

Creating a circuit with OrCAD Capture CIS

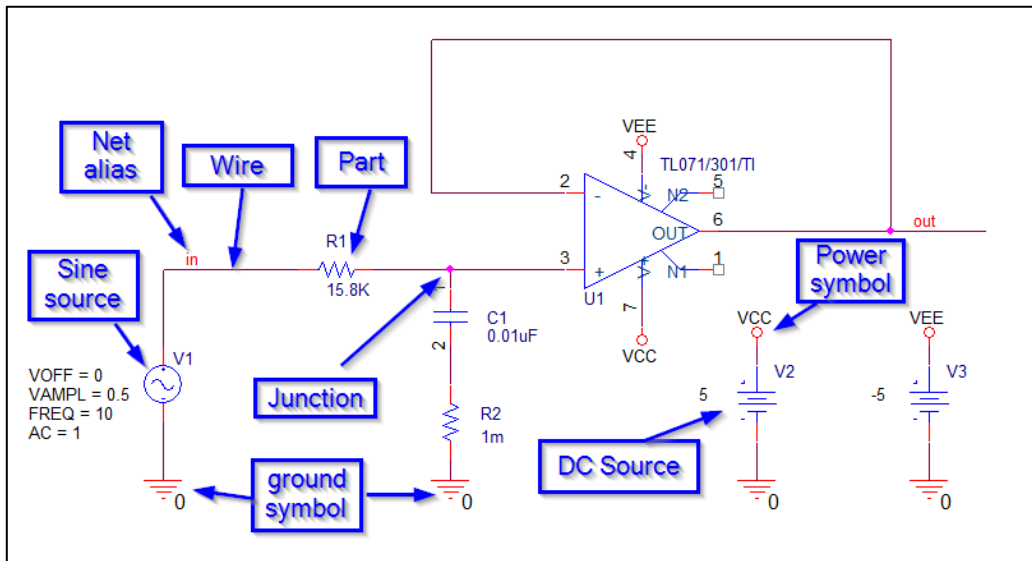
In this part of the workshop you will be creating the schematic for a lowpass filter including the parts necessary for simulating the circuit with PSpice.

Table of Contents

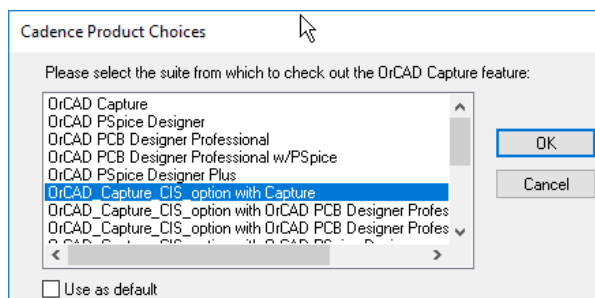
- Creating a circuit with OrCAD Capture CIS 1
 - Schematic Capture 1
 - Placing Parts from the component database 3
 - Placing parts using PSpice Search 5
 - Placing sources for PSpice simulation 7
 - Placing power and ground symbols 8
 - What did you learn? 9
- Component databases 10
- Keyboard shortcuts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

