

Autodesk Eagle

Software Instructions

Software used			Autodesk Eagle
Version #			1
Created by	Logan F	Date	04/02/21
Reviewed by	James T	Date	04/02/21

Software Instructions: Autodesk Eagle

Acknowledgments

We would like to acknowledge the following references used to compile these instructions for students:

https://www.youtube.com/watch?v=v979MUCKVoo

https://www.youtube.com/watch?v=bDwTjWpU584

https://www.youtube.com/watch?v=SgT2aUhJQHA&t=2821s

https://www.youtube.com/watch?v=JtytRPefUxk

COMMONWEALTH OF AUSTRALIA

Copyright Regulations 1969

WARNING

This material has been reproduced and communicated to you by or on behalf of The Engineering Institute of Technology pursuant to Part VB of the *Copyright Act 1968* (the Act).

The material in this communication may be subject to copyright under the Act. Any further reproduction or communication of this material by you may be the subject of copyright protection under the Act.

Do not remove this notice.



Table of Contents

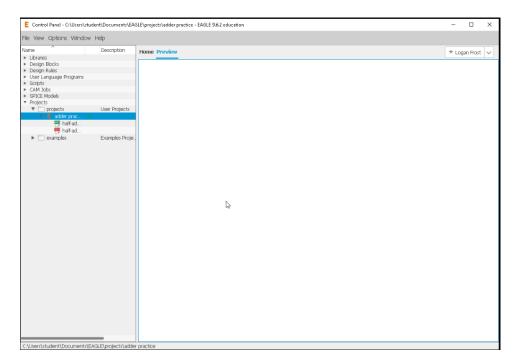
- 1. Intro
- 2. Creating a Schematic
- 3. Creating the PCB
- 4. SPICE Simulation

Software Instructions for Autodesk Eagle

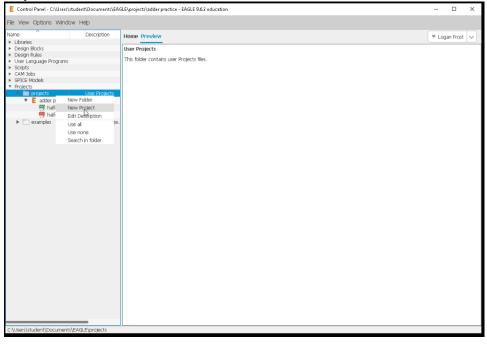
1. Intro:

Starting your project

When you open your new project, you will be presented with the following screen.

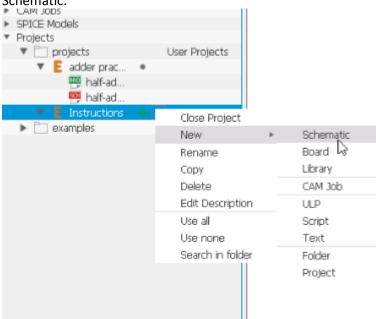


To create a new project, drop down the projects tab, right click on the new projects folder, and select New Project.

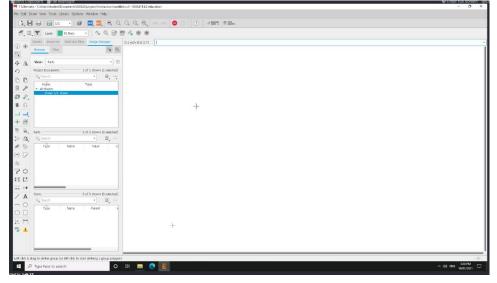




To begin working on your new project, right click on the project you just created and select new -> Schematic.



Once you have created a new schematic, you will be presented with this blank workspace.



The Help menu is very useful in this software. Following the general or documentation help menus, you can find out about any commands or functionality you need.

彈 1 Schem	atic - C:\Us	ers\student\D	ocuments\EAGLE\	projects\Ins	tructions\untitled.sch - EA	GLE 9.6.2 educ	ation
File Edit (Draw Vie	w Tools Lib	orary Options	Window	Help		
B, E			•	SCR UL	? General		a 🗢 🕒 💿
	_				? Context F1	•*	_
🛃 #	ŧ , ⊤	Layer:	91 Nets *	- A	Documentation	-3@5	
	Sheets	Inspector	Selection Filter	Design I	Schematic Editor About EAGLE	.6 3.5)	Click or press Ctrl+L ke
(i) (i)	Browse	r Filter			₽ @		
с.,							
+ ⊿⊾	View:	Parts			• ?		

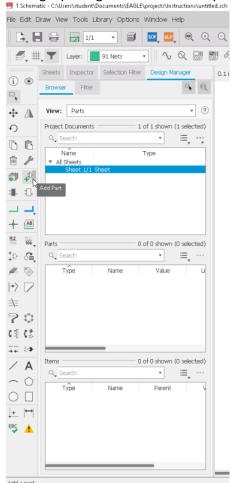
Here is a preview of the option and tabs you have available to you in the help menu.

