Contents < Capture >

1.	OrCAD		6
2.	Capture		8
	Capture		
3.	Capture Window -		10
4.	Capture		18
	Hot key		
	Capture Schem	atic Design Flow	
5.	Schematic Design		20
	Schematic Design		
	Annotate (Update Part	Reference)	
	Design Rules Check		
	Create Netlist		
6.	Footprint		25
	Property Editor		
	UPD file		
	Layout		
7.	Option		27
	Preference		
	Design Template		
~	Schematic page prope	rties	20
ð.	(Hierarchical Str	ucture)	30
	(One sr	neet structure)	
	(Flat Hi	erarchical Structure)	
	(Hierard	chical Structure)	• •
9.	Schematic Library		36
10.	. Program Install		41

OrCAD Capture for Windows Software version 9.2

- Basic Educational Book -

1. OrCAD

가

		EDA(Electronic	Design
Automation)			
가	EDA		
	EDA가		
Computer			
•	Compute	er	
CAD(Computer Aided Design	- Computer)	
CAD	Electronic CAD	Mechanical CAD가	,
Electronic CAD	가	OrCAD ,	,
3	CAD	가 AutoCAD .	
OrCAD Inc. 1985	EDA(Electronic D	esign Automation)	
	DOS S	SDT(Schematic program)
, Windows Capture	Schematic D)esian	/
. 1995 PCB	Artwork	N	Aassteck
PCB Artwork Tool(La	ayout)		, 1998
Analog & Digital M	ixed mode Simula	ation	
Pspice , PC	level EDA		
-1	ED	/A	
· /f		/ ,	

.

Time - To - Market OrCAD 9.xx EDA Tool

.

Capture / Capture CIS for Windows

- , PC Board Windows Analog/Digital PCB SMD PCB 가 SmartRoute Tool CAM Tool Gerber file Editing

PSpice (A/D) for Windows

- PC Analog/Digital Tool.

2. Capture

Capture

OrCAD Capture program (Schematic) , Netlist file . OrCAD Capture program 가 Netlist Auto CAD, Genertic CAD DXF Netlist , EDIF, VHDL, Verilog HDL Netlist . Pspice Library 44,000 library 가 . Wire, Bus (Auto Logo File Junction), Macro Toolbar Tool Palette , , database . Windows On-line Help Interactive Tutorial , Excel, Lotus . Install OrCAD program 9.x version CD , Install 10 Capture OrCAD Capture Program C: \ Program Files \ OrCAD Release 9.x \ Capture \ **OrCAD** Release OrCAD program Windows Capture 9.x . Capture Program Windows가 .



	File menu	New
Project		

File menu New Project



OrCAD Program

Analog Analog/Digital Pspice Project option.

PCB OrCAD Layout Project option.

CPLD FPGA design, Digital simulate OrCAD Express Project option. Schematic Capture Project option.

Editing , .

Key point

3. Capture Window

OrCAD 9.x version Capture program *.opj . Pspice Express program Capture Schematic file *.dsn , *.dsn file file *.opj , *.opj file Project Design file

.



cache), (Bill of Materials, Cross Reference part)

(Design Rules Check)

Schematic Page Editor Window , manager (Project manager Window) (Schematic Page Editor Window) Schematic . Parts(Symbols) Part editor Windows ✓Session Log Window (U1, U2...) Annotate(Update Part Reference) ERC(Error Rules Check) (DRC : Design Rules Netlist Check) ≪Status Bar 가가 가 Bar . ∠Title Bar **OrCAD** Capture Bar , , . ∠Menu Bar File Tool Options Place . ∠Tool Bar open, save, , Annotation, , Design Rules Check, Netlist Bar . [new file, open. save, print] [Help] [Zoom] [cut, copy, paste] MRU window [project manager] [undo, redo] [snap to grid] 12 🗃 🖬 🎒 X 🖻 🖬 B. B. B. B. U? 11 080 v N X

- Annotate (Update Part Reference) :

- Back Annotate : Gate Pin
- Design Rules Check (DRC) :
- Create Netlist :
- Cross Reference Part :
- Bill of materials :
 - file

.

≪Tool Palette

Bar .

