Library Management for an Ever-Evolving Diverse EDA Tool Industry

Michelle Gleason
Plexus Corporation
Neenah, WI

Abstract
Any company that designs PCBAs that has gone through the throws of an acquisition, CAD tool change or even an EMS company with a varied customer base has to deal with more than one EDA tool and library. Many questions arise. How can the data be managed and re-used so parts built and validated in one tool can be leveraged for future designs in any tools available? How can leaders manage a global library team, share work between teams, monitor and prioritize work based on current status information? This paper will illustrate how multiple libraries can be designed to accommodate a variety of tool flows, and demonstrate a custom tool developed to control the library creation process as well as track revisions that are made to library models. The tracking, history, datasheets, supporting data and communication of this information is also managed within this browser based tool. Many CAD vendors offer specific solutions that are aligned to their tool sets. This solution is independent of any EDA vendor. This web-based tool also provides functionality that allows a global library team to seamlessly share work with workflows designed for custom part creation processes.

Introduction
EDA Library management today is a constant challenge. EMS companies with a varied customer base are tasked with providing libraries efficiently in an ever-growing EDA tool and vendor industry space. Mergers and acquisitions of companies/organizations also present a challenge for existing business units to efficiently combine and/or leverage their investments in EDA tools and libraries. EDA vendors today offer library management tools for their specific tool flows, but there is not a solution in the market place today that allows for users to manage libraries independent of EDA tool vendor. In this paper, the challenges of managing multiple EDA libraries will be presented as well as a possible path to a long term solution.

Typically within a design firm there may be one EDA tool flow that all designers use, and there may or may not be a formal library process leveraged to create and control that library. Many companies spread the library creation task between the design engineers and the PCB designers.

These companies may have either a single project approach or an enterprise oriented approach to library development. A company with a single project approach would apply one library for a specific project/product. This solution is effective in a singular sense, however the ability to leverage existing symbols and footprints between designs, and even root causing and solving yield issues becomes a project by project challenge without the ability to leverage the accomplishments on one project to the next project. Within this context, there can be less of an effort put forth to make symbols or footprints consistent between separate projects. Any consistency that does exist will be constrained to within the single project.

An example of an enterprise solution would be to implement a corporate library and have specific trained resources assigned to the librarian task. In this case, the EDA library is created for the whole design community to utilize. The corporate library allows the ability to leverage parts that may be used throughout all products. As this library evolves, and the products of the company evolve, the quality of the library is proven and the EDA components can be reused with higher confidence because these schematic symbols and PCB land patterns have been proven not only in the design of the products but also in the manufacturing assembly process. Because these parts can be used across multiple products, any issues found along the development and manufacturing process in one product can be fed back into the corporate library ensuring that future use of these parts will avoid the same issues. The challenge then becomes not only the initial creation process but also the traceability and tracking of these EDA components and along with any revisions that may need to be made. Once the library creation process is documented, a process to determine how to share the librarian work among the team is the next challenge. The last layer adding complexity to this scenario is the company that needs to support many different EDA tools within this corporate library. The management of different EDA tool flows, the libraries that support those flows, and the need to keep the libraries consistent within a given tool and between tools really drove the need for the solution described in this paper.

This paper and Presentation ‘Library Management for an Ever-Evolving Diverse EDA Tool Industry’ was first presented at the 2018 IPC Apex Expo Technical Conference and published in the 2018 Technical Conference Proceedings.

Defining the Standard

While industry standards exist for guidance to the librarian resources, none of the standards provide an all-inclusive definition of schematic symbols or PCB land patterns.

Schematic symbols require a clear standard for the definition of pin name and pin number. These two pieces of the symbol are critical to the functionality of the symbol within the design. Specifics of how to number mounting pins, thermal pads need to be defined. Pin numbering when either not specified on the datasheet, or when only pin 1 is defined is another item that needs to be clearly documented in a standard. Of less importance to symbol creation, is the location of pins and any artwork or drawing meant to imply logical function of the component. All symbols can be defined as boxes with pin names and numbers and define the connections of the pins within the schematic. Pin location/orientation on the symbol contributes to the readability of the schematics, but it is not critical to the functionality of the schematic symbol. In addition, a symbol that does not contain graphics visually explaining the logical function of the component will still operate as required in a schematic, though it will make the schematic much more difficult to interpret. A symbol standard that documents fully the pin naming and pin numbering scheme and also gives guidelines for the general look of each symbol type is the first step to the consistency of a library and of the resulting schematics.

PCB land patterns have many more details that are critical to the success of the design. The land pattern by its simplest definition should contain the shape of the area where the pins of the component will be soldered to the PCB. Beyond that definition, some organizations will add more detail to ensure the footprints will be easier to use within in a PCB design and/orto provide guidance in a manufacturing assembly process. Details like solder mask openings, paste mask apertures, assembly outlines, component height, and specific placement requirements, are critical to the success of the yield of the assembly to which they are applied and should be included in any standard for footprint creation. These items need to be fully defined before a company or library team can develop a strategy for their library creation.