? 2 EAGLE Help - EAGLE 9.6.2 educati	on — 🗌	>
ile Edit View Window		
$\blacklozenge \Rightarrow \boxplus \boxdot$		
3 Find:	Command Line Options	
'opic	You can call up EAGLE with command line parameters. Use the following format:	
General Help		
Configuring EAGLE	eagle [options [filename [layer]]]	
Command Line Options	Under Windows EAGLE is also available as a console application (see below).	
Quick Introduction	Under Withows Exact is also available as a console application (see below).	
 Control Panel 	Orthogo	
Keyboard and Mouse	Options	
 Editor Windows 	-Axxx Assembly variant -Cxxx execute the over Command	
 Editor Panels 	-DXXX execute the given command -DXXX praw bolerance (0, 1 = 10%)	
 Editor Commands 	- Example to the contract (0.1 = 0.0 k)	
 Generating Output 	-Fxxx Flash tolerance (0.1 = 10%)	
Group Default ON	-N- no command line prompts	
Autorouter	-0+ Optimize pen movement	
 Design Checks 	-Pxxx plotter Pen (layer=pen)	
 Cross-references 	-RXXX drill Rack file -SXXX Scriptfile	
User Language	-SXXX SATISTIE -UXXX User settings file	
Writing a ULP	- WAXX aperture Wheel file	
Executing a ULP	-X- ekecute CAM Processor	
▼ Syntax	-c+ positive Coordinates	
Whitespace	-dxxx Device (-d? for list)	
Comments	Emulate apertures	
Directives	-f+ Fil pads -bxxx page Height (inch)	
Keywards	-nxxx page megnic (incr)	
Identifiers	-xxx Output fierame	
Constants	-para Pendiameter (mm)	
Punctuators	-g- Quick plot	
 Data Types 	-r- Rotate output 90 degrees	
 Object Types 	-sxxx Scale factor	
 Definitions 	-u- output Upside down	
 Operators 	-vxxx poet velocity -wxxx poet width (inch)	
Expressions		
 Statements 	-yxxx offset ((nch)	
Builtins		
▶ Dialogs	where xxx means that further data, e.g. a file name or a decimal number needs to be appended to the option character (without space or separated by a space), as in	
Supported HTML tags	-Waywheel.whl	
Automatic Backup	- anywheel whi - W nywheel whi	
File Locking	- Aperture enulation on	
Forward&Back Annotation	-et dto.	
EAGLE Editions - License	-e- Aperture emulation off	
	For flag options, a '-' means that the option is off by default, while '+' means it is on by default.	
	Flag options (e.ge) can be used without repeating the '-' character:	
		_

2. Creating a Schematic:

Placing Components

To begin placing the electrical components for your scematic, select the Add Part button in the toolbar.



In the parts menu, your search term will be searched for exact matches in either the name or the description of the parts. To get around this, you can use wildcards to broaden your search. Using a "?" acts as a wildcard and will search for results with any character in that position. In this example, the search will return results for any 4-character name starting with "LED" eg. "LED1", "LED5", "LED3"

► C0	on-amp-n	nto 🔳	Eagle PCD	AMP Connecto	-0 / - /	
► C0	on-amp-o	🖻	Eagle Pcb	AMP Connecto		
► cc	on-amp-t	e 💼	Eagle Pcb	AMINTE Conne		
		✓ Pads	Smds	 Description 	Hide Unpopular Parts	Preview
earch	h 🖸	LED?				*
ttribu	utes 🛚					*

Using an "*" acts as a wildcard as well, representing any number of characters. The search in this example will return any string starting with "LED" eg, "LED", "LED1", "LED55", "LED12345678987654321"

 con-amp-q con-amp-ti 		Eagle Pcb Eagle Pcb	AMP Connecto AMP TE Conne		
	Pads	Smds	 Description 	🗌 Hide Unpopular Parts	Preview
Search 🛛 🗵	LED*		Ĩ		*
Attributes 🗵					•

Once you have found the part you are looing for, click the name to select it. On the right of the window is a preview of the parts and their description. With the part selected, click the OK button to begin placing it. Click anywhere on the scematic to place the parts. You can place as many as you'd like without needing to reselect the part.

