

Importing Symbols

1. Find part on DigKey

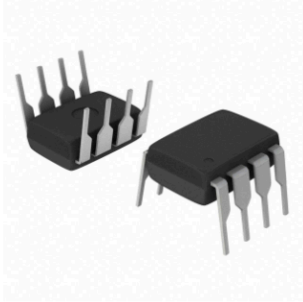



Image shown is a representation only. Exact specifications should be obtained from the product data sheet.

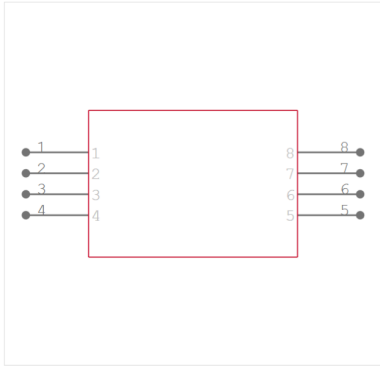
LM555CN

DigiKey Part Number	LM555CNFS-ND
Manufacturer	onsemi
Manufacturer Product Number	LM555CN
Description	IC OSC SINGLE TIMER 8-DIP
Customer Reference	<input type="text"/>
Detailed Description	555 Type, Timer/Oscillator (Single) IC 8-DIP
Datasheet	 Datasheet
EDA/CAD Models	LM555CN Models

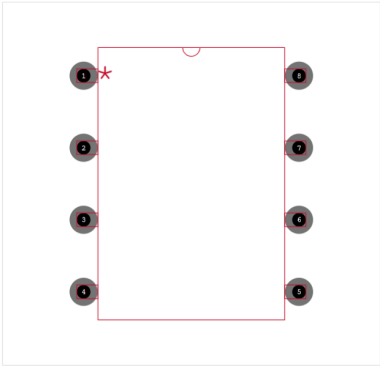
2. Navigate to EDA/CAD Models
3. Preview EDA/CAD models for accuracy to part

Ultra Librarian

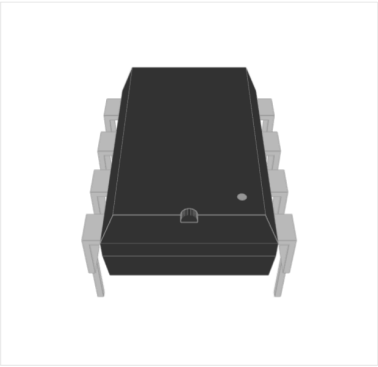
Symbol



Footprint



3D Model



[Select Download Format](#)