Schematic Capture

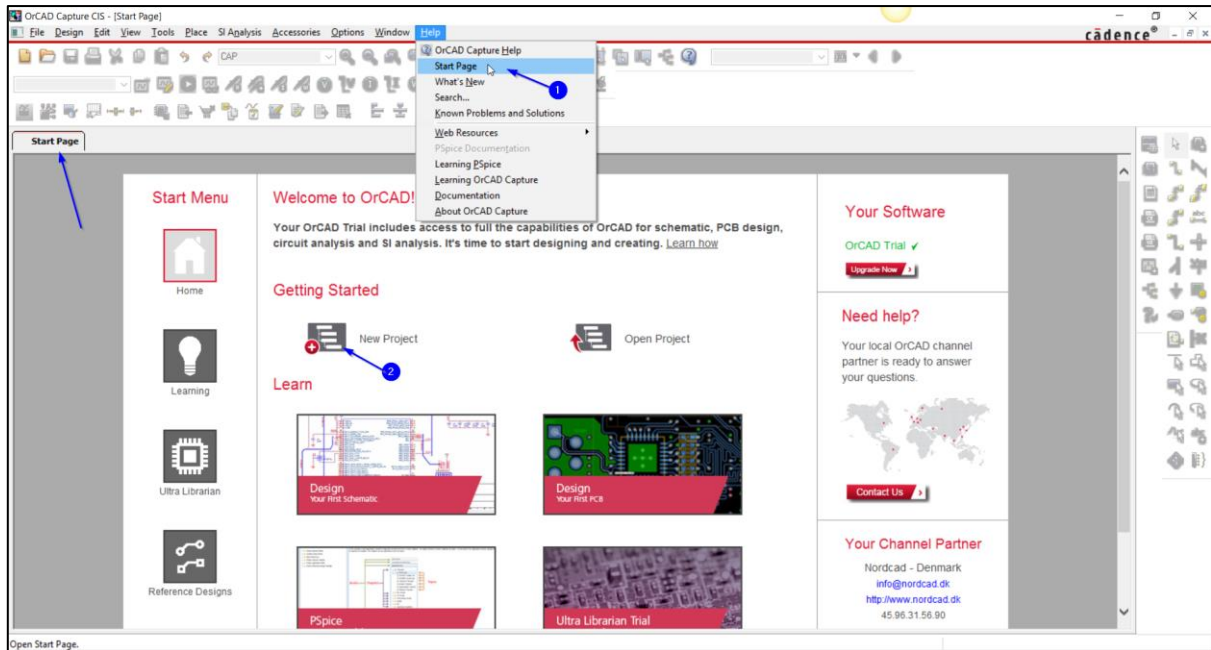
The final schematic design is shown below.



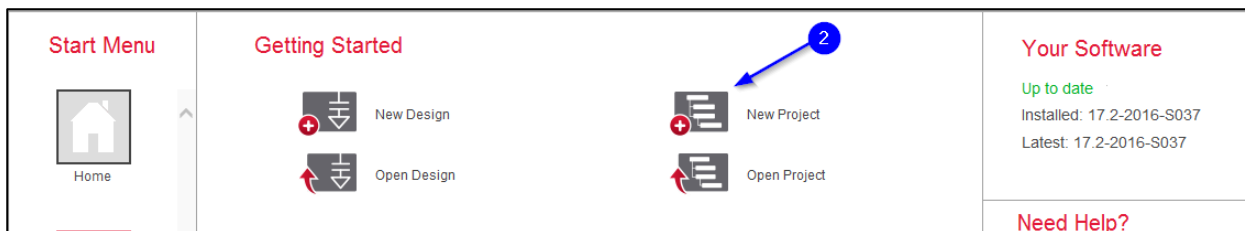
1. Be sure to start OrCAD Capture with the CIS option



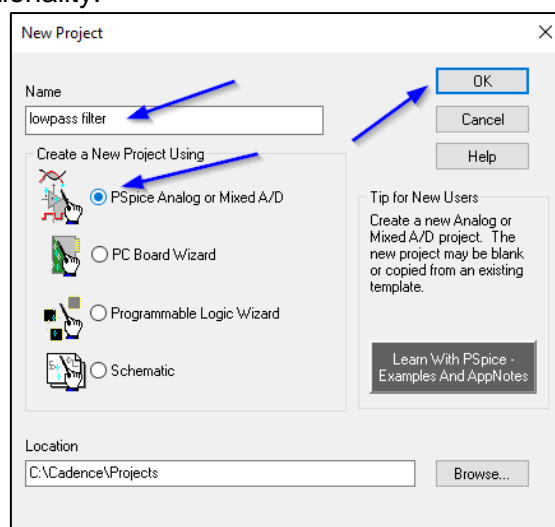
2. At startup the Start Page will be shown. If not select **Help** → **Start Page**.
3. On the Start page select **New Project**.