Process

The process of creating a library is another critical item to library development that may be overlooked. The process of creating a schematic and layout model for a component is a necessary chore of this industry. The importance of building these “molecules” of the EDA design world correctly and concisely is often undervalued. Undervalued until an assembly with either a land pattern or a net connectivity issue is on the assembly line and parts do not fit or function. The process at a minimum should include a building process, a verification process, and a release process to a write protected location. The building process and verification process should be the result of documenting the standard and then building and verifying the standard is followed. The release process is critical to move the correctly built part to a location not editable by the user community. Preserving the state of the symbol and footprint as they have been verified provides confidence to the users that those parts will function as needed in their designs, and will assemble properly in a manufacturing environment. The write protected location needs to be easily accessed for an engineering team. This may mean that there are multiple write protected library copies placed locally to each of the engineering teams with a single master location defined. In this scenario, a propagation process of the master to all the copies also needs to be defined.

In addition, verifying the manufacture part number is valid and is available for new designs should be additional steps in the process that would keep the library at a very high quality. The orderable part number can be verified with the supplied datasheets for each part. To determine if the manufacture part is available, a check with an approved supplier list/service can be done to ensure that the part can be purchased. If the library team has an established process that includes building, checking, and a process to move that verified component into a read only library, the process will result in a high quality and consistent library.

Naming

There are many EDA tools in the market place but all of the tools require a schematic that contains logical symbols for components, and a PCB layout tool where land patterns developed for each physical component are leveraged for a PCB design.

What is not consistent is the use of these EDA tools in the user community. The tool flow which defines the specific Schematic tool/PCB tool that is used for their designs can vary greatly. Some companies follow the EDA vendor's playbook and use the schematic tool and layout tool from the same vendor in the flow that the EDA vendor intended. Other companies, for any number of reasons, will have a preferred schematic tool and a preferred layout tool that do not come from the same EDA vendor. These tool flows are possible and because these tool flows cross between EDA vendors, the library support and management becomes a challenge. Figure 1 below is an example of the complexity when multiple EDA tool flows are supported.

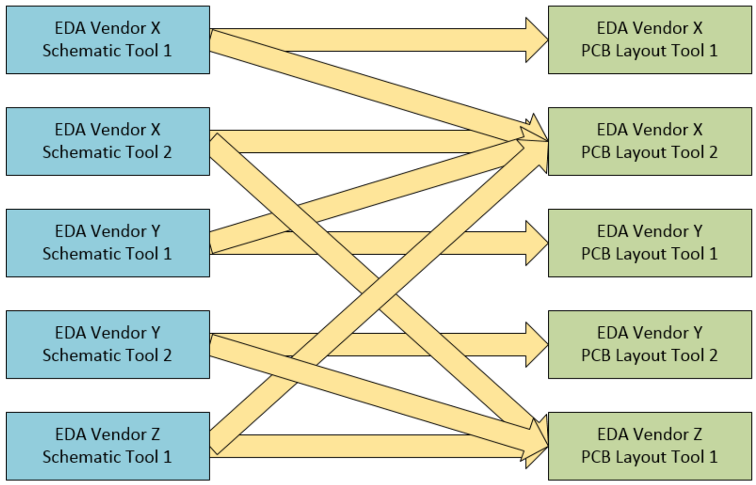


Figure 1: Example of Possible EDA Tool Flows within and between multiple EDA vendors.

In order to be successful in managing EDA libraries with variations shown in Figure 1, a few factors must be considered.

To support the possible variations of targeted layout tools, a library can leverage a consistent naming convention for the PCB footprints. By naming a footprint the same across EDA layout tools, the name of the footprint that is built into the schematic symbol is PCB layout tool independent. When the naming convention for the footprint is consistent between the layout tools, the targeted PCB layout tool for a given schematic symbol is variable until that symbol is placed on a schematic and the schematic is pushed to PCB layout.

Another factor that is critical when implementing this naming convention is to ensure the details within the footprint must be consistent in all ways possible. The size of the land pattern, the order and assigning of pin numbers, and the number of pins are the first critical items to be kept consistent between footprints named the same in two different PCB layout libraries. In addition, consistent design of package keep out, silkscreen, assembly outline, route keep out, via keep out, and any other details should also match when a footprint of the same name is created in a different PCB layout tool. The list of items that would need to be equivalent should be documented based on each PCB layout tools capabilities. Enforcing this convention allows a schematic symbol calling out a specific land pattern constructed in one schematic tool to be available for use in multiple tool flows just by creating a footprint of the same name in multiple layout tools. An example application has a footprint for the 0603 resistor created in multiple layout tools, a single 10K resistor schematic symbol calling for that 0603 footprint can be used in a schematic and pushed to any of the layout tools that contains the targeted 0603 footprint in its library.