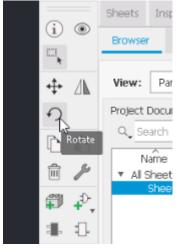
lame ^		Managed Folde	r Description	Popularity		
₹ 40xx		Eagle Pcb	CMOS Logic De			
▶ 4019	_	Logio i co	Quad AND-OR			Z>VALUE
▼ 4073			Triple 3-input			ZP VALUE
407						
407.			DIL14			
▼ 4081			Quad 2-input		P - Svalue - Svalue	10mm
408	-40		S014		P	
408.					$P = \frac{1}{2} $	
▶ 4082		13	Dual 4-input AND		6 Janue 10 Janue	
# 45xx		Eagle Pcb	CMOS Logic De		VALUE VALUE	m
▶ 4506	_	Logio i co	Dual expandabl			
▶ 4519			4-bit AND/OR			
74ac-logic		Eagle Pcb	TTL Logic Devi			
▶ 74AC11.		Logio i co	Quadruple 2-in			
▶ 74AC11.			Triple 3-input		4081 (Version 6)	
▶ 74AC11.			Dual 4-input AND			
74ttl-din		Eagle Pcb	TTL Devices wi		Quad 2-input AND	
7408N	10	Logio PCD	Ouad 2-input			
7400N	<i>A</i>		Ouad 2-input		Footprint: SO14 (Version 1)	
7403N	10		Triple 3-input		Small Outline package 150 mil	
7421N	#		Dual 4-input A		Small outline package 150 mil	
7451N			AND-OR-INVER		3D Package: SO14 (Version 2)	
7474N	10		Dual D type po			
7476N	10		Dual J-K FLIP F		Small Outline package 150 mil	
7478N	10		Dual J-K FLIP F			
74109N			Dual J-K positiv			
74109N			Dual J-K negati			
74112N			10-line to 4-lin		Attribute Value	
7414/N			10-line to 4-lin		POPULARITY 4	
74148N 74240N			Octal BUFFER a		POPULARITY 4	
74240N			Octal BUFFER a			
			Octal BUFFER a			
74244N 74390N			Dual 4-bit deca			
74390N 74393N			Dual 4-bit deca			
74393N 74540N			Octal BUFFER a			
-	Pads	🗹 Smds 🔍	Description	🗌 Hide Unpopular Parts 🛛 🗹 Preview		
earch 🛛 🛛	AND			*		
ttributes 🛛				*		
					L	

Once you have finished placing your first component, you can press the ESC key on your keyboard to open up the parts menu again to choose the next part you'll place.

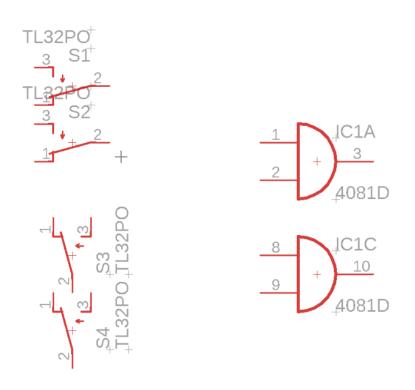


lame ^	Managed Folder	Description	Popularity		
ELMA-0	Hanagoa Foldor	Rotary Switche			
ELMA-0		Rotary Switche			>NAME
ELMA-0		Rotary Switche			
ITT-RT		ROTARY SWIT.			
ITT-RT		ROTARY SWIT			
ITT-RT		ROTARY SWIT.			
LPV-1		TOGGLE SWIT			
M251		SLIDING SWIT			
M9040P		TOGGLE SWIT			
M9040P2		TOGGLE SWIT			
MARO6		DIL SWITCH			VALUE
MS243		TOGGLE SWIT			
RDT1034		PUSH SWITCH			1 5mm - 1
REKN		PUSH SWITCH			U.an '
SKHMP		6.2 X 6.5mm T			TL32PO (Version 2)
SKH		SKHMPXE010			
SKH		SKHMPXE010			TINY SWITCH ON - MOM
SKH		SKHMPXE010			Source: http://www2.produktinfo.conrad.com/datenblaetter/
SRRM1		12-position Ge			700000-724999/705152-da-01-de-Subminiaturschalter TL 36YO.pdf
SRRM2		12-position Ge			/obbod//24999//05152-da-01-da-5dbininatdischarter_nc_5010.pdr
SRRM2		12-position Ge			Footprint: TL3XPO (Version 1)
SRRM2		12-position Ge			
SRRM3		12-position Ge			TINY SWITCH
SRRM4		12-position Ge			Source: http://www2.produktinfo.conrad.com/datenblaetter/
 SSRM1 		12-position Ge			700000-724999/705152-da-01-de-Subminiaturschalter TL 36YO.pdf
TL32PO		TINY SWITCH			
TL32WO		TINY SWITCH			
TL32YO		TINY SWITCH			Attribute Value
TL36PO		TINY SWITCH			POPULARITY 2
TL36WO		TINY SWITCH		-	
TL36YO		TINY SWITCH			
TL37PO		TINY SWITCH			
TL37WO		TINY SWITCH			
TL37YO		TINY SWITCH		2	
TL38PO		TINY SWITCH		15	
	Pads 🗹 Smds 🗹	Description	Hide Unpopular Parts	Preview	v
arch 🖸				*	
tributes 🔀				*	

When placed, the parts will all be the same orientation. You can rotate the parts using the rotate tool in the toolbar.



