cadence™

Electrical Design, Mechanical Design Incremental Data Exchange in Allegro PCB Editor: Best Practices

Product Version 23.1
September 2023

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Contents

Electrical Design, Mechanical Design Incremental Data Exchange (IDX)

Introduction

The exchange of electrical and mechanical CAD data has had many import-export formats, such as, DXF, and IDF. Each format has maintained a set of standards accepted by the CAD industry. These formats have served well over the years, with one common underlying issue, the exchange of data is considered, “all or nothing”. If a board outline, constraint areas, and component placement is used initially, all the same data is continually exchanged bidirectionally, even if just one object is modified. This exchange format makes it difficult to manage design impact and track changes.

The EDMD schema, a new XML based data exchange format, was created to aid in the exchange of ECAD-MCAD data by introducing the concept of passing incremental changes. This implies that both the ECAD and MCAD tools begin at the same starting point, or baseline, and any change from the baseline line is considered an incremental modification of the data. The incremental data is then passed from one CAD tool to the other, not the entire CAD interface data set.

Additional capabilities in the EDMD schema — commonly referred to as Incremental Data Exchange (IDX) format — enhances the ability of design collaboration. The options provided in the EDMD schema, such as ability to add comments, and accept or reject modifications, improve communication between the two design disciplines. For example, after the design is baselined, the PCB Editor may wish to move a component whose location was defined in the baseline, the designer would export this change, and through a GUI, add a comment describing the reason for the modification. The Mechanical Designer would preview the incremental data, accept the change and import the data, or reject the change, and reply with a comment explaining the reason for rejection, along with a possible new location. The PCB Editor previews the new proposal, accepts, rejects, and so on.

Note: The EDMD schema is managed by ProSTEP iViP ECAD/MCAD Implementation Collaboration Project Group (http://www.prostep.org). Cadence Design Systems is a member of this project group and continues to review and recommend updates and modifications to this standard to address technology and design process requirements.

The IDX import/export is available in the SPB 16.5 release and beyond, in the XL and GXL suite of tools. By default, the SPB 16.5 release supports IDX version 2.0. IDX 1.2 is also

supported be enabling the `idx_set_default_version_1.2` parameter in the User Preferences Editor window.

To maintain a certain level of consistency, many of the practices used in creating IDX data are similar to the practices used for creating IDF data. The processes and objects used in tasks such as, creation of areas, holes, and component mapping, are the same as used in IDF. However, in preparing to use the EDMD Schema, the user must be aware that the commonly used IDF format and the new IDX format cannot be interchanged. If IDF is used initially, and IDX is then imported, the user is given a warning that all IDF properties associated to the design will be removed. If IDX is used initially, and IDF is imported, the user is given a warning that all associated IDX properties and baseline data will be removed. Table 1-1 on page 6 compares features of IDF 3.0 and IDX 2.0.

Table 1-1 IDF 3.0 vs IDX 2.0

Feature	*IDF 3.0*	*IDX 2.0*
Board Outline	✓	✓
Component Placement (XY Location, Rotation, Top/Bottom layer)	✓	✓
Component Height (max height only)	✓	✓
Component Properties (REFDES, Part number, symbol name)	✓	✓
Place Keep-in/Keep out (Keepout with height)	✓	✓
Route Keepin/Keepout	✓	✓
Via Keepin/Keepout	✓	✓
Rooms	✓	✓
Incremental changes	✗	✓

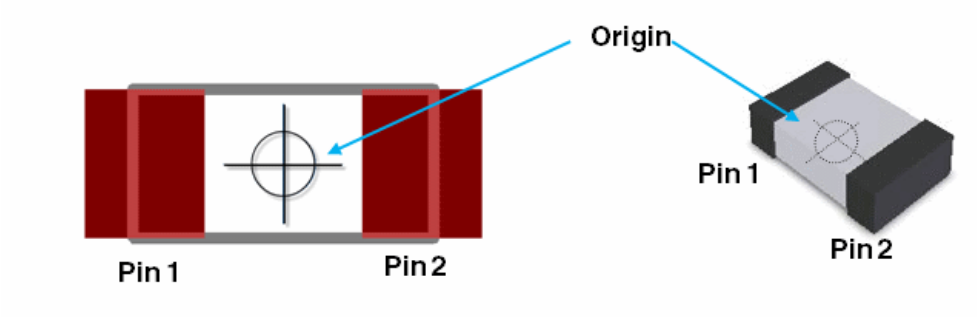
Table 1-1 IDF 3.0 vs IDX 2.0, *continued*