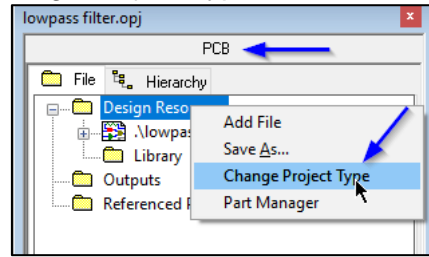
OR



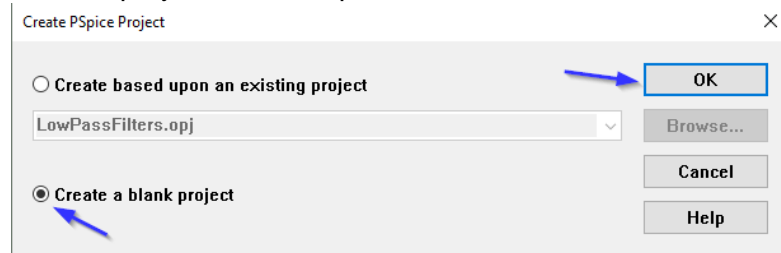
4. When you create the new project specify a name. In this case name the project **“Lowpass filter”** as shown.
 - a. Make sure to **select “PSpice Analog or Mixed A/D”** project in order to get all the PSpice functionality.



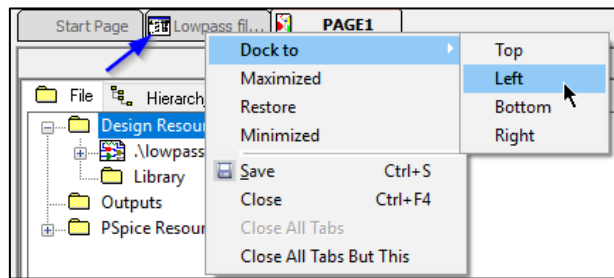
TIP: In case your project type by mistake was set to something else then it is easy to change from the project manager using the Change Project Type function shown below.



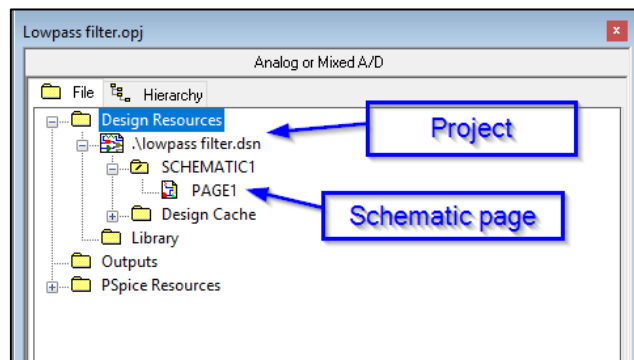
5. Click the create a blank project and then press ok.