		•
1	1	
Ì	N	1
	1	
1	ŧ	
5	1	I
F	ų	R
6	N	0
4		P
Free.	1	H
Y	81	2
ł	×	<
	~	
11	2	/
[]
0		>
10000	2)
-	A	1

Select :					
Place part :					
Place wire :					
Place net alias :		()		
Place bus : Multi					
Place junction :					
Place bus entry :	wire				
Place power :					
Place ground :					
Place hierarchical block :					
Place [hierarchical] pin :					
Place [hierarchical] port :					
Place off-page connector :					
Place no connect :	pin				
Place line :				•	
Place polyline :					
Place rectangle :					
Place ellipse :					
Place arc :					
Place text :		Т	ext		

.

-

, , ,

file

. file .(가)

Tool bar

Place Part



Part Search × Eart: OK Part Search X (al Part Part IBE ary. Als?? .ibrary Cancel Libraries rch. Part Search... SEL Begin Search eDriverReceiver, alb eDriverReceiver, alb eDriverReceiver, alb -Help part search Librarie .(hot key; P) ate, olb ate, olb -MF wild card(character) Library Bath C:WOrcad9, 2WCaptureWlibrary Browse ... Type: Extension character(*)가) *ls00 *character part search ls 가 ls00 Question character(?)가 74ls?? . 가 가 74ls 가 (? 가) *ls?? 가 가 가 ls가



Ex) Net Alias name A1,A2

Change,... Use Default | Arial 7 (default)

Fart

.





Place hierarchical block



. Scalar wire Bus 가 Descend hierarchical block name port가 project manager folder가 .

.

가 .

Reference namenameimplementation typenameblock.(8)..



Place hierarchical Port

Place	hierarchical	port(🔎)
Conne	ect	

Place Herarchical Port		×	Port	
Surrool: PORTLEFT-R PORTLEFT-R PORTLEFT-L PORTNO-R PORTNO-R LUraries: CAPSYM Decian Cache SOURCE	Njamet PORTLEFT-R	ок Cancel Add Ubrary Bernove Ubrary Belp	. Off-p t type	age connect
Dest		Fares	output	port
Port	input	Error		

4. Capture

Ľ

≤.opj (OrCAD Project file) - OrCAD Capture , Design file link . ≤.dsn (Schematic Design file) -≤.olb (OrCAD Library file) - Capture ∠ .upd (Property Update file) - PCB library Footprint Library ≤.swp (Back annotation file) -Gate pin swap . ≤.drc (Design Rules Check report file) -DRC report ≤.bom (Bill of materials report file) report . ≤.xrf (Cross Reference part report file) report . ∠.mnl (Netlist file) -Netlist .

Hot key

Zoom in	l I
Zoom out	0
Zoom to all	+
Zoom center	С
Cut	Ctrl +X
Сору	Ctrl + C
Paste	Ctrl + V
сору	Ctrl + mouse

Find	Ctrl + F
Repeat	F4
Rotate	R
Mirror Horizontally	H
Mirror Vertically	V
(Alt
	Shift
Redraw active window	F5

Schematic Design Flow

, PCB Footprint Library 6 .

. Schematic Design

- Capture , Schematic , Design .

. Annotate (Update Part Reference)

- (:Reference name)

. DRC (Design Rules Check)

- Wire Port ERC(Error Rules Check) matrix Port type .

.

. Footprint Layout Footprint 가.

- . Create Netlist
 - Schematic Design Netlist file

5. Schematic Design

🖉 Schematic Design

Capture	File	New	Design		
, Schematic page editor window	menu b	Project bar	manager	window	Schematic
Project manager window					
,		me	nu가	, Sch	ematic page
editor window menu가			,	,	

Schematic	PCB
Part (Library)	Component (Footprint)
Wire	Net (Ratsnest, Connection / Track)

<i>⊯</i> Library Part	Symbol	···· 💬 ··· 🛫 💡 ··· 🛠 ··OFFPAGELEFT-L·····
, Title Block, Svmbol	Port	PORTLEFT-L PORTLEFT-R VCC_CIRCLE COFPPAGELEFT-R
library	, Part	Tītļe <tītlę></tītlę>
		Size Document Number Rev A3 <doc> <rev code=""></rev></doc>
		Date: Wednesday, February 23, 2000 Sheet 1 of 1

✓ PCB Component , PCB Footprint . , PCB (Routing) 가 Connection Ratsnest , Routing Track .