Solution

The first step in implementing a global team to address the EDA library creation need is to document the process as outlined above. Once a robust process is defined to handle these challenges, the next step was to determine how much of this process can be automated, and how much of the data can be controlled and re-used to allow for efficient library creation.

As alluded to earlier, the solution for this problem needs to meet the following criteria.

1) High Quality - Process and Output
2) Web based tool for an international team
3) Seamless work-sharing
4) In-a-moment workload assessment
5) Full Traceability within the creation process
6) Revision history
7) EDA Vendor Neutral Solution

A system has been developed that provided the functionality to request EDA models for manufacture parts, to facilitate the creation of those models, and manage this library information in a single tool. The company has developed a web-based EDA part request system that allows engineers at different global design centers to enter a request or ticket for an EDA schematic symbol and footprint for a specific manufacturer part number and attach any data needed for the creation of the EDA components in EDA tool flow. Shown below in Figure 2 is an example of a request from an engineer. The ticket indicates a schematic symbol tool and a footprint tool so the librarian knows what EDA tools to use in the model building. Each ticket has a workflow associated with it that steps the librarians through the part creation process, as well as ensuring that there is a check and a release process that is followed.

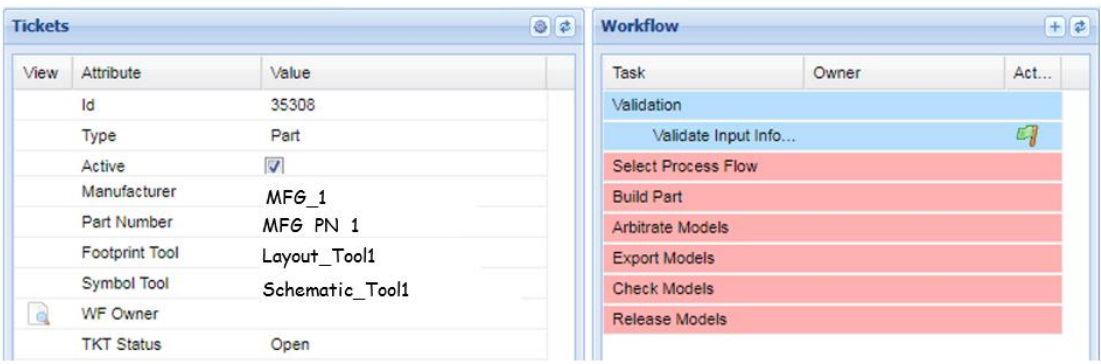


Figure 2: Example of Part Ticket

The workflow provides the librarian the ability to claim a request, fulfill a portion of the workflow, complete and release the request for the next librarian to continue the workflow. An example of the workflow is shown below in Figure 3. The workflow steps the single ticket through the process of creating the schematic symbol and the PCB footprint and verification of those new models. The colors shown at each step of the workflow give a clear visual of the status. Steps marked in green are completed steps. Steps marked in blue are in process, and steps marked in red have yet to be done. The flag is the icon that the librarian clicks on to claim that step of the workflow.

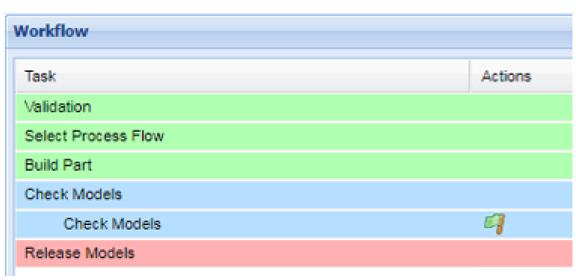


Figure 3: Example Workflow

Each open request or part ticket is shown in the ticket queue (See Figure 4) from which a team of librarians are able to draw work.

View	Manufacturer	Part Number	TKT Status	WF Status
	MFG_1	MFG_PN_1	Open	Check Models
	MFG_2	MFG_PN_2	Open	Select Proces...
	MFG_3	MFG_PN_3	Open	Validation
	MFG_4	MFG_PN_4	Feedback	Validation
	MFG_5	MFG_PN_5	Open	Check Models
	MFG_6	MFG_PN_6	Open	Check Models
	MFG_7	MFG_PN_7	Open	Check Models
	MFG_8	MFG_PN_8	Open	Check Models
	MFG_9	MFG_PN_9	Open	Select Proces...
	MFG_10	MFG_PN_10	Open	Check Models

Figure 4: Example Ticket Queue

The ticket queue shows the ticket status and the workflow status for each request. The view of the ticket queue shows the present workload for the librarians. This view allows a quick assessment of workloads, and estimated lead-times for part availability in an EDA library. If there are many tickets at a similar step in the workflow the interface can also show there may be a potential bottleneck in the process that needs to be addressed. This bottleneck can be communicated to organization leadership where a prioritization can be provided before multiple projects encounter significant schedule delays. The requestor can also quickly assess the status of the parts they requested. Once a ticket is completed, the ticket is closed and the requestor is notified via email. The tool also tracks communication between requestor and librarian to document the decisions and/or special requirements made as the part is created. The ticket queue shown in Figure 4 above has one ticket with a ticket status of feedback. When a ticket status is set to feedback, the requestor is notified that the process is stalled by an automated email indicating more information is needed. Additionally, the next available step in the workflow is displayed. Once a librarian claims that step in the workflow the ticket queue will reflect that the librarian is working on the ticket so that the rest of the librarian team will know that ticket is in process.

