



# Designing with OrCAD: PCB Editor Tutorial

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

## Table of Contents

Getting Started.....	2
Video 1: Assigning PCB Footprints.....	3
Video 2: Netlisting.....	4
Video 3: PCB Setup.....	5
Video 4: Mechanical Symbols.....	6
Video 5: Component Placement.....	7
Video 6: Differential Pairs and Constraints.....	8
Video 7: Copper Pours.....	10
Video 8: Routing.....	11
Video 9: DRC.....	12
Video 10: Manufacturing Preparation.....	13
Video 11: Manufacturing Export.....	15
Video 12: 3D Modeling and Collision Detection.....	16

## 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, a completed design file, a 3D PDF of the completed design, the instruction guide, a PCB library, and a STEP library.*

3. Copy all the files in the PCB Footprints and Padstack Library folder (.dra, .psm and padstacks) to the standard OrCAD PCB Editor Library folder using the path below:

C:\Cadence\SPB\_17.2\share\pcb\pcb\_lib\symbols

4. Copy all the files in the 3D Step Model Library folder (.step) to the standard 3D Model library folder using the path below:

C:\Cadence\SPB\_17.2\share\local\pcb\step

5. To find your OrCAD PCB Editor Software, select the Start Menu → Cadence → PCB Editor.
6. Click to open.
7. Select OrCAD PCB Designer Standard.
8. Set the paths for netlisting. Select Setup → User Preferences → Paths → Library.
9. Select "padpath".
10. Click Add New.
11. Browse for the directory of the pads library: C:\Cadence\SPB\_17.2\share\pcb\pcb\_lib\symbols.
12. Click OK.
13. Select "psmpath".
14. Click Add New.
15. Browse for the directory of the footprint library: C:\Cadence\SPB\_17.2\share\pcb\pcb\_lib\symbols.
16. Click OK.
17. Close the User Preferences Window.

*Note: To follow along with the tutorial, open the finished capture design by opening the "Capture Tutorial.opj" file and selecting OrCAD Capture from the pop-up window.*

## Video 1: Assigning PCB Footprints

1. Open Capture Tutorial.opj in OrCAD Capture.
2. In the project files, right click on Page One and select Edit Object properties.

*Note: This will show you the properties for all your components on Page 1.*

3. Open OrCAD PCB Editor Standard.
4. Select Place→Components Manually.
5. Choose the Advanced Settings tab and check Library.
6. Go back to the Placement List tab and select Package Symbol from the pull-down menu.
7. Search for “\*0402\*”.

*Note: This will bring up any footprint with 0402 in the name.*

8. Check the box next to the footprint to preview the 0402 Capacitor footprints.
9. Go back to the schematic and add “0402-CAP” as the PCB Footprint for the remaining capacitors.

*Note: You can use the shortcuts CTRL +C and CTRL +V to add the footprints.*

10. Back in the PCB, preview the 0402 resistor footprints.
11. Close OrCAD PCB Designer Standard.
12. In the schematic properties window in Capture, add the PCB footprint “0402-RES” to the resistors.
13. In the project files, select Accessories→ Ultra Librarian→ Open.
14. Log in to Ultra Librarian for OrCAD.
15. Search for “LT1965”.
16. Preview the symbol, footprint and 3D Model for part “LT1965EMS8E-1.5”.
17. Click the arrow next to preview in the footprint column and select Download.
18. Select OrCAD PCB Editor Standard to complete the footprint generation.
19. Open the folder where the Ultra Librarian footprints are saved.

*Note: To check the path where your Ultra Librarian part was downloaded, go to Edit Download Settings. Here you can view and change the path for your Ultra Librarian symbols, footprints, padstacks and 3D Models.*

20. Copy the MSOP-8\_MS.dra and MSOP-8\_MS.psm files.
21. Paste them into the standard OrCAD PCB Editor Library:  
C:\Cadence\SPB\_17.2\share\pcb\pcb\_lib\symbols
22. Open the folder where the Ultra Librarian padstacks are saved.
23. Copy “rx55y17d0t.pad”, “rx67y19d0t.pad” and “rx43y15d0t.pad” files.
24. Paste the .pad files into the standard OrCAD PCB Editor Library.
25. Going back to Ultra Librarian for OrCAD, download the 3D Model for “LT1965EMS8E-1.5”.

