

ORCAD 10.5, ORCAD 15.7 & ORCAD 16.0

Document Created – August, 2006

Note:- *This document serves as a tutorial for OrCAD 10.5, OrCAD 15.7 and OrCAD 16.0. The 16.0 version is being used in the 2008 PHYS333 and PHYS334 courses.*

The process is identical for all versions, in that the same method is followed, the same screens and icons are seen, and the end results are the same.

Table of Contents

ORCAD 10.5 Tutorial.....	1
Objectives of the tutorial and Introduction	5
Creating a New Project and Schematic Diagram	6
Instantiating circuit components, connecting them together and setting component values and properties AND saving the schematic diagram ..	8
Instructions to set up a circuit	10
Creating a New Simulation Profile and setting up the simulation.....	14
Simulating the circuit and observing the simulation results	17
Opening a Saved Project	20
Appendix A – Component Libraries.....	21
Appendix B – Suggested Project Storage	22

List of Figures

Figure 1: Schematic window	7
Figure 2: Toolbars.....	8
Figure 3: Place menu.....	8
Figure 4: Circuit diagram – R-C circuit	13
Figure 5: PSpice menu	14
Figure 6 : New simulation profile.....	14
Figure 7 : Analysis TAB	15
Figure 8 : Options TAB	16
Figure 9: Simulation results	17
Figure 10: ICONs in the PSpice A/D simulation results window	18
Figure 11: Compact PSpice A/D window	18
Figure 12: Compact PSpice A/D window	19
Figure 13: Project Folder Windows.....	20

List of Tables

Table 1 : Commonly used short cut keys	9
Table 2: Simulation short cut keys	17
Table 3: Libraries and Components	21

Objectives of the tutorial and Introduction

In this laboratory experiment you will

Become familiar with the simulation software

Learn to use schematic capture

Setup parameters, simulate and analyze the waveforms obtained

If you are in the LAB the software is already installed on the LAB computers.

If you are at home, download and install PSpice (Demo Version 10.5 ORCAD)

There are FOUR main steps involved in circuit simulation using PSpice. They are:

1. Creating a New Project and Schematic Diagram
2. Instantiating circuit components, connecting them together and setting component values and properties AND saving the schematic diagram
3. Creating a New Simulation Profile and setting up the simulation
4. Simulating the circuit and observing the simulation results

Creating a New Project and Schematic Diagram

Start ORCAD 10.5

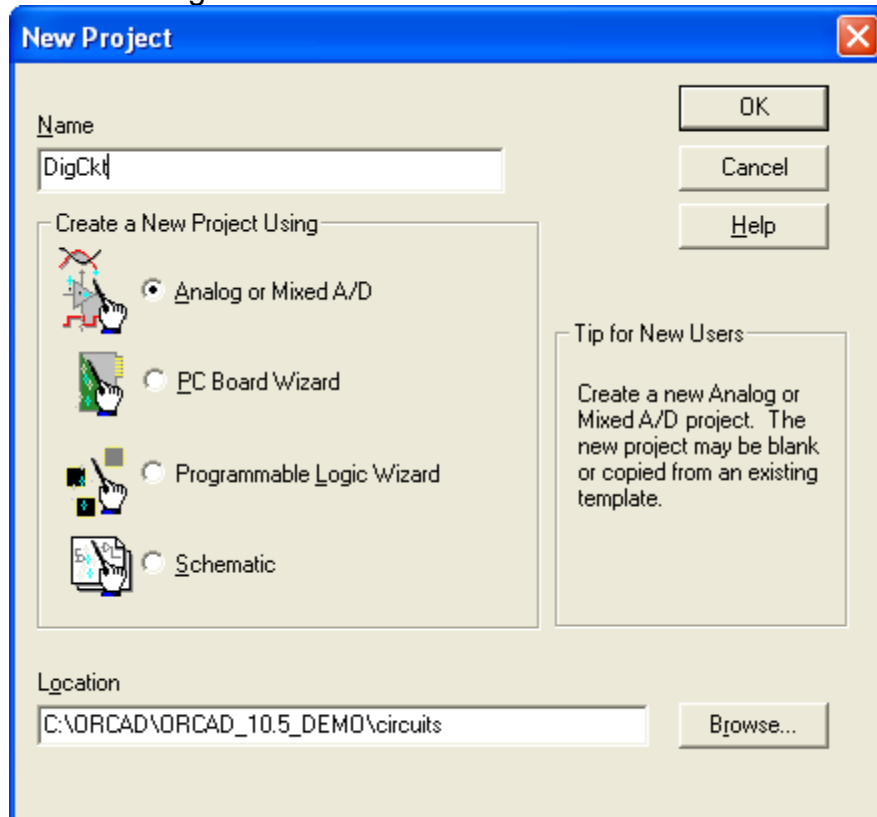
Start→All Programs→OrCAD 10.5 Demo → Capture CIS Demo