Place Part



가 , Gate.olb library file - , Library !- []

- Place Part
- 가 , [OK]

(hot key; H)





Rotate (hot key; R) Mirror Horizontal Mirror Vertical (hot key ; V)

- 21 -

Part Search	
Library Pla	Ce Part OK
Part [Part Search] . Part Search	Part Part Part Part Part Part Part Part
Search] library *.olb file [OK] .	JIII I Idole's OS FEOL/SO/AusDitiverTranscelver.olb Beain Search
Library Path !	C:WOrcad3.2WCaptureWilbrary Browse
Part serch Wild Card Extension Character (*) : MOO Question Character :	(EX ; *LS00 , NE5* , C*21 ,) (EX ; SHC?01 ,
IL?54	4M,?LP161,ZSR800/S?,…)
Place wire	
Menu Bar Place Wire . schematic window	, Tool Palette Place Wire
	pop up menu
End Wire . , , , Shift'	가
≪Annotate (Update Part Refere	ence)
Annolate E Scope C Update gentre design C Update gelection Cancel	가
Action C Incremental reference update C East patr references to ?" C Add Intersheet References	U1, U2 R1, R2 C1, C2
C Delete Intersheet References Mode C Update Occurrences C Update Instances (Preferred)	(Part Reference Name) .
Physical Packading Combined property string: (RValus)(Source Package)	
 Beset reference numbers to begin at 1 in each page Do not change the page number 	

Reference Name Option menu	Preference	Annotating, Miscellaneous Referen	Update Pa Auto Ref ce Name	rt Reference erence'
Сору		,		
Edit property		,		
Annotate		Reset (U?, R?)		
Scope()				
Update entire design :		update		
Update Selection :	upda	te		
Action				
Incremental Reference upo	date : F	Reference name	가	
		(? Mark)	upda	ate.
Unconditional Reference u	update :	Reference name	가	
Reset part Reference to ?	: Refer	ence name		4 - 4
Add Intersheet References	5.		on -page conn	ector
Delete Intersheet Reference	ces : Intershee	et Reference		
Mode				
Update Occurrences : Ref	erence updat	e	Ref	erence
L	Jpdate.			
Update instances (Prefer	red):		Reference upo	late. PSpice,
~ Docian Pulso Chock				
Design Rules Check		×		
HARRING THE T NOT Y SERVER MARKING				

Scope C Check gritine design C Check gelection	Mode C Use occurrences G Use instances (Proterred)
Action Check design rules Check design rules Check design rules	
Report Create DRC markers for <u>w</u> arnings	To Paraly managemented parts
 Check off-page connector connections 	Check SDT compatibility
Report identical part references Report invalid packaging Report Invalid packaging Report Interarchical ports and off-page Report Elie:	F Report off-grid objects Report all get nemes connectors
D:WAkkimWsampleWSample2_90WSchl	#SCHILDRC Browse

Error Check , Electronic CAD

가 .



- EDIF Electric Data Interface Format
- Spice Simulation Program with Intergrated Circuit Emphasis
- VHDL Very high speed Hardware Description Language
- Verilog Verilog-Hardware Description Language
- Other ? Mentor(mentor.dll), Cadstar(racalred.dll, rinf.dll), Pads(pad2k.dll, padspcb.dll), Pcad(pcad.dll), Protel(protel2.dll),...

6. Footprint

			PCB	
PCB desig	Footprint n program Layc	PCB out Library	가 .	, OrCAD
F	Footprint		3가	
⊯ Proper	ty Editor			
Footprint manager	Prop Footprint	erty Editor copy	, Property Editor	Edit property Layout program library Footprint
Lib Too Property Editor New, App 1 p: BCD : BCD 2 p: BCD : BCD 3 + BCD : BCD	rary manager ol menu My Display Delete Prope 1947 - 200717093 U47 1948 - 215060997 U48 1948 - 215060997 U48	Layout		Library Manager X Librarics METRIC EX. gui Add Remove Footprints DIP.100/24/W.300/L1.250 BLKCON.100/VH/TM120FW.20 BLKCON.100/VH/TM20E/W.20 BLKCON.100/VH/TM20E/W.20 BLKCON.100/VH/TM20E/W.300/H.1.250 DSUB/RP.318/TM/15 JUMPER200 OSC8H4P PLCC44 QUAD.65M/160/WG30.00 QUAD.65M/140/WG22.00 QUAD.65M/160/WG30.00 SIMM.050/VS_RP/TM/1.300/S SIP/TM/1.300/S SIP/TM/L.300/S SIP/TM/L.300/S SIP/TM/L.300/S SIP/TM/L.300/S
Parts 6	Schematic Nets (Pins (TI		Delete Footprint



🗷 Layout



7. Option

∠ Preference

grid'

.