*Note: We will use this later in the tutorial.*

26. Going back to the schematic property window, assign the PCB Footprint of U1 as MSOP-8\_MS.

*Note: Always assign the PCB footprint as the name of the .dra file.*

27. Save and close the properties window.

*Note: The footprint, padstacks and 3D Model for this part have been included in the provided “PCB Footprint and Padstacks Library” and “3D Model Library” folders.*

## Video 2: Netlisting

1. In your project files, select the .dsn file.
2. Select Tools → Create Netlist.
3. Check Create PCB Editor Netlist.
4. Select the location you want the netlist to be exported.
5. Check create or update a PCB Editor Board.

*Note: If you do not want to create a board at this time, leave that option unchecked. You can always create your board file and import the netlist later.*

6. Select a location for the output board file.

*Note: We are creating a new board, so I will leave the input file blank.*

7. Select Open Board in OrCAD PCB Editor.

*Note: If this is your first time netlisting, you may need to set up the configuration file. To do so select Set up and browse for the path for the allegro.cfg file.*

8. Click OK to netlist your schematic.

*Note: If the netlist completed with warnings, the warnings will be visible in the session log. If there is an error netlisting you may need to set up the paths for footprints and padstacks in OrCAD PCB Editor. View the [Getting Started](#) section of this tutorial.*

9. Select OrCAD PCB Editor Standard from the pop-up window and the new board file will be generated.

## Video 3: PCB Setup

*Note: We are going to use the Design Workflow on the left of the screen to complete our PCB Design. If this window is not visible, select Display→Windows→Design Workflow.*

1. Select Setup from the Design Workflow.
2. Select Design Parameters.
3. Select the Design tab.
4. Set the Units to Millimeters.
5. Click OK.
6. Select Grid from the Design Workflow.
7. Set the spacing to "1" under "Non-Etch" in the "X" and "Y" directions.
8. Click OK.
9. Select Colors.
10. Leave the defaults and close out of the window.
11. Select Database Preparation from the Design Workflow.
12. Select Board Outline→Create→ Automatically.

*Note: Make sure "Create" is checked in the Design Outline window.*

13. Enter 0.3 MM for the Design Edge Clearance.
14. Select Place Rectangle.
15. Add the following dimensions:

Height: 100 MM

Width: 50 MM

16. Click to Place the Rectangle.
17. Click OK.
18. Select Cross Section (Stack-up) from the Design Workflow.
19. Right Click and choose Add Layers.
20. Select Plane.
21. Click Add.
22. Click Exit.

*Note: This has added the plane and dielectric layer.*

23. Select the "Layer\_1" text.
24. Right click and select Rename.
25. Rename the layer to "POWER PLANE".
26. Click OK to close out of the cross-section window.

*Note: If you did not select the option to create a new PCB when netlisting, choose Netlist→Import. Select Design Entry CIS and set the import directory as the folder location of your netlist. Click Import.*

## Video 4: Mechanical Symbols

*Note: If you do not want to create the mechanical symbol. Copy the .bsm and .dra files included in Video 4: Mechanical Library and paste in the standard PCB footprint library. C:\Cadence\SPB\_17.2\share\pcb\pcb\_lib\symbols*


1. Select File→New.
2. Name the file mounting\_hole1.
3. Select Mechanical symbol under Drawing Type.
4. Browse and save in the PCB library.
5. Select Layout→Pins.
6. Browse for a padstack and select "Hole 150".
7. In the command window type x 0 0 to place the padstack at the origin.