Create a New Project

File→ New → Project

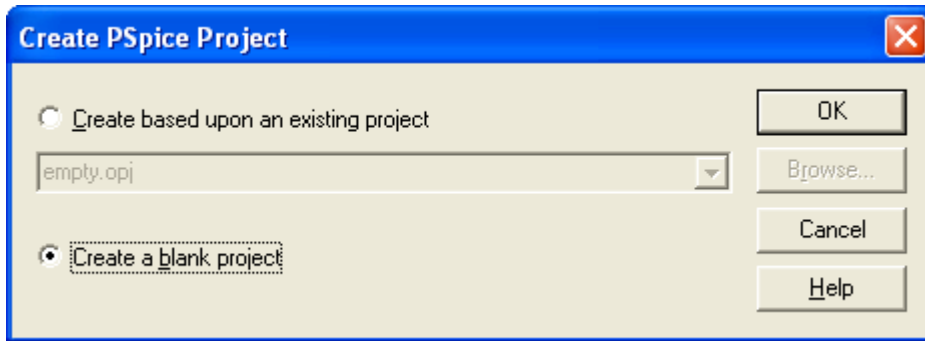
Enter Project Name and Location

Select Analog or Mixed A/D



Click OK

Note that clicking the Browse button in the New Project window allows you to set up a folder for your project (see Appendix B for suggested project storage)



Select “Create a blank project” and Click OK
You should now see a blank schematic diagram.

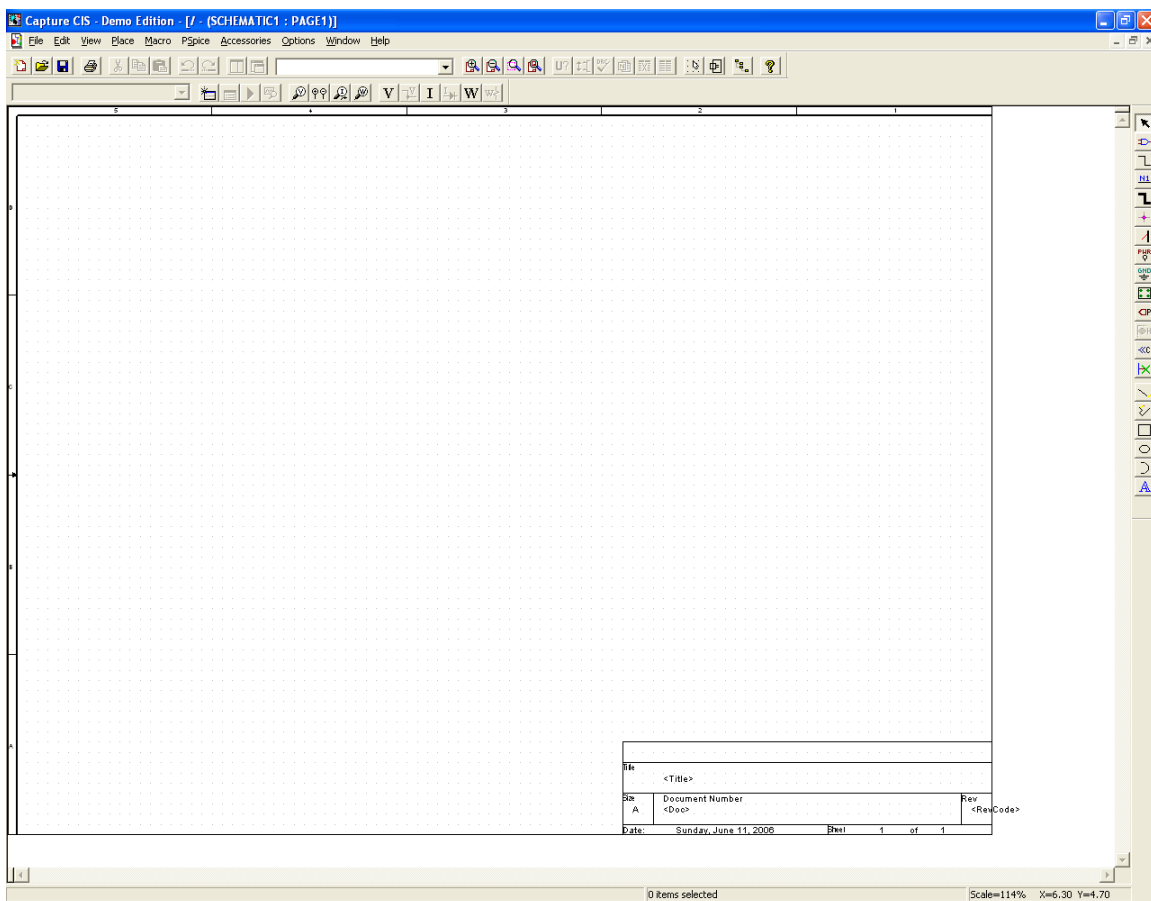


Figure 1: Schematic window

Instantiating circuit components, connecting them together and setting component values and properties AND saving the schematic diagram