While this ticketing system is not unique, this system provides a connection to a database that is structured to relate these tickets to a manufacturer part number requested and to the EDA models that are created in response to that request. As a new manufacture part is requested, the EDA tool flow is also initiated. This EDA tool flow directs the EDA librarians what tools to use to build the schematic symbols and the PCB footprint. Once the workflow is completed, the ticket is closed and an entity is left in the database that links the following together:

1) The orderable manufacturer name and manufacturer part number
2) All of the information used to build the EDA models
3) All of the related EDA models that were created

Figure 5 shows the result. This entity is used in the future where the component is being selected again but for another tool flow. The new ticket will be linked to this entity which allows easy access to all of the documentation used to initially build the models for this manufacturer part number. In addition the database displays the other EDA models that have already been created for this same manufacturer part number. The linking of data at this level easily allows the librarians to keep the symbols and the footprints and the naming consistent across all EDA tools.

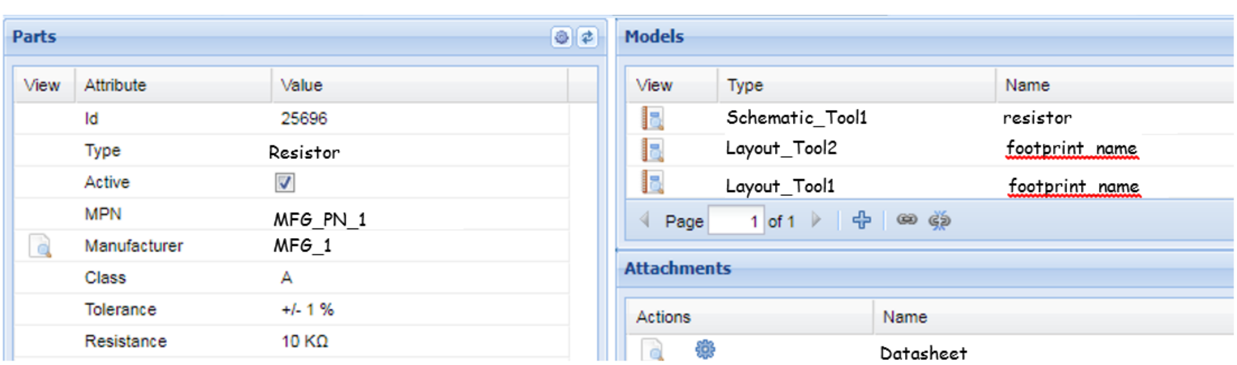


Figure 5: Example Part showing related EDA symbol and footprint models.

This paper and Presentation 'Library Management for an Ever-Evolving Diverse EDA Tool Industry' was first presented at the 2018 IPC Apex Expo Technical Conference and published in the 2018 Technical Conference Proceedings.

Regardless of how much process and quality are used to create EDA library symbols and footprint, there are issues that arise that require updates to symbols and footprints that have already been created. This tool allows the reporting engineer to open an Administrative ticket. This admin. ticket is also linked to the manufacturer part entity. The admin. ticket is used to document that a change is requested, and allows the engineer to attach any additional information that is needed to support the issue. This admin. ticket can then be claimed by a librarian and addressed. The librarian becomes responsible to investigate what is needed to resolve the issue. Any of the data the reporting engineer attached to the admin. ticket can be pushed to the manufacturer part number as part of the history of that entity. The challenge is the change requested may impact existing schematics or board layouts or other existing manufacturer part numbers in the library that also use this symbol or footprint. The change also may also impact symbols or footprints in multiple EDA tools. Once the change and its impact is fully understood, the change is made in the required libraries and the needed updates are available for all to use. The admin. ticket is updated with the results of the change, and the admin. ticket is closed yet remains linked to the entity in the database.

The database discussed here, with all of the linked data, is fully available to the librarians. Each librarian can review all the input documentation, any communication that occurred between the engineers and the librarians in the process of the EDA library model creation very quickly. The database also allows the librarians insight to where the process can fall down, if training is needed either for the engineer or the librarian, or if additional definition or documentation is needed.

