Creating a circuit with OrCAD Capture CIS

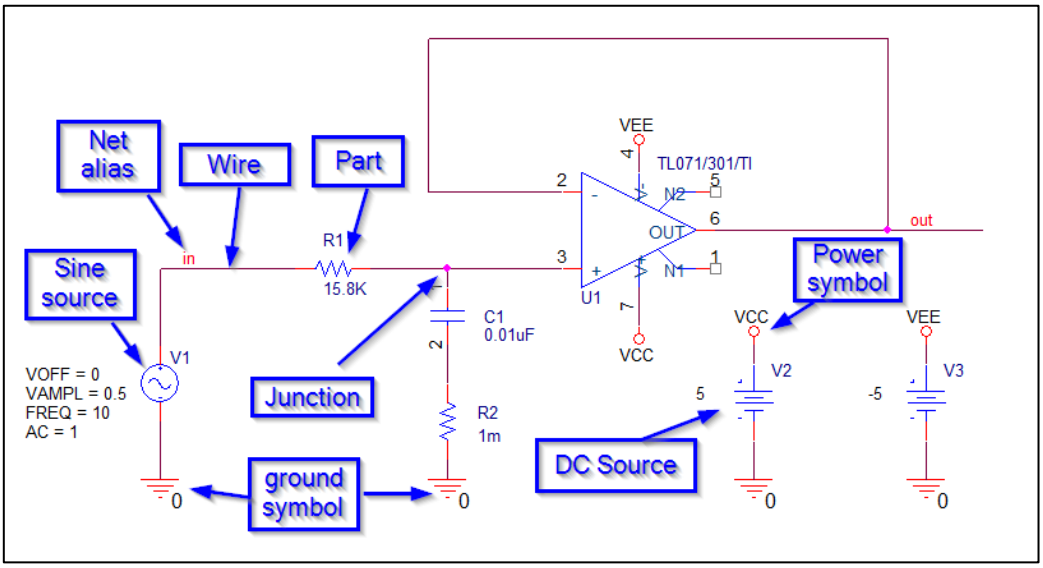
In this part of the workshop you will be creating the schematic for a lowpass filter including the parts necessary for simulating the circuit with PSpice.

Table of Contents

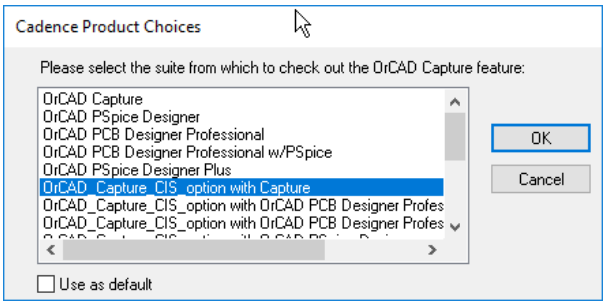
Creating a circuit with OrCAD Capture CIS 1
Schematic Capture 1
Placing Parts from the component database 3
Placing parts using PSpice Search 5
Placing sources for PSpice simulation 7
Placing power and ground symbols 8
What did you learn? 9
Component databases 10
Keyboard shortcuts 11

Schematic Capture

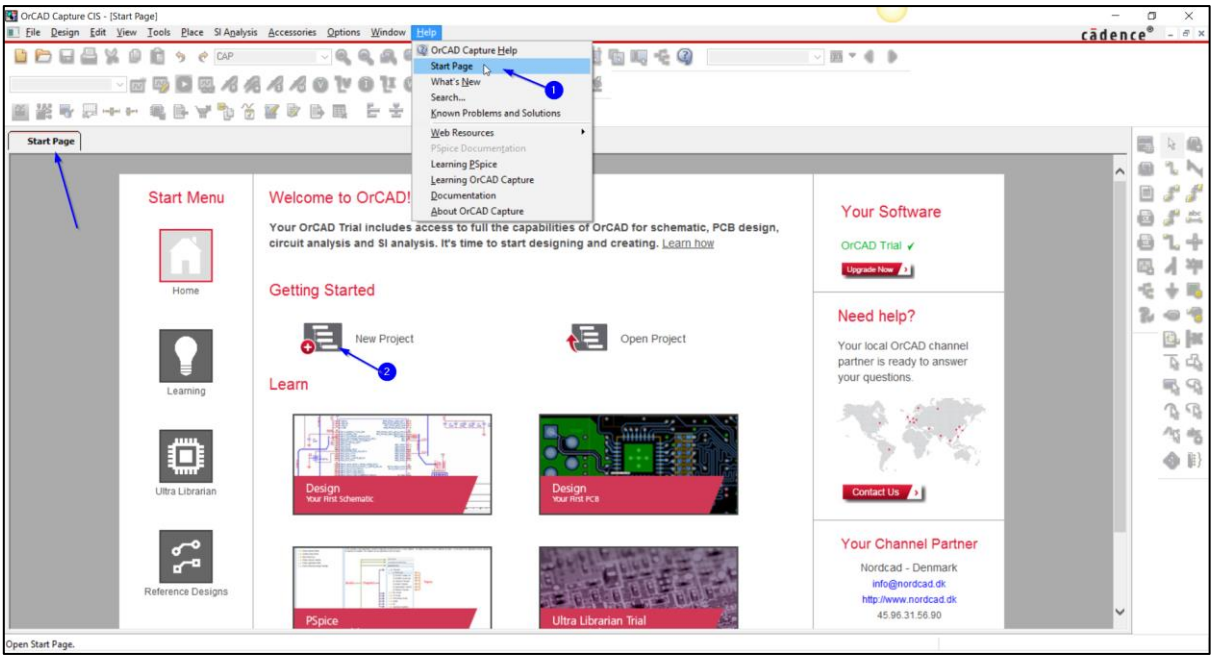
The final schematic design is shown below.