Click the central cross of the part you want to rotate. This will rotate the part by 90° in a directiopn. Repeat this until the parts are rotated to your liking.



Another part you can add is a supply part. This includes power and ground components. These supply components can be placed any number of times in the schematic and will be treated as though they are linked.

stm32xx		Eagle Pcb		
supply1	-	Eagle Pcb	Supply Symbols	
			SUPPLY SYMBOL	
+5V			SUPPLY SYMBOL	
+12V	3		SUPPLY SYMBOL	
+15V	-		SUPPLY SYMBOL	
+18V			SUPPLY SYMBOL	
+24V			SUPPLY SYMBOL	
-5V			SUPPLY SYMBOL	
-12V			SUPPLY SYMBOL	
-15V			SUPPLY SYMBOL	+3V3 (Version 1)
-18V			SUPPLY SYMBOL	
-24V			SUPPLY SYMBOL	SUPPLY SYMBOL

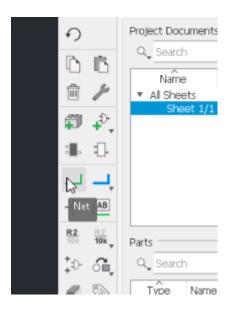
These can be found under the Supply1 tab in the full components menu.

244					SUPPLY SYMBOL
-24V		SUPPLY SYMBOL			OUT FET STITUDE
OV		SUPPLY SYMBOL			
AGND		SUPPLY SYMBOL			
GND		SUPPLY SYMBOL			
GNDA		SUPPLY SYMBOL			
GNDI 😞		SUPPLY SYMBOL			
GNDINT		SUPPLY SYMBOL			
GNDIO		SUPPLY SYMBOL			
PE		SUPPLY SYMBOL			
TH		Thermal			
V+		SUPPLY SYMBOL			
V-		SUPPLY SYMBOL			Attribute Value
VCC		SUPPLY SYMBOL			
VCC/2		SUPPLY SYMBOL			
VCCINT		SUPPLY SYMBOL			
VCCIO		SUPPLY SYMBOL			
VDD		SUPPLY SYMBOL			
VSS		SUPPLY SYMBOL			
🕨 supply2 🛛 🛱	Eagle Pcb	Supply Symbols			
Pads	Srnds	Description	Hide Unpopular Parts	Preview	
earch 🛛				*	
ttributes 🛛				*	
					OK Open Library Manager Cancel
					OK Open Library Manager Carice

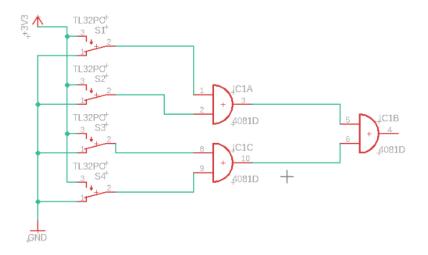
Connecting Your Parts

Using the Net tool in the toolbar, you can begin connecting your parts.

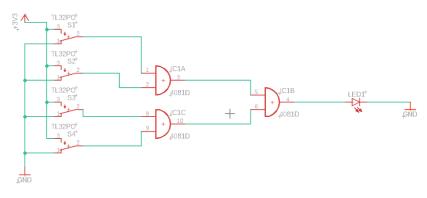




Click the node on your components you want to start connecting from and then click on another node to complete the connection. You can also begin drawwing a net anywhere but you will need to end to net on a node.



You can place source and ground components anywhere and as many times as you like. All ground connection will be connected when brought into the PCB builder.



3. Creating the PCB:

Once you have your circuit prepared, click the "Generate/switch to board" to transfer the circit to a PCB.

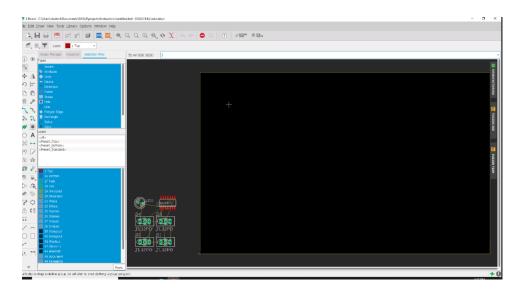
_ r seneme	see c. (osers (seaderic))	ocumento (c-tore g	projects (instructions (and a	Caser Ender Stere cau	cation
File Edit D	Draw View Tools Lib	prary Options \	Window Help		
B, E	1/1	•	ب 🔍 💭 🚛		← ≈ 😑 💿
Ø ,#	La Generate,	/switch to board		U º+ 🌸 🕸	
	Sheets Inspector	Selection Filter	Design Manager	0.1 inch (7.3 2.2)	s <mark>tript</mark>
(i) (i)	Browser Filter		₽ @		
♣⊿⊾	View: Parts		• ?		
Ð	Project Documents	10	of 1 shown (1 selected)	▲	TL32PO ⁺
D B	Q_ Search			++	3 S1 ⁺
ê 🎤	Name ▼ All Sheets	Т	ype		1
₽ + ⁰ ,	Sheet 1/1 Sl	heet			TL32PO ⁺ 3 S2 ⁺

When you do this the first time in your preject, you will be asked if you want to create from schematic. Click yes.

