Designing with OrCAD: Schematic Tutorial

This document is intended to provide step by step instructions to accompany the OrCAD Capture Tutorial Video Series. Short cuts are found in **(BOLD)**. For this Tutorial, use the provided design files in "Capture Tutorial Design Files.zip". Starting design files are provided for each video. To view the completed design, select "Completed Capture Tutorial.OPJ". Use the table of contents below to skip to a specific video.

Table of Contents

Getting Started

1. Download the zipped practice files.
2. Save the file to your computer and open.

Note: Included in the zipped file is a starting design file for every video (beginning with video 2), a completed design file, a Smart PDF of the completed design, the instruction guide and two library files.

3. Copy the library (.OLB) files to the standard OrCAD Capture Library folder using the path below:

   C:\Cadence\SPB_17.2\tools\capture\library

Video 1: Start a Schematic Project

1. Open Capture.

Note: You can locate your OrCAD software by going to the Start Menu → Cadence →OrCAD.

2. Click on New Project in the Start page.
3. Add a name for your project.
4. Select Schematic as the project type.
5. Browse for the location to save. Click ok.

Note: You can change the project type at any time by right clicking on Design Resources in the project directory and selecting change project type.

6. Select Options→Preferences from the menu.

Note: Here you can set preferences for colors, grid, pan and zoom, selection, miscellaneous items and text.

7. Close the Preferences Window.
8. Select Options→ Design Template.

Note: Here you can set the page size, grid reference and title block information.

9. Select the Title Block tab.
10. Add your information and set TitleBlock as the Title Block Name.

Note: Adding this information here will automatically populate the title blocks on any new pages.

11. Click ok to close.
12. Select File→New→Library.
13. Right click on the library. Choose "Save As" and save your library.

Video 2: Add Libraries and Parts

1. Select File→Open→ Library.
2. Use the following path to access the default libraries provided in Capture:

    C:\Cadence\SPB_17.2\tools\capture\library

3. Open the "CAPSYM.OLB" library.

Note: Right click on the library to dock the window to different locations on your screen.

4. Select Ground_Power and Titleblock3.

Note: To select multiple items just hold down the CTRL button and click.

5. Use CTRL-C to copy.
6. Back to the project directory, right click on the library and select paste.
7. Right click on a part and select Rename.
8. Rename the Parts to "GND" and "TitleBlock".
9. Right click on the library and select New part.
10. Fill in the following information:

    Name: USB-MicroB
    Part Reference Prefix: X
    PCB Footprint: USB-MicroB

11. Click Ok.
12. Click and drag to adjust the boundary box.
13. Select Place→Rectangle from the menu.
14. Click and drag to draw the rectangle.
15. Right click and choose End Mode **(ESC)**.
16. Select Place→ Pin from the menu.
17. Use the pin information below:

    Name: GND
    Number: GND
    Shape: Short
    Type: Input

18. Click OK.
19. Click to Place.
20. Right Click and select Edit Properties.
21. Use the pin table below to complete the pin placement.

Name	Number	Shape	Type
USBID	ID	Short	Input
D+	D+	Short	Input
D-	D-	Short	Input
VBUS	VBUS	Short	Input
MT1	MT1	Zero Length	Bi-Directional
MT2	MT2	Zero Length	Bi-Directional
MT3	MT3	Zero Length	Bi-Directional
MT4	MT4	Zero Length	Bi-Directional
P_1	P_1	Zero Length	Bi-Directional
P_2	P_2	Zero Length	Bi-Directional

Note: When the pin ends in a number the next pin placed will be sequential. You can continue to click and place MT1 to MT4 and P_1 to P_2 without editing properties in between.