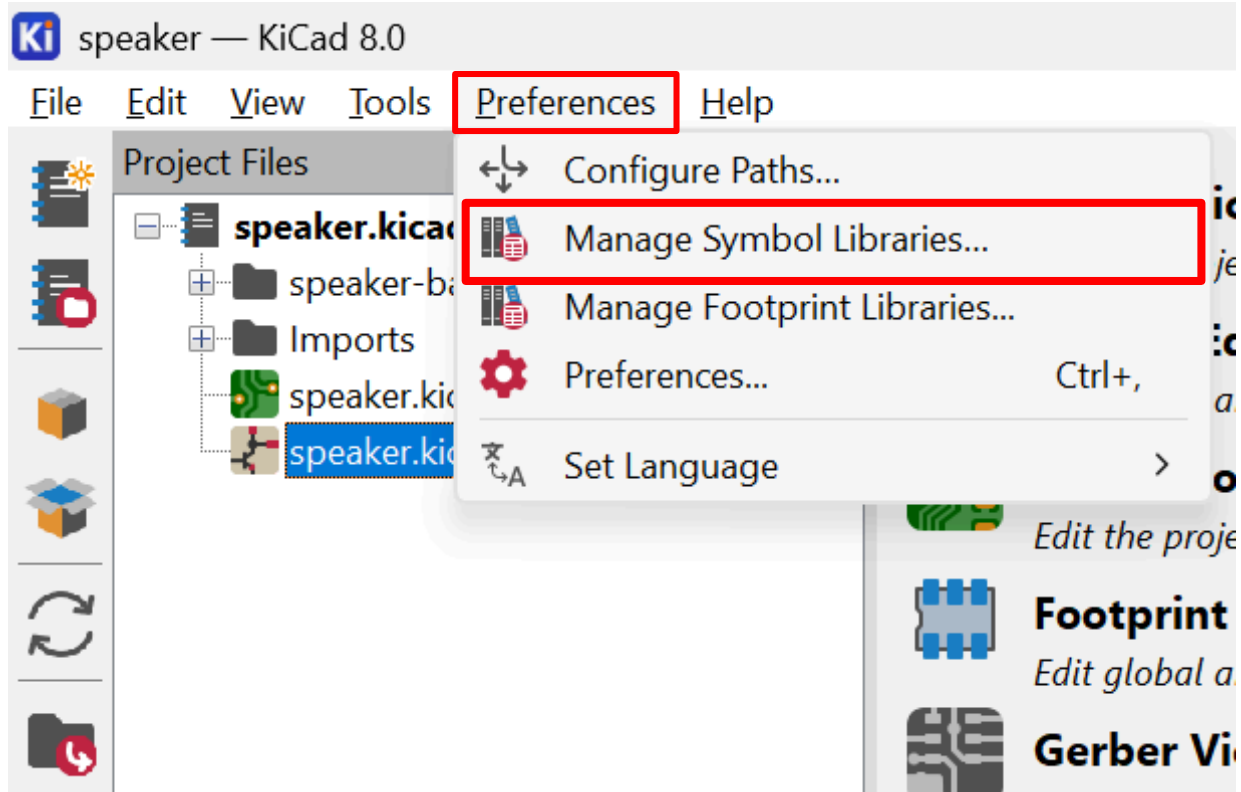
Choose Download Format

3D CAD Model

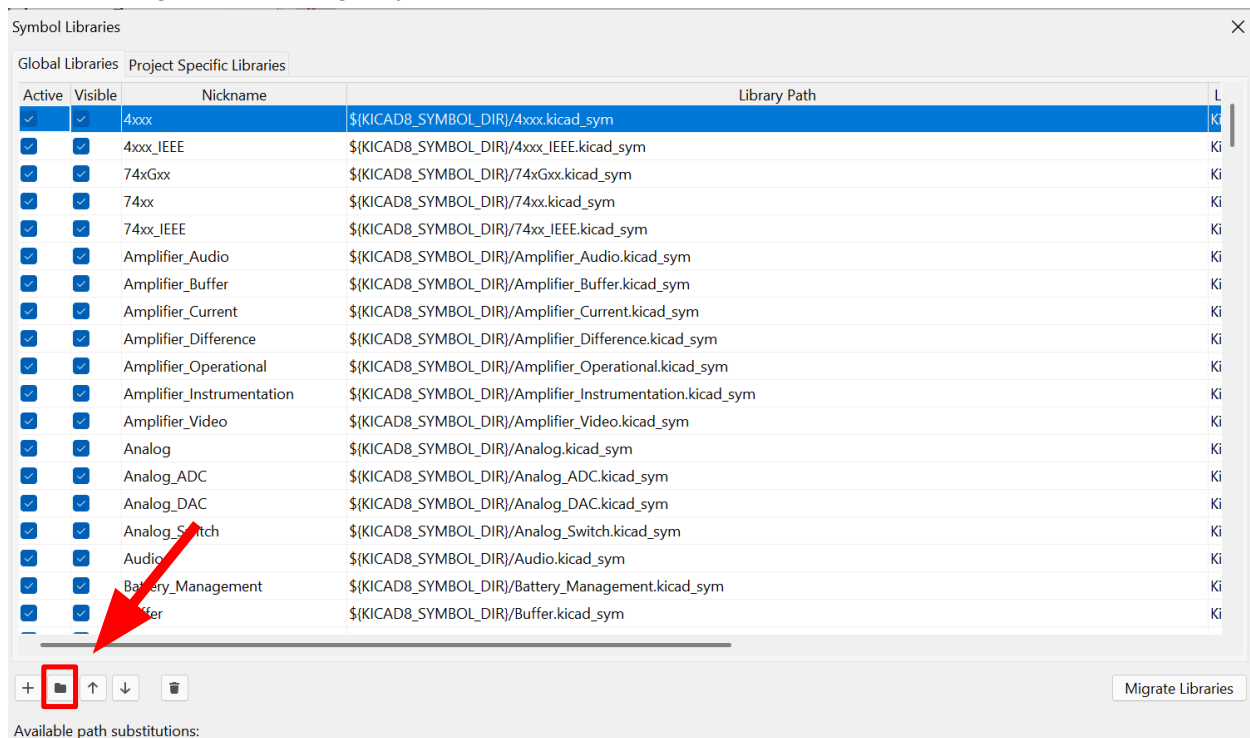
- ☐ IGES v5.3
- ☐ STEP
- ☒ STL

- ☐ eCadstar
- ☐ Fusion360 PCB
- ☐ KiCAD v5
- ☒ KiCAD v6+
- ☐ OrCAD Capture v17.2