Figure 2: Toolbars

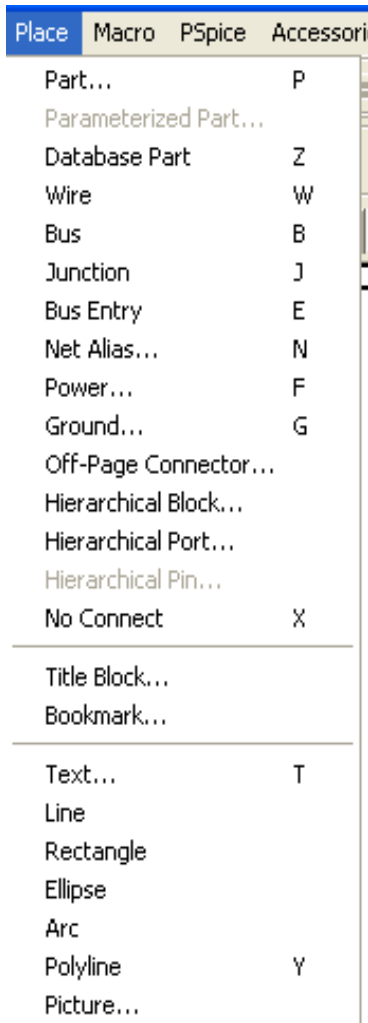


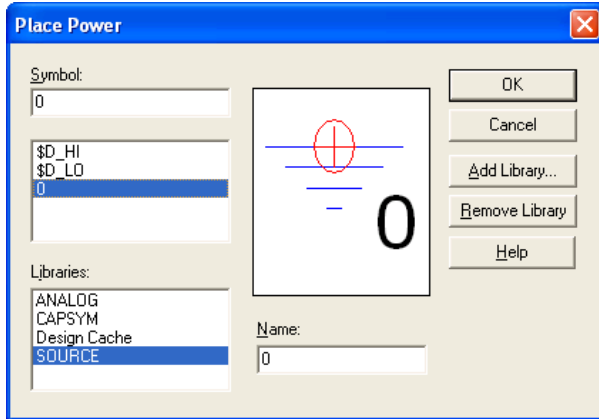
Figure 3: Place menu

Table 1 : Commonly used short cut keys

Short cut keys	Operation
P	Place Part
W	Place Wire
F	Power
G	Ground
T	Text
N	Net name or alias
I	Zoom IN
O	Zoom OUT
Control+F	Find
Control+C	Copy
Control+V	Paste
Control+X	Cut
Del	Delete
Control+A	Select All
Control+Z	UnDo
R	Rotate Component
H	Mirror Horizontally
V	Mirror Vertically

Instructions to set up a circuit

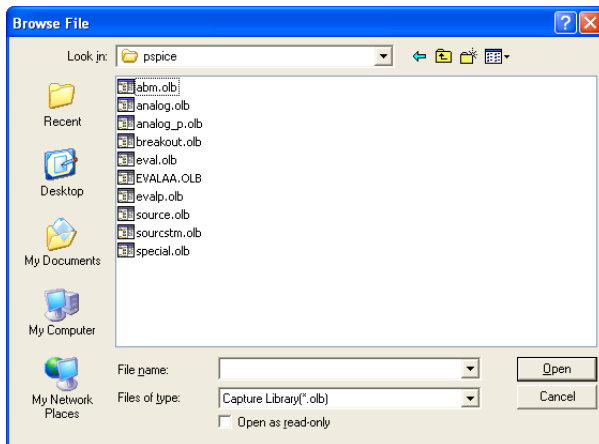
To obtain a part, click on Place/Part to get the following window:-



When starting from scratch, libraries will have to be added. So, if necessary, click on the “Add Library” box, to get the “Browse File” folder shown below, and get into the pspice folder.

Note that this screen is shown with the SOURCE library selected, and then the 0-Ground. This represents Node 0 in the PSpice netlist, and is

necessary for all the circuits you will be using in this course.



Select the appropriate folder, click on the OPEN box, to return to the Place screen above. Find and select the appropriate part, and then click on OK to return to the Schematic window and place the part.


Note that many copies of a part may be placed in the schematic (subject to the limitations of the Demo version), by moving the cursor and Left Clicking. When the required

number of parts have been placed, Right Click, and then Left Click in the pop-up window on “End Mode”, or press the Esc key.

To rotate or mirror parts – Left Click to select the part, then Right Click to get a pop-up window, which allows you to mirror or rotate

Note again the following Libraries that you will be using:-

SOURCE	--	Power, 0-Ground, VDC, VAC, VSIN
ANALOG	--	R, L, C components
EVAL	--	ICs, e.g. 74xx series, LF411 op-amp, μ A741 op-amp, etc
ABM	--	the GAIN function

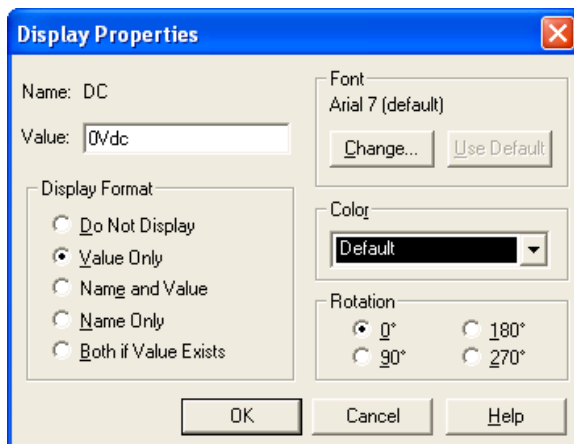
Once parts have been placed, use Place/Wire, or click on the  “Wire” button to allow you to wire the circuit. The cursor changes to a crosshair, which

may be used to connect between valid connection points. Hold the left mouse button as you move the cursor to the other connection point, and then release it. If the “wire” continues beyond the intended termination point when you move the cursor away, return the crosshair to the termination point and left-click the mouse.

