

## Cleanup PLX file exported from Library Expert

- Change (fileUnits mil) to (fileUnits mm)
- For SM pads
  - Delete lines:
    - (holeDiam 0 mm)
    - (isHolePlated False)
    - (startRange 1)
    - (EndRange 2)
  - Delete padShape lines with
    - (layerType Signal)
    - (layerType Plane)
    - (layerType NonSignal)
  - Padstack name shall not contain solderpaste pattern subname (e.g. "s140" instead of "s140p99" for the thermal pads)
    - Rename padstack in the whole file
    - Add missing solder paste layer for thermal pads (layer 22). Copy it from solder mask layer (layer 20)
    - Delete unnecessary solder paste elements (layer 22)
  - Rectangular padstack names shall have longer side shown in the name first
  - Corner radius shall be in 0.01 mm steps
- For TH pads
  - Change (EndRange 2) to (EndRange 16)
  - Delete padShape lines with
    - (LayerNumRef 20) - top mask
    - (LayerNumRef 22) - top paste
    - (layerType Signal)
    - (layerType Plane)
    - (layerType NonSignal)
  - For PTH pads
    - Check that (isHolePlated True)
  - For NPTH pads (mounting holes)
    - Check that (isHolePlated False)
    - Change name from "c50hn100" to "hn100"
    - Change sizes on (LayerNumRef 1) and (LayerNumRef 16) to 0 mm
- Check pad names vs actual pad dimensions
- Delete textStyleDef section
- Make patternDef and originalName value lowercase
- Check if patternDef and originalName are not truncated
- Check for odd pad coordinates (e.g. 2.733333333333)
- Check for odd line coordinates and correct line width
  - Layer 24 (top courtyard), line width 0.01 mm
  - Layer 26 (top component outline), line width 0.001 mm

- Layer 28 (top assembly), line width 0.05 mm
- Layer 18 (top legend), line width 0.1 mm
  - Or delete it
- Change layer 28 (top assembly) attr "RefDes" text style to "assembly 0.4 mm"
- Delete attr "Type" on layer 28 (top assembly)
- Change layer 18 (top legend) attr "RefDes" text style to "legend"
  - Or delete it
- Delete compDef section

#### Cleanup footprint in Pulsonix

- Import into Pulsonix
- Delete Pin names
- Edit "For Use By" settings of the padstacks
- Check padstack dimensions vs padstack name
  - Check rounded rectangle padstack corner radius to be consistent with padstack name
  - Check plated/non-plated for TH
- If necessary convert appropriate SM and TH pads to mounting holes
  - For SM: check if layer is <Top Side>
  - For TH: check if layer is <Through Board>
- Convert courtyard shape to area
  - Change shape type to "Area"
  - Check that "Shape > Layer" is <Top Side>
  - Check that "Line Style > Style" is "courtyard"
  - Set "Area > Placement Clearance" checkbox
  - Clear "Area > Body" checkbox
  - (Optional) Set "Area > Copper Pour Avoid" checkbox
- Convert outline shape to area
  - Change shape type to "Area"
  - Check that "Shape > Layer" is <Top Side>
  - Check that "Line Style > Style" is "component"
  - Clear "Placement Clearance" checkbox
  - Check "Area > Body" checkbox, enter appropriate height with units (e.g. "1mm")
- Check that assembly line style is "assembly"
- If necessary check that legend line style is "legend"
- Add/Delete unnecessary legend elements
- Change {<Component Name>} text alignment to "middle center" on assembly layer
  - Check that text style is "assembly 0.4 mm"
- If necessary change {<Component Name>} text alignment to "middle center" on legend layer
  - Check that text style is "legend"
- Check if "pin 1" notch is present on assembly layer and add it if necessary
  - For connectors:
    - Add first and last pin numbers on assembly layer
- For connectors

- Round-off courtyard to 0.05mm
- For plated TH pads draw a filled circle on layer "top paste" (pin-in-paste technology is assumed)
  - Paste circle diameter shall be the same as copper pad diameter
  - Layer class "paste" is set to generate openings for SM pads only to avoid unnecessary openings in paste stencil but any explicitly drawn shape will be included too
- For non-plated TH pads draw a filled circle on layer "top mask" and "bottom mask" to create solder mask clearance
  - Solder mask circle diameter shall be 0.2 mm larger (0.1 mm per side)
- Check if pads are numbered correctly