Schematic design			option
menu .			
Preference option			
Schematic Page Editor		,	Part and
Symbol Editor			
Preference option		object	color ,
Grid	, Zoom	Scroll	Display % ,
option			
Preferences	×	cole	or/print
Colors/Print Grid Display Pan and Zoom Select Miscellaneous Ter Print Print Print	et Editor		

Print		Print	a and the second second	Print	V
7	glias		Hiererchical Pin	7	Bn
	Background	₽	Hierarchical Port	R	Ein Name
	Bookmark		Hier, Port Text	P	Pin Number
F	Bus	F	Junction	F	Eower.
	Connection Square	2	No Connect	P	Power Text
F .	Display Property	₽	Off-page Connecto		Selection
	QRC Marker	P	QH-page Cnch Tex	t 🖂 🔤	Jest
F	Graphics	P	Part Body	9	Die Black
Г [Grid	₽	Pert Body Rectangi	*F	Title Text
P	Herarchical Block	2	Part Reference	2	19 the
R	Hier, Block Name	12	Eart Value		Use Detaults
			8991	철소	5.89





가.

<i>ಹಹ</i> displayed	: schematic	part edit window	grid	
<i>ട്ടെ</i> grid style : grid	d	()		
ese pointer snap te	o grid :			

Schematic Page Editor Fill Style : None Line Style and Width: .	Text Rendering	vith strakes	► Text	
Color: Unsent Part and Symbol Editor Fill Style: None Une Style and Width:	Auto Recovery		→ .	
Session Log Foot: Avial 11	Intertool Communication	nication		Poforonoo
			name	Reference
Anno	tate 가	,	сору	
erence name				
rtool communicati	on Schema	atic Design	PCB Layout	



Cgarri D C Abgateli r Secondag G r Ngmerc r Ogscandro r Note: D bobes W	augt (<mark>6)</mark>
and the second s	ktth: 0.1 Inches
Conter Visible - Ge Displayed Displayed	ki Neterance Middle IP Displayed IP Pointer
다 Displayed 및 Distaid 다	AVE(grid references

Font Title block setting, Hierarchy option , SDT tool

.

Grid Reference

✓ Schematic page properties

Schematic page properties Design template

option schematic design Design template option

Schematic page window 7

Menu Bar Option pull down

Page Properties

chemetic Page Prop	erties			×	Schematic Page Properties	
Page Size Grid Re	aterence M	iscellaneous			Page Size [Cirid Pleterence Miscellane	eous)
Units (* jnches	с <u>м</u> і	Imeters			Horizontal Cgunt: (5	Vertical Couge 4
New Page Size	Width 9,700	Height 7,250	inches		C Abbabelic C Ascending C Nymeric C Discending Width: 10.1 Inches	← Aphabate ← Ageanding ← Numeric ← Desgending Width: 10.1 → Inches
C E C B	20,200	15,200 20.200	inches inches		Border Visible I Displayed I Printed	Grid Reference Viable IF Displayed IF Printed
⊂ E ⊂ Cystom	42,200)9,700	92,200 [7.200	inches Inches		The Block Visible I Displayed I Printed	P ANSI gid referances
Pin-to-Pin Spa	cing;	0,100	inches			
			19 19	취소 도움말		확인 취소 도움방

.

page size tap

unit

tapgrid Referenceschematicgridreference()(). Widthgrid Reference.

0 grid Reference가 .

8. (Hierarchical Structure)

가 가

,

,

3가 .





Manager Schematic



Page가

Z

Ľ

Off-page connector

Schematic page off-page connector

wire off-page connector net name off-page 가 bus connector off-page connector off-page connector name bus name off-page connector bus bus

(Hierarchical Structure) - 🗆 × sch3 PCB 🗀 File 🔩 Hierarchy 🖃 🛅 Design Resources Schematic 📥 🎇 ,₩sch3,dsn 가 🚊 🖄 SCHEMATIC1 🛄 🔁 PAGE1 dimultiflx 3가 🕂 🛅 Design Cache 🛅 Library 🗄 🛅 Outputs AHierarchical Block .₩sch3,drc E Referenced Projects AHierarchical Pin

