# Description

Generic High Energy Physics Parts Library (GHEPlib) is a CAD library to manage data across multiple electronic CAD tools.

# Dependencies

* Supported CAD tools
  + Siemens PADS VX2.3 or higher
  + Siemens DxDesigner (assumed import ability from PADS)
  + Altium Designer 23 or newer (tested)
  + KiCAD
* Symbol library
  + None
* Management tools
  + Python 3

# Used On

* Caribou v2.0 project
* Internal (confidential) projects at contributor sites

# Primary Developers

* Brookhaven National Laboratory
* Carleton University Particle Physics Instrumentation Group

# IP Control

* See License File
* Data can be imported from manufacturers into this library only if that data is accompanied by a waiver of ownership (stated as “free to use for any purpose including sale and open source”)
* Imported data without modification is kept in separate import libraries. If data is modified or customized, it should first be copied to the custom libraries. The indicated author should also change.

# Conventions

## Pin Mapping for CAD Data Reuse

There are multiple ways to handle generic symbols and footprints:

* part + generic symbol + pin remapped footprints
  + E.g. PART\_NUM, BJT, SOT23\_123
  + In practice this is very challenging to manage
* part + pin remapped symbol + footprint
  + E.g. PART\_NUM, BJT\_123, SOT23
* part + generic symbol + generic footprint + pin map file to associate pin numbers, pin names, or pin IDs between the symbol and footprint
  + Different CAD tools use different values to link schematic and layout.
* Part + one symbol + one footprint, 1 to 1 map
  + PART\_NUM, BJT\_123, SOT23\_123

Symbols and footprint pin mapping is dictated by the CAD tool (pin names or pin IDs are sometime used) and symbols and footprints must comply with this.

However, within this library, linkage by a positive integer is generally preferred. The reasons for this are

* This transcends 2D design limits (column+row format)
* This also applies to virtual / logical pins
* This is compact to store

The conventions for this are:

* Pin numbering provided by manufacturer should generally be preferred
* Alphanumeric pin IDs (such as for BGAs) should be converted by sorting alphanumerically and ascending
* Pins missing a number on manufacturer datasheets (E.g. power pads) should be assigned am unused number

**Selected strategy:**

For parts with 3 pins or less (transistors, potentiometers, etc.):

* generic symbols are available (pin remapping is done at the symbol level).
* 2 pins gives 2 unique symbols
* 3 pins gives 6 unique symbols and one unique footprint

123 mapping conventions:

* BJT CBE E.g. C→ 1 B→ 2 E→ 3
* MOSFET DGS
* Potentiometer top wiper bottom
* TVS diodes vcc signal gnd

For parts with more than 3 pins:

* It is considered a coincidence that the pinout is identical between two given parts. This even applies to fairly standard parts such as quad opamps, comparators, etc. since the pinouts usually still vary.
* If a high (>3) pin count part is found to have an identical pinout to an existing symbol and/or footprint, this CAD data can be directly reused without renaming any files and only changing properties as needed. This linkage should be indicated in the database.

A “drop in” replacement occurs when:

Manufacturer: May vary  
Manufacturer PN: May vary  
Symbol (includes pin numbering / connectivity): No change  
Footprint: No change  
Part specifications: May vary but typically must be close

A “near drop in” replacement occurs when:

Manufacturer: May vary  
Manufacturer PN: May vary  
Symbol (includes pin numbering / connectivity): No change  
Footprint: May vary  
Part specifications: May vary but typically must be close