Preview with Accept Reject changes	✗	✓
Change with comments	✗	✓
Panelization	✓	✗

ECAD/MCAD Library Synchronization

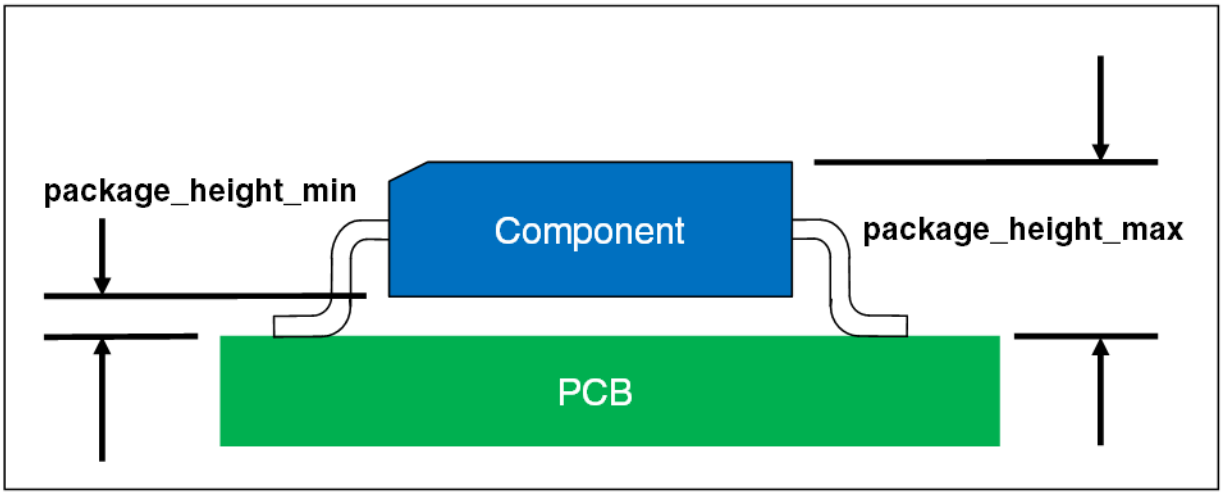
The key to successful implementation of ECAD and MCAD interfaces is the complete synchronization of the respective libraries. The ECAD and the MCAD libraries must be validated to ensure that the following are true:

- Origin – The ECAD and MCAD symbols of the same object must have the same symbol origin, body center, pin 1, and so on. A mismatch results in incorrect placement of component.
- Orientation – The orientation must be identical to ensure that component rotations are correctly matched.



- Mounting View – ECAD libraries are typically developed from the top layer or top plane view. A mismatch in the plane layers will result in incorrect layer placement (TOP vs. BOTTOM) and component mirror flags.
- Height – Maximum and minimum height values should match to provide for proper clearance. Changes in symbol height values must be managed at the library-level

not the design-level. The following figure shows the maximum and minimum package height, as defined in Allegro PCB Editor.



EDMD Schema Design Data Cycle

The EDMD Schema Design cycle consists of three phases:

- Creating initial baseline
- Incremental modifications
- Final baseline synchronization.

The exchange flow contains two files, the data file, which contains the object data, and the transaction files.

MCAD to Allegro with MCAD Baseline Initiation

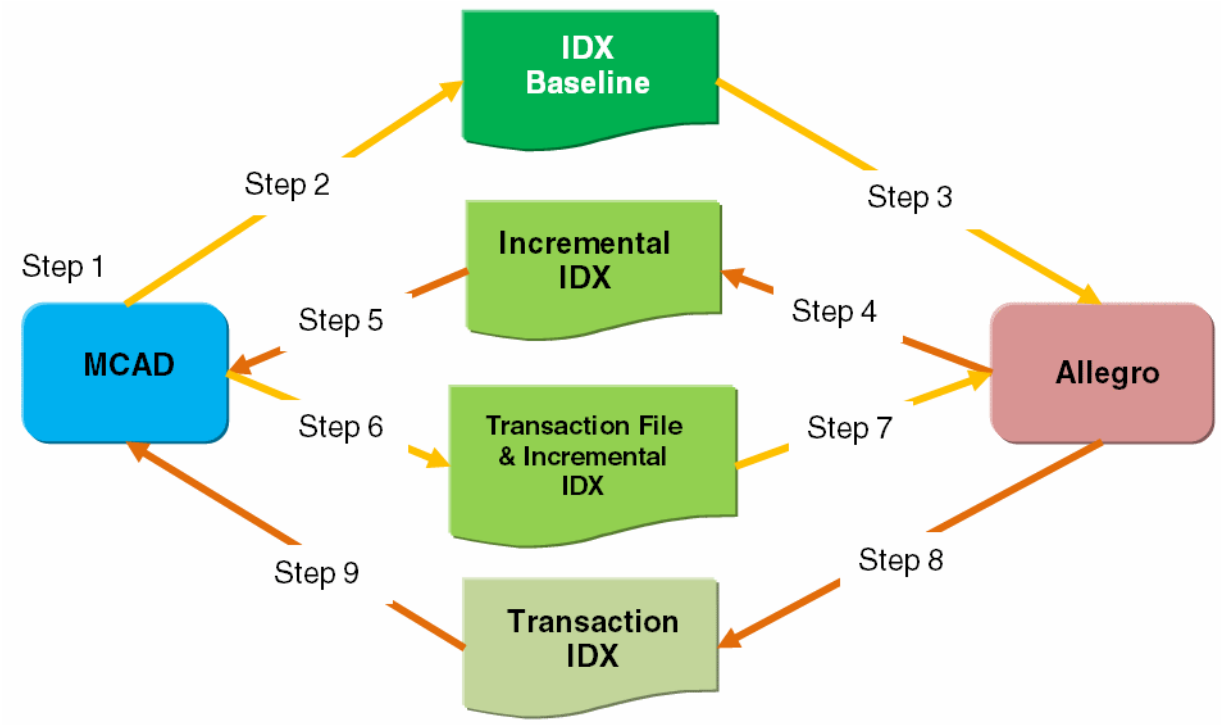
This section describes the creation of the baseline from the MCAD tool, which is then passed to Allegro PCB Editor for import. Key components (with assigned reference designators) are placed in the Allegro drawing and an incremental IDX file exported from Allegro to the MCAD tool. The key components exported to the IDX file are imported, and then placed in proper locations by the MCAD designer. A new incremental IDX file with the updated placement is passed back to Allegro for import.