To change the direction of a wire, left-click at the point where the direction is to change. This is useful when you wish to get around awkward corners, or to make the circuit more presentable.

Components come with preset values, which may be changed easily.

Resistors -- 1k for 1 kΩ Capacitors -- 1n for 1 nF
DCPower -- 0Vdc



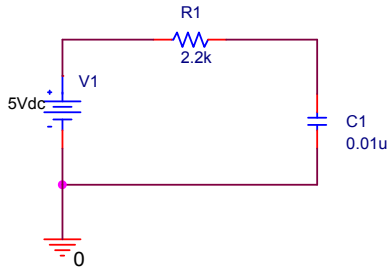
To change the values, double click on the value itself (not on the component), and in the resulting window, alter the value, e.g. in the value window, change the 0 to 5 (for 5Vdc).

Standard “electronic” values are written using actual value, the symbol or the exponential form according to the table below.

Unit Name	Symbol	Exponential Form	Value
Femto	F (or f)	1E-15	10 ⁻¹⁵
Pico	P (or p)	1E-12	10 ⁻¹²
Nano	N (or n)	1E-9	10 ⁻⁹
Micro (μ)	U (or u)	1E-6	10 ⁻⁶
Milli	M (or m)	1E-3	10 ⁻³
Kilo	K (or k)	1E3	10 ³
Meg	MEG (or meg)	1E6	10 ⁶
Giga	G (or g)	1E9	10 ⁹
Tera	T (or t)	1E12	10 ¹²

- *Example:* to denote -7.8×10^{12} , you can enter -7.8T instead of -7.8E12.
- To enter the resistance value of $1\text{k}\Omega$, you can enter 1000, 1E3, 1K or 1k.
- Enter the attributes for all three resistors and the DC voltage source.
- **Note:** you do not have to include the units when entering the values.
- **Caution** – the scale symbols are not case-sensitive; M means milli, not Mega.

[Ref: PSpice Tutorial by David Lay – for Table and Example above]



Draw and complete the circuit shown and save it. Finally, Save and Close the schematic, and then Close the Project.

The lab handouts will explain how to set up and activate a simulation.

Or, a sample simulation profile and run are shown below.

Sample Simulation Set-up and Run

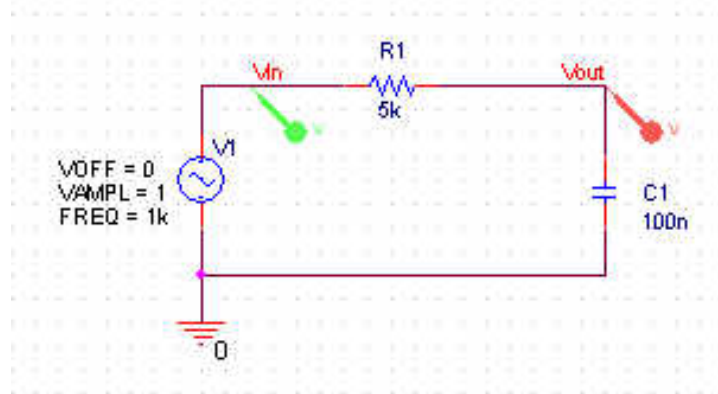


Figure 4: Circuit diagram – R-C circuit

Place Net Names (Vin, Vout) -- Short cut key is N

Place Voltage/Level Markers



Save the schematic



Creating a New Simulation Profile and setting up the simulation



Figure 5: PSpice menu

PSpice→New Simulation Profile

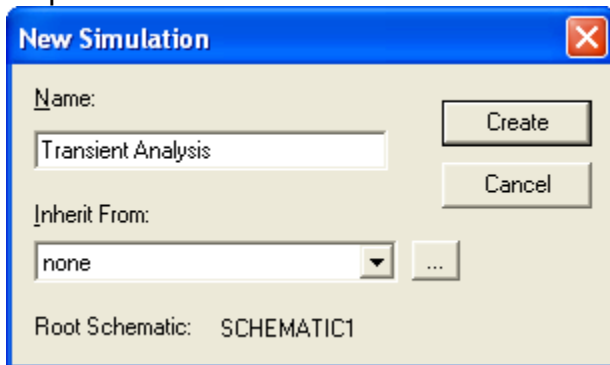


Figure 6 : New simulation profile

Give the New Simulation Profile a name and click Create

ANALYSIS TAB

Select analysis Type: Time Domain (Transient)