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**

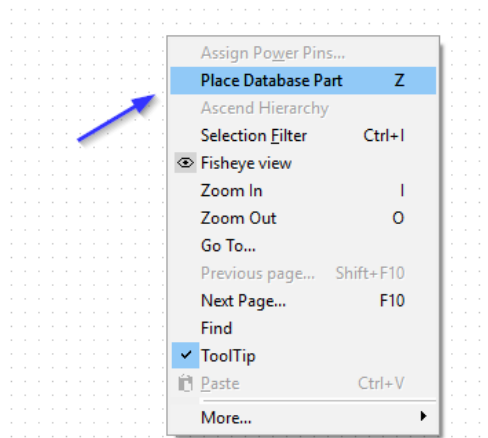


7. To find your schematic inside OrCAD Capture, look under your project DSN file.

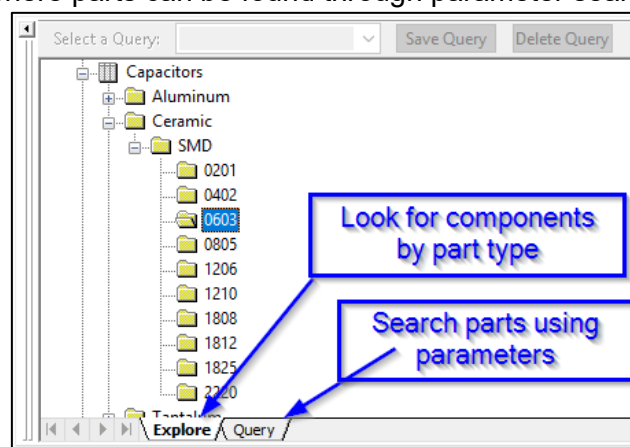


Placing Parts from the component database

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

1	Value	=	10nF
2			

Find value in the drop down box (2)

Search for 10nF or 0.01uF (3)

Explore Query (1)

4	Table	PART_NUMBER	Part type	Value	Description	Power	Tolerance
	Capacitors	CIS-1		0.01uF	CAP CER		1%
	Capacitors	CIS-10		0.01uF	CAP, Cera		10%
	Capacitors	CIS-1045		0.01uF	CAP, Cera		20%
	Capacitors	CIS-11		0.01uF	CAP, Cera		10%

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

	Property	Compare	Value
1	Value	=	15.8k
2			

Table	PART_NUMBER	Part type	Value	Description	Power	Tolerance
Resistors	CIS-2889		15.8kOhm	RES, Thick		0.01%
Resistors	CIS-00001		15.8K	RES SMD		0.05%

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

	Property	Compare	Value
1	Value	=	1m
2			

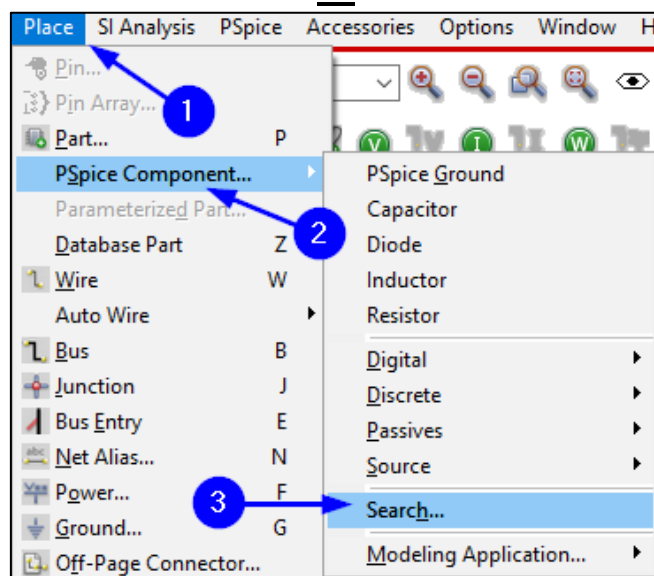
Table	PART_NUMBER	Part type	Value	Description	Power	Tolerance
Inductors	CIS-2790		1000uH	Inductor, 1		10%
Resistors	CIS-00002		1m	RES SMD		0.05%

Placing parts using PSpice Search

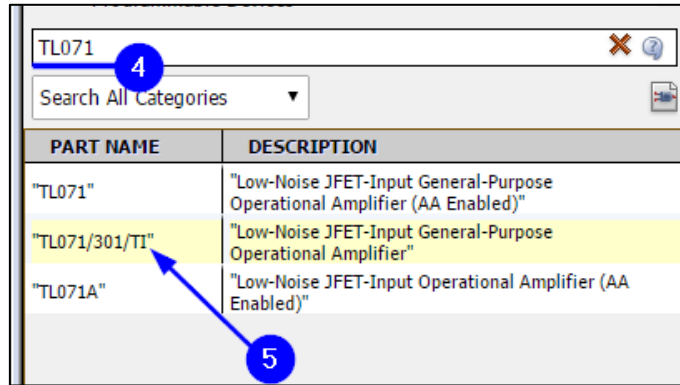
14. To place the opamp we're going to search in the built in library of parts with PSpice models.