Recommended Flow Steps MCAD initiation

This section outlines the preferred flow for establishing the first steps in defining a baseline with the MCAD tool as the initiator. This flow assumes that board outline, restricted areas, mounting holes and components locations are to be defined.

1. The MCAD designer creates the initial board with the following items:
    a. Board outline – the physical board outline with routed cut-outs.
    b. Mounting holes – Drill holes, such a mounting and tooling-holes not associated with any physical part, placed by the ECAD tool.
    c. Component boundary areas – One component keep-in area. Usually, this area is within the bounds of the board outline but may extend beyond the board outline. Various package keep out areas may be defined where component placement and component height restrictions may apply for top, bottom, and all other layers. Definitions such as, *external layers apply only*, and *any internal layer*, which are valid for an ECAD tool, may be ignored or converted to “ALL” in the MCAD tool.
    d. Conductor boundary Areas – Only one conductor keep-in area (ROUTE_KEEPIN) may be defined. Various conductor keep-out (ROUTE_KEEPOUT) areas where placement of conductor material is not allowed may be defined. Restrictions may apply for top, bottom, and all layers. Definitions such as, *external layers apply only*, and *any internal layer*, which are valid for an ECAD tool, may be ignored or converted to “ALL” in the MCAD tool.
    e. Via Keep out Area – Areas where via locations are restricted. Restrictions may apply for top, bottom and all layers. External layers apply only, any internal layer definition may be ignored or converted to “ALL” in the MCAD tool.
    f. Key parts or components that do not require reference designators may be placed at this time. A mapping file may be required in the MCAD tool, to map the MCAD part model name with the Allegro symbol model name. Any component or part that requires a reference designator should not be placed at this time.
2. The MCAD designer exports an IDX baseline. The newly generated baseline is passed to the Allegro PCB Editor.
3. The Allegro PCB Editor imports the IDX baseline.
4. The Allegro PCB Editor places parts that require a reference designator within the board outline. The Allegro designer exports the new placement data (selecting only those placed parts as incremental export data) as an incremental IDX file. The incremental IDX file is then sent to the MCAD designer.

5. The MCAD designer imports the incremental IDX file. After import, the parts are placed into their correct position.
6. The updated part placement is exported as incremental data (moved components only). This incremental file is sent to the Allegro Designer.
7. Allegro imports the new incremental file showing the adjusted placement.
8. The Allegro Designer sends the transaction IDX file to the MCAD designer which contains acceptance transaction data.
9. The MCAD designer reviews the transaction data and the initial baseline process is complete. Any alteration from this point is considered a change.



Allegro to MCAD with Allegro Baseline Initiation

This process involves placement of key components in PCB Editor (with assigned reference designators), followed by creation of baseline IDX file from Allegro PCB Editor. This file is then passed to MCAD for import. Changes made by the MCAD designer are passed as incremental IDX file, exported from MCAD tool to the Allegro PCB Editor.

Recommended Flow Steps Allegro Initiation

This section describes the preferred flow for establishing the first steps in defining a baseline with Allegro as the initiator. This flow also assumes that board outline, restricted areas, mounting holes and components locations are to be defined.

1. The PCB Editor creates the initial board in Allegro PCB Editor with following items:
    a. Board outline – the physical board outline with routed cut-outs.
    b. Mounting holes – Drill holes such a mounting and tooling-holes not associated with any physical part that will be placed by the ECAD tool.
    c. Component boundary areas – One component keep-in area (usually an area within the bounds of the board outline but may extend beyond the board outline). multiple package keep out areas where component placement and height restrictions may apply for top, bottom and all layers may be defined. External layers apply only, any internal layer definition may be ignored or converted to “ALL” in the MCAD tool.
    d. Conductor boundary Areas – Only one conductor keep in area (ROUTE_KEEPIN) may be defined. Various conductor keep out (ROUTE_KEEPOUT) areas where placement of conductor material is not allowed may be defined. Restrictions may apply for top, bottom and all layers. External layers apply only; any internal layer definition may be ignored or converted to “ALL” in the MCAD tool.
    e. Via Keep out Area – Areas where via locations are restricted. Restrictions may apply for top, bottom and all layers. External layers apply only; any internal layer definition may be ignored or converted to “ALL” in the MCAD tool.
    f. Key components may be placed at this time.
2. The Allegro designer exports an IDX baseline. The newly generated baseline is passed to the MCAD Designer.
3. The MCAD designer imports the IDX as a baseline.

Incremental Changes

This section follows a scenario where:

- Changes are made in the MCAD tool
- Change proposal is sent to Allegro PCB Editor
- PCB Editor accepts some changes, rejects others
- PCB Editor proposes new changes
- MCAD designer accepts new changes

Important

Initiating changes from Allegro PCB Editor is an identical process. The most important step to remember for the PCB Editor is to read in the transaction file when no other proposed changes are being passed from the MCAD tool. The import of this transaction file gets recorded into the Allegro baseline data, maintaining a current up-to-date status.