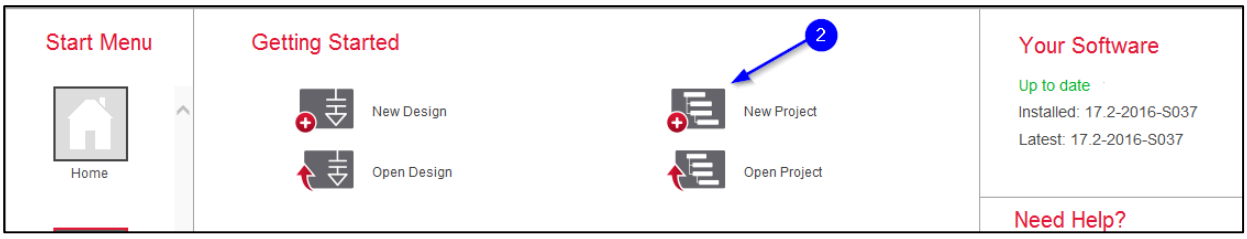
1. Be sure to start OrCAD Capture with the CIS option



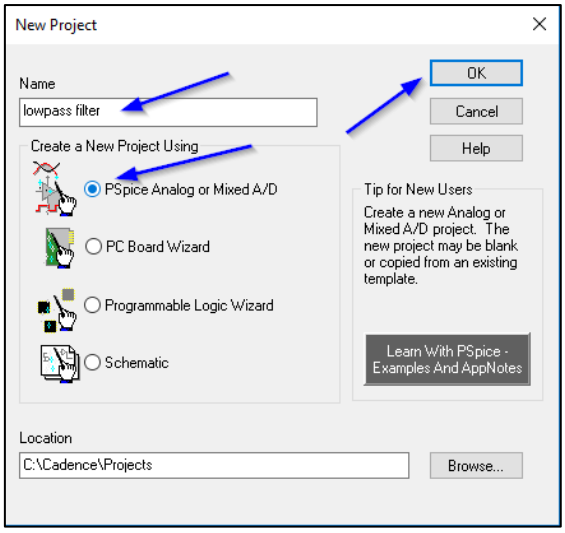
2. At startup the Start Page will be shown. If not select ***Help → Start Page***.
3. On the Start page select ***New Project.***



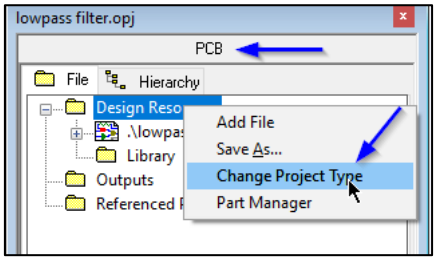
or



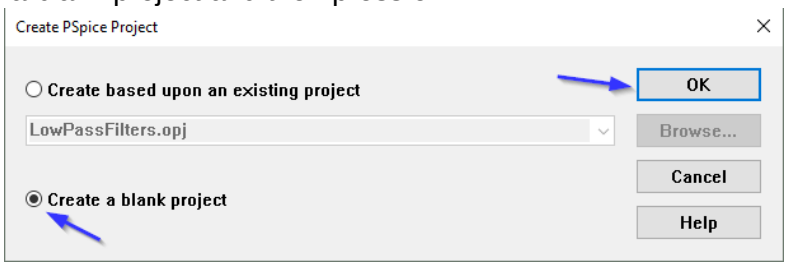
4. When you create the new project specify a name. In this case name the project ***"Lowpass filter"*** as shown.
    a. Make sure to ***select "PSpice Analog or Mixed A/D"*** project in order to get all the PSpice functionality.