*Note: If you receive an error that the pin cannot be placed outside of the drawing extents you need to change the origin. Go to Setup→Design Parameters→Design. Type 100 in the x and y to move origin.*

8. Right click and select Done.
9. Select Shape→Circular from the menu.
10. In the Options tab, choose Route Keepout under Active Class and Subclass.
11. In the command line type x 0 0 to place at the origin.
12. Type x 3 in the command window.

*Note: This sets the radius of the circle. You can also drag the circle to the desired size.*

13. Choose Package Keepout under Active Class and Subclass.
14. Select Place Circle.
15. Set the radius as 3.
16. Type x 0 0 in the command window.
17. Right Click and select Done.
18. Save the file.
19. Open your PCB.
20. Select Place→ Mechanical Symbol.
21. Check the box next to mounting\_hole1.
22. Click to place on the board.
23. Place 4 Mounting Holes.
24. Close out of the placement window.

*Note: Make sure to Close the Window. Do not 'x' out the window or the last hole will not be placed. To move the mounting hole, select the Placement Edit button  on the toolbar. Click to select the hole and click to place.*

## Video 5: Component Placement

*Note: Reference the provided PCB Editor Tutorial\_Placement.png or PCB Editor Tutorial\_Placement.docx files for placement of components.*

1. Open the Capture Tutorial.opj file.
2. Select Options→ Preferences.
3. In the miscellaneous tab, check Enable Intertool Communication.
4. Set up a split screen with OrCAD Capture and OrCAD PCB Editor.
5. In OrCAD PCB Editor, go to Place→Interactive.

*Note: This enables the communication between Capture and PCB.*

6. Select Place→Components Manually.
7. Click IC1 in OrCAD Capture.
8. Move the mouse to the OrCAD PCB Editor Screen.
9. Click to place the footprint.
10. Do the same for JP9, JP2, and X1.

*Note: To rotate a part during placement right click and choose Rotate.*


11. In OrCAD PCB Editor, close the placement window.
12. In the Design Workflow, select Placement→Manual.

*Note: The already placed components have been removed from the list.*

13. Check the box next to “Components by refdes” in the placement window.

*Note: This will select all the components left to place.*

14. Place the remaining components.
15. Close out of the placement window when finished.
16. In the Design Workflow, select Report.

*Note: This will report any unplaced components. If any components need to be moved, select the Placement Edit button  from the toolbar. Click and move the parts as needed.*

17. Going back to OrCAD Capture select a component.

*Note: The selected component will be shown in OrCAD PCB Editor.*

18. Close out of OrCAD Capture.

## Video 6: Differential Pairs and Constraints

*Note: If you did not define your differential pairs at the schematic level, go to Set up → Add differential pairs. Click Auto Generate. Add DP\_ for the name and set the polarity to positive (+) and negative (-). Click Generate.*

1. In the Design Workflow, select Constraints.
2. Select Modes.

*Note: In the Modes window, select which constraints will be checked by setting the constraints “On” or “Off”.*

3. Close out of the Mode window.
4. Select Physical.

*Note: This opens the constraint manager window. In the constraint manager, you can assign electrical, physical, and spacing constraints. Assign constraints by creating Csets or assigning values to each individual net.*

5. In the constraint manager window, select the Physical Worksheet.
6. Select Physical Constraint Set → All Layers.
7. Right click, select Create → Physical CSet.
8. Name the constraint set POWER\_GND.
9. Click OK.
10. Set the following values:
  - Minimum line width: 0.381
  - Minimum neck width: 0.127
  - Maximum neck length: 5.08

11. Right click, select Create → Physical CSet.
12. Name the constraint set DP.
13. Click OK.
14. Set the following values:
  - Line Width: 0.127
  - Primary Gap: 0.127

15. In the worksheet selector, click on Net → All Layers.
16. In the column Referenced Physical CSet, assign the Cset POWER\_GND to the following nets:
  - 3.3V
  - GND
17. In the column Referenced Physical CSet, assign the CSet DP to the following nets:
  - DPD