The following steps explains the transaction process between MACD tool and Allegro PCB Editor:

1. After a modification is made within the MCAD tool.
    a. An incremental IDX file is exported
    b. The incremental IDX file is sent to the Allegro PCB Editor.
2. The Allegro PCB Editor Imports the incremental IDX file and views the changes, and performs one of the following tasks.
    a. Accepts some proposed changes.
    b. Rejects some proposed changes, and adds comments to rejected changes.
    c. Closes IDX incremental import.
3. The Allegro PCB Editor makes modifications to the rejected items.
4. Exports new incremental IDX file.
    a. Adds comments to the items modified.
    b. Makes incremental available to the MCAD designer
5. MCAD designer Imports new incremental IDX file.

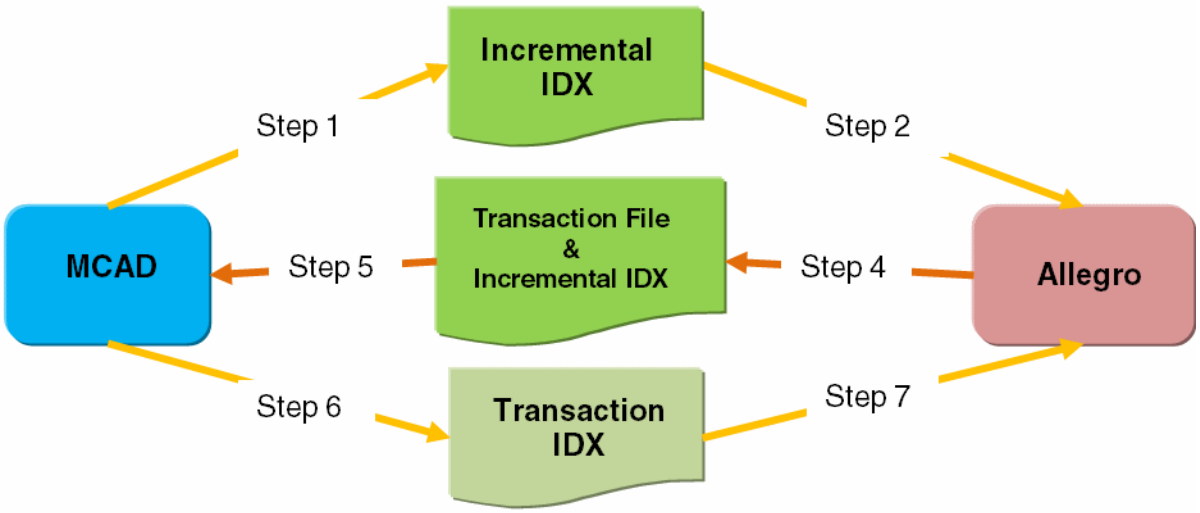
   a. Views proposed changes

   b. Accepts some and rejects some proposed changes.

   c. IDX tool exports processes transaction file (`design1_incremental_data1_processed.idx`).

6. The IDX transaction is sent to the PCB Editor.
7. The IDX transaction file is imported into the Allegro PCB Editor tool.
8. The transactions are complete.

This process may continue multiple times as the design progresses. The MCAD designer and the PCB Editor may create new proposals, accept or reject proposed changes, and continue the cycle until all modifications are complete.



Final Baseline - Re-baseline

When the design is complete, and there are no further planned modifications, run a final comparison between the Allegro PCB design file and the MCAD tool design. The latest transaction files should be imported and verified for accuracy between the two design systems. There must not be any outstanding changes pending in either tool.

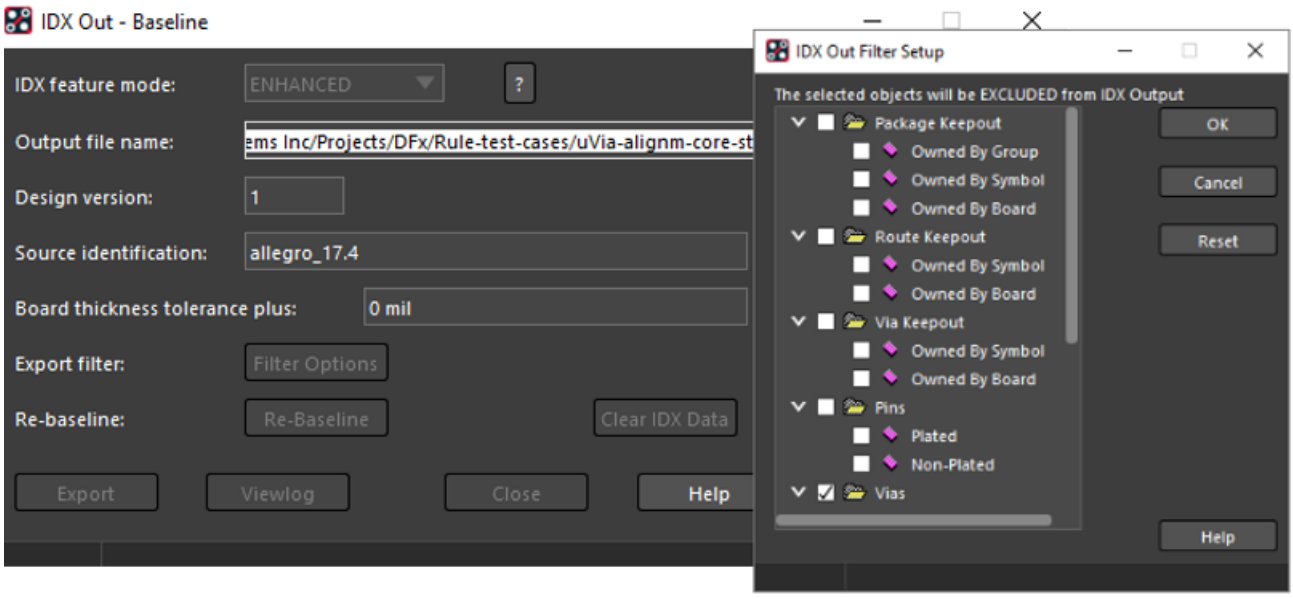
When running the re-baseline process, or importing as a baseline, all previous baseline data attached to the Allegro drawing and all previous transactions are removed. The new-baseline data for the design is attached to the Allegro drawing, and is now used as the base for comparison of any changes.