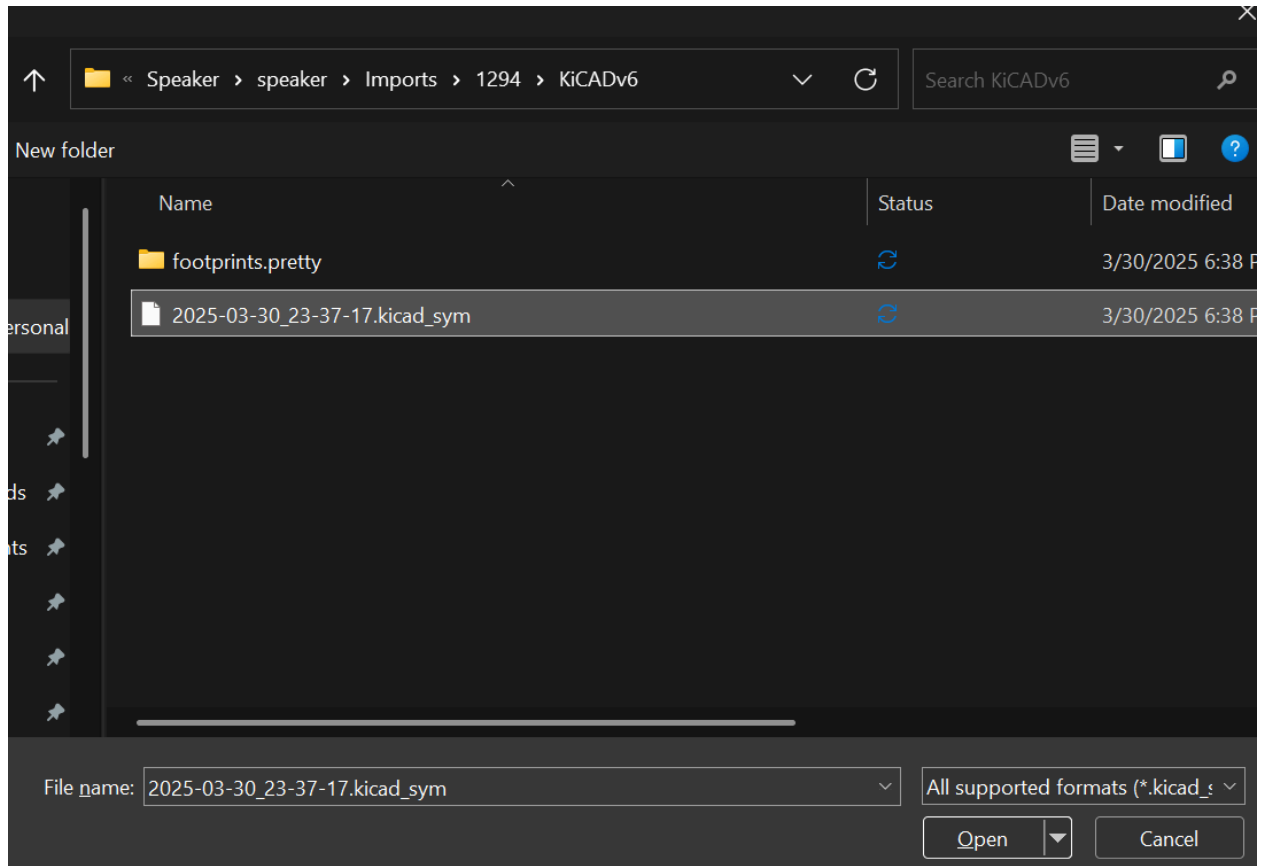
4. Choose download format (KiCad v6+)
5. Unzip into desired file location making sure to take note of the file path



6. To import the symbol open the current version of KiCad installed on your device and navigate to “Manage Symbol Libraries”