*Note: Assign the differential pair Cset (DP) at the top level to apply to each net.*


18. Select the Electrical Worksheet.
19. Select Electrical Constraint Set → Routing.
20. Right click and select create → Electrical CSet.
21. Name the CSet DP.
22. Click OK.
23. Add the following values to the constraint set:
  - Gather Control: Ignore
  - Uncoupled Length: 5.08



Static Phase Tolerance: 0.508

24. In the worksheet selector, click on Net→Routing.
25. In the column Referenced Electrical CSet, assign the Cset DP to the following nets:  
DPD
26. Select Properties from the worksheet selector.
27. Select Net→General Properties.
28. Assign 0V to Net "GND".
29. Assign 3.3 V to Net "3.3V".
30. Close out of the Constraint Manager Window.



## Video 7: Copper Pours

1. In the Design Workflow, select Interconnect.
2. Select Shape→Rectangle.
3. In the options tab, Select Etch under Active Class.
4. Select Bottom under Active Subclass
5. Under Assign Net Name, Browse for the net.
6. Select GND.
7. Click OK.
8. Click to draw the rectangle on half of the board.
9. Click to draw an overlapping rectangle.
10. Right click and select done when finished.
11. Select the Shape Edit button  from the toolbar.
12. Right click on a shape and select Merge Shapes.
13. Click on the shape you would like to merge with.
14. In the Design Workflow, select Shapes→Polygon.
15. Keep the Active Class and Subclass the same.
16. Assign the net as GND.
17. Click and draw your polygon to cover the remainder of the PCB.
18. Right click and select Done.
19. Right click on a shape and select Merge Shapes.
20. Select the shape you want to merge.

*Note: OrCAD PCB has dynamic healing for copper pours. You can move your components and mechanical symbols and the copper will heal itself.*

21. In the visibility tab click off of “Conductor”.
22. Select Edit→Split plane from the menu.
23. Select Power Plane as the layer and click create.
24. Assign 3.3V.
25. Click OK.


## Video 8: Routing

1. Choose the Color button  from the toolbar.
2. Uncheck Bottom, Power Plane, Pastemask, Soldermask and Film Layers.
3. Select the Net tab.
4. Click a purple color and click the Net 3.3V.
5. Click a Blue color and click the Net GND.
6. Click OK.
7. Select Route→Create Fanout from the menu.
8. In the options tab, select Outward under Via Directions.
9. Click on the components with both pins connected to either ground or power.
10. In the Find tab, uncheck symbols and find only pins.
11. Click to fanout the remainder of the ground and power pins.
12. Right click and select done.
13. Select the Etch Edit button  on the toolbar.
14. Click on a DRC marked trace on IC1 and drag to move the via and trace on the inside of the IC.
15. Complete this for all the DRC error traces on IC1.
16. Select Edit→Change Objects.
17. Check line width.
18. Enter a line width of 0.28.
19. Click on all the traces routed for IC1 to change the width.
20. Right click and select done.

*Note: The DRC errors are now resolved, and the markers have been removed.*

21. In the Design Workflow, Select Interconnect →Manual Routing→Add Connect.
22. Click to place a trace.
23. Route the PCB.

*Note: In the options tab you can set the angle of the trace, change the line width, choose to shove or hug a trace, shove vias, smooth traces and auto blank all other rats. Click on one of the traces for the differential pair and they will be routed together. To add vias, right click and add via at any point during your trace. If you make mistake while routing, right click and select "Oops".*




24. Select the Etch Edit button  on the toolbar.
25. Click and drag to slide a trace.
26. Select Route→Custom Smooth from the menu.
27. Click on the trace you would like to smooth or highlight a selection and OrCAD PCB Editor will automatically smooth the trace.
28. In the Design Workflow, select Utilities→Display Status or Check→Design Status from the menu.
29. Ensure all the nets and connections are routed.
30. Close the Design Status Window.

