Types of Symbols in KiCad

Symbol Type	Symbol	Footprint	Part	MPN	Symbol Type	Explanation	Symbol Name	Examples
1	✔	☐	☐	☐	Generic Symbol	A generic type of Part. A Generic Symbol requires the user to add BOM information and the Footprint assignment sometime during the schematic design process.	Symbol Name is the Name of the Type of Part. Generic Symbols are owned by the 'Device' Library'.	Resistor_THT Capacitor_THT
2	✔	✔	☐	☐	Partially Specified Symbol	A common type of Part	Symbol Name is the Name of the Type of Part. The Type of Part is described by the Library Name.	*Resistor.*THT_1W
3	✔	✔	✔	☐	Specified Symbol	A specific type of Part supplied by many manufacturers.	Symbol Name is the Name of the Type of Part qualified by the Part ID.	*Resistor.*SMD_0603
4	✔	✔	✔	✔	Fully Specified Symbol	A specific Part supplied by one manufacturer	Symbol Name is the MPN.	*Resistor.*Y162510K0000T9R