Component management

In OrCAD Capture CIS you can easily manage components and their related information. Assembly variants and Bill of Materials (BOM) for these can also be defined.

The built in Part Manager window makes it easy to work with the components with easy sorting, linking to alternative parts numbers and creating BOMs. The basis is the core design which represents the schematic as it was drawn.

Table of Contents
Component management.........................................................................................................................1
Starting Part Manager ..............................................................................................................................1
Creating variant groups and assigning alternate values (part numbers) ..............................................2
Define the BOM Variants.........................................................................................................................4
View the variants on the Schematic.........................................................................................................6
Assigning part number to the undefined opamp....................................................................................6
Adding connector for power nets and filter input/output ......................................................................8
Creating Bill of Materials (BOM) ...........................................................................................................8
Creating intelligent PDF Documentation ...............................................................................................11
What did you learn? .................................................................................................................................13

Starting Part Manager
This part of the tutorial continues the work previously made, where a Lowpass filter was drawn using parts from the component database.

1. Go to the project manager

2. Select the line ‘Lowpass filter.dsn’ and right click → Part manager
The part manager shows the 4 components in the design. The 2 resistors and the capacitor has a Part Number since they were placed from the database. U1 has no part number but it is still possible to create BOMs and go to PCB Design. Since there are no part number there is probably no purchasing information on this part. We will solve this later in this tutorial.

The Part Status shows how well the part data on the schematic page fits with the part data in the database. At any time a status update can be run and any discrepancies between the data in the schematic and the component database will be highlighted.

Creating variant groups and assigning alternate values (part numbers)

C1 (0.01uF) and R1 (15.8K) have a large influence on the cutoff frequency. Changing C1 to 0.03uF and R1 to 1K will result in a cutoff frequency = 5kHz.

Instead of drawing another circuit with these values for C1 and R1 it is possible to make 2 different assembly variants. For this purpose C1 and R1 is put into a group which can be mounted in 2 different alternatives. The following will show how this can be done very simply.

3. **Right click on Groups** and create a new group named ‘CutoffFrequency’

4. Now create the 2 mount alternatives (subgroups) ‘1kHz’ and ‘5kHz’ by **Right click on CutoffFrequency → New Subgroup**

5. To make variants on C1 and R1 they need to be moved into the CutoffFrequency group. **Go back to the schematic page and select R1 and C1**, then **Right click → ‘Add Part(s) to Group’**
6. Now select CutoffFrequency and click Add. Click OK to the information that popup. Now both the components are present in the 1kHz and 5kHz subgroups of CutoffFrequency in the part manager.

7. Go back to the Part Manager and select the 5kHz subgroup.

8. This is the place where C1 and R1 is changed to get a 5kHz cutoff frequency

9. To replace R1 with a 1k resistor either Right click R1 → Link Database Part or double click on R1

10. In CIS Explorer type 1k in the Value field and press Enter to do the search and select the resistor with Part_Number=CIS-00003 by double clicking on CIS-00003
**TIP:** Remember sorting can be done by double clicking a column header.

11. Perform the same operation on C1 to replace it with Part Number CIS-001 with a value of 0.03uF

12. Now both parts inside the subgroup 5kHz has a green checkmark which symbolizes that these parts do not use the same part number as the core design.

**Define the BOM Variants**

In the BOM Variants section of the part manager the different variant Bill of Materials can be defined.

13. **Right click BOM Variants → Create BOM Variant** and create 2 BOM Variants named ‘FC 1kHz’ and ‘FC 5kHz’

With the BOM Variants defined it is necessary to specify which components are mounted and how in each of these BOM Variants.
The components that are common between the 2 BOM variants are in the ‘Common’ group. Dragging ‘Common’ into both BOM variants will ensure that those components are mounted as in the core design. From the CutoffFrequency group it is necessary to select which mounting subgroup represent the assembly of a certain BOM Variant.

14. To specify the content of BOM Variant ‘1kHz’ drag ‘Common’ and ‘CutoffFrequency→1kHz’ into the BOM Variant named ‘FC 1kHz’

15. Repeat the operation above where you drag ‘Common’ and ‘CutoffFrequency → 5kHz’ into the BOM Variant ‘FC 5kHz’

16. Select BOM Variant ‘FC 5kHz’ to see how the parts are mounted in that BOM

It can be seen that R2 and U1 are mounted without any change from the core design since there are no green checkmark and no red cross which would symbolize that the component should not be mounted. Also R1 and C1 are mounted using another part number than in the core design, this is shown with the green checkmark.
View the variants on the Schematic

17. Go to the schematic page and display your design variants using View → Variant View Mode or use shortcut Shift + m

18. Select either one of the Variants and notice how the schematic changes into showing the correct values used in the selected variant.