## Video 9: DRC

1. In the Design Workflow, select Utilities→Design Status.
2. Click Update DRC.
3. Close out of the Design Status Window.
4. Select Edit→ Change Objects from the menu.
5. Check line width and add a value of 0.508.
6. Click on one of the traces for IC1.
7. Right click and select Done.
8. In the Design Workflow, select Utilities→Design Status.
9. Click the yellow button next to DRC.
10. Click on the coordinate to find your error or look at the PCB for the DRC marker.
11. Close out of the Design Status Window.
12. Select Edit→ Change Objects from the menu.
13. Change the line width back to 0.28.
14. Click on the trace with the DRC Error.
15. Right Click and select Done.

*Note: The DRC error is resolved, and the markers are removed.*

## Video 10: Manufacturing Preparation

1. Select the Color button  from the toolbar.
2. Click on the Nets tab.
3. Select Clear all Nets.
4. Click "Yes".
5. Click OK.
6. Click on the General Edit button  from the toolbar.
7. In the Visibility tab, select "On" for global visibility.
8. In the Find tab, select "All On".
9. Highlight the board.
10. Click on the Move button  on the toolbar.
11. Click and place your board at a new location.
12. Zoom in to the bottom of the board.
13. Select Setup→Change Origin.
14. Right click the corner of the board and choose Snap Pick to→Segment Vertex.
15. Select Place→Component Manually.
16. In the Advance Settings tab, select library.
17. In the Placement List tab, select format symbols from the pull-down.
18. Check the box next to ASIZEH.
19. Click to place the title block.
20. Close the placement window.

*Note: To edit any text in the title block, right click on the text and choose text edit.*

21. In the PCB, select the text for MAIN\_PWR and U1 that is off of the board.
22. Right click and select delete.
23. Click and drag to move any text.

*Note: To rotate select the text, right click and choose spin.*


24. Select Edit→ Change Object from the menu.
25. Click off of line width.
26. Check the Text Block box.
27. Change the Text Block to 2.
28. Click on the reference designator text for each component.
29. Move and edit the text as needed.

*Note: To add any reference designators to the design, select Add→ Text from the menu. Make sure the "Ref Des" is selected under Active Class and Subclass. Click on the component and click to place the text. Add the text. Right click and select Done.*

30. Select Setup→User Preferences.
31. Choose File Management→Output\_dir.
32. In the first line add the text: ".\artwork".

*Note: This will create a folder for your exported artwork.*

33. Select Manufacture→Customize Drill Table.
34. Click on Validate.
35. Click Auto generate Symbols and Yes.

36. Click OK to close the window.
37. In the Design Workflow, select Manufacturing Preparation→Documents→ Drill Chart.
38. Leave the defaults and click ok to create drill chart.
39. Click to place the drill chart.
40. In the Design Workflow, select Cross Section Detail.
41. Set the text block to 3.
42. Click OK and click to place.
43. Select Manufacture→Dimension Environment from the menu.
44. Right click.
45. Select linear dimension.
46. Click corners of the board to dimension length.
47. Click to place the dimension.
48. Click corners of the board to dimension height.
49. Click to place the dimension.
50. Right Click and select done.
51. Create the Fabrication Film. Select the Color button  from the toolbar.
52. Select off for Global Visibility.
53. In Geometry, select Design Outline and Dimension.
54. In Manufacturing, select NClegend1-3 and Xsection chart.
55. In Drawing Format, select All.
56. Click ok to close the window.
57. In the Design Workflow, Select Artwork (Film Records) Setup.
58. Right click in the film window and select Add.
59. Name this film FAB.
60. Click OK.
61. Change the Undefined line width to 0.127.
62. Close the Artwork Setup Window.
63. Create the remaining Artwork Films using this process and the table below.