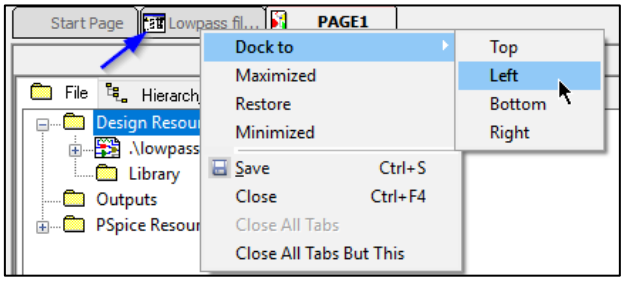
TIP: In case your project type by mistake was set to something else then it is easy to change from the project manager using the Change Project Type function shown below.



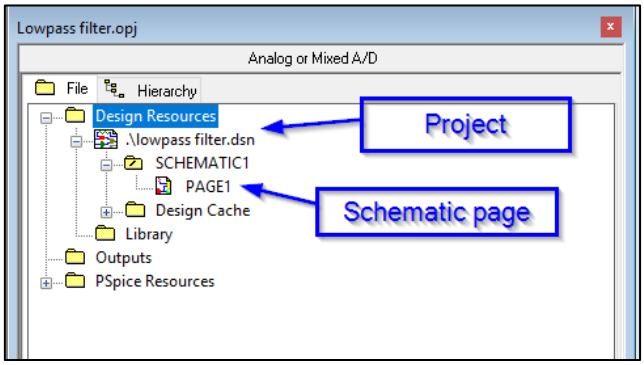
5. Click the create a blank project and then press ok.



6. It is often an advantage to be able to see the project manager. Right click the tab named "Lowpass filter" and select ***Dock to → Left***

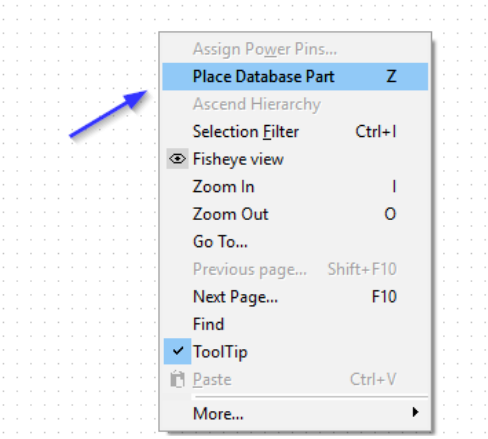


7. To find your schematic inside OrCAD Capture, look under your project DSN file.

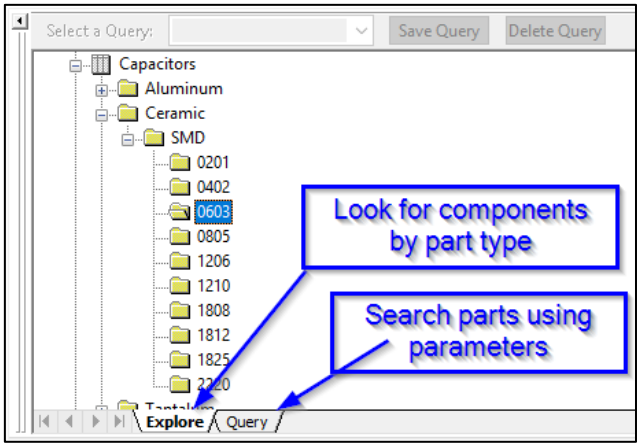


Placing Parts from the component database

8. Now start placing parts needed for the Lowpass filter. On the empty schematic page ***right click → Place Database Part***. (Shortcut key: z)

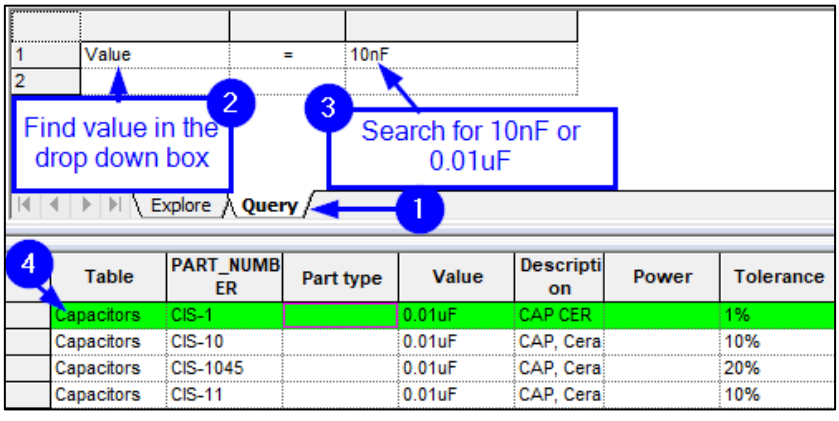


9. A component database is shown in the ***CIS Explorer***. Parts can be found either through
   a. "Explore" where parts can be identified from the categorization into part types
   b. "Query" where parts can be found through parameter search