19. Press Shift + m and select <Core Design> to go back to the core design

Notice: While showing the variants most functionality is disabled. However, documentation can be made, like PDF creation.

Assigning part number to the undefined opamp

The opamp TL071 is still not defined with a part number. From a BOM and PCB perspective it is possible to continue without assigning it a real part number from the component database.
However to create proper purchasing information it is highly recommended to assign it to a part number in order to have correct purchasing information in the BOM.

20. Select the opamp ‘U1’ and Right click → Link Database Part or use the shortcut Ctrl + L which will open CIS Explorer.

21. The Link Database function will look for part with the same value as the selected opamp, change this to ‘TL071’ and press Enter to make a search

22. The part on the schematic page will be replaced with the official part from the component database, set the checkmark to accept this and click OK

Note that the circuit will still simulate, the only difference is that the part now has purchasing information.
Adding connector for power nets and filter input/output

23. Open schematic page ‘PAGE1’ and **press ‘z’** for Place Database Part, then find and place CIS-1553

24. After placing the part connect all external connections as shown and use a NoConnect symbol on pin 4 (shortcut ‘x’)

Note: The circuit will still simulate with PSpice, but a warning will occur since there is no simulation model for the connector and for that reason it will be ignored during simulation.

The connector will also automatically be added to the common group and hence be part of both assembly variants.

Creating Bill of Materials (BOM)

Previously 2 assembly BOM variants was defined in Part Manager. In this section the actual BOM file will be created.

25. Start or go to Part Manager like previously
26. Select the BOM Variant ‘FC 1kHz’ and Rightclick → Reports → Standard BOM

In the Standard BOM it is possible to define a list of standard reports that can be made at any time during the design cycle. If the component database contains pricing, then it would be easy to get a price indication at any point or a list of obsolete parts.

27. Make the following settings in the Standard Bill of Materials dialog
a. Change Template Name to ‘My Report’
b. Add the following from Report Properties into Output Format and reorder as shown
   i. Part Number
   ii. CIS Manufacturer Parts: Manufacturer
   iii. CIS Manufacturer Parts: ManufacturerPN
c. Change Max Rows to ‘2’
d. Select ‘FC 1kHz’ at the bottom
e. If you have Excel on your PC then enable ‘Export BOM report to Excel’
28. Click **OK** to create the BOM

Unless the Excel option is selected the BOM is generated directly inside OrCAD Capture CIS and can be saved in a number of different formats with File → Save As or by closing the report.

Since the number of Max Rows was set to 2 there can be up to 2 Manufacturer parts per part in the BOM. As can be seen from the BOM it is possible to use either a Murata or Kemet part for this capacitor. Providing a BOM with alternatives minimizes the risk of component shortage and also offer the possibility to have cheaper alternatives mounted.

29. Select the report tab and **Right click → Save**
This will save the report and add it to the Output section of the project manager.

30. Close the report tab.

31. If you wish to do so, repeat the previous steps to create a BOM for ‘FC 5kHz’.

Creating intelligent PDF Documentation

PDF documentation can be created for each of the variants. When working in Variant View Mode the selected Variant can be printed to PDF.

32. Select the Schematic page tab named ‘PAGE1’.
33. **Press Shift + m** and **select the variant ‘FC 5kHz’**.
34. Now the schematic is shown for the variant 'FC 5kHz' like before.

35. To create a PDF of this variant select **File → Export → PDF**.
36. Click **OK** to create the PDF document. The document will be shown after creation, left click a component to see all the information like part number, footprint etc.

37. Repeat the steps if you want to create a PDF document for ‘FC 1kHz’

38. Back on Schematic page ‘PAGE1’ do a Shift + m and select <Core Design>

**What did you learn?**

✓ How to use the Part Manager to create assembly variants
✓ How to assign part numbers to an undefined part
✓ How to create Bill of Materials with alternative manufacturer data
✓ How to create PDF Documentation for each variant