DRAWING	COLOR CATEGORIES				
	STACKUP	GEOMETRY	COMPONENTS	MANUFACTURING	DRAWING FORMAT
FAB		Design Outline		NClegend1-3	All
		Dimension		Xsection Chart	
ASSEMBLY	Top: Pin	Assembly Top	AssemblyTop: RefDes		All
		Design Outline			
SILKSCREENTOP		Design Outline	Silkscreen_Top: RefDes		
		Silkscreen_Top			
SOLDERMASKTOP	Soldermask_Top	Design Outline			
		Soldermask_Top			
PASTEMASKTOP	Pastemask_Top	Design Outline			
		Pastemask_Top			
SOLDERMASKBOTTOM	Soldermask_Bottom	Design Outline			
		Soldermask_Bottom			

*Note: Remember to set the undefined line width to 0.127 for each film created. If you need to make changes to existing films, expand the film name. Add or cut layers as needed. To view a film in the PCB Window, right click and select display for visibility.*

64. Close out of the Artwork setup window.

## Video 11: Manufacturing Export

1. In the Design Workflow, select Manufacturing Deliverables.
2. Select IPC2581.

*Note: All the layers have been generated since we already set up the Gerber films.*

3. Select Layer Mapping edit.
4. Select Assembly, Fabrication and Silk as Documentation Layers.

*Note: All Other layers should be correct.*

<i>Soldermask/Solder paste:</i>	<i>Soldermask top</i>
	<i>Soldermask bottom</i>
	<i>Pastemask top</i>
<i>Inner Layers:</i>	<i>Power Plane</i>
<i>Outer Layers:</i>	<i>Top</i>
	<i>Bottom</i>

5. Click OK.
6. Check compress output file.
7. Select export.
8. In the Design Workflow, select Artwork (Gerber).
9. Choose general parameters tab.
10. Select Gerber RS274X and use the defaults.
11. Select the Film Control tab.
12. Choose Select All.
13. Click Create Artwork.
14. Click OK to close out of the Artwork Window.
15. In the Design Workflow, select NC Drill.
16. Click on Parameters.
17. Use the defaults and click OK.
18. Click Drill.
19. Close the window.

## Video 12: 3D Modeling and Collision Detection

*Note: Included in the folder for Video 12\_3D Modeling and Collision Detection is a starting design file (PCB Editor Tutorial.BRD) and a design file with completed STEP Mapping (PCB Editor Tutorial\_STEP.BRD).*

1. Copy any downloaded Ultra Librarian 3D Models to the standard 3D STEP Model Library using the following path:

C:\Cadence\SPB\_17.2\share\local\pcb\step

*Note: These models have been included in the zipped 3D Model Library in the PCB Editor Tutorial Files. If you need more information, view the "Getting Started" Video.*

2. In OrCAD PCB Editor, select Setup→Step Mapping.

*Note: If you are unsure where your STEP models are stored, select Path under Available STEP packages. This will give you the path for the STEP Model Library.*

3. Select 1210 package from the Available Package window.
4. Select the 1210.step model from the Available STEP Model window.
5. Click Save.
6. Complete the STEP Mapping for all the components using the table below.

*Note: If needed, set the x, y, or z rotation and the x, y or z offset according to the provided table. To set the rotation and offset, either type the numbers in the desired location or use the increment button. If any PCB footprints have been downloaded from Ultra Librarian, the STEP Mapping is already completed for these components (LED\_LNJ and MSOP-8\_MS).*

Package	STEP Model	Rotation			Offset		
		X	Y	Z	X	Y	Z
1210	1210CAP.step	0	0	0	0	0	0
0402-CAP	0402CAP.STEP	0	0	0	0	0	0
0402-RES	0402RES.STEP	0	0	0	0	0	0
1x06_Lock	conn1x6.step	0	0	0	0	0	0
CRYSTAL-SMD-5x3	Crystal4.step	0	0	0	0	0	0
LED_LNJ	LED_LNJ.STEP	0	0	0	0	0	0
MSOP-8_MS	MSOP-8_MS.step	0	0	0	0	0	0
SQFP-S-10x10-64	QFP64-16x16_PO_5.step	0	0	0	0	0	0
USB-MicroB	USB_MicroAB.STEP	0	0	0	-4	3.5	2.25
XTEND-DIGI-MODEM	xtend_digi_modem.step	90	180	0	31	18.4881	6

7. Click Add Mechanical.
8. Name the new mechanical item, STEP3D\_MECH\_USB.
9. Click OK.
10. Map USB.STEP to the new mechanical item using the table below.