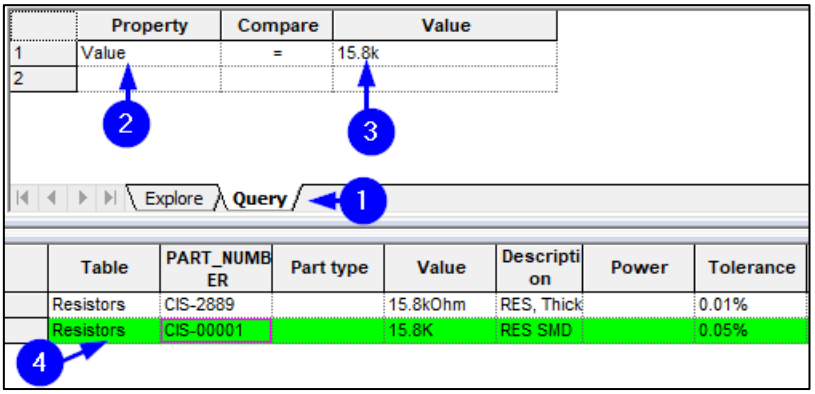
10. Find the part with part number CIS-1 either through the Explore tree "Capacitors→Ceramic→SMD→0603" or using Query as shown below

Capacitors

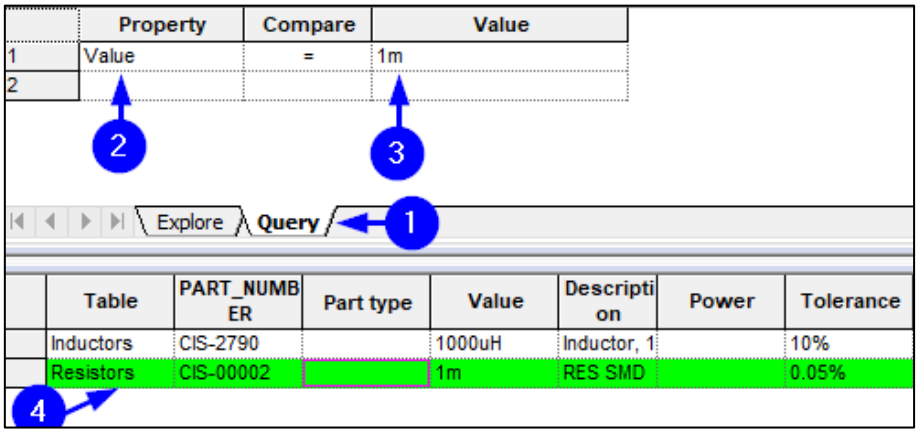


11. Double click the line (4) above to place the part on the schematic page
12. Press 'z' again to get back into the CIS Explorer and search for the 15.8k resistor and place the resistor on the schematic page

Resistors



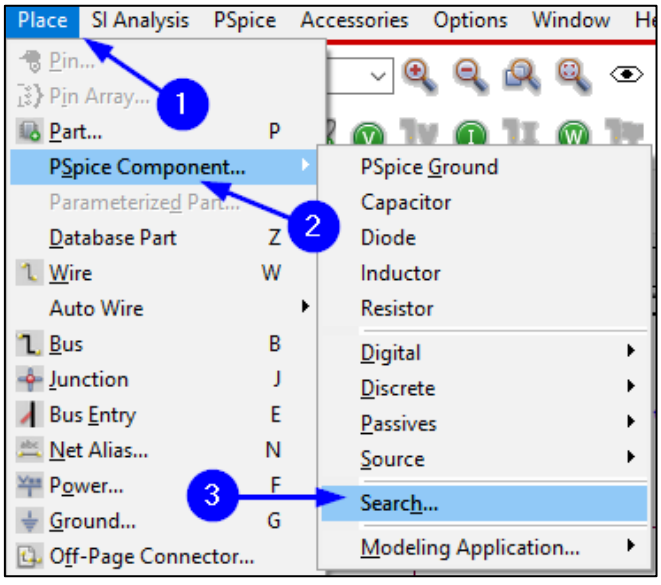
13. Use the same methodology to find and place the resistor with the value of 1m