IDX Filter

When exporting IDX incremental data, a filter is available to identify what objects are not to be exported. The objects that are not to be exported should be set before the baseline. It is recommended that once the filter is set, no changes are made to the filter during the remainder of the design process or until a new baseline is established. Changing the export filter during the design cycle will result in unexpected results in the export.

As an example, a baseline is imported into Allegro PCB Editor with the default filter values. The Allegro PCB Editor moves a placed component defined by the baseline. When exporting, the designer changes the filter to exclude pins from the export. When the GUI for changed objects is displayed, each pin for the component is now displayed as deleted. This occurs because on import of the baseline, the pins for the components were identified as being added to the database and recorded in the baseline data. With the filter set to exclude pins, the pins do not appear in the export which is interpreted as deleted pins.

The default filter settings are *vias* and *unplaced components*. The Filter can be set by selecting the Filter Options button in the IDX Out form.



IDX Data

The Baseline File

When a baseline is established from the Allegro PCB Editor, or imported into Allegro PCB Editor, the data defined as baseline is attached to the Allegro drawing. This baseline is referenced each time an IDX import or export is run. When exporting a proposed change, the current data in the design drawing is compared to the baseline. If new object is added, or an existing object is changed when compared to the baseline, those objects are listed in the export GUI.

As objects are imported or selected for exported IDX specific properties are used to maintain a transaction history of that object. For detailed information on IDX properties, see IDX Properties.

When Allegro PCB Editor is used to initiate a baseline or propose a change to the MCAD tool, it is required that the transaction file created by the MCAD tool be imported when no new proposals are returned. It is not required to import the transaction file if counter-proposals are passed as incremental changes, as the current status of the preceding proposed changes are contained within the incremental file.

Incremental Files

Incremental files contain the changes to a design after comparison with the baseline. The items exported from Allegro PCB Editor are selected by the Allegro PCB Editor in the export GUI. The incremental file may contain new objects, changed objects, deleted objects, and previous transaction status when applicable. When Allegro PCB Editor exports incremental data, an increment value will be embedded into the exported file name.

Transaction Files

The Transaction file records all actions taken during the Allegro/MCAD data exchange process. As each incremental file is imported, the status or action taken on each item in the proposals is recorded into the transaction file as well. In a sense, a transaction history is being kept throughout the exchange process.

When Allegro initiates a baseline or proposes a change to the MCAD tool, it is required that the transaction file created by the MCAD tool be imported when no new proposals are returned. It is not required to import the transaction file if counter-proposals are passed as incremental changes as the current status of the preceding proposed changes are contained within the incremental file.

After each IDX import into the Allegro PCB Editor, the transaction file is updated with the status for any proposed change items. When no further changes proposals are to be exported, the transaction file must be sent to the MCAD tools to report the status of the most recent changes.

In all cases, it is not required that change proposals be accepted. If a proposal is rejected, the status is reported in the transaction file as rejected along with any comments added for the rejected items. The latest transaction history may be viewed through a web browser as the file has the `.html` file extension.

IDX Properties

There are specific properties used in the IDX process that may be viewable to the user:

- ***IDX_OBJID*** – This property is assigned with the value of the IDX object identifier. This value is created through the creation of the IDX file. For some objects, the value will indicate the origin tool and the type of object being imported/exported.
- ***IDX_OWNER*** – The value of this property can affect how the import process will function. When importing an IDX file and the imported object property value is set to MECHANICAL, Allegro will assign the FIXED property to that object upon import. This property may be changed; legal values are MECHANICAL or ELECTRICAL.

Object Mapping

The following table lists the objects that are exchanged through IDX.

Table 1-2 Object Type Definitions

Object	Description
Board Outline	Closed polygon defining board outline (cut-outs outline)
Keep-in Areas	Place keep-in, Route keep-in (ALL)
Keep-Out Areas	Place keep-out, route keep-out, via keep-out (with min/max height), (TOP/BOTTOM/ALL)
Vias	Typically filtered out
Pin	Passed as child of the component definition (Through hole only)
Mechanical Hole	Hole from MCAD mapped as mechanical hole symbol in Allegro PCB Editor.