Select Place → **PSpice Component** → **Search**

IC

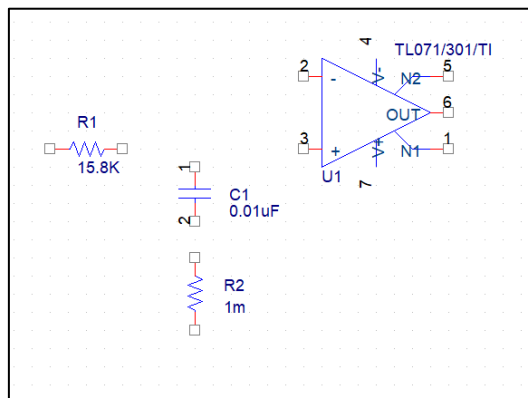


15. Type in 'TL071' to search for the opamp



16. Double click the line "TL071/301/TI" above in order to place the component

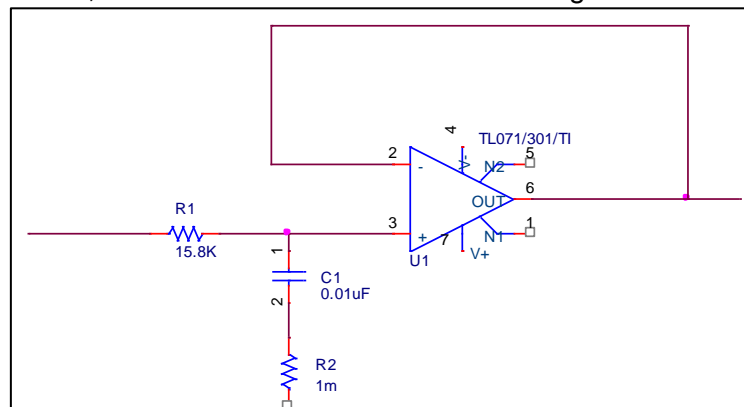
- a. The opamp can be mirrored to have the negative pin on top either by **Right Click** → **Mirror Vertically** or using shortcut 'v'



