

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			Surface mount transfoi
<input type="checkbox"/>	Transformer_THT			Through hole transforr
<input type="checkbox"/>	Transistor_Power			Power Transistors
<input type="checkbox"/>	Transistor_Power_Module			Transistor Power Modu
<input type="checkbox"/>	Valve			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	

Message



Wählen Sie eine oder mehrere Zeilen mit Bibliotheken aus, die im aktuellen KiCad-Format gespeichert werden sollen.

OK



Bibliotheken zurücksetzen

Bibliotheken migrieren

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\

\${KIPRIMOD} H:\Buffer\Einzel-FET-Buffer\Einzel-FET-Buffer

OK

Abbrechen