7. Click the folder icon in the bottom left corner
8. Now navigate to the place that you unzipped the EDA/CAD files

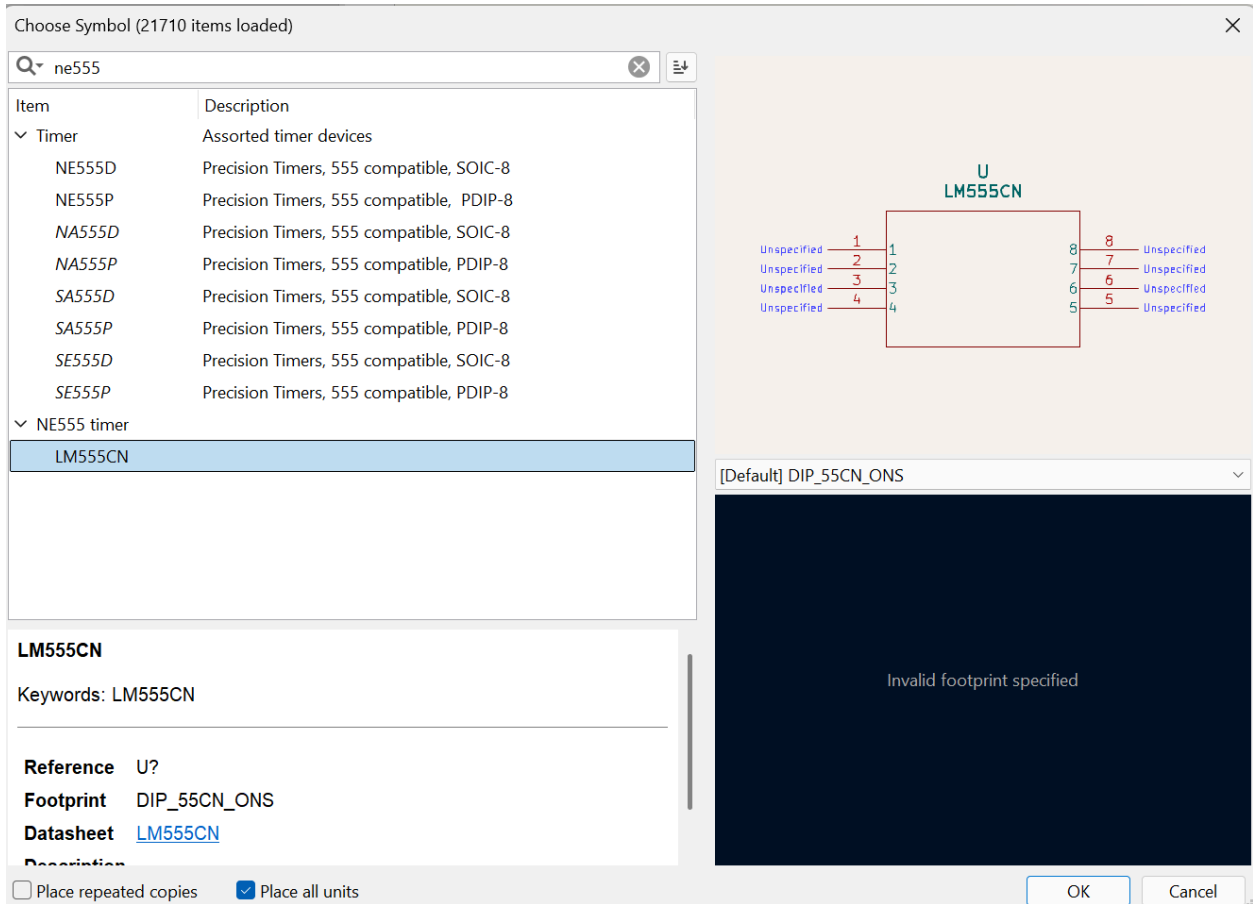


9. Select the “.kicad_sym” file and select open

Global Libraries		Project Specific Libraries	
Active	Visible	Nickname	Library Path
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Sensor_Touch	\$(KICAD8_SYMBOL_DIR)/Sensor_Touch.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Sensor_Voltage	\$(KICAD8_SYMBOL_DIR)/Sensor_Voltage.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Simulation_SPICE	\$(KICAD8_SYMBOL_DIR)/Simulation_SPICE.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Switch	\$(KICAD8_SYMBOL_DIR)/Switch.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Timer	\$(KICAD8_SYMBOL_DIR)/Timer.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Timer_PLL	\$(KICAD8_SYMBOL_DIR)/Timer_PLL.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Timer_RTC	\$(KICAD8_SYMBOL_DIR)/Timer_RTC.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Transformer	\$(KICAD8_SYMBOL_DIR)/Transformer.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Transistor_Array	\$(KICAD8_SYMBOL_DIR)/Transistor_Array.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Transistor_BJT	\$(KICAD8_SYMBOL_DIR)/Transistor_BJT.