Package	STEP Model	Rotation			Offset		
		X	Y	Z	X	Y	Z
STEP3D_MECH_USB	USB.STEP	90	90	0	19.5	58.75	5.5

*Note: Under View, select preset views to ensure the STEP file is mapped correctly.*


11. Click Save.
12. Click Add Mechanical.



13. Name the new mechanical package, STEP3D\_MECH\_ENCLOSURE.
14. Click OK.
15. Map Enclosure1.STEP to the new mechanical package using the table below.

Package	STEP Model	Rotation			Offset		
		X	Y	Z	X	Y	Z
STEP3D_MECH_ENCLOSURE	Enclosure1.STEP	90	0	0	50.5	25	4

*Note: Using the 3D Canvas in OrCAD we will be able to turn this off and on to view our board.*

16. Click Save.
17. Close the STEP Mapping Window.
18. Select the 3D button  from the toolbar or Display→3D View from the menu.
19. In the symbols tab, click off of STEP3D\_MECH\_ENCLOSURE.

*Note: You can select which layers are visible in the visibility tab. You can select which models are visible in the symbols tab.*

20. Turn the enclosure back On.
21. Select the Collision Detection tab.
22. Click Calculate.

*Note: The problem components will appear in the window. The first item “X1” is reporting the “collision” between X1 on the PCB and the USB. This can be ignored.*

23. Right click on an item and choose Locate.

*Note: This will make the component flash for easy detection. You can also select the component from the list. This will highlight the component.*

24. Select Setup→ Preferences.
25. Set the color preferences or leave the default.
26. Under Interactive, select Enable.
27. Select Cutting Plane.
28. Select Enable Cutting Plane.
29. Select X-Axis.
30. Move the Offset bar to view the cross section of the board.
31. Uncheck Enable Cutting Plane.
32. Close out of the Preferences window.
33. Set up a split screen between the PCB and the 3D View.
34. Click to move the connector JP2 in the PCB to line up with the existing cutout on the enclosure.

*Note: The changes will appear automatically in the 3D view once the part is placed.*

35. Re-route the connections to JP2.

*Note: Delete the existing traces for JP2. In the Design Window, select Interconnect→Manual Routing→Add Connect. Click to place the trace. Right click and select Done when finished.*

36. Back in the 3D Window, select View→Camera→Back.

*Note: To rotate the board hold down the shift key and middle mouse button or you can also choose from preset views under View→Camera in the menu.*

37. Right click and select measure.

38. Click each point to measure the required opening for the USB.

*Note: The measurement will appear in the options tab.*

39. Back in the PCB, select Setup→Step Mapping.

40. Select the STEP3D\_MECH\_ENCLOSURE in the Available Package Window.

41. Map Enclosure2.STEP to the mechanical package using the table below.

Package	STEP Model	Rotation			Offset		
		X	Y	Z	X	Y	Z
STEP3D_MECH_ENCLOSURE	Enclosure2.STEP	90	0	0	50.5	25	4

42. Click Save.

43. Close the Step Mapping Window.

44. Select Display→3D View.

45. In the Collision Detection tab, select Calculate.

*Note: The errors have been resolved and only the collision between X1 and the USB is reported.*

46. Select the symbols tab.

47. Uncheck STEP3D\_MECH\_Enclosure and STEP3D\_MECH\_USB.

48. Select File→Export.

49. Select 3D PDF from the pull-down menu.

50. Name the file.

51. Click OK.

52. View the 3D PDF.