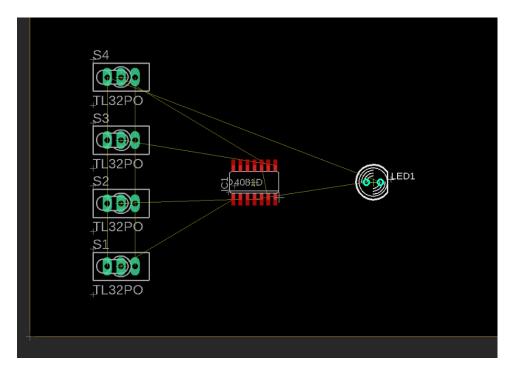
¢	🥮 Warn		×	E 9.6.2 education
Í.	- warn	ng	^	
F		The board C:\Users\student\Documents\EAGLE\projects\Instructions \untitled.brd does not exist.		• •
		Create from schematic?		
		Yes No		8 8 0.5 2.8) a
	i) (1)	Browser Filter		0.3 2.8)

You will be presented with a screen of one colour, and a rectangle of a darker colour. The darker rectangle is the PCB.

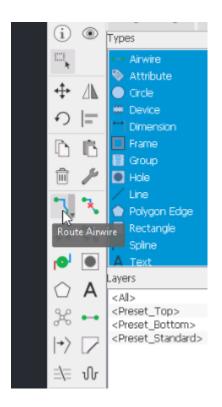




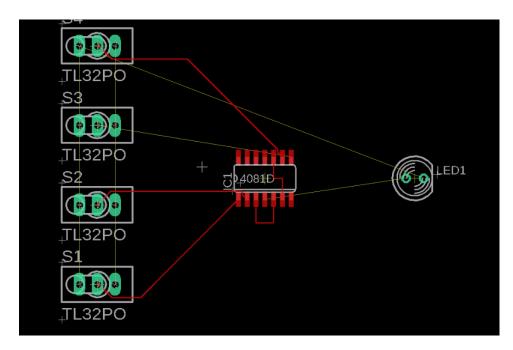
You can move the components from the left into the board. Adjust their positions to how you'd see them on the PCB. Try to avoid the lines crossing too much. The connections can be made later to ensure nothing gets tangled.



To begin connecting the components on the PCB, select "Route Airwire".

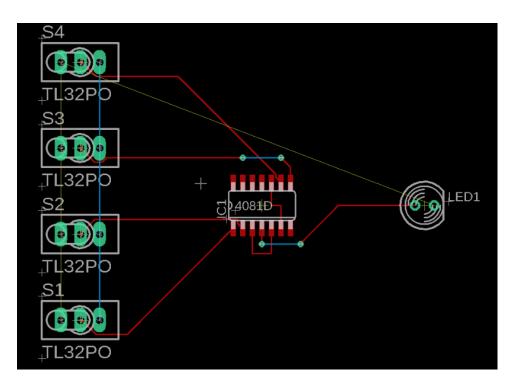


Click the node you want to start drawing from and draw towards the correct connection. The lighter lines will update as you draw to show where the connection should reach. There are two options in the upper toolbar for drawing these traces. One pushes asside aleady placed traces, and the other lets the trace you are drawing walk around the other components.

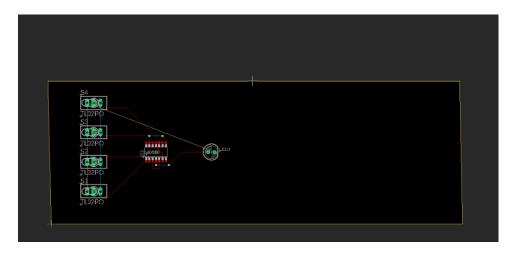


When drawing the traces, you can press the spacebar to switch the trace to the opposite side. This is useful for letting lines cross over eachother.