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Transistor_FET	\$(KICAD8_SYMBOL_DIR)/Transistor_FET.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Transistor_FET_Other	\$(KICAD8_SYMBOL_DIR)/Transistor_FET_Other.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Transistor_IGBT	\$(KICAD8_SYMBOL_DIR)/Transistor_IGBT.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Transistor_Power_Module	\$(KICAD8_SYMBOL_DIR)/Transistor_Power_Module.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Triac_Thyristor	\$(KICAD8_SYMBOL_DIR)/Triac_Thyristor.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Valve	\$(KICAD8_SYMBOL_DIR)/Valve.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Video	\$(KICAD8_SYMBOL_DIR)/Video.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1294 9v Battery Holder	C:/Users/cvw07/OneDrive/Documents/KiCad/projects/Speaker/speaker/Imports/1294/KiCADv6/2025-03-30_23-37-17.kicad_sym
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	2025-03-30_23-37-17	C:/Users/cvw07/OneDrive/Documents/KiCad/projects/Speaker/speaker/Imports/1294/KiCADv6/2025-03-30_23-37-17.kicad_sym

Migrate Libraries

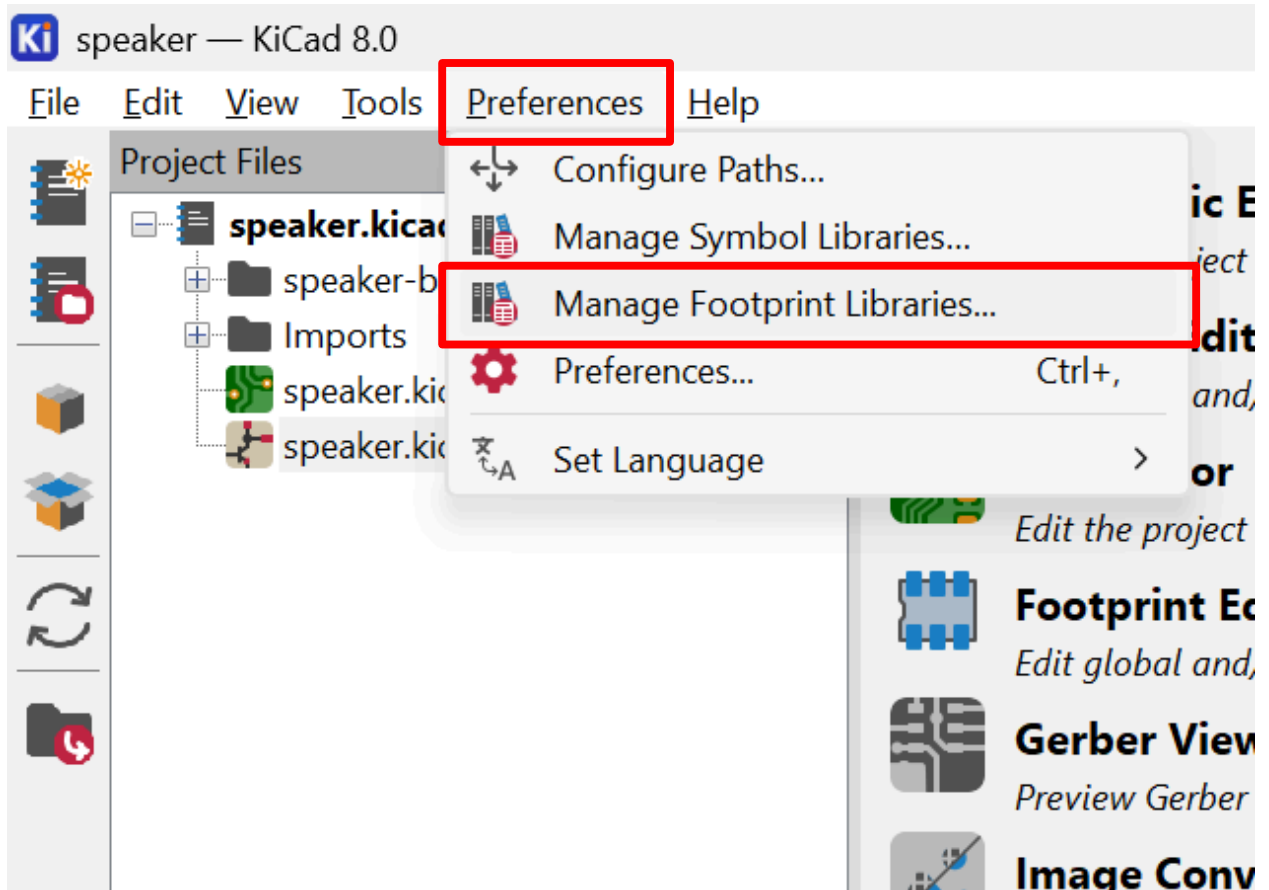
10. Rename the symbol to something relevant (e.g. NE555 timer)