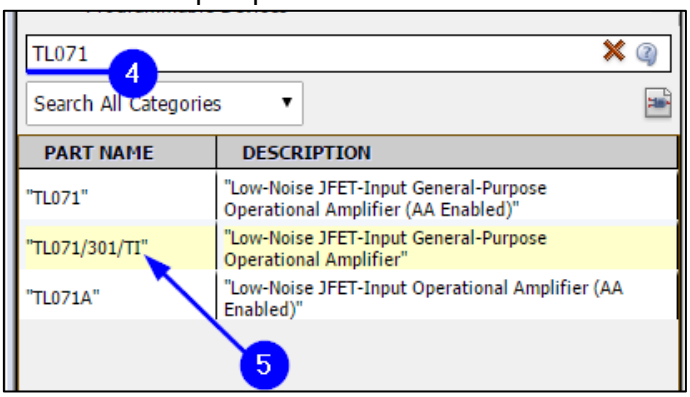
Placing parts using PSpice Search

14. To place the opamp we're going to search in the built in library of parts with PSpice models.
Select Place → PSpice Component → Search

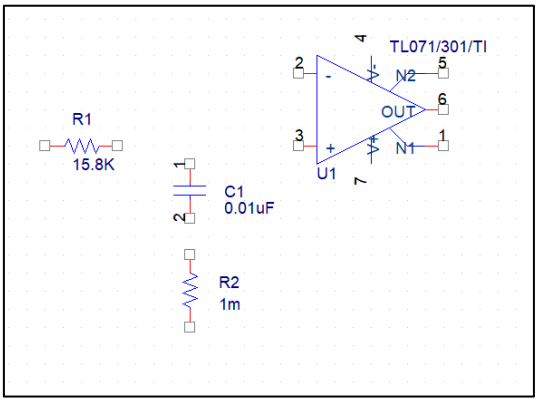
IC



15. Type in 'TL071' to search for the opamp

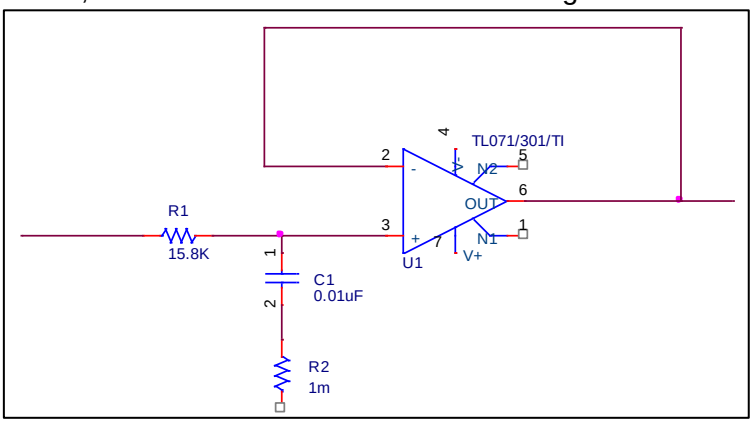


16. Double click the line "TL071/301/TI' above in order to place the component
    a. The opamp can be mirrored to have the negative pin on top either by ***Right Click*** → ***Mirror Vertically*** or using shortcut ***'v'***



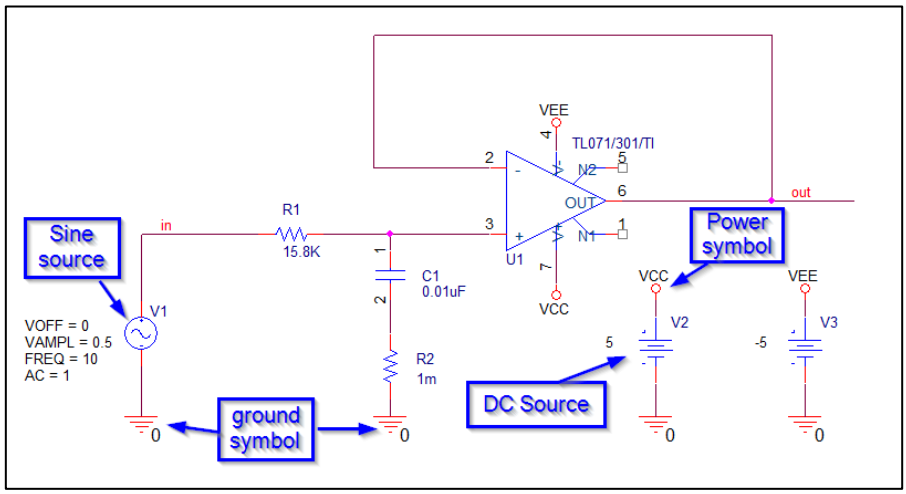
17. Now the wiring can begin. The shortcut for placing a wire is "W"