## **General Rules for CAD Data Reuse**

* Note that, due to JEDEC, IEC, ANSI, ISO and other mechanical standards, footprint pin numbering is unlikely to change and footprint reuse is much more likely than symbol reuse.
  + It should not be assumed that manufacturers are standards compliant: pin numbering on footprints should always be verified before using a footprint
* A symbol which is a superset of another symbol is considered a completely different symbol
  + This is clearer in the schematic and prevents design errors
  + This is to target DRC cleanliness
  + E.g. DFN and TSSOP parts have the same pin numbering but the DFN has an additional PAD connection
* A symbol with multiple sub sections (units, heterogenous parts) is considered a single symbol
  + This is partly driven by CAD tool limitations: many CAD tools (Altium, KiCAD) actually have the symbols grouped within the database.
  + While some CAD tools (Siemens) have piece-meal symbols that could be reused, the symbols must still be linked together with properties or other data entry and it is considered best if this is included in the symbol.
* Separate footprints are allowed to accommodate different assembly methods (wave, reflow, bonding, etc.)
* Separate footprints for different design densities are allowed but typically only recommended for parts that are very frequently used (passives). Low volume footprints can be manually modified as needed. E.g. Silk screen can be removed.

## Schematic Symbols

* Imperial dimensions – no metric (yet)
* 0.100" pin-pin and primary grid spacing
* 0.020” drawing grid
* Digital symbols
  + 0.020” radius bubbles
* 2 pin component dimensions
  + Siemens < 200mil wide, 400mil pin to pin (per conventions of existing parts)
  + Altium <200mil wide, 300mil pin to pin (per conventions of existing parts)
* Pins should include pin numbers for ease of use

## Layout Footprints

* Dimensions metric
  + Imperial is often what is given by the manufacturers or required by a standard. Use the following rules to convert:

|  |  |  |  |
| --- | --- | --- | --- |
| Conversion(s) | Typical Applications | Metric grid (rounding) | Imperial equivalent |
| Strict tolerance | Drill sizes  RF  Wafer scale dimensions | 1um | 39.37e-6 inches |
| Standard imperial to metric | Connectors | 25um | ~1mil=0.001 inches |
| Rounded imperial to metric | General Purpose | 100um | 3.937 mil |
| Large rounded imperial to metric | Copper Fills  Mechanical Outlines | 1mm | 39.37mil |

* + The conversion strategy appropriate for the application should be used. Tight tolerances require smaller rounding errors. Note that the courser rounding is usually an integer multiple of the finer options.
  + For drill size matching between imperial and metric sizes, drills should always use strict (1um) conversion
  + Standard passive components footprint names should include whether they are imperial or metric to avoid confusion in duplicate names
* Silkscreen
  + Polarity indicators included but may be covered after assembly
  + All footprints should include a bounding box. The primary reason for this is to show which pads are grouped when a part is not installed. This also serves as a type of layout keepout.
* Soldermask
  + Dedicated shapes
* Solder paste
  + Dedicated shapes
* Assembly layers
  + Dedicated shapes
* Drill layer
  + Preferred drill size: 1mm
  + Other acceptable drill sizes
    - 40mil
    - 60mil

General design rules that footprints should meet:

|  |  |  |  |
| --- | --- | --- | --- |
| Rule | Min. | Typ. | Max. |
| Copper 1 (top)  to  Top silk screen | 0.4mm |  |  |
| Copper 1 (top)  to  Copper 1 (top) | ? |  |  |
| Silk screen text height |  | 2mm |  |
| Via ring width |  | 500um |  |
| NPTH copper ring to hole clearance | 0 | 10um |  |
| PTH copper ring hole coverage |  | 10um | Full hole |