Another benefit of this database structure is that it provides additional information to a librarian when a change to a schematic symbol or PCB footprint is needed. Before this database existed, the concept of updating a footprint for a single manufacture part number that may or may not also be related to other manufacture part numbers was a difficult task. The change was either made based on only the manufacturer part number with the issue. Or an investigation of impact was abandoned, and a new footprint or symbol was created by default. The thought being, that creating a brand-new footprint or symbol ensures no other part is impacted. The issue is that the library and maintenance quickly can become inconsistent or difficult to manage with all the different exceptions. Additionally, any other manufacturer part that should be impacted because the change was needed across more than the initial part will not get the needed update. While creating a new symbol or footprint to address a specific issue is an acceptable solution, the possibility of fixing a more general problem is lost and the library is forced to be larger than necessary. If the librarian can easily determine all the manufacturer parts that were impacted by a specific symbol or footprint, a more complete and comprehensive fix could be put into the library driving in better quality (see Figure 6). This is a view of the data from the footprint perspective. For the given footprint, the librarian can easily see all the manufacturer part numbers that have been assigned.

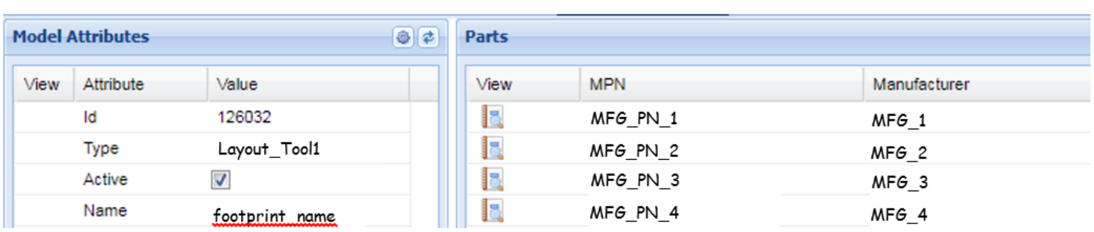


Figure 6: Example of a specific EDA footprint with a list of Manufacture part numbers assigned to it.

A change to a footprint within a regular library structure would be risky without knowing all of the other part numbers that are assigned to this symbol or footprint. With this database tool and the fully linked information between manufacturer part numbers and EDA models, a suggested change to a symbol or footprint can be fully investigated for all manufacturer parts that would be impacted. Allowing the librarians to make a fully informed decision on what changes should/could be made and to understand the impact to each manufacturer part that is related. If it is decided that the change needed would adversely affect other related manufacturer parts and the need for a specific footprint or symbols is required, this update can be made, the database can be updated and the reasons for this uniquely created symbol or footprint is documented.

Conclusion

In summary, this library management tool and the processes explained within this paper allow librarians to manage manufacturer part numbers and the associated EDA schematic symbols and PCB footprint across any EDA tool flow. In today's product development environment, where global teams are the state of the industry, this tool allows a global engineering team to interact with a global library team efficiently and effectively. Implementing support for the next EDA tool that hits the market is not as big of an obstacle once a library management tool of this capability is in place.

Library Management for an Ever Evolving Diverse EDA Tool Industry

Michelle Gleason

Plexus Corp.

Neenah, WI

Introduction

- Who?
  - *Companies as a result of a merger or acquisition*
  - *EMS, and Design and Development Companies supporting multiple customer needs*
- What?
  - *Tasked with supporting multiple EDA tools possibly from multiple vendors*
  - *Wanting to protect their current investment in EDA tools and libraries*

Introduction

- How?
  - *Single Project Approach*
    - Project specific library
    - Project team creating library parts
    - Consistent symbols and footprint within the project
    - Issues solved for specific design only
    - Duplicating work if library is not leveraged for other designs
  - *Enterprise Approach*
    - Corporate Library
    - Specifically trained library team
    - Consistent symbol and footprints
    - Issues solved for all future projects
    - Library development leveraged for all future projects

An Enterprise approach will be discussed here.

Defining the Standard

- Schematic Symbol Standard
  - *Required*
    - Pin Names
    - Pin Numbering
    - Pins without Names or Numbers
      - *Ex – Thermal pads, mounting pins/pegs, Shield pins*
  - *Suggested Guidance*
    - Pin Location
    - Artwork/graphical notation

Symbol standards beyond the required definition are mainly to create more readable schematics and to provide a consistent look within a library.

Defining the Standard

- PCB Land Pattern Standard
  - *Required*
    - Padstack design
    - Courtyard/Placement Boundaries
    - Orientation/Pin 1 Indicator
  - *Suggested Guidance*
    - Assembly Outline
    - Keepouts (Route/Via)
    - Silkscreen
    - Component Height

Land pattern standards can originate from industry standards, but additional requirements need to be defined for PCB fab house or electronic assembly/inspection needs.

