PAC phas	e	
Delaytim	e	
Offset vol	Itage	
Amplitude	e	Vtran V
Initial pha	ise for Sinusoid	
Frequenc	у	freq_tran Hz
	-	Hide Cancel Defaults Help

AC magnitude is used for AC simulation, Amplitude and Frequency are used in transient simulation. Here we used variables Vtran and freq_tran rather than fixed values.

For the capacitance value use a variable named C.

You should have the following schematic:



In the ADE window, we can add the variables used in the schematic by right-click at the Design Variables area, and selecting "Copy from Cellview":

생		ADE L (2) - c	adence_fi	les tut	orial sch	emat	ic		_ 0	×
Launo	:h S <u>e</u> ssion S	et <u>up A</u> nalyses <u>V</u> ariable	s <u>O</u> utputs	<u>S</u> imulatio	n <u>R</u> esults	Tools	<u>H</u> elp		cāden	ce
11	🌮 💦 27	/ 📓 ⊁ 🖨	🖪 🖻							
Design	Variables		Analyses						? 5 ×	AC
	Name	Value	Туре	Enable	l.		Argun	nents		Citrans
	IVUITE	Voide	1 dc	⊻	t					98
1		Find	1							
		Edit								~
		Copy From Cellview								~
		Copy To Cellview								
		Delete			3000/			2		0
			Outputs						7 6 X	hA.
			Na	me/Signal	/Expr	Value	Plot	Save	Save Options	4.0
			a i P0 dc				×			
			a RO/PLUS						Wes	
			E				-	-		
			100		1100					
			<u> </u>		Auto	-	_	_	Deplace	
> Res	ults in /home/ff.	/ee105/simulation/tutorial	Plot after s	mulation:	Auto		Plotting	; mode:	Replace M	
(11) (1)					1	1			1	-
4(8)	Load State		S	atus: Rea	dy T=27 (Simi	lator:	spectre	State: spectre_state	21

Set C=10nF, transient amplitude of 0.5V (1Vptp) and frequency of 1KHz:

aunch Session	Setup Analyses Vari	iables Outputs	Simulatio	n Results Tools	Hein		cādeņ
darien session	seral Ferdiyaea Adri	aucs <u>o</u> utputs	2	1 10013	Tech	_	Lauent
1 100 14	27 👌 🎾 🖞	3 🖪 🗁					
anime Maniablan		Analyses			_	_	7 5 ×
esign variables		Туре	Enable		Argur	nents	8
Name	Value	1 dc	~	t			5
C	10n		1				
freq_tran	1K						
Vtran	500m	7					
		1<0)	
		Outputs			_	_	? 🖥 🗙
		Na	ame/Signal	/Expr Value	Plot	Save	Save Options
		1 out_dc			-	100	
		2 i_R0_dc				6	
		3 R0/PLUS				~	yes
				100			
				Auto	01-44		Renlace
Results in /home	⊻/ff/ee105/simulation/tut	torial/ Plot after s	imulation:	Auto	Plotting	g mode:	Replace

AC simulation

Add an AC (ac in ADE) simulation. We will sweep the frequency in a logarithmic scale between 1Hz and 1MHz:

-	Choosin	g Analys	es ADE I	L (2)	×
Analysis	🔾 tran	🔾 dc	🖲 ac	🔾 noise	
	🔾 xf	🔘 sens	O dcmatch	 acmatch 	
	🔾 stb	🔾 pz	🔾 sp	🔾 envip	
	🔾 pss	🔾 pac	🔾 pstb	O proise	
	🔾 pxf	🔾 psp	🔾 qpss	🔾 qpac	
	🔾 qpnoise	🔾 qpxf	🔾 qp sp	🔾 hb	
	🔘 hbac	hbnoise	🔾 hbsp		
		AC Analys	is		
Sweep Var	iable				
Freque	ency				
O Design	Variable				
C Tempo	erature				
Comp	onent Paramet	er			
O Model	Parameter				
O None					
Sweep Rar	ige				
Start-S	top ,				_
Center	-Span	start 1	Sto	ρ 1 Μ	- 1
Super Tur					
sweep typ		e Point:	s Per Decade	11	
Logarithm	IC Y	🔾 Numi	er of Steps		
Add Specific	Points				
Specialized	i Analyses				
None					
Enabled	2			Options	
	ОК	Cancel	Defaults	Apply	lelp

To add the transfer function output:

	Selected Output	Table Of Outputs			
	,	Name/Signal/Expr	Value	Plot	Save Option
Name (opt.)	vo_over_v1_RC_ac	1 out_dc	2		
Expression	(VF("/out_RC") / VF("/in_RC"))	2 i_R0_dc	100u		
	Hombesgr	3 RO/PLUS		no	yes
Calculator	Open GetExpression Close	4 vo_over_vi_RC_ac	wave	yes	
Will be	✓ Plotted/Evaluated				
		<u> </u>			
Add	Delete Change Next New Expression				