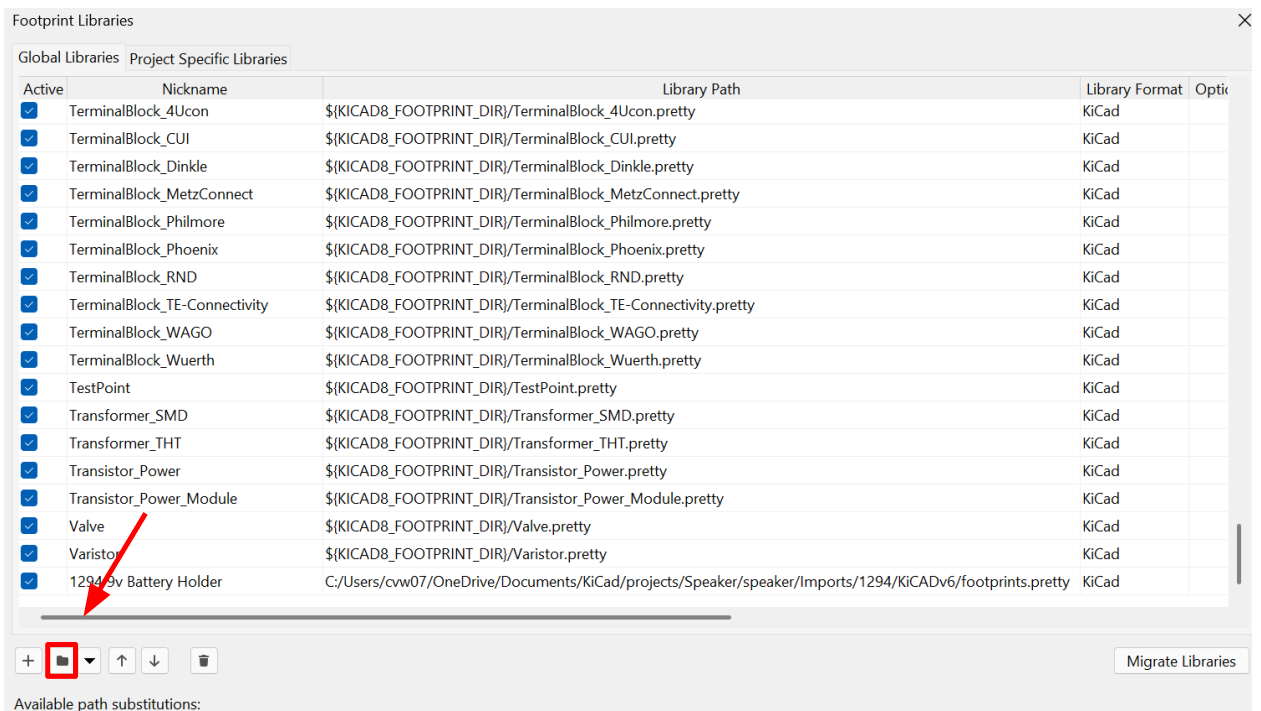
11. Inside KiCad Schematic Editor, add a symbol to your schematic and search for the part you just created