22. Right Click and select End Mode when finished **(ESC)**.
23. Select Options→Part Properties.
24. Under Pin Name Visible select False.
25. Click ok.
26. Select the "Value" text. Click and Drag.
27. Right click and select Rotate.
28. Double click the text or right click and select edit properties.
29. Change the value to USB-B.
30. Click OK to close the window.
31. Right click the part tab and Save.
32. Right click the part tab and Close.
33. Select Accessories→ Ultra Librarian→ Open from the menu.
34. Log in with your information.
35. In the search window, enter LT1965.
36. Click Search.
37. Click the Preview button under part "LT1965EMS8E#PBF" for the schematic, PCB Footprint and 3D Model.
38. Select the arrow next to preview in the schematic column.
39. Click Download.
40. Save the created library.
41. Back to the project directory, select Open→Library.
42. Select the newly created Ultra Librarian library.
43. Copy the part **(CTRL-C)** and paste into your Library.
44. Right click to rename the part to LT1965.
45. Right click and select Edit or double click the part.
46. Click and drag to move pins and edit text.
47. Right click on the parts tab and select Save.
48. Right click on the parts tab and select Close.

Note: To view the paths where your Ultra Librarian parts will be saved, select Edit Download settings. Here you can set the paths for symbols, footprints, padstacks and 3D Models. You can also set your preference for functional or sequential pin layout and footprint units.

Video 3: Search and Place Parts

1. In the project directory, right click on Library and select Add File.
2. Select CaptureTutorial2.OLB from the downloaded files.
3. Right Click on the original library and select Cut **(DELETE)**.

Note: The CaptureTutorial2.OLB library includes all of the custom parts for this design.

4. Go to page 1 of the schematic.
5. Select Place→Part **(P)**.
6. Select LT1965.
7. Click the place part icon or double click on the part.
8. Click to place.
9. Right Click and select End Mode **(ESC)**.

Note: To rotate a part during placement use "R" on the keyboard or right click and rotate. You can do the same once the part is placed.

10. Place all the parts in the Capture Tutorial 2 library according to the provided "CaptureTutorial.PDF" file.
11. In the parts window, click the plus sign to expand "Search for Part".
12. Search for "CAP".
13. Set the following path to search the default Capture libraries:

> Cadence > SPB_17.2 > tools > capture > library

14. Click Search part.
15. Select the discrete library: CAP/Discrete.olb
16. Click Add.

Note: This has added the library and the library can now be searched in the parts window.

17. Double click to place or click the place part icon.
18. Click to place the capacitors in the schematic.
19. Right click and end mode **(ESC)**.

Note: The reference designators are automatically annotated. If you do not want the parts automatically annotated, go to Options→Preferences→Miscellaneous. Click off of Auto Reference. Any parts with question marks in the part reference can be annotated at the end of the design.

20. Search for "Resistor" in the parts window.
21. Double click RESISTOR/DISCRETE to place or click the place part icon.
22. Click to Place.
23. Right Click and select End Mode **(ESC)**.
24. Search for "LED" in the parts window.
25. Double click LED/DISCRETE to place or click the place part icon.
26. Click to place.
27. Right click and select End Mode **(ESC)**.
28. Select Place→Ground from the menu **(G)**.

Note: If you do not see the "CaptureTutorial2" library available, Click Add Library and Select "CaptureTutorial2.OLB".

29. Select the ground symbol and click OK.
30. Click to place on the schematic.
31. Right click and select End Mode **(ESC)**.

32. Select Place➔Power from the menu **(F)**.
33. Select VCC and click ok.
34. Click to place on the schematic.
35. Right click and select End Mode **(ESC)**.

Note: To add multiple libraries click the add library button. Select all of the default libraries. This will allow you to search all of the libraries in the place part window. Click the "X" button to delete a library from the parts window.

Video 4: Connect Parts

1. Select Place→Wire **(W)**.
2. Click each connection to add a wire.
3. Wire the schematic according to the provided "CaptureTutorial.PDF" file.

Note: To repeatedly place a wire hit F4 on the keyboard. Add wires to pins 1, 5 ,6, 7, 9, 10 of JP9, pin 7 of IC1, and pins 1 and 3 of JP2. We will be connecting these to buses or adding a net alias later in the tutorial.

4. Right click and select End Mode **(ESC)**.

Note: You can easily move sections of your schematic by highlighting and dragging the selection, a component or a wire.

5. Select Place→Bus **(B)**.
6. Click to place your bus.

Note: To add a bus at an angle hold down the "Shift" key.