Table 1-2 Object Type Definitions

Electrical Component	Passed to MCAD as an object containing part number, REFDES, package boundary with height, symbol name, IDX properties, and pins. Requires Symbol definition on import
Mechanical Component	Passed to MCAD as an object containing part number, REFDES (if available), package boundary with height, symbol name, and pins (if available). Requires Symbol definition on import.

Note: Component height (package_height_min/package_height_max) changes passed to Allegro are not applied to the design. The height values are considered to be a library synchronization issue and must be addressed at the library level.

Board Outline

The board geometry should be a contiguous or closed polygon (BOARD GEOMETRY/ OUTLINE). When exporting the board outline, the board thickness value defined in the cross section is exported. Other board geometry, such as cut-outs or non-circular drilled holes may be exported as outline geometry. Board cross-section and cavity definitions are not exported.

Keep-in Areas

The maximum extents for component placement and conductors are defined through Allegro PACKAGE KEEPIN/ALL, and ROUTE KEEPIN/ALL geometries. Only one of each may be defined.

Keep-Out Areas

Regions that define exclusion or restricted areas are defined by a combination of geometry definitions and properties. Component keep-out areas are defined by a geometry figures on Allegro drawing class PACKAGE KEEPOUT, and limited to subclasses TOP, BOTTOM, ALL. With no height property associated with these, object assumes no component may be placed within the defined boundary. To set height restrictions see Height Restriction Management.

Restriction areas for conductor are defined by geometries in the Allegro drawing class ROUTE KEEPOUT, and limited to subclasses TOP, BOTTOM, ALL. Any exported route keep-out area defined on any other subclass may be exported as subclass ALL.

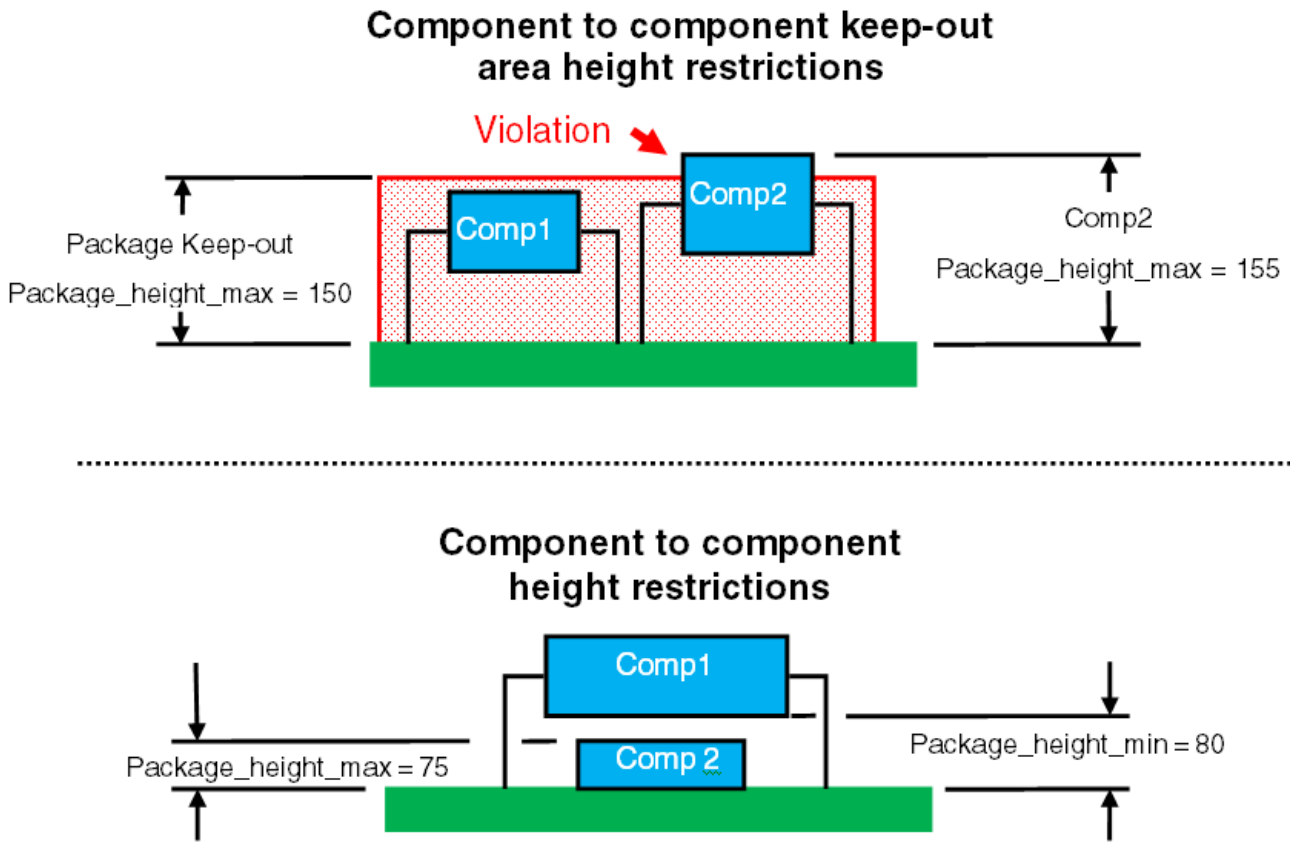
Restriction areas for vias are defined by geometries in the Allegro drawing class VIA KEEPOUT, and limited to subclasses TOP, BOTTOM, ALL. Any exported via keep-out area defined on any other subclass may be exported as subclass ALL.

