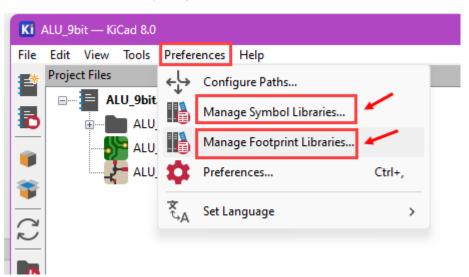
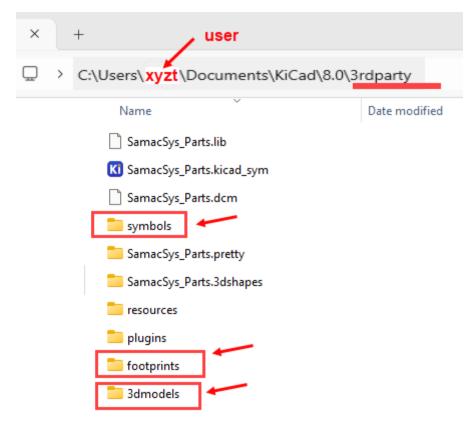
### Environment variables: xyzt is your user name

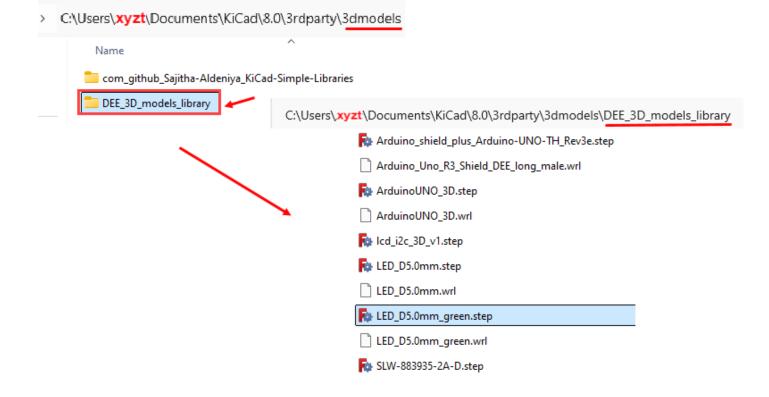


Configure Paths				
Environment Variables				
Name				
KICAD8_3DMODEL_DIR	C:\Program Files\KiCad\8.0\share\kicad\3dmodels\			
KICAD8_3RD_PARTY	C:\Users\ <b>xyzt</b> \Documents\KiCad\8.0\3rdparty			
KICAD8_FOOTPRINT_DIR	C:\Program Files\KiCad\8.0\share\kicad\footprints\			
KICAD8_SYMBOL_DIR	C:\Program Files\KiCad\8.0\share\kicad\symbols\			
KICAD8_TEMPLATE_DIR	C:\Program Files\KiCad\8.0\share/kicad/template			
KICAD_USER_TEMPLATE_DIR	C:\Users\franc\Documents\KiCad\8.0\template\			

Recommended folders for your libraries. Unzip and save your libraries in these locations:



Location and example of some of your 3D models:



# Location of your DEE\_Library footprints and example tuned and modified component models:

Footprint Libraries										
Global L	ibraries Project Sp	pecific Libraries								
Active Nickname			Library Path S{KICAD8_3KU_PAKTY}/footprints/com_github_dawidcislo_alternate-kicad-library/Potentiometer_THT_US_AKL_U	0						
PCM_Resistor_SMD_AKL			\${KICAD8_3RD_PARTY}/footprints/com_github_dawidcislo_alternate-kicad-library/Resistor_SMD_AKL.pretty							
PCM_SL_Mechanical			\${KICAD8_3RD_PARTY}/footprints/com_github_Sajitha-Aldeniya_KiCad-Simple-Libraries/SL_Mechanical.pretty							
DEE_Library			\${KICAD8_3RD_PARTY}/footprints/DEE_Library.pretty							
	your footprints library									
Available path substitutions:										
\${KICAD8	_3DMODEL_DIR}	C:\Program Files\KiCad\8.0\share	\kicad\3dmodels\ your libraries location							
\${KICAD8_3RD_PARTY} C:\Users\xyzt\Documents\KiCad\8.0\3rdparty			\8.0\ <mark>3rdparty</mark>							
\${KICAD8	\${KICAD8_FOOTPRINT_DIR} C:\Program Files\KiCad\8.0\share\kicad\footprints\									
\${KIPRJM	OD}	C:\CSD\P4\ALU_9bit\PCB								
			Your current project folder OK Cano	:el						

# C:\Users\xyzt\Documents\KiCad\8.0\3rdparty\footprints\DEE\_Library.pretty

- Innicadel_ixo1_iz.54mm_vertical_iong.kicad_mod
PinHeader_1x02_P2.54mm_Vertical_long.kicad_mod
PinHeader_1x02_P5.08mm_Vertical_long.kicad_mod
PinHeader_1x03_P2.54mm_Vertical_long.kicad_mod
C_Disc_D3.0mm_W1.6mm_P2.50mm_long.kicad_mod
C_Disc_D6.0mm_W2.5mm_P5.00mm_long.kicad_mod
C_Radial_D6.3mm_H11.0mm_P2.50mm_long.kicad_mod
DIP-8_W7.62mm_long.kicad_mod
HLCP-J100_long.kicad_mod
IDC-Header_2x20_P2.54mm_Vertical_long.kicad_mod
LED_D5.0mm_green_long.kicad_mod

## Location for the symbols DEE\_Library:

Symbol Lik	braries			$\times$
Global Li	ibraries	Project Specific Libraries		
Active	Visible	Nickname	Library Path	
	<ul> <li>Image: A set of the set of the</li></ul>	PCM_Transistor_BJT_Darlington_AKL	\${KICAD8_3RD_PARTY}/symbols/com_github_dawidcislo_alternate-kicad-library/Transistor_BJT_Darlington_AKL.kicad_sym	
~		PCM_SL_Resistors	\${KICAD8_3RD_PARTY}/symbols/com_github_Sajitha-Aldeniya_KiCad-Simple-Libraries/SL_Resistors.kicad_sym	
		PCM_SL_Screw_Terminal	\${KICAD8_3RD_PARTY}/symbols/com_github_Sajitha-Aldeniya_KiCad-Simple-Libraries/SL_Screw_Terminal.kicad_sym	
~		PCM_SL_Transistors	\${KICAD8_3RD_PARTY}/symbols/com_github_Sajitha-Aldeniya_KiCad-Simple-Libraries/SL_Transistors.kicad_sym	
	<u>~</u>	DEE_Library	<pre>\${KICAD8_3RD_PARTY}/symbols/DEE_Symbols_library/DEE_Librarv.kicad_svm</pre>	I
+		stitutions:	Migrate Librar	ies
{KICAD8	_3RD_PAF	RTY} C:\Users\ <b>xyzt</b> \Documents\KiC	ad\8.0\3rdparty	
KICAD8	_SYMBOL	_DIR} C:\Program Files\KiCad\8.0\sha	re\kicad\symbols\	
S{KIPRJM	OD}	C:\CSD\P4\ALU_9bit\PCB		
			OK Canc	el

### Symbols library