7. Use Escape on the keyboard to end the bus.
8. Choose Place→Bus Entry **(E)**.
9. Click to place the bus entry to pins 5, 6, 7, 9 and 10 of JP9 to connect the wires and bus.

Note: Use "R" on the keyboard to rotate the bus entry.

10. Choose Place→Autowire→Connect to Bus.
11. Select the pins you want to connect then select the bus.
12. Type the name of the bus and click OK.

Note: If your nets have different names just type the first net and we will name the other nets at a later time. If your bus has sequential nets add the name, a bracket, and the set of numbers.

13. Finish wiring the buses according to the "CaptureTutorial.PDF" file.
14. Select Place→No Connect **(X)**.
15. Click on the pins that are not connected according to the "CaptureTutorial.PDF" file.
16. Right click and select End Mode **(ESC)**.

Video 5: Name Nets

1. Select Place → Net Alias **(N)**.
2. Type the name and click OK.
3. Click to place on the net.

Note: To rotate use the "R" key on the keyboard.

4. When all nets have been placed with that net name, right click and edit properties.
5. Add the new net name and select ok.
6. Place all the nets according to the "CaptureTutorial.PDF" file provided.
7. Right click and select End Mode **(ESC)**.

Note: By adding the same net alias to multiple nets, you do not need to physically connect them in the schematic. If your net has a number at the end of the name, this will be incremented every time you place the net alias.

Video 6: Define Differential Pairs

1. Go to the project directory.
2. Select the design file (capture tutorial.dsn).
3. Select Tools→Create Differential Pair from the menu.
4. Select either the manual or automatic generation below:

Manual

5. Select D+.
6. Click the > button.
7. Select D-.
8. Click the > button.
9. Click Create.

Automatic

5. Click Auto Setup.
6. Enter the following information:

   > Prefix: DP
   > + Filter: +
   > -   Filter: -

7. Click on the window to generate the differential pair.
8. Click Create.
9. Close out of the Auto Setup window.

10. Close out of the window.

Note: Defining differential pairs can be completed in either the schematic or PCB.

Video 7: Assign Part Information

1. Select all of the VCC components.

Note: To select multiple components hold down the control key.

2. Right click and choose edit properties.
3. Select the "Name" row, right click and edit.
4. Add 3.3V and click ok.
5. Right click and close out of the properties window.

Note: You can also assign the value during placement. Go to place → power. Select VCC. Before placing right click and choose edit properties. Add 3.3V for the value and click ok. Now the component will be placed with the correct value.

6. Highlight the section of series capacitors. Right click and edit properties.

Note: Make sure to select the parts tab.

7. Right click the "Value" row and select Edit.
8. Add 0.1uF and click ok.
9. Right click and close out of the properties window.

Note: Double clicking a component will also bring up with properties window. You can also double click on the text and change the value there.

10. Finish adding the values according to the provided "CaptureTutorial.PDF" file.
11. Right Click JP2 and select Edit Properties.
12. Right Click on Value and select Display.
13. Select Do Not Display.
14. Click OK.
15. Right click and close out of the properties window.

Video 8: Annotate Your Design

1. Go to the project directory.
2. Select the design (capture tutorial.dsn) file.
3. Select Tools→Annotate.
4. Select Reset Part References and Click OK.

Note: If you didn't need specific names assigned to components and just wanted to annotate your design, choose Unconditional Reference update. This will reset all the components and re-annotate your design. If you only wanted to annotate those components with question marks in the name, choose incremental reference update.

5. Go back to the schematic.
6. Add the following names by double clicking and editing text:
    > JP9
    > MAIN_PWR
    > JP2

Note: Reference "CaptureTutorial.PDF". The remainder of the components can be annotated automatically.

7. Back in the project directory, select Tools→ Annotate.
8. Choose Incremental Reference Update.

Note: Make sure the entire design is selected.