Importing Footprints

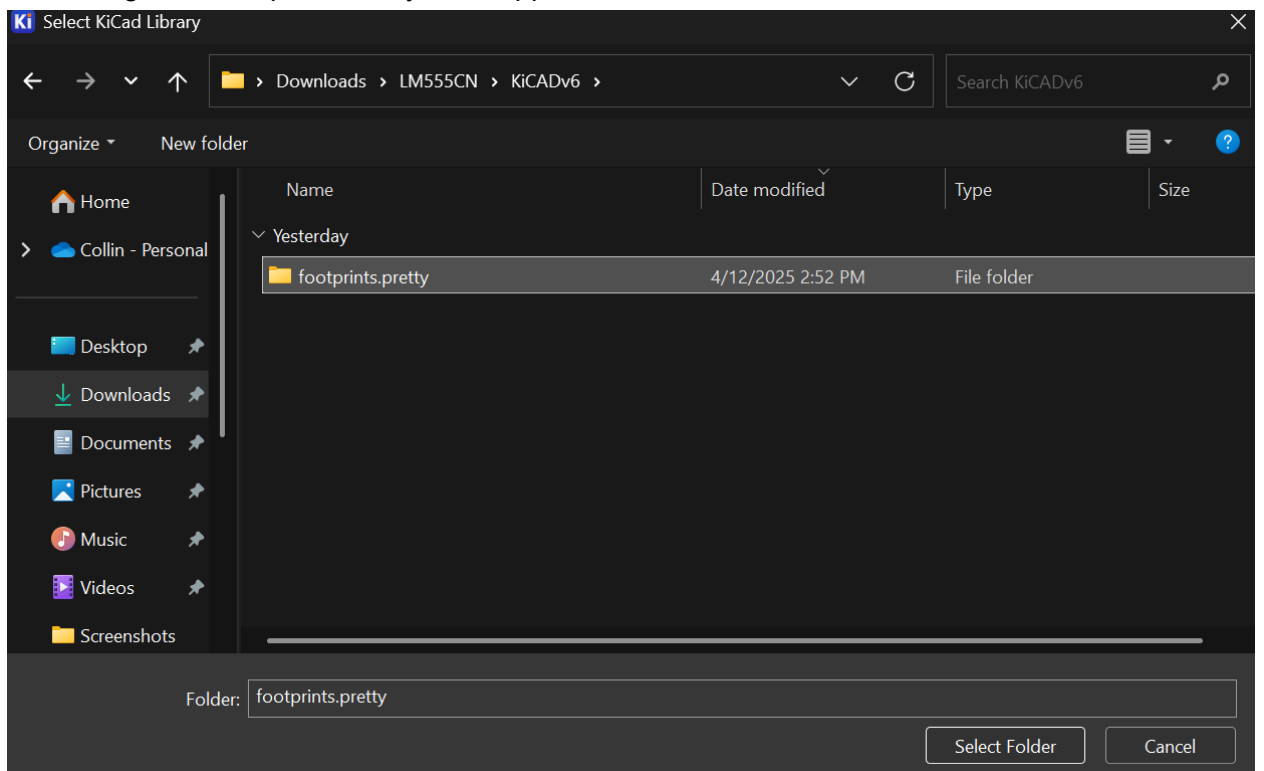
1. Perform steps 1 through 5 of [Importing Symbol](#) section



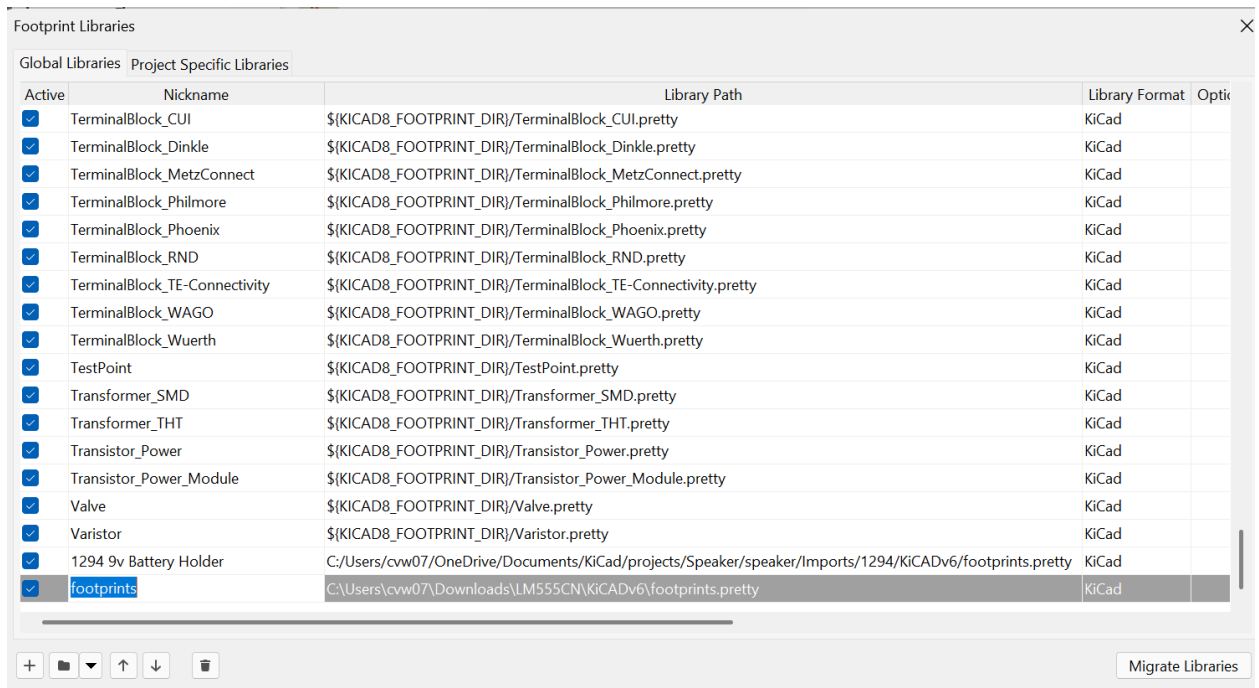
2. To import the Footprint open the current version of KiCad installed on your device and navigate to “Manage Footprint Libraries”