Project Manager Schematic

- 31 -

Hierarchical Port

	,	Sc	hematic Page
Hierarchical Block	Block	Hierarch	ical Pin
Hierarchical Block			
Descend Hierarchy			
	Hierarchic	cal Pin	Hierarchical
Port가 .			
Hierarchical Block	Schematio	c Window	가
Tool palette 가			
Place Hierarchical Block		×	
Reference: Schematic2	ОК		
C Yes	Cancel		Reference
Default	User Properties	s	Implementation
	<u>H</u> elp		Type Schematic View
Implementation			, Implementation
Schematic View			name
Implementation name:			Page Schematic
Path and filename			
	<u>B</u> rowse,,	<u>. </u>	
	je		71
Enter	т.		가
. BIOCK	IC	ol palette	e Hierarchical Pin
	П		lical block
	Schomatic		lierarchical Part7
	Schemalic	page i	
1 / - (SCHEMATIC1 : PAGE1)	X		`
	🗐 /Sche	matic2 - (d	dimultiflx : PAGE1) 🗖 🗖 🗙
Schemaro2		5	
		x1 x1	
s 🕞 x1 y1 📕			
		-	
dimutifix	- K	./	n sulah sulah sulah .
			<u>)</u> //.



<Simple Hierarchy>



<Complex Hierarchy> block

.



Project manager

가

HALFADD.SCH

.





.

.

Reference name

Annotate

Mode Occurrence mode

9. Schematic Library

OrCAD Capture for Windows Library 30,000 가 library가 Part Symbol .

Part Library Design

New part libraryFileNew LibraryProgramLibraryFileOpen.New Library.





Part Reference Prefix : Reference name type . , Gate type ? U, Register type - R, Condenser type - C...

PCB FootPrint :

Footprint library가 New Part Properties Footprint PCB

Footprint Create Convert View : menu bar ?View ?convert Multiple Part Package : Gate가 Package type Library Part per Package , Gate Library Homogeneous, Heterogeneous parts per package가 4, 74LS00 , 74LS00 Not gate 6. .





가 part .

place pin Library Pin Pin name, Pin number, Pin type

Pin	Design Rules Check			
ce Pin		×		
<u>V</u> ame: Nu <u>m</u> ber:	Shape: Line Type: Passive	OK Cancel		
Width S <u>c</u> alar C <u>B</u> us	I Pin ⊻isible	<u>U</u> ser Properties <u>H</u> elp		

Name : name

Number :	Package		
Shape :		가	가
Type :			

가 .

.

3가

() Pin



Place Pin Array

•

Place Pin Array		×			
Starting N <u>a</u> me:	<u>S</u> hape:	ОК		,	
li Starting N <u>u</u> mber:	Type:	Cancel			
	Passive 💌	Help		Тс	ool
Number of			palette	Place Pin Array	
Increment:	₽ Pins <u>V</u> isible				
Pin S <u>p</u> acing: [1					

•

Starting Name	: Place	e Pin Array			
Starting Numbe	r :				
Number of :					
Increment :	1	2,2	3	가.	
Pin Spacing :				Pin Spaci	ng

EX)





Libr	ary		Data book		Pin	
		PCB				,
DRC	error가			Pin name, number		
	3	Pin type	Passive type			

Design save save as . Save library C: \ Program File \ OrCAD \ Library . ,

Save As

3 Library Part Symbol Symbol Title block, off-page connector, Hierarchical Port, Symbol library Part Library , Tool Palette Symbol .

; 1. ; library manager . Menu bar File ? New ? Library Project manager Library

가 .

2. ; Edit Part , Part edit copy .

3.Part & symbol Edit ; part library Net part New part properties 가 .



Part Edit Menu bar Option - Package Properties .

3. ; Part & symbol Edit Full Down menu가 Ctrl + V

4. ; copy save save as

.

10. Program Install

🖉 Stand-alone

- 12 Key Code Key Code

- Network Interface Card(LAN Card)가 Computer Program 가 Install authrize.orcad.com Computer , Key Code Code Request Authorization Code Install Install Locking Method Lock] [Rainbow Lock(green, or gray for legacy OrCAD product) .

Lock			!!		
Prog	ram	가 Computer	Program		
Install	Authorization	Code	가	()
02)517-37	′07				

- * Network Install

, Server	License Manager	Install	Program		Computer
IP		Moniter	Program	Install	

Program Install .

- Server Computer Program Install Installation Setup 47 Install OrCAD applications and libraries

- Client Computer Program Install Install Server Computer License Manager (C: \Program Files \OrCAD \LIC_MNGR), Client Computer Server Computer License Manager Install.

Server Computer Program Install Client Computer
 Program Install , Server Computer Program Install
 Client Computer Program Install
 Server Computer

Program Client Computer Install Local / Remote Install Remote , Client User 가 Server Computer , 가 Client Program Install - Server Computer License Manager Program

* Windows 95 System Program Install , Install CD Dcom dcom95.EXE . ***