Defining the Process

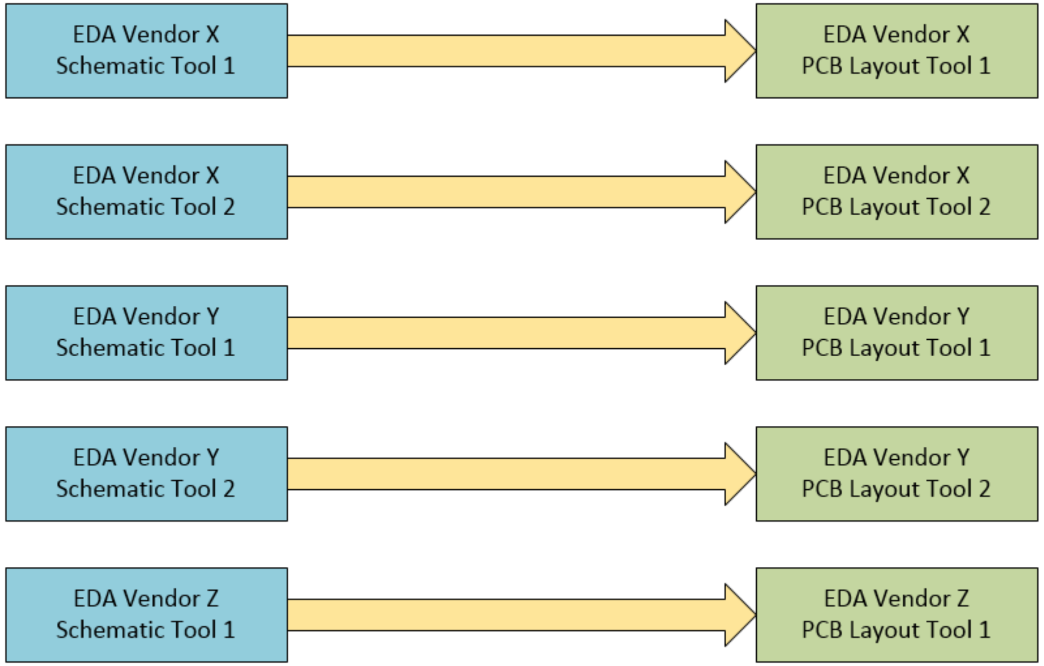
- Minimum
  - *Building*
  - *Verification*
  - *Release*
- Suggested
  - *Part Number Validation*
  - *Part Availability*

A library creation process that includes a verification step reduces the chance of human error that can be introduced into the library.

Releasing the verified parts to a write protected library ensures that the parts will stay as verified for all users of the information.

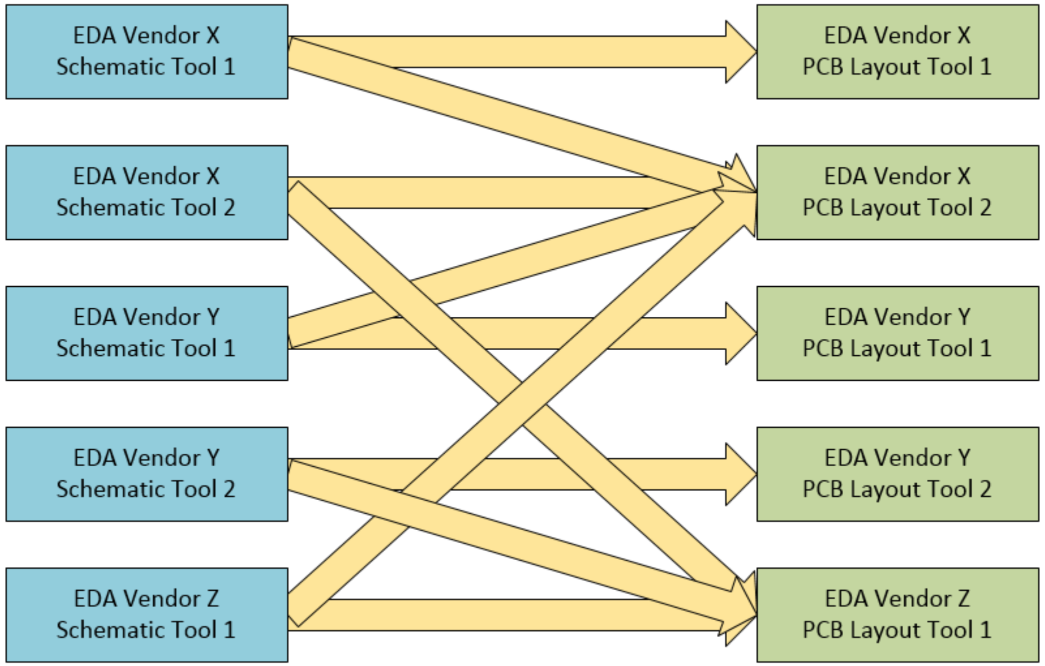
Defining the Naming

- Tool Flow Typical
  - *EDA Vendor expected data flow*
  - *A single company may support multiples of these flows*



Defining the Naming

- Tool Flow Alternatives
  - *For EDA Vendors supporting these flows library management is the challenge*
  - *A schematic symbol built in one tool may need to be used in different PCB layout tools.*



Defining the Naming

- PCB land pattern naming is the important factor
- Name a land pattern consistently between all PCB layout tools
- Keep the details within that land pattern consistent between all PCB layout tools

This practice allows a symbol to be created without being specific to the PCB layout tool that is being targeted.