18. After the wiring is done, the schematic should look something like this.

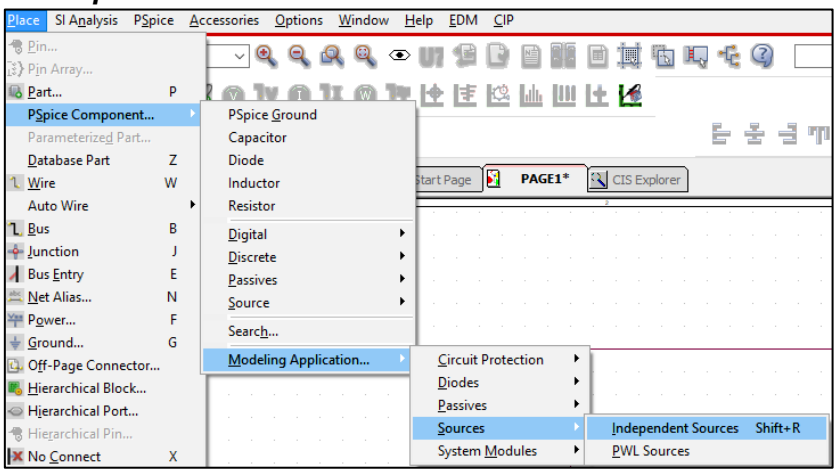


Placing sources for PSpice simulation

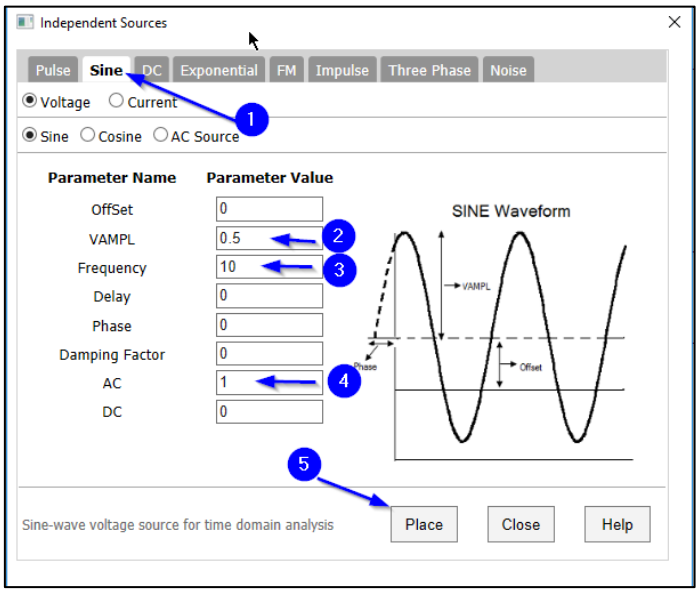
19. We will place power/ground symbols and sources for PSpice simulation.



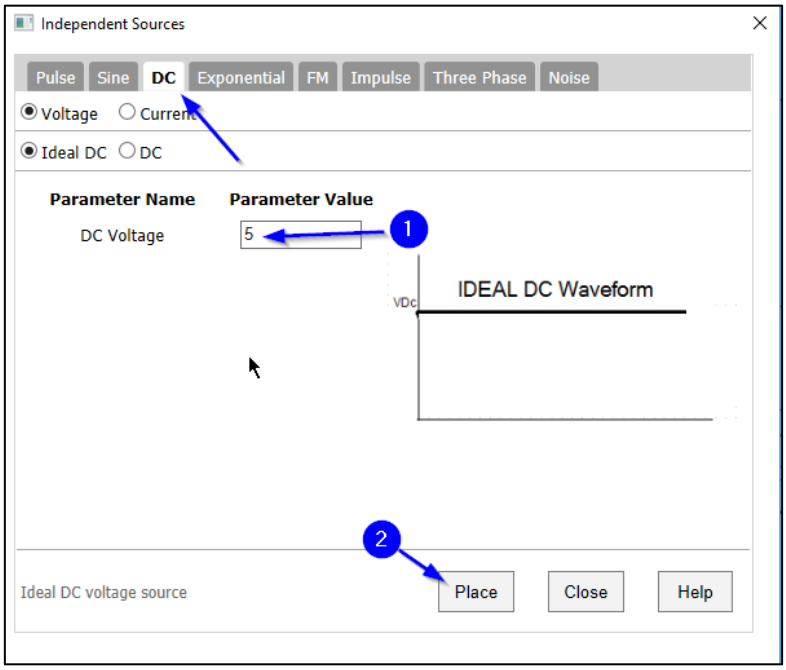
20. To place a sine source, go to ***Place → PSpice Component → Modelling Applications → Sources → Independent Sources***