Run to time: Set the simulation stop time here

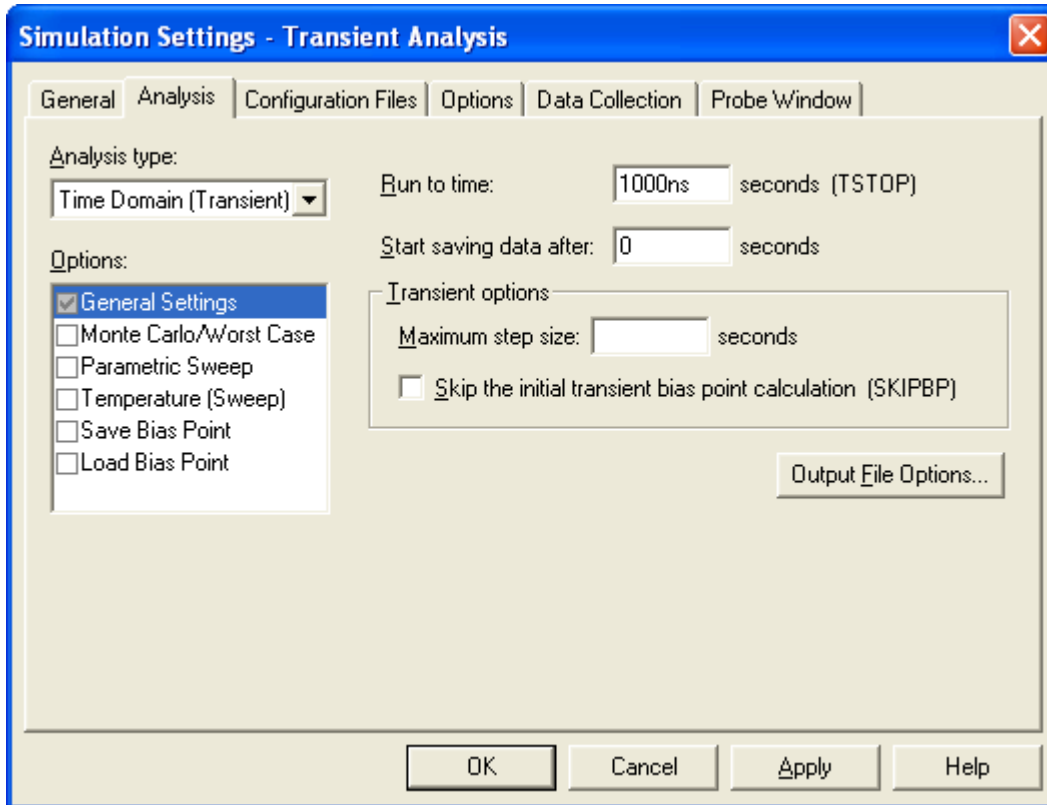


Figure 7 : Analysis TAB

Change the “Run to Time” to the desired time (say 5 ms in this case)

OPTIONS TAB – Required for Digital circuits, NOT for Analogue Circuits
Category: Gate-level Simulation

Timing Mode: Typical
Initialize all flip flops to “0”

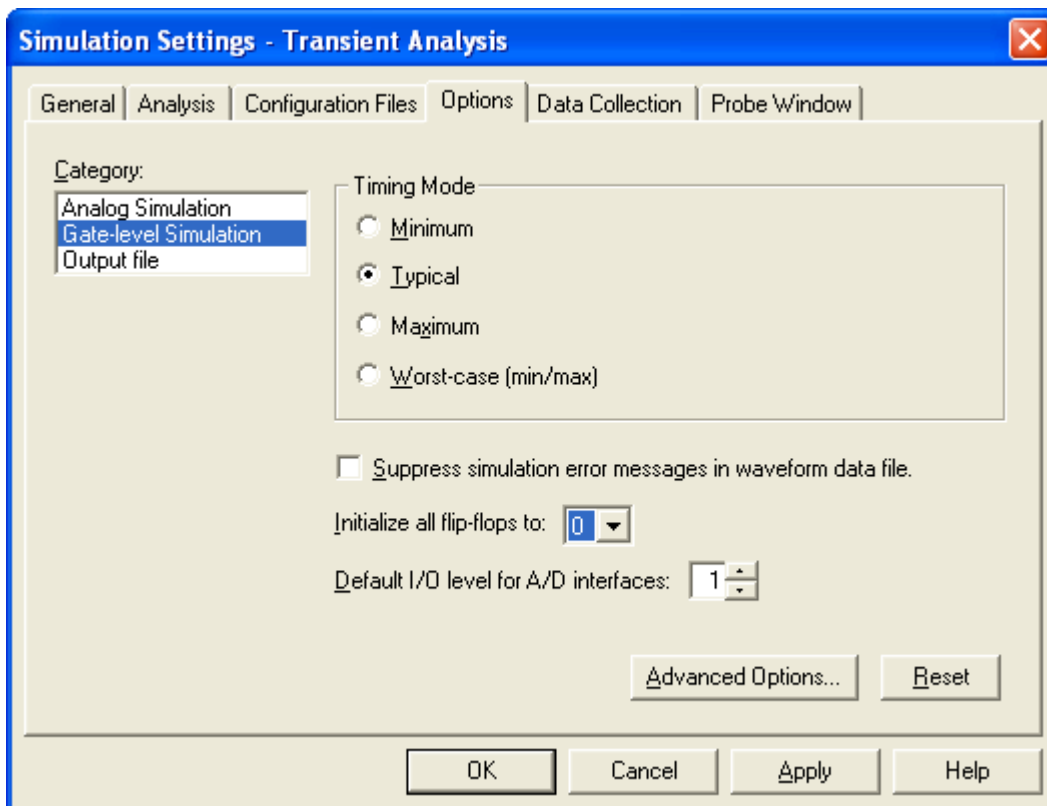


Figure 8 : Options TAB

Apply and click OK

Simulating the circuit and observing the simulation results

Table 2: Simulation short cut keys

F11	Run Simulation
F12	View Simulation Results

Run and Display Simulation Results (Press F11 or F12)