Once there is a matching footprint in each PCB layout tool, the schematic being designed is layout independent until it is packaged in that specific layout process.

Solution

- Goal
  - *High Quality*
  - *Web-based tool*
  - *Seamless Work-sharing*
  - *Easy Workload assessment*
  - *Full Traceability*
  - *Revision History*
  - *EDA Vendor Neutral Solution*

The solution needs to meet the goals above and allow for efficient library creation and maintenance.

The following system has been developed.

Tickets and Workflow

- A Ticket or Part Request is created from an engineer including Manufacturer, Manufacture Part Number, and datasheet of needed part.
- Example Ticket and associated workflow

Tickets

View	Attribute	Value
	Id	35308
	Type	Part
	Active	☑
	Manufacturer	MFG_1
	Part Number	MFG_PN_1
	Footprint Tool	Layout_Tool1
	Symbol Tool	Schematic_Tool1
	WF Owner	
	TKT Status	Open

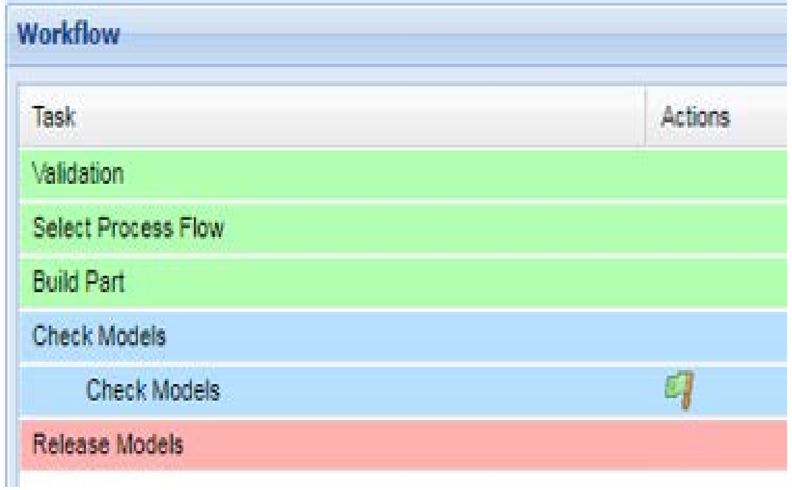
Workflow

Task	Owner	Act...
Validation		
Validate Input Info...		
Select Process Flow		
Build Part		
Arbitrate Models		
Export Models		
Check Models		
Release Models		

Ticket contains manufacture part information and tool flow information. The workflow walks the librarian through the EDA Part creation process.

Tickets and Workflow

- Each step of workflow is claimed, performed, and completed.
- Completing a step releases the ticket for another librarian to claim the next step.
- Colors indicate status of steps –
  - *Green = Completed*
  - *Blue = In Process*
  - *Red = Left to do*



Workflow allows easy assessment of the state of the creation of the request.

Ticket Queue

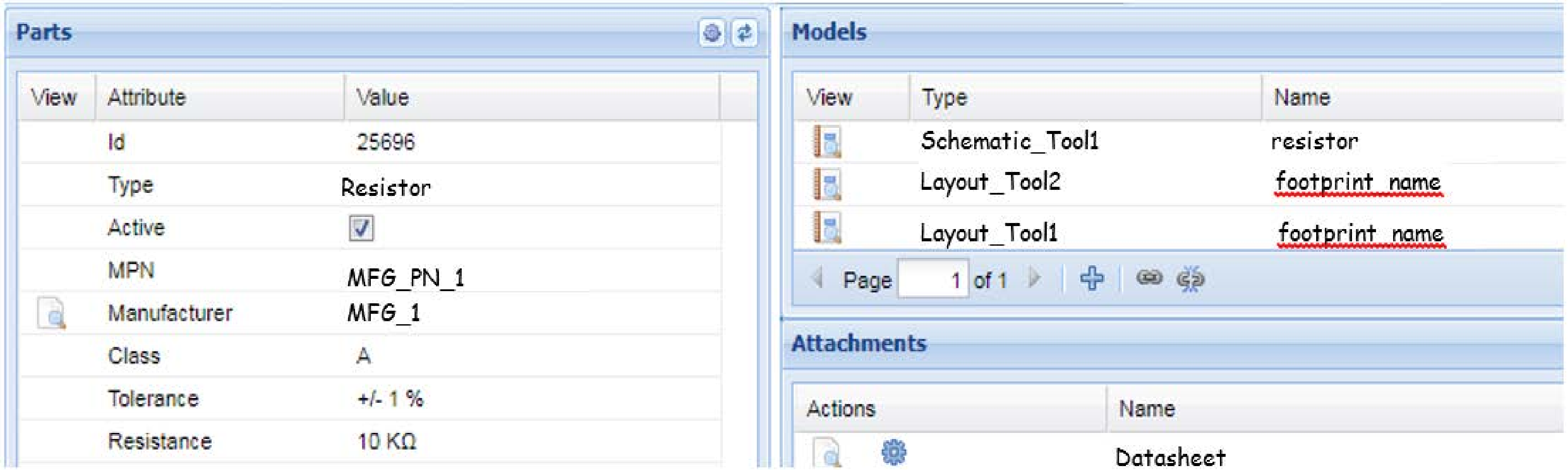
- List of tickets and their current workflow status.
- Ticket status shows requestor one ticket in “Feedback” indicating more information is needed.