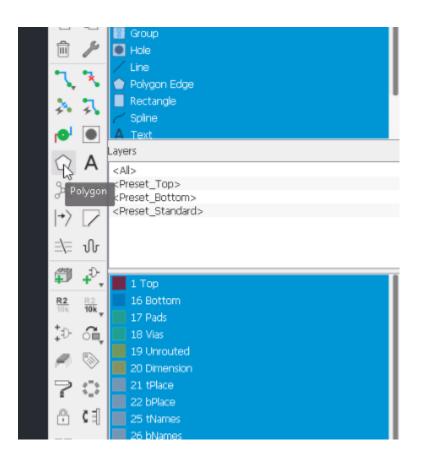




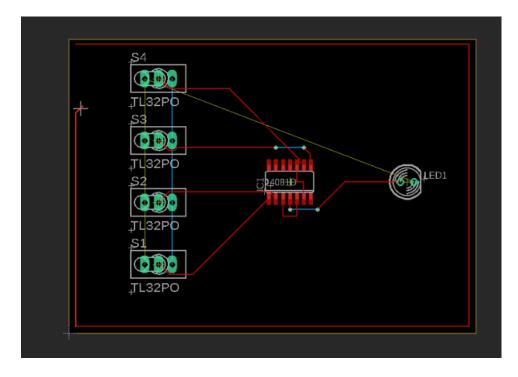
You can click and drag the outlines of the PCB board to adjust the size of the PCB itself.



Another connection that needs to be made is with the power source. Select the polygon tool.

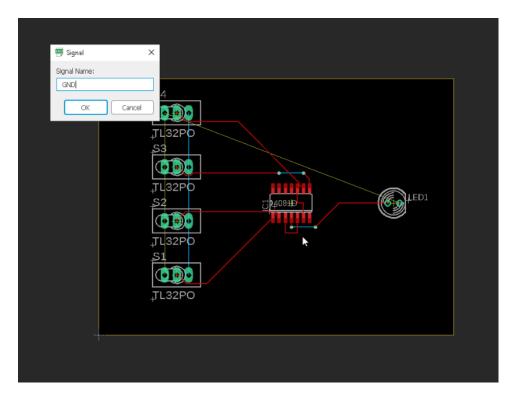


Draw an outline using the tool which covers all of the necessary components.

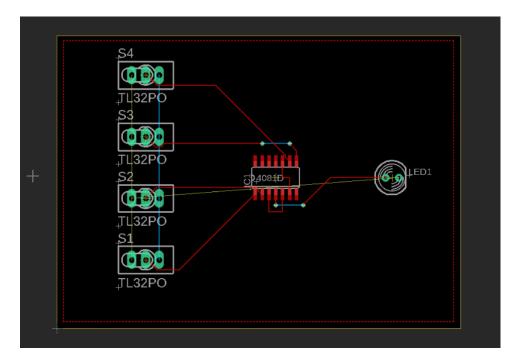


Once the drawing is complete, select either the ground or power connections to set the polygon to.

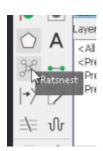




Here is what the complete polygon looks like.



Click the ratsnest tool to complete the polygon. This will fill the polygon with contacts connecting all of the nodes of the connection type you chose earlier. This fill will stay separate from the other traces on that layer.



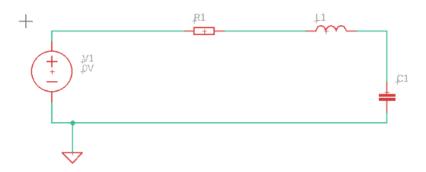


4. SPICE Simulation:

A				
Name	Manageo		Popularity	
national-se				
ngspice-digi				
 ngspice-sim 	🖨 🛛 Eagle Pc			
AMMET		AMMETER.		
BJT_NPN		Bipolar NPN Tr		
BJT_PNP		Bipolar PNP Tra		
C		CAPACITOR		
CCCS		Linear Current		
CCVS		Linear Current		
CURRENT		Independent C		
DIODE		Diode		
FET_N		N-Type MOS Fi		
FET_P		P-Type MOS Fi		
FN_ABS		Voltage contro		
FN_ACOS		Voltage contro		ngspice-simulation (Version 18)
FN_ASIN		Voltage contro		
FN_ATAN		Voltage contro		SPICE compatible library parts
FN_COS		Voltage contro		
FN_EXP		Voltage contro		
FN_LN		Voltage contro		
FN_LOG		Voltage contro		
FN_SIN		Voltage contro		
FN_TAN		Voltage contro		
GND		Simulation gro		
L		INDUCTOR		
LED		Diode		
NONLIN		Nonlinear depe		
OPAMP		Operational am		Attribute Value
R		RESISTOR		
TRANS		Coupled induct		
VCCS		Linear Voltage		
VCVS		Linear Voltage		
VOLTAGE		Independent		
V_SWI		Voltage contro		
on-semicon	Eagle Pd	b ON Semicondu		
	Pads 🛛 Smd		🗌 Hide Unpopular Parts 🛛 🗹 Preview	
		o goodpan		
Search 😰			•	
Attributes 🖾			*	
				OK Open Library Manager Cancel

In the add components menu, open the drop down labled "spicesim" to access simulation parts.

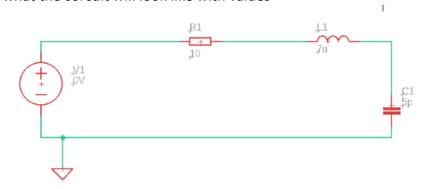
Using the components in that menu, build the circuit you'd like to simulate.