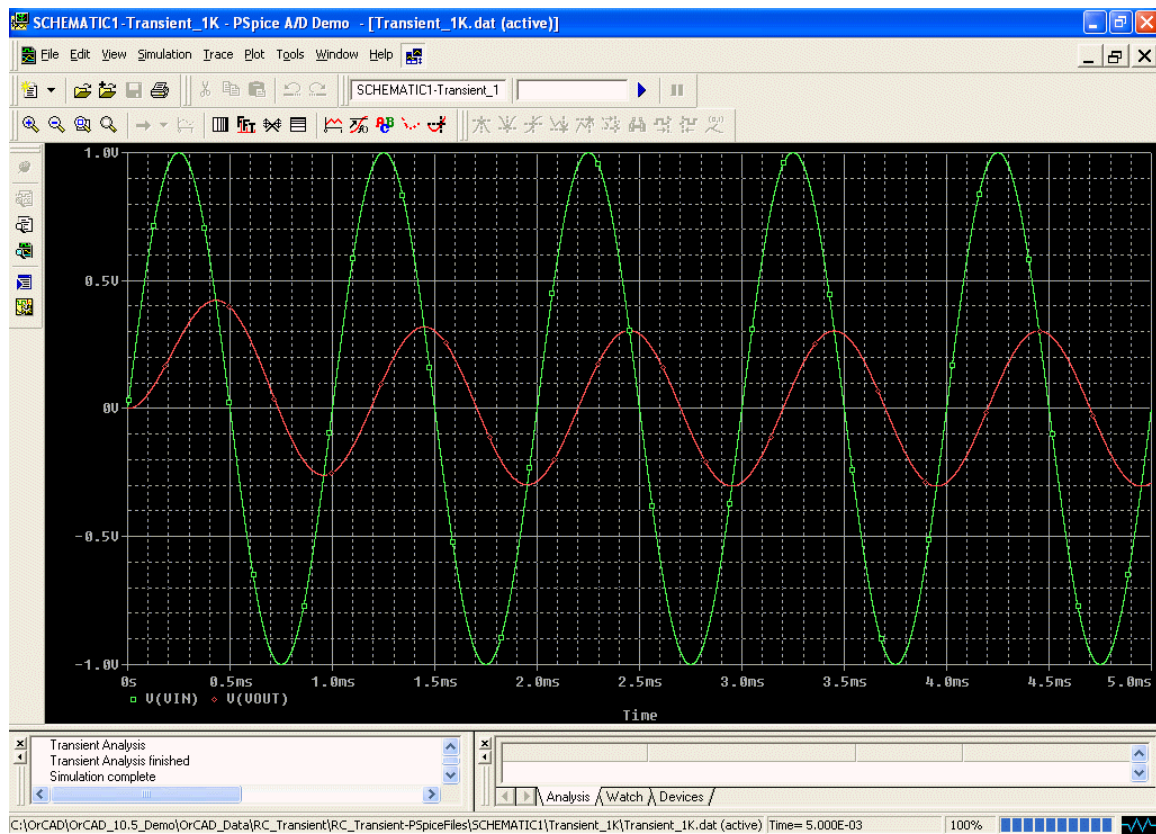


Figure 9: Simulation results



Figure 10: ICONs in the PSpice A/D simulation results window

Alternate Compact Simulation Display Window

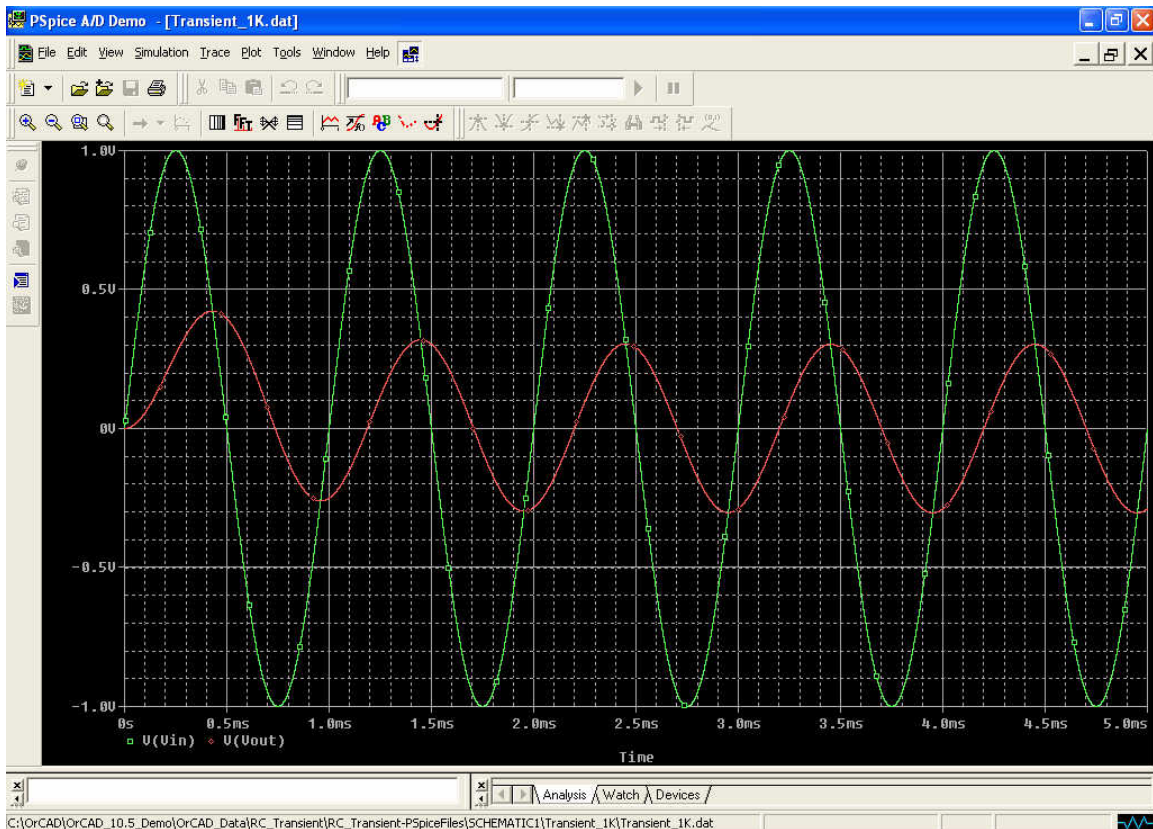