9. Click ok.

Video 9: Perform a Schematic DRC

1. In the project directory, select Tools→Design Rule Check.
2. In the Design Rule Options tab, select the following:

    Scope: Check entire design
    Mode: Instances
    Action: Check design rules
               Create DRC Markers for Warnings
    Design Rules: Electrical rules
                     Physical rules

3. Under Electrical Rules tab select the following items to check:

    Electrical Rules: Check single node nets
                      Check unconnected bus nets
                      Check unconnected pins

    Reports: Report all net names

4. Under the Physical Rules tab select the following items to check.

    Physical Rules: Check Power pin visibility
                     Check missing pin numbers
                     Check device with zero pins
                     Check power ground short
                     Check Name Prop consistency
    Reports: Report visible unconnected power pins
                     Report unused part packages
                     Report identical part references

Note: Typically, a check for missing and illegal footprints is performed; however, we haven't assigned footprints to our design yet, so leave this unchecked.

5. Click ok to run the Design Rule Check.
6. Double click on the output generated (capture tutorial.drc) in the project directory to view the report.
7. Go to the schematic.
8. Double click on a marker to see more information on the warning.

Note: To remove the markers go back to the project directory. Select Tools→Design rule check. In the Design Rule Options tab, under Action, select Delete Existing DRC Markers. Click Ok.

Video 10: Generate BOM

1. In the schematic, right click and choose selection filter **(CTRL-I)**.
2. Click Clear All.
3. Check the box next to Parts and Click OK.
4. Highlight the schematic.

Note: Only the parts will be selected.

5. Right click and edit properties.
6. To add a field, click the new property button. Add a name and click OK.

Note: Fields have already been added for Manufacturer, Manufacturer Part Number, Part Number and Vendor.

7. Fill in the information using the table below:

Item	Quantity	Reference	Part	Vendor	Manufacturer	Manufacturer Part Number
1	3	C1,C3,C9	10uF	Digi-Key	Murata	GRJ155R60J106ME11D
2	1	C2	100uF	Digi-Key	Murata	GRM32EC80J107ME20L
3	5	C4,C5,C6,C7,C8	0.1uF	Digi-Key	Samsung	RCS1005F151CS
4	1	IC1	PIC32MX440F512H	Digi-Key	MicroChip	PIC32MX440F512H-XMR
5	1	JP2	PIC 32 PROG	Digi-Key	TE Connectivity	640456-6
6	1	JP9	XTEND-DIGI-MODEM	Digi-Key		
7	1	MAIN_PWR	RED	Digi-Key	Lite-On, Inc.	LTST-C191KRKT
8	1	R1	150	Digi-Key	Samsung	RCS1005F151CS
9	1	R2	56k	Digi-Key	Panasonic	ERA-2AEB563X
10	1	U1	LT1965	Digi-Key	Linear Technology	LT1965EMS8EPBF
11	1	X1	USB-B	Digi-Key	Amphenol	10104110-0001LF
12	1	Y1	8MHz	Digi-Key	Abracon	ABM3-8MHZ-CL-R500-B1G

Note: Use CTRL-C and CTRL-V to copy and paste quickly in the properties window.

8. Right click and close the properties window when finished.
9. In the project directory, select the design (capture tutorial.dsn) file.
10. Select Tools→Bill of Materials.
11. Use the Header and Combined Property text boxes to customize the Bill of Materials or copy and paste the following text strings:

    Header: Item\tQuantity\tReference\tPart\tVendor\tManufacturer\tManufacturer Part Number

    Combined Property:
    {Item}\t{Quantity}\t{Reference}\t{Value}\t{Vendor}\t{Manufacturer}\t{Manufacturer Part Number}

Note: Make sure the precisely matches the properties added included spaces and case.

12. Check the box Open in Excel.
13. Click ok to generate the Bill of Materials.

Note: You can also view the bill of materials by double clicking the (capture tutorial.bom) in the project directory outputs.

Video 11: Generate Smart PDF

1. In the project directory, select File→Export.
2. Choose PDF.

Note: If this is your first time generating a smart PDF you may need to download Ghostscript. You will see this red text. Click on the link and download Ghost script.

3. With Ghostscript installed, use the path to select the gswin64c file as the converter path:

   C:\Program Files\gs\gs9.21\bin

4. Click OK.

Note: View part information by clicking on the part in the PDF. Click on a reference designator in the bookmark, the PDF will zoom to that part. The same can be done for pins and nets.

Continue with the PCB Tutorial Series to learn about netlisting, PCB footprint assignment and PCB Design.