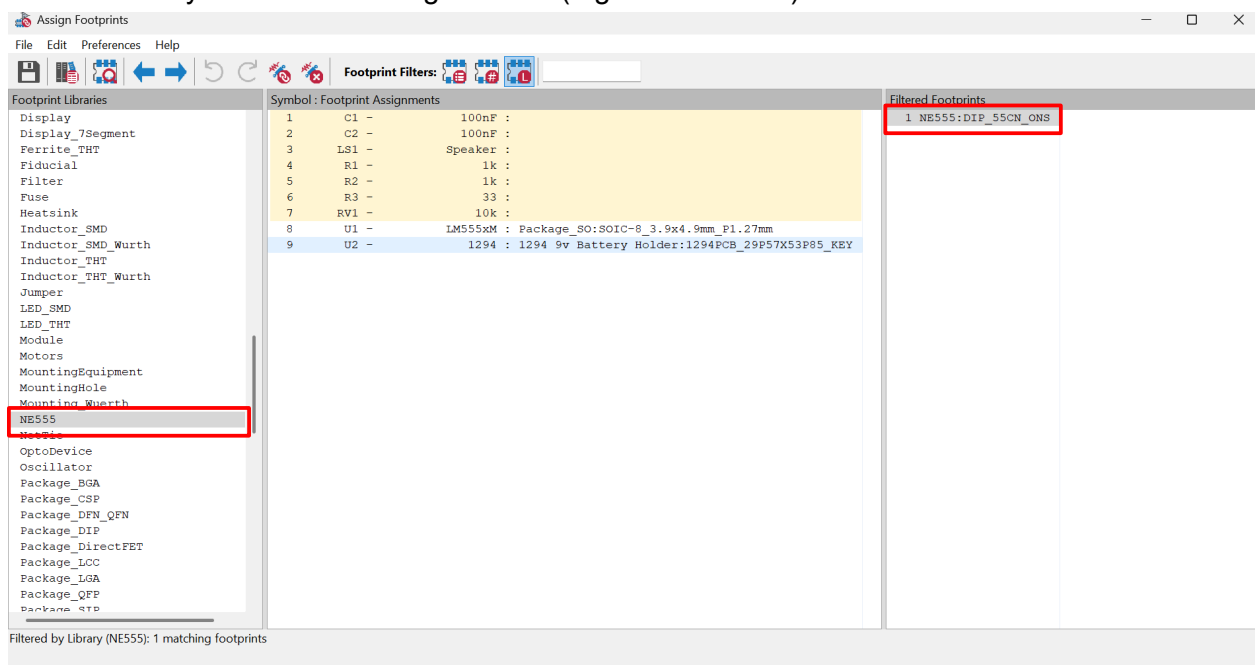
3. Click the folder icon in the bottom left corner
4. Now navigate to the place that you unzipped the EDA/CAD files



5. Navigate to the “.pretty” folder and then Select Folder



6. Rename the symbol to something relevant (e.g. NE555 timer)



7. The footprint should now appear in your footprint library when assigning footprints