Figure 11: Compact PSpice A/D window

If the PSpice A/D screen output is sent to the printer, the black & white output appears as:-

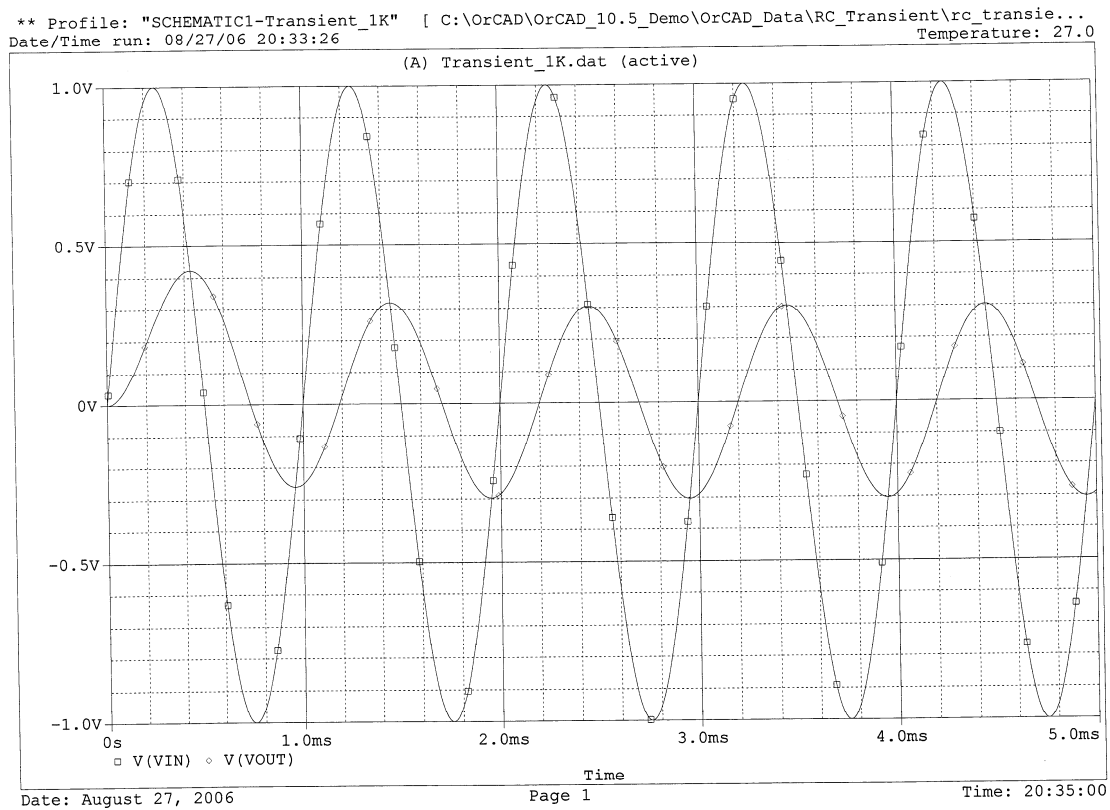
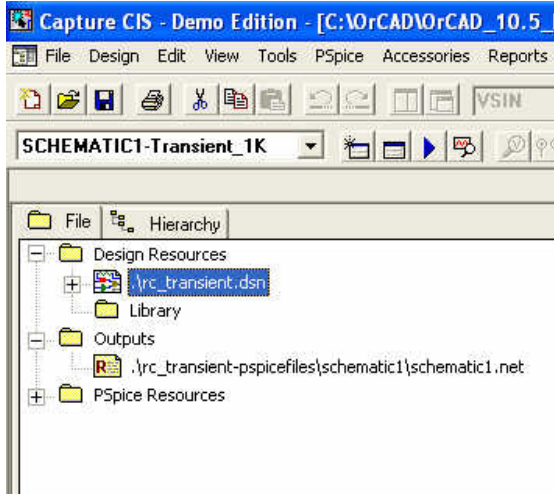