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

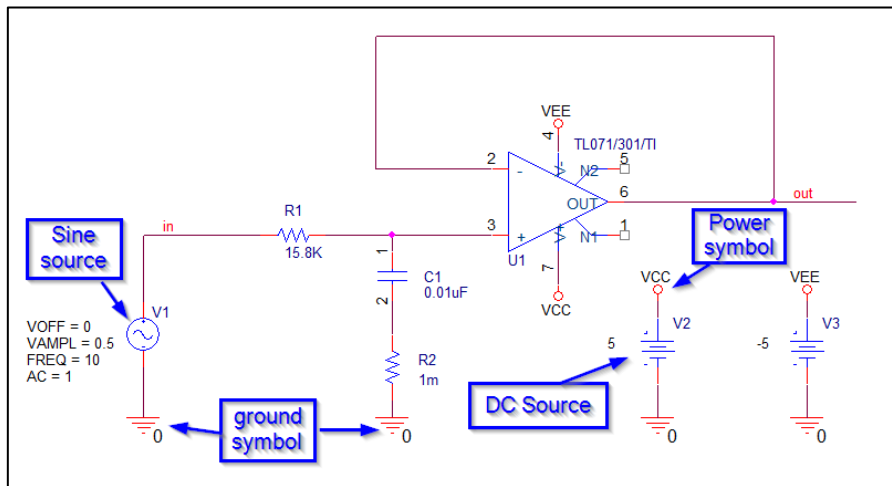


18. After the wiring is done, the schematic should look something like this.

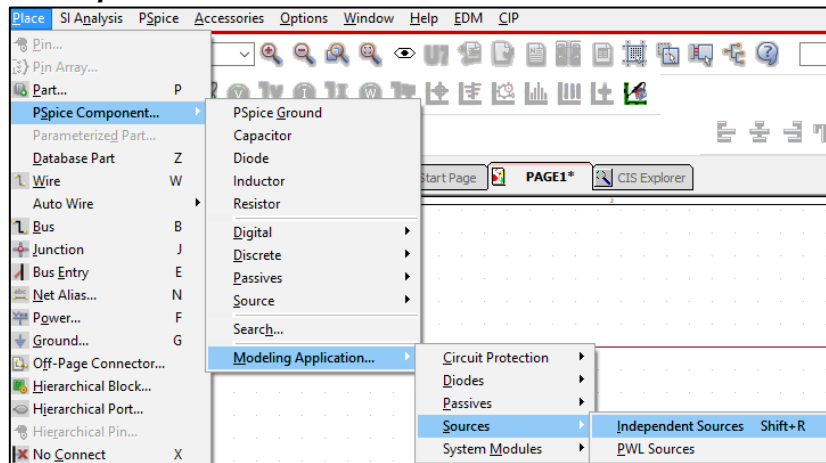


Placing sources for PSpice simulation

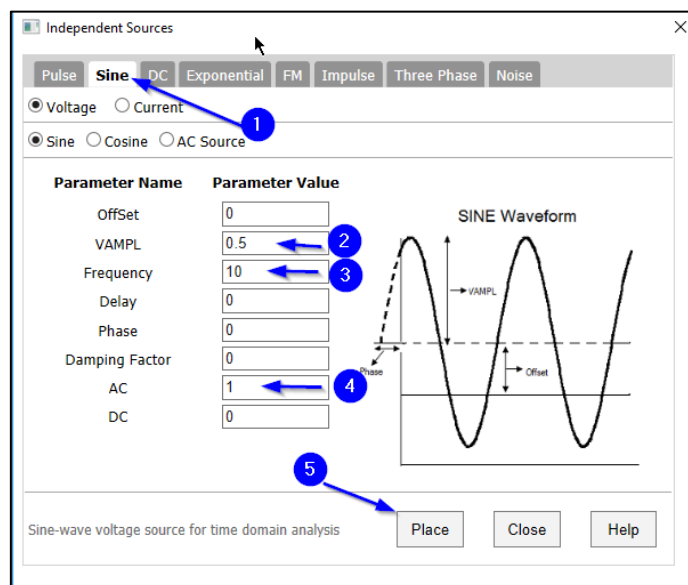
19. We will place power/ground symbols and sources for PSpice simulation.



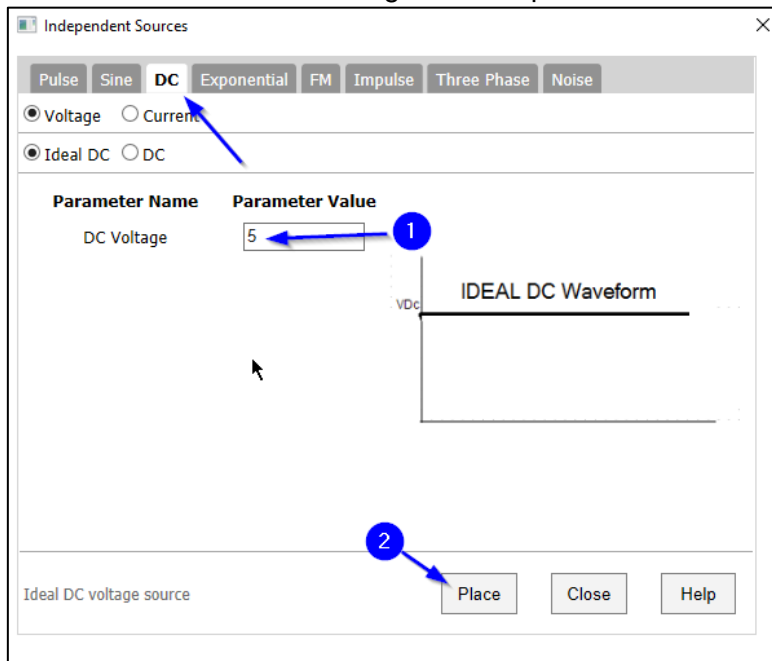
20. To place a sine source, go to **Place** → **PSpice Component** → **Modelling Applications** → **Sources** → **Independent Sources**