# Device Property Conventions

* Multiplier Format
  + “DNI” = “DNP” = “NP” = 0
  + multiplier (integer) = 1xmultiplier
  + “” (empty) = 1
* Unique Resistance, Capacitance, Inductance, … properties
  + Siemens tools typically prefer a single VALUE property that is displayed. This is also more convenient for BOM export. However, this lacks clarity since the exact property being referred to may need to be assumed (e.g. diode forward vs. reverse voltages)
* Siemens
  + Convention for DEVICE = “Symbol Name” without any suffixed that are used to identify a part of a heterogenous symbol
    - Formerly, DEVICE = “Symbol Name”\_PN. As of 2023-03-08, the MANUFACTURER\_PART\_NUMBER property was added to enable the use of different symbols for the same part within the same design. This property name is also more explicit, clearer, and CAD tool independent
    - DEVICE must be unique since the tool uses this property for layout-schematic linkage
  + MANUFACTURER\_PART\_NUMBER= “PART\_NUMBER” before completion
  + Property names target the Netlist PADS/DxDesigner flow. The PADS documentation indicates that the cases for the properties vary between the Netlist and Intgrated design flows. E.g. [Part Number (PART\_NUMBER)](../../MentorGraphics/PADSVX.2.10/docs/htmldocs/attr/topics/General_PartNumberPartNumber_idee3e0fd4.html" \l "idee3e0fd4-f115-4b2d-a24a-fb65ff02efa8__General_PartNumberPartNumber_idee3e0fd4.xml%23idee3e0fd4-f115-4b2d-a24a-fb65ff02efa8)
  + HETERO property is fully completed if applicable. Symbol grouping is not explicitly stored in the database
  + PKG\_TYPE=”PKG\_TYPE” (unlinked – determined at PCB layout start)
    - PCB only parts do not have this property
  + Only properties necessary for display in the symbol and basic packaging are added to the symbol
    - This is faster to enter and allows direct addition of other parameters at the schematic level as preferred by designers, assuming that mapping to database fields are managed correctly
    - This is typically, DEVICE, PKG\_TYPE, PART\_NUMBER, REFDES
    - Note: For some reason, properties can only be moved if they don’t contain the default value

# Verification Flows

* PADS example project holds all newly developed symbols
  + Visual review
  + Tools→ PCB Interface… (Packaging)
  + Tools → Diagnostics
    - Shows if schematic library is out of date
  + PADS Databook → New hierarchical Verification Window
    - Shows if library items can’t be found
* Tested on the following CAD tools
  + Altium 23
  + PADS VX.2.10

# Debugging Tips

## Siemens

* Symbols can “break” and exhibit strange behavior when instantiated. This may occur if the text of the symbols is directly edited.
  + One observed symptom of this is that the tool may automatically (per an ambiguous / unknown rule set) change the case of characters in symbol names in schematics however, references to symbol filenames with different case characters still work
  + Another observed symptom of this was the inability to delete or add specific properties, especially ones that are not present in the properties list.
  + The solution to this is to copy and paste the symbol graphics to a new symbol – PADS will generate a new file and therefore clean errors
* All symbols and symbol instances in a HETERO part must have exactly the same properties, otherwise packaging fails
  + The visibility of the property can vary between symbols
* After Tools→ Update Libraries, pins may become corrupted and fail to package. To resolve this, the part must be full replaced: right click→ Replace Symbol→ Search for same symbol name→ Replace
* PADS layout libraries are **not** compatible with DxDesigner

# Table Descriptions

Primary keys in each table are indicated with “PK”

* CAD\_table
  + Relates parts to the **preferred** CAD files. A single part can have many CAD models or data files but only a few are preferred and/or tested.
  + MFG = Manufacturer
  + PN = Part Number. This is the primary manufacturer part number (not distributor, not internal)
  + All symbols, footprints, and simulation models are references to CAD files in the Data\_table
* Data\_table
  + path: This can be a path to a file or a section of a file.
    - If this refers to sections of a file, identifiers relevant to each CAD tool should be used.
    - Symbols recognized by the CAD tool can be used. This is useful for default libraries.
  + author: The most recent editor of the data. As soon as the data or a file is modified, the author is considered to have changed. This is for liability tracing
  + release\_version:
    - If a default library is used, the exact release version should be indicated.
    - If a file was directly downloaded from a manufacturer, the date of the download should be indicated.
  + sym\_group:
    - Some parts are best represented by multiple separate symbols. This indicates that they should be grouped together.
  + deployment\_history: A short description of the harshest environmental deployment that the design within the file has survived. Examples include “not deployed”, “functional”, “functional after shock and vibration testing…”, “functional after high temperature testing…”

# TODO

* Table generators
  + database to Altium database with dblink file
  + database to DxDatabook database
* Cleanup functions
  + CAD entry deduplicator with reference updates
* Import functions
  + Digikey BOM
  + Mouser BOM
  + General BOM