View	Manufacturer	Part Number	TKT Status	WF Status
	MFG_1	MFG_PN_1	Open	Check Models
	MFG_2	MFG_PN_2	Open	Select Proces...
	MFG_3	MFG_PN_3	Open	Validation
	MFG_4	MFG_PN_4	Feedback	Validation
	MFG_5	MFG_PN_5	Open	Check Models
	MFG_6	MFG_PN_6	Open	Check Models
	MFG_7	MFG_PN_7	Open	Check Models
	MFG_8	MFG_PN_8	Open	Check Models
	MFG_9	MFG_PN_9	Open	Select Proces...
	MFG_10	MFG_PN_10	Open	Check Models

Ticket Queue allows seamless work-sharing for a global library team and easy/quick assessment of workload for library team.

Result

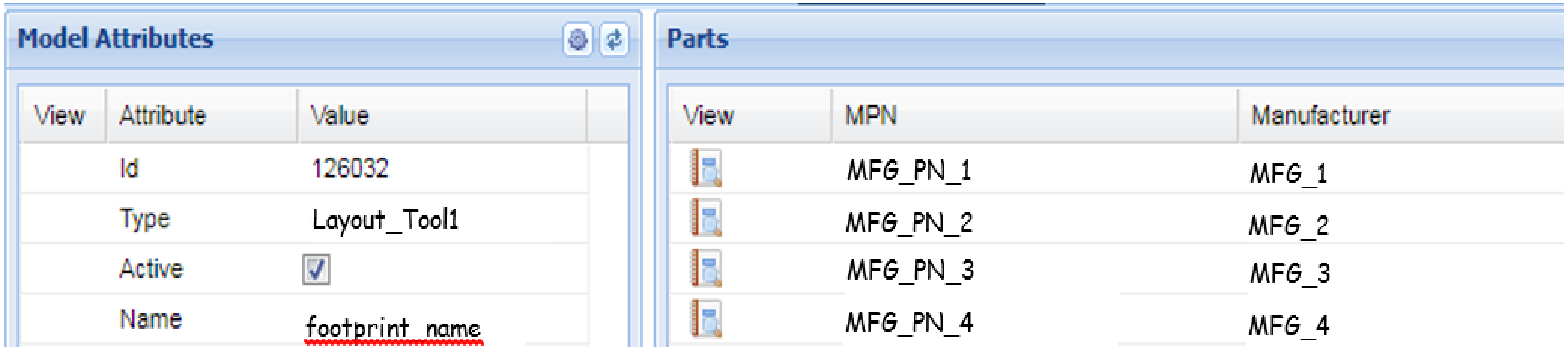
- EDA models are built per the request
- This entity is the result and leveraged for future use.
- The datasheet and all the resulting schematic symbols, and PCB land patterns are associated.



The history and model data is valuable when an issue arises and maintenance to the library is required.

Library Maintenance

- Occasionally updates to a released library part are necessary
- Ability to understand impact to all affected parts is critical
- This system allows librarians to quickly view all manufacture part numbers that will be affected by the change.



Model Attributes

View	Attribute	Value
	Id	126032
	Type	Layout_Tool1
	Active	☑
	Name	footprint_name

Parts

View	MPN	Manufacturer
	MFG_PN_1	MFG_1
	MFG_PN_2	MFG_2
	MFG_PN_3	MFG_3
	MFG_PN_4	MFG_4

The example here shows a PCB land pattern and a partial list of all the manufacture part numbers assigned to that land pattern

Conclusion

- Present new way of defining and managing a library that is independent of EDA tool vendor.
- Illustrate a system that was developed to ensure quality is built into the library and to automate and document the library creation process.

Thank you!

Questions

?