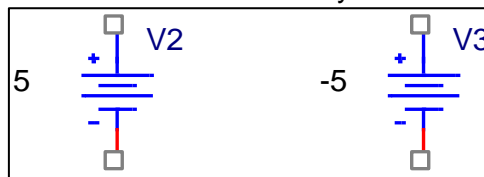
21. Set the parameters for the source as shown below and place it on the page



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

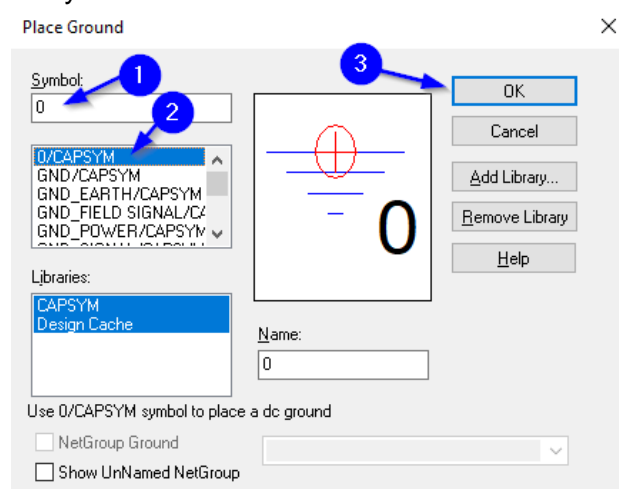


24. Copy the DC source that was just placed using ctrl+c and ctrl+v
25. Change the value of the new DC source to -5V by double clicking the value



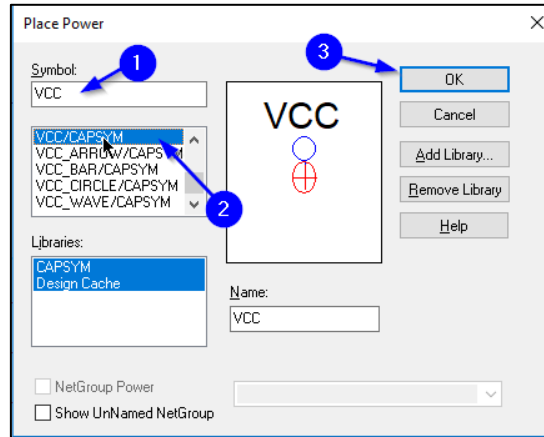
Placing power and ground symbols

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

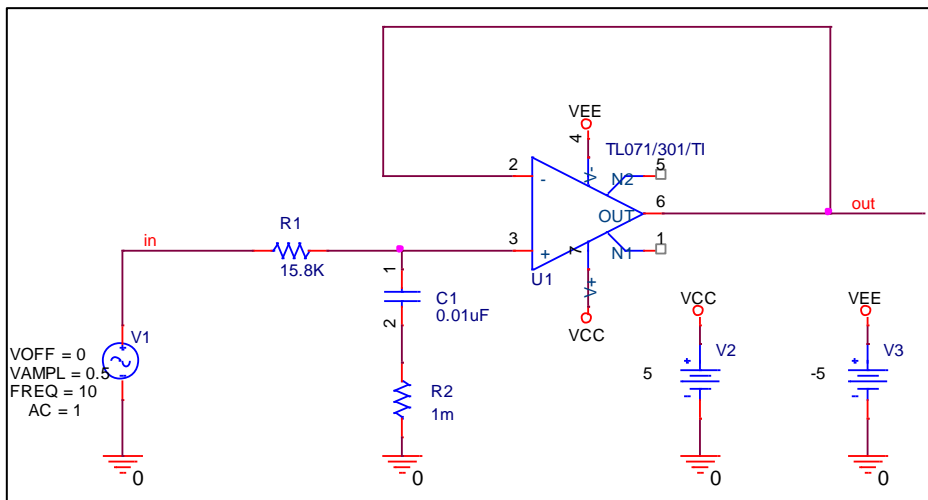
30. Select VCC and click OK to place it.