Height Restriction Management

A maximum height restriction from the MCAD tool passed to Allegro is defined through a package keep-out area with an associated max height value (`Package_Height_Max`). The MCAD designer should pay special attention in how the restriction value is represented. Allegro PCB Editor uses the `Package_Height_Max` property attached to the package keep-out geometry to identify the maximum allowable height allowed in the defined area. If the Allegro symbol maximum height value is greater than the keep-out area maximum height value, a violation will be reported. The modeling of a height restriction defined in an MCAD tools may vary.

The methodology used by PCB Editor based on max/min height values permits the placement of symbols beneath other symbols, also known a shadowing. If an Allegro symbol is defined with a max height (`Package_Height_Max`) value less than the min height

(*Package_Height_Min*) value of another symbol, and pin to pin rules are not violated, no violation will be reported.



Vias

Vias are exported as through holes with X-Y location and drill size values. No pad or net structures are associated to the hole. Vias are filtered from export/import by default.

Hole Mapping

Plated and non-plated holes passed to Allegro PCB Editor through IDX that are not identified through the MCAD tools. Mapping file are generated in Allegro PCB Editor as a mechanical symbol. When these symbols are created, they include keep-out areas, pad stack, and are given a default name.

The symbol name is derived from the plating status, hole type, and drill diameter, using the following syntax.

<plating><type><drill_dia in mils>.bsm

Plating Designation		Hole Type Designation	
P	Plated	MTG	Mounting Hole
N	Not Plated	TOOL	Tooling Hole
		UNKNOWN	Not Specified

Examples:

- *PMTG125* – plated mounting hole 125 (mils) diameter drill
- *NTOOL130* – non-plated tooling hole 130 (mils) diameter drill

The pad stack used in the board symbol may be derived from a pad stack mapping file, or from a pad stack selection GUI. The mapping file — `idxHolePadstackMapping.txt` — is a text file that maps the derived board symbol name to a specific pad stack definition. Each entry in the file must contain two values. The first value is the derived board symbol name, and the second entry is the pad stack used in the symbol definition.

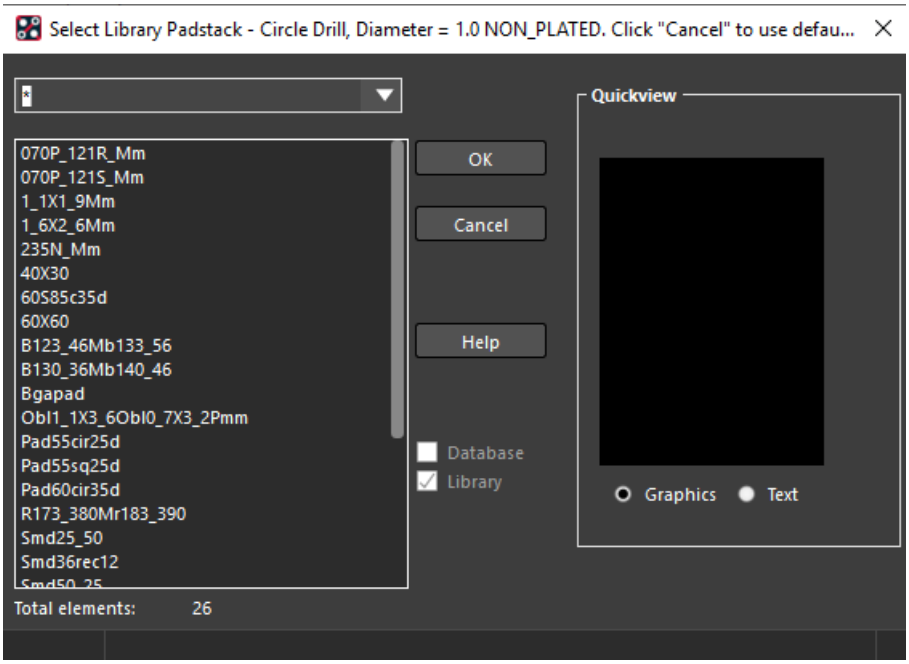
Examples:

```
NMTG125          c120n125
NMTG250          c200n250
PUNKNOWN137      c150p137
NTOOL130         c125p130
```

By default, the IDX import tool searches for the mapping file in the local directory, but you can use the `IDXHOLEPADSTACKPATH` environment variable to point to a specific directory containing the mapping file. The value of the environment variable is the directory path where the mapping file is contained.

```
IDXHOLEPADSTACKPATH = /Cadence/mysite/pcb/settings
```

If the IDX import cannot determine the pad stack, the pad stack selection GUI will open requesting the user to select a definition from the pad stack library to be used to create the hole symbol.



Allegro Symbol Exported Data

Allegro exports IDX symbol data using the symbol geometry and properties. The exported symbol data are:

- Symbol origin XY location
- Placement Layer (Top, Bottom)
- Rotation
- Symbol geometry extents
  - PLACE_BOUND_TOP/PLACE_BOUND_BOTTOM (combined if multiple)
  - Derived place bound outline (no PLACE_BOUND present)
- Package_Height_Max value: (in order of precedence)
  - Package property height value.
  - Place bound top/bottom maximum height value if multiple.