Using the Value tool, assign all of the components their values. Do not use the value tool on the voltage source.

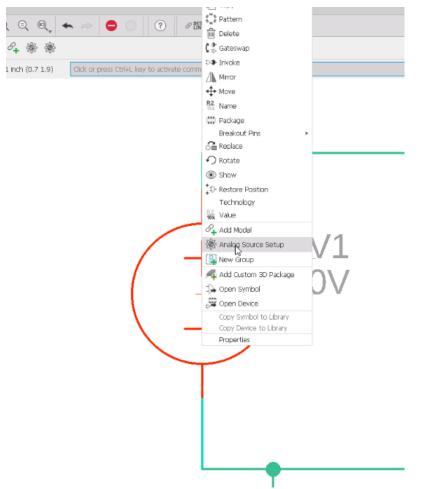
😁 1 Schem	atic - C:\U:
File Edit (Draw Vie
🕒, E	
Ø, #	ŧ, T
(i) (i)	Sheets
Q. O	Browse
÷ //	View:
• 2m	Project
66	Q, SI
ê /	۹ IA ۳
1	
∎ 0.	
<u> </u>	
+ 😬	
R2 1045	Parts -
÷ि ी	Value . Si
# >	тур зэ
→ [Z	4 ()
≋ ?≎	
(1) (1) (1) (1)	
¥-0,¥0- ⊒⊒ 0\$-	
/ A	Items
	Q_ 54
$\circ \Box$	
<u>↓+</u> ⊷	
erç 🙏	
Define the	value of ar
	D Type I

Here is what the corcuit will look like with values

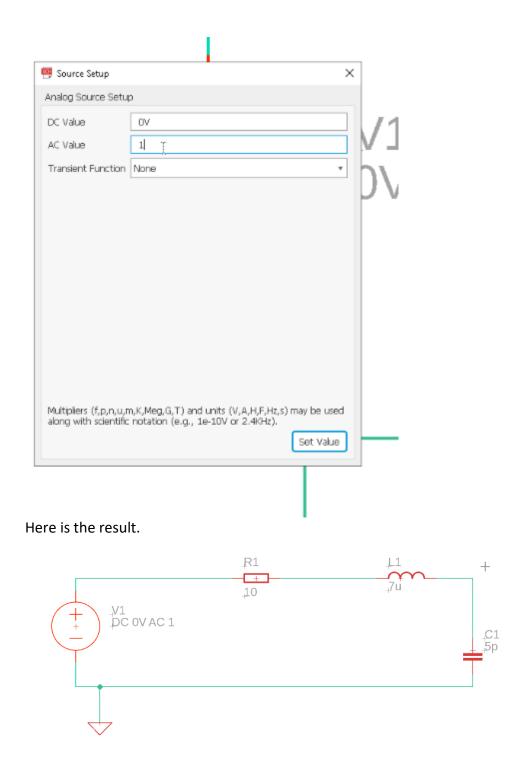


To assign a value to the voltage source, right click the voltage source and select Analog Source Setup.





Input your values and click "Set Value". In this case there is no DC value because we are doing an AC sweep simulation.

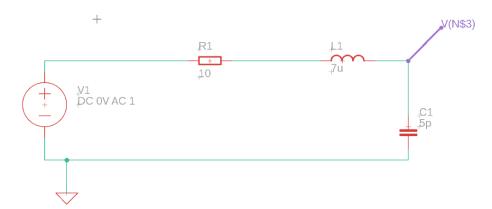


To gain data in the simulation, you will need to add probes. In the toolbar, select Voltage Probe.



Options Window Help	
🎬 🔤 🔍 Q Q Q 🔍 🖛 🗢 😑	0
₅ • ∿ ℚ 碶 웹 4 * *	
tion Filter Design Manager Voltage Probe3 3.3) Delete	
□ • @	
• ?	

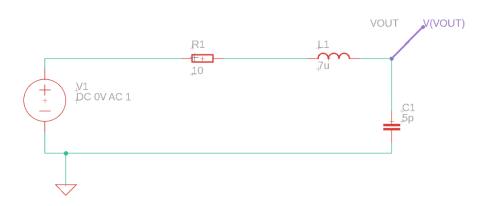
Choose where to place your probe in the circuit.



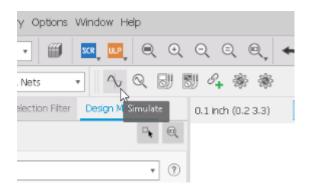
N\$3 isn't the most descriptive name, so you can use the "name" tool in the toolbar ro change the name of that node in the net.



Here is my outfit named as VOUT.



