

# Tutorial: Component Library

A new schematics library that will contain the schematics symbol and component parameters is created `f → n → l → l`, and a new PCB library that contain footprints, silkscreen graphics and mechanical outlines is created `f → n → l → y`. The schematic symbol will later be associated with a footprint from the PCB library. This ensures footprint re-use, and is especially useful for common footprints such as chip capacitors and resistors.

Table 1: Example capacitor specifications.

Parameter	Value
Type	MLCC
Capacity	12pF
Tolerance	±2%
Temp. Coefficient	NP0
Package/Footprint	0402

The first step in the creation of a new component is to find a suitable component on one of the distributor's websites. DigiKey was used in the thesis, and has therefore been selected here as well. The component serving for the example will be the Bosch IMU. From DigiKey's website <https://digikey.com>, navigate to *Products* → *[Semiconductors]Sensors, Transducers* → *Motion Sensors - IMUs (Inertial Measurement Units)*. Then apply filters according to component specifications. This will be very useful for

components such as chip capacitors, but in this example we have already decided upon the IMU. The *In Stock* and *Normally Stocking* filter options at DigiKey can be helpful as they ensure availability of all design components in the future.

Having found a suitable component, specifications and data are now imported directly from the Digikey listing. From within the schematics library environment navigate to Panels → Part Search → [Icon]Options and ensure that Digikey is enabled in the suppliers column. Search using Digi-Key's product number, and import the component with the desired packaging RMB → Import Selected. All component specifications and values—including pricing and link to supplier for BOM—will be imported.

Next the schematic symbol is created. All pins are positioned. For this project the convention has been to position supply voltage pins in the top left corner, ground connections and NCs at the bottom left, main buses on top right and interrupts on the lower right. Always ensure that all pins are aligned to a 100mil grid for proper alignment. The final schematic symbol can be seen in Figure 2.

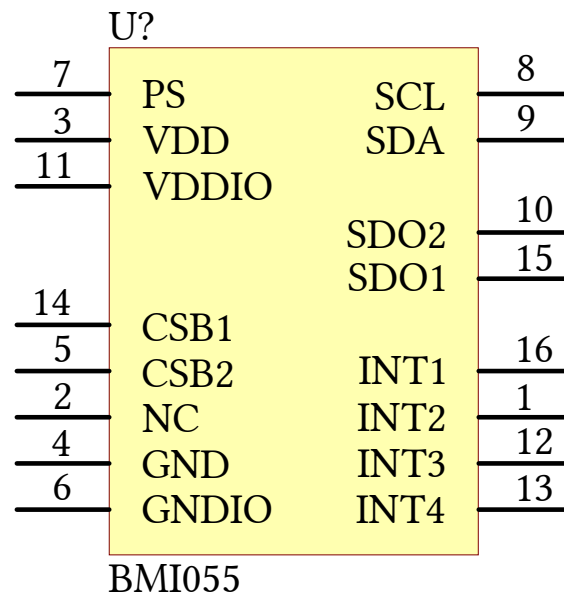


Figure 1: Schematic symbol for IMU.

The final part that constitutes a component is the footprint with a 3D model. Footprint recommendations are dimensioned and shaped as specified in the component's datasheet. From the PCB library environment a new component is created **CTRL + N**, and the footprint is drawn by placing pads **p → p**. Silkscreen and mechanical outlines are drawn using the set of drawing primitives also available via the placement menu **p → . . .**.

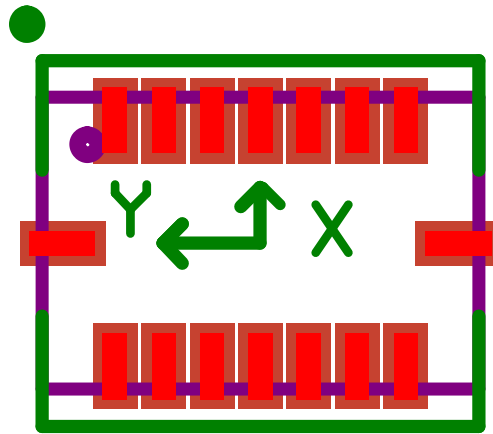


Figure 2: PCB Footprint for IMU.

The 3D model in the STEP is imported by **p → b**. Align imported 3D models with **t → b → f**, select the component, and then finally click on the component's surface that should be aligned with the footprint. If the 3D-model contains pads, this is a great opportunity to verify that the footprint is correct. Note that if the 3D-model has been obtained from third party sources, there error could potentially be in the 3D-model.

As footprints and schematic symbols are created separately, the final step is to associate the schematic symbol with the correct footprint. This is done by navigating to the schematic symbol in the SCH Library browser, then navigate to Add Footprint and select the associated footprint.