- Board drawing default height

- Package_Height_Min value: (in order of precedence)
  - Place bound top/bottom minimum height value if multiple.

Property and other symbol data exported include:

- Symbol name
- Symbol REFDES (if applicable)
- User part number (If Applicable)
- Placed/not placed status (See: IDX Filter)
- IDX_OWNER (when assigned)
- Through hole pin hole data

  Allegro exports IDX hole data using the pad stack definition and properties. The exported drill data include:

  - Drill diameter (Circle only)
  - Drill XY location
  - Plating status
  - Pad stack name

  Any drilled hole or keep out area (via, package, etch) defined within a symbol is also exported as a group of related objects. Each of these objects is identified as a child of the parent symbol. The receiving tool may view these objects as individual items.

Allegro IDX Settings

Allegro preference settings are available to allow the user more flexibility in the use of the IDX import/export interface. Use the User Preferences Editor dialog box, to view and modify the IDX settings.

The options available are listed below:

Preference	Description
idx_ignore_comp_height	Ignores the HEIGHT property and exports the symbol package_height_max value or, if not defined, the default symbol height value defined in the board drawing preferences.
idx_ignore_part_number	Ignores the PART_NUMBER property for export, and export the component device type definition
idx_ignore_processed_data	By default, IDX export appends new transactions to the processed transactions, thus maintaining an exchange history. Enabling this preference, only new transactions are exported with no history tracking.
idx_place_bounds_bottom	Package Geometry subclass to be used to calculate the bottom placement layer component outline. `idx_place_bounds_top` should also be used
idx_place_bounds_top	Package Geometry subclass to be used to calculate the top placement layer component outline. `idx_place_bounds_bottom` should also be used
idx_set_default_version_1.2	IDX 2.0 is the default version, enabling this preference allows the user to create IDX in older 1.2 version

A

PTC's Creo Parametrics Configuration Settings

Configuration Variables

Creo Parametrics environment variables	Description
ecad_area_default_import	Defines how imported ECAD areas are treated. Cosm_area treats imported ECAD areas as cosmetic features; 3d_volume imports ECAD areas with z-heights as a 3D enclosed quilt.
ecad_board_csys_def_name	Specifies the default coordinate system name add to an ECAD board being imported. If not set, the system prompts for a name.
ecad_comp_csys_def_name	Specifies the default coordinate system name add to an ECAD component being imported. If not set, the system prompts for a name
ecad_comp_layer_map	Allows layer mapping for the ECAD components import into assembly.
ecad_create_hint_map	Helps create an *ecad_hint.map* file. YES – automatically renames components, if necessary, each time the library of component outlines is imported. This does not create an *ecad_hint.add* file. It controls whether the *ecad_hint.map* file is created
ecad_default_comp_height	Set the default value and units for an ECAD component being imported. Units can be: inch, mil, thou, cm, mm, micron, dsu. If not set, the system uses the current components units.

ecad_default_comp_place_status	Sets the default component placement status for export(parameter setting overrides the default)
ecad_export_holes_as_cuts	Exports holes as “cut-outs” to ECAD system.
ecad_exp_both_two_areas	Supports the export of ECAD areas with different “above board”, “below board” conditions. Yes – enables the export of both sided keep-in and keep-out ECAD areas as two individual areas (TOP and BOTTOM)
ecad_import_holes_as_features	Imports sections specified as DRILL_HOLE as through-all holes. Boards created with Creo Parametrics drilled holes export with the default value of NPTH for the ECAD_HOLE_TYPE parameter. Create this feature parameter if value of PTH is required.
ecad_mapping_file	Specifies the *ecad_hint.map* file that will be used for ECAD operations
ecad_other_outl_csys_def_name	Specifies the default coordinate system name for the OTHER_OUTLINE section of an IDX ECAD component being imported. IF not set, user is prompted for a name.
template_ecadasm	Specifies the model used as the default ECAD assembly template
template_ecadpart	Specifies the model used as the default ECAD part template

Configuration Parameters

Parameter	Optional Value	Description
ECAD_OWNER	ECAD, MCAD, UNOWNED	Owner of ECAD area.
ECAD_HOLE_TYPE	VIA, MTG, PIN, TOOL. OTHER	The type of hole feature.
ECAD_PLATING_STYLE	PTH,NPTH	Plated or non-plated through hole.

ECAD_HOLE_OWNER	ECAD, MCAD,UNOWNED	Owner of the hole feature
BOARD_SIDE	TOP, BOTTOM, BOTH	Side associated with the place region, place keep-in or place keep-out ECAD areas.
ROUTING_LAYERS	TOP, BOTTOM, ALL	Shows which ECAD board routing layers in the electrical system are included in a Creo Parametrics routing area ECAD feature. No actual conductor features are supported through IDX.
COMPONENT_GROUP_NAME	ANY	Allows you to add or keep a name for the place region ECAD area feature
ECAD_ASSOCIATED_PART	BOARD, NOREFDES, reference designator	In the ECAD system, this is the part to which the Creo Parametrics hole feature exists.
ECAD_NAME	ANY	The ECAD system name of the component which becomes the <model_name>.prt file
ECAD_ALT_NAME	ANY	The ECAD system alternative name for a component
ECAD_REF_DES	Reference Designator	The ECAD reference