A window with the plots will open:



The transfer function is a complex number. Cadence is always plotting the magnitude by default. We will switch to bode-style (log-log) plot, by right-click on the y axis and selecting "Log Scale":

22		Virtuoso (R) Visualization	& Analysis XL			_ 0 ×
<u>File Edit View G</u> raph <u>A</u> o	xis <u>T</u> race <u>M</u> arker M	easurements Tools <u>W</u> in	dow <u>B</u> rowser <u>H</u> el	р			cādence
🐨 • 🗃 • 🗁 🔒 🐇	B 💼 I 🤊 🤞	🖌 🖸 🖻 🗙 🛛 🤇		Q 🖤 🛄 🎠 G	family	· X X X X 19 10	
📝 🧐 🗄 🔲 Subwi	indows 1 🔽 🛛	🖬 🚺 📲 💁 Da	ita Point		Classic	- 5. 😞	
Cadence_files tutorial sche	matic × 🗹 caden	ce_files tutorial schematic	×		-		
vi_over_vi_RC_a						Fri Aug 5 15:	19:54 2016 1
Name Vis	< I		_				
vi_over_vi_RC_ac 🧔	1.1						
	1.0						
	0.9						
	0.8 -						
	0.7						
	Axis Propertie	·S					
	0. Select Attache	d Traces					
	⊖ Show Axis Nu	mber					
	Major Grids						
	0. 🗸 Minor Grids						
	0. Show Units						
	Allow Any Uni	ts					
	0						
	0.1 -						
	0.0						
	-0.1						
	100	101	102	103		105	1.00
	10	10	10-	freq (Hz)	10	10	10

We can add a marker by pressing "m" and double-clicking the marker to select the frequency or the desired y value (or moving the marker with the mouse). For 1KHz:



To add a marker value to the output expressions we can use value() function. The value() function is an x-axis marker. It returns the function value for a specific x-axis (frequency in this case) value. The syntax:

	Selected Output	Table Of Outputs			
	-	Name/Signal/Expr	Value	Plot	Save Option
Name (opt.)	vo_over_v1_RC_1KHz	1 out_dc	2		
Expression	value(abs(vo over vi RC ac) "freg" 1K)	2 i_R0_dc	100u		
	Hombesign	3 RO/PLUS		no	yes
Calculator	Open GetExpression Close	4 vo_over_vi_RC_ac	wave	yes	
Will be	✓ Plotted/Evaluated	5			
		6)	<u></u>
Add	Delete Change Next New Expression				
			ancel	AP	ply C

So far, we plotted the magnitude of the transfer function. Add another output expression for the phase of the transfer function, using the phase() function.

Parametric sweep

Now we will sweep the capacitance value and look at the simulation result for each value. In the AD window Tools -> Parametric Analysis. Fill the following sections: Variable: C, From: 5n, To: 20n, Step Mode: Linear Steps, Step Size: 5n:

Transition in the second se		Para	ametric Analy	sis - spec	tre(1): ca	dence_files	tutorial sch	ematic	
<u>File A</u> nalysis <u>H</u> elp							/		
Parametric Simula	ition Compl	leted.				~			
🖻 🔒 🍓 🖉) × (🕜 🎹 🕇 Run Ma	de:Sweeps&	Ranges 🔽	0 🗧 🛛			
Variable	Value	Sweep?	Range Type	From	To	Step Mode	Step Size	Inclusion List	Exclusion List
C	10n	¥	From/To	5n	20n	Linear Steps	5n		

To run the parametric sweep, click on the "play" button in the parametric sweep window. Attach the following parametric sweep plots to your lab worksheet:

- The transfer function magnitude vs frequency (in log-log scale)
- The transfer function magnitude value at 1KHz

- The transfer function phase vs frequency
- The transfer function phase value at 1KHz.

Transient simulation:

Add a transient (tran in ADE) simulation:

Choosing Analyses ADE L (2) ×				
Analysis	🖲 tran	🔾 dc	🔾 ac	🔾 noise
	🔾 xf	🔾 sens	🔘 dcmatch	 acmatch
	🔾 stb	🔾 pz	🔾 sp	🔾 envip
	🔾 pss	🔾 рас	O pstb	O proise
	🔾 pxf	🔾 psp	🔾 qpss	🔾 qpac
	🔾 qpnoise	Q qpxf	🔾 qp sp	ight hb
	🔘 hbac	hbnoise	🔘 hbsp	
Transient Analysis				
Stop Time 3m				
Accuracy Defaults (errpreset)				
Transient Noise				
Dynamic Parameter				
Enabled	2			Options
OK Cancel Defaults Apply Help				

The stop time is 3msec (3 time periods), and the accuracy is "conservative" (usually slow for large circuits, but OK for small circuits like ours).

Add an output expression for the out_RC node, and attach the plot to your lab worksheet.

--- End of The Document ---