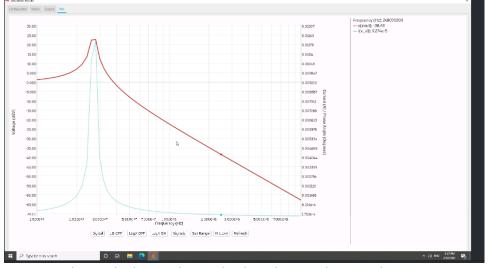
Once you've placed all of your probes, you can click the "Simulate" button.



In this case we are doing an AC sweep, so fill in your simulation details and click simulate.

Configuration Netfits Output Plot Simulation Type DC Sim Options Operating Point Is 2010 ASSTOL GMUN PIVREL DC Sweep Source Start Value End Value O C Sweep Start Value End Value Is 2010 Is 2010 O AC Sweep Type Start Freq End Freq Is 2010 Is 2010 Is 2010 Transient Start Time Stop Time It Max) Is 30 Is 60 OFF Is 2010 # Points Stop Max # Points 2000 Temperature (C) 25.0 METHOD SRCSTEPS TRTOL Multipliers (fig.n.m.m.K/Meg.G,T) and units (V.A.H.F.H.L.s) may be used along with scentific notation (e.g., 1e-10V or 2.4KH2). Reset Reset Reset Reset Reset Reset	×
Operating Point ABSTOL GMIN PIVREL DC Sweep Source Start Value Construct Transient Start Treg End Value End Value AC Sweep Type Start Treg End Preg Intervention Intervention AC Sweep Type Start Treg End Preg Intervention Intervention Transient Start Time Stop Time (TMAX) Transient Start Time Stop Time (TMAX) Multipliers (f.p.n.u.m,K.Meg.G.T) and units (V.A.H.F.Hz,s) may be used along with scoentific notation (e.g., Ia-IOV or 2.4KHz). Reset Reset	
Operating Point DC Sweep Source Start Value End Value AC Sweep Type Lin Start Freq End Freq IG Transient Start Time Start Time Stop Time Points S00 Multipliers (f, p, n, u, m, K, Meg, G, T) and units (V, A, H, F, Hz, s) may be used along with scientific notation (e.g., 1e-10V or 2.4(Hz)). Reset	DC Sim Options
Octore Statt Preq Statt Preq Statt Preq AC Sweep Type Statt Time Statt Time	
AC Sweep Type Start Preq End Freq II	
Transient Start Time Stop Time (TMAX) # Points 500 Max # Points 2000 Temperature (C) 25.0 Multipliers (f,p,n,u,m,K,Meg,G,T) and units (V,A,H,F,Hz,s) may be used along with scientific notation (e.g., 18-10V or 2.4KHz). Temperature (C) 25.0 Reset Reset Reset	Start Freq End Freq 1e-3 1e-6 OFF * 10Meg 1G IG Integration Integrateaaaaaaaaaaaaaaaaaaaaaaaaaaaaaaaaa
# Points 500 Max # Points 2000 Temperature (C) 25.0 Multipliers (f,p,n,u,m,K,Meg,G,T) and units (V,A,H,F,Hz,s) may be used along with scientific notation (e.g., 1e-10V or 2,4KHz). Ie-15 10 Reset Reset Reset	Ston Time (TMAX)
Reset Reset	
	,r,n,z,s/may be used along with
Update Netlist Simulate	Reset
	Update Netilst Smulate

A graph of the results will appear. In this graph there is a VSource and a VOUT line.



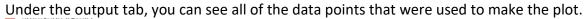
You can adjust which signals are displayed using the Signals Button.

00e+7	3.000e+7	5.000e+7 7	7.000e+7 1.000 Frequer	De+8 ncy (Hz)	2.000e+8	3.000e+8	5.00
	Digital dB	OFF LogX OF	FF LogY ON	Signals	Set Range His	Refresh	





You can click and drag to select regions of the graph to zoom in on either vertically or horizontally. Any of these adjustments you make can be undone using the refresh button.



Configuration	Newst 0	utput Plot	
Title: * spice Date: Tue Fe Piotname: AC Flags: comple No. Variables No. Points: 5 Variables:	eb 02 15:27:30 I Analysis ex : 3	2021	
Values:	0 1 2	frequency v(vout) i(i(v_v1))	frequency voltage current
0	1.160312229	790391e+000)0.000000000000000+000),-4.229659269336404e-003),-3.645228376979886e-004
1	1.247539607	7479614e+000	,0.00000000000000000000000000000000000
2	1.369031239	491624e+000	",0.00000000000000000000000000000000000
3	1.542173364	312726e+000	;0.000000000000000+000),-1.191940930180368e-002),-7.728506788500663e-004
4	1.799998114	761635e+000	;0.00000000000000000000000000000000000
5	2.213317728	3019904e+000	,0.000000000000000e+000),-3.066243068887257e-002),-1.385094721066322e-003