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

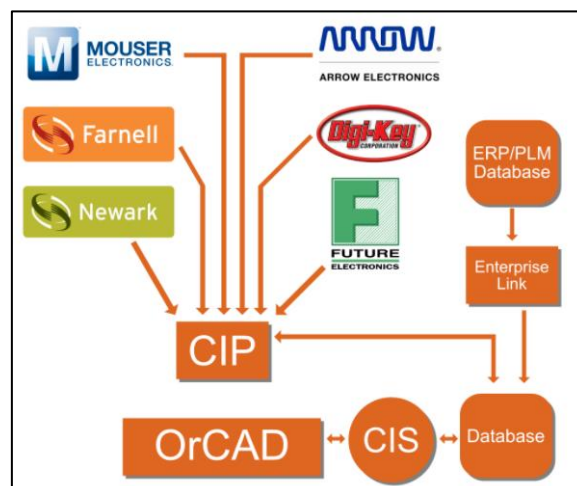
Component databases

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.

Component Information Portal		Welcome Ole [Log Off] [Help] [Download CIPClient]	
Admin	Components	CIS DB Search	Distributor Search
Compliance	Temp. Parts	BOMs	search <input type="text"/> GO
Capacitors		EMA-00000137V22	
Part Information	Mechanical Parts	History	
PART_NUMBER	EMA-00000137V22	Part Type	EMA\Ceramic\SMD\0805
Description	CAP, Ceramic, SMD, 1.0 uF, 10 %, 16 V, 0805	Value	1.0 uF
PCB Footprint	CAPC2012X145N	Schematic Part	CAP <input type="button" value="Place"/>
Number of Pins	2	Operating Temperature Maximum	125 C
Operating Temperature Minimum	-55 C	Package Size	0805
Package Height	1.45 mm	Package Type	SMD
Company Part Status		Dielectric Type	Ceramic
Equivalent Series Resistance		Temperature Coefficient	X7R
Tolerance	10 %	Rated Voltage	16 V

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

General

Ctrl + C	→ Copy
Ctrl + V	→ Paste
Ctrl + X	→ Cut
Ctrl + Z	→ Undo
Ctrl + Y	→ Redo
Ctrl + F	→ Find
Ctrl + E	→ Edit Properties

Schematic Page & Part Manager

Ctrl + L	→ Link Database Part
Ctrl + D	→ View Database Part

Part Manager

Ctrl + U	→ Update All Part Status
Shift + U	→ Update Selected Part Status
Shift + S	→ CIS Standard BoM
Shift + V	→ Variant Report

Schematic Page

C	→ Zoom Center
i	→ Zoom in
O	→ Zoom out
R	→ Rotate
V	→ Mirror Vertically
H	→ Mirror Horizontally
Z	→ Place Database Part
B	→ Place Bus
E	→ Place Bus Entry
W	→ Place Wire
N	→ Place Net Alias
G	→ Place Ground
F	→ Place Power
J	→ Place Junction
T	→ Place Text
X	→ Place No-connect
U	→ Place No-connect

F5	→ Redraw
F6	→ Full screen cursor
ESC	→ Deselect all
Shift + M	→ View Variant
Ctrl + F8	→ View Variant