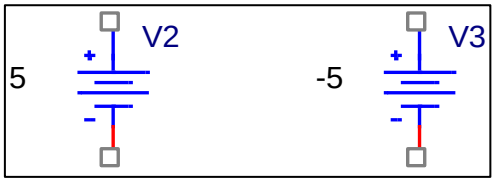
21. Set the parameters for the source as shown below and place it on the page



22. To place a DC voltage source use a similar function. Go to ***Place*** → ***PSpice Component*** → ***Modelling Applications*** → ***Sources*** → ***Independent Sources***
23. Select the tab ***'DC'*** and set the DC Voltage = 5 and place the source

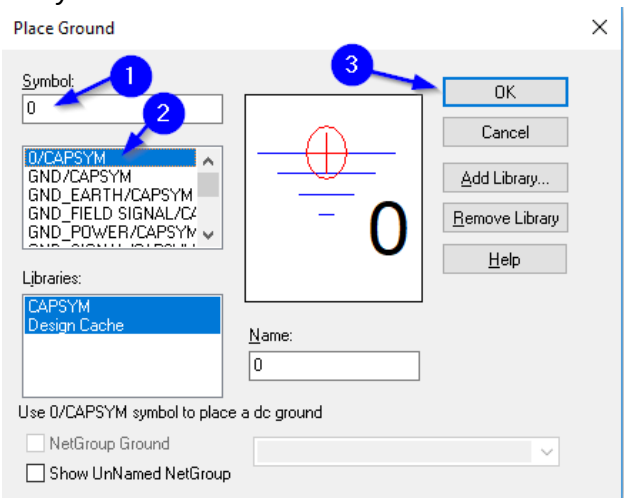


24. Copy the DC source that was just placed using ctrl+c and ctrl+v
25. Change the value of the new DC source to -5V by double clicking the value



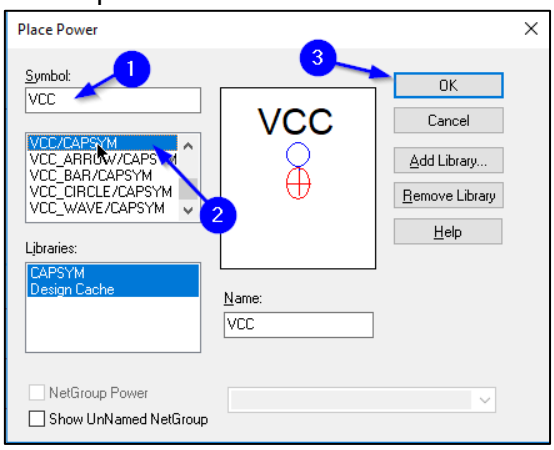
Placing power and ground symbols

26. To place power and ground symbols use ***Place*** → ***Power/Ground*** or shortcut ***'g'***
27. PSpice simulations always needs a 0 reference and to place this start typing '0' in the Place → Power/Ground dialog
28. ***Place 4 x '0'*** ground symbols

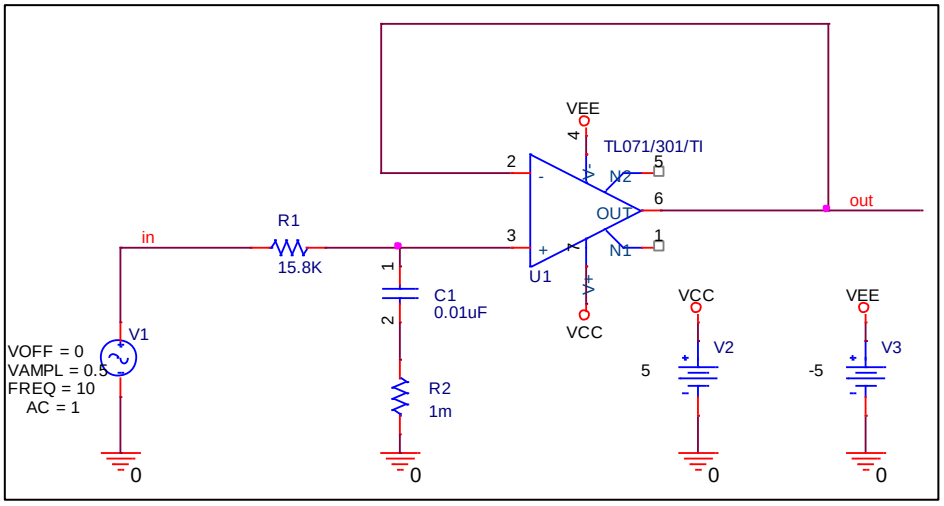


29. In the Place power/ground dialog type ***'VCC'*** to see the power symbols with names starting with VCC

30. Select VCC and click OK to place it.



31. Make 3 copies of the VCC symbol just placed and rename 2 of those to 'VEE'
32. Now finish up the schematic to look like the one below
    a. To name the nets ***'in'*** and ***'out'*** select Place → Net Alias or use shortcut ***'n'***. Then type the name of the net and place it on the wire.