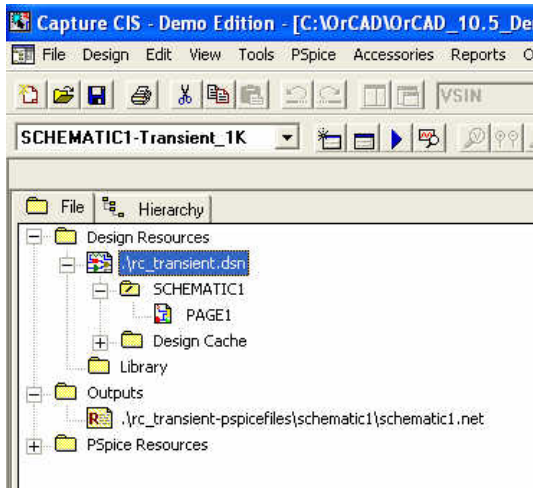
Figure 12: Compact PSpice A/D window

Opening a Saved Project

From the Capture CIS window, File/Open/Project



Find the correct project, and click on the .opj file to get a screen with the project folder



Click on the .dsn icon to open up the folder and then on SCHEMATIC1 and finally on PAGE1. This will bring up the saved schematic.

You can work on the schematic, as before – modify it, edit the simulation profile, run the simulation, etc.

Figure 13: Project Folder Windows

Appendix A – Component Libraries

Table 3 lists all the components (and their corresponding libraries) needed for simulating circuits in the text book and the laboratory experiments for phys333/334

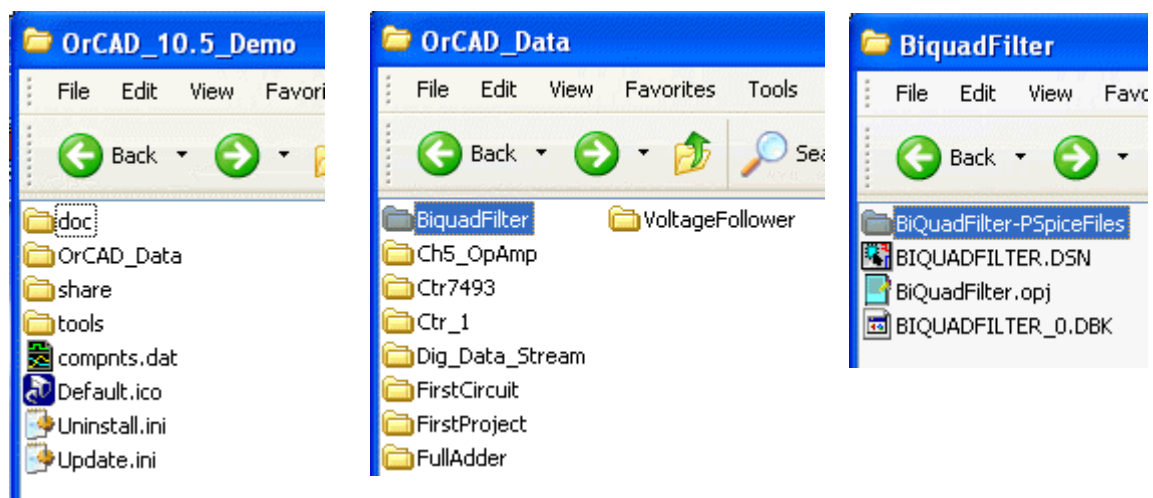
Table 3: Libraries and Components

Component	Library	Comments
DigClock	Source	Digital clock input
VDC	Source	DC supply – 5V
0	Source	Ground
7493A	EVAL	Binary Ripple Counter – 4 bit
7473	EVAL	JK FlipFlop
...
Logic Gates	Library	Comments
74XXX Series	EVAL	Basic Logic Gates
Flip Flops	Library	Comments
74XXX Series	EVAL	Counters, Registers, Memories, Flip Flops and Latches
		Analog to Digital Converters
		Digital to Analog Converters
Passive Components		
R	ANALOG	Resistor
L	ANALOG	Inductor
C	ANALOG	Capacitor
Switches		
DIP switches		
AC Sources		

Sinusoidal		
Pulse		
Rectifier Diode		
Light Emitting Diode		
Operational Amplifiers		
LM741		
LM324		
LM747		
LM348		

Appendix B – Suggested Project Storage

It is suggested that your project files are saved in a systematic manner, so that you can find them and re-open the project, or use the saved data at a later time.



When installing ORCAD 10.5 Demo, you will be given the opportunity to set up a folder for project files – in the left screen shown, this is the OrCAD_Data folder (my name on my PC).

In the OrCAD_Data folder, set up a separate folder for each project, as shown in the middle screen above – I have selected the BiquadFilter project, in order to show how the project files are stored by ORCAD – right screen.

When it is necessary to open this project later, first open Capture CIS Demo. Then File/Open/Project, and work your way to the appropriate project file (.opj) and click on this – in my examples above, the *BiQuadFilter.opj* in the right screen.