



Layer verwenden  Via: Netzklassengrößen verwenden  F.Cu (PgUp)  0,0500 mm (1,97 mils)  Zoom 13,00

Footprintbibliotheken

Globale Bibliotheken **Projektspezifische Bibliotheken**

Aktiv	Aliasname	Bibliothekspfad	Bibliotheksformat	Optionen	
<input type="checkbox"/>	TerminalBlock_Philmore	\${KICAD9_FOOTPRINT_DIR}/TerminalBlock_Philmore.pretty	KiCad		Philmore terminal bloc
<input type="checkbox"/>	TerminalBlock_Phoenix	\${KICAD9_FOOTPRINT_DIR}/TerminalBlock_Phoenix.pretty	KiCad		Phoenix Contact termi
<input type="checkbox"/>	TerminalBlock_RND	\${KICAD9_FOOTPRINT_DIR}/TerminalBlock_RND.pretty	KiCad		RND terminal blocks
<input type="checkbox"/>	TerminalBlock_TE-Connectivity	\${KICAD9_FOOTPRINT_DIR}/TerminalBlock_TE-Connectivity.pretty	KiCad		TE Connectivity termin
<input type="checkbox"/>	TerminalBlock_WAGO	\${KICAD9_FOOTPRINT_DIR}/TerminalBlock_WAGO.pretty	KiCad		WAGO terminal blocks
<input type="checkbox"/>	TerminalBlock_Wuerth	\${KICAD9_FOOTPRINT_DIR}/TerminalBlock_Wuerth.pretty	KiCad		Wuerth Elektronik terr
<input type="checkbox"/>	TestPoint	\${KICAD9_FOOTPRINT_DIR}/TestPoint.pretty	KiCad		Test points, measurem
<input type="checkbox"/>	Transformer_SMD	\${KICAD9_FOOTPRINT_DIR}/Transformer_SMD.pretty	KiCad		Surface mount transfo
<input type="checkbox"/>	Transformer_THT	\${KICAD9_FOOTPRINT_DIR}/Transformer_THT.pretty	KiCad		Through hole transfor
<input type="checkbox"/>	Transistor_Power	\${KICAD9_FOOTPRINT_DIR}/Transistor_Power.pretty	KiCad		Power Transistors
<input type="checkbox"/>	Transistor_Power_Module	\${KICAD9_FOOTPRINT_DIR}/Transistor_Power_Module.pretty	KiCad		Transistor Power Modu
<input type="checkbox"/>	Valve	\${KICAD9_FOOTPRINT_DIR}/Valve.pretty	KiCad		Valve
<input type="checkbox"/>	Varistor	\${KICAD9_FOOTPRINT_DIR}/Varistor.pretty	KiCad		Varistor
<input checked="" type="checkbox"/>	01_WI-AEL_SMD	H:/kicad/footprints/01_WI-AEL_SMD.pretty	KiCad		
<input type="checkbox"/>	02_WI-AEL_THT	H:/kicad/footprints/02_WI-AEL_THT.pretty	KiCad		
<input type="checkbox"/>	03_WI-AEL_elektromechanik	H:/kicad/footprints/03_WI-AEL_elektromechanik.pretty	KiCad		



Verfügbare Pfadersetzungen:

\${KICAD9\_3DMODEL\_DIR} E:\Program Files\KiCad\9.0\share\kicad\3dmodels\  
 \${KICAD9\_FOOTPRINT\_DIR} E:\Program Files\KiCad\9.0\share\kicad\footprints\  
 \${KIPRJMOD} H:\Buffer\Einzel-FET-Buffer\Einzel-FET-Buffer