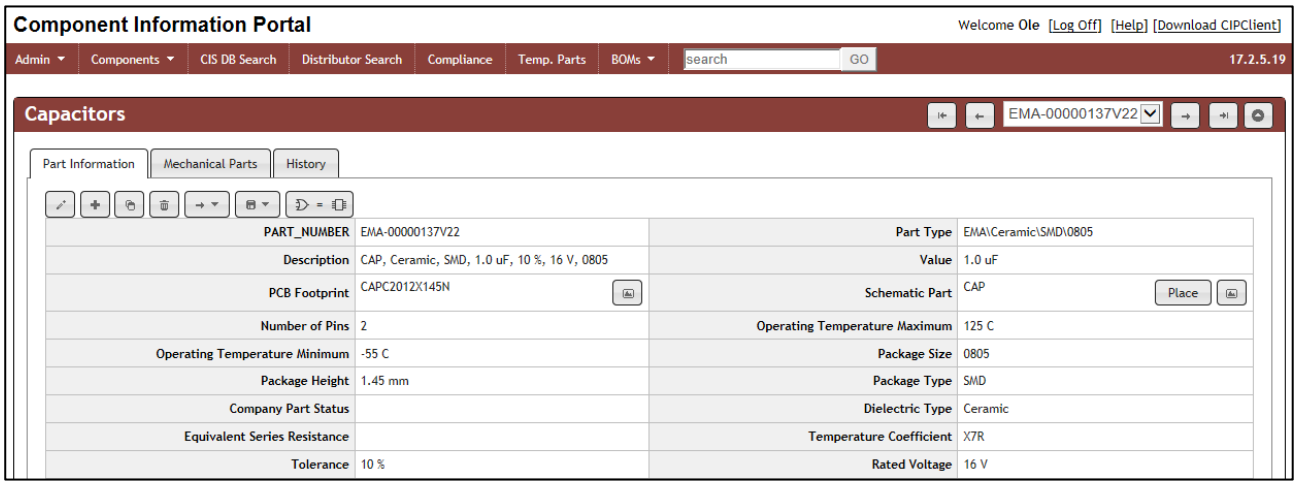
33. The Circuit is now ready.

What did you learn?

- ✓ How to create schematic design for PSpice simulation and PCB Design
- ✓ How to search for and place parts using a component database
- ✓ How to search for PSpice components
- ✓ How to place sources for PSpice

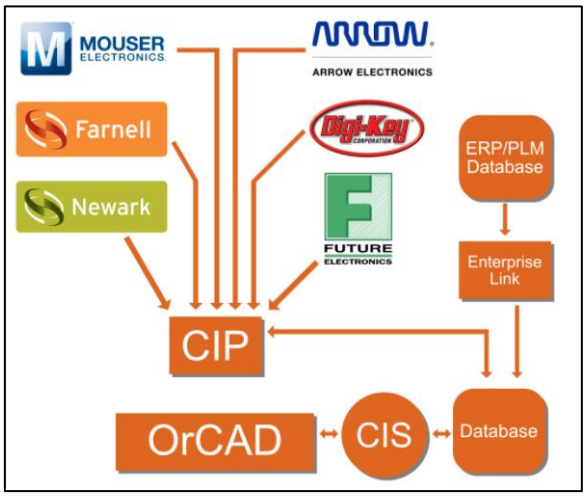
Component databases

For most companies a good component database is the basis for a solid design flow. Most OrCAD solutions are delivered with OrCAD Component Information Portal (CIP) which is a component database solution targeted at the electronics development engineer. It contains all the technical information relevant for designing electronic circuits like tolerances, temperature coefficients, ratings etc. The system can easily be integrated with existing purchasing/ERP/PLM/PDM systems for updated information on pricing and availability.



On top of this CIP also delivers key functionality like

- Correct Bill of Materials with relevant information
- Ability to handle Alternate Vendor Lists (AVL)
- Component history for traceability
- Directly integration with component distributors like Digi-Key, Arrow, Farnell, Mouser, RS Components and Future for error free new part introduction
- Web based user interface with highly configurable user roles
- Fully supported database



Do not hesitate to contact us if you would like to get more information on OrCAD Component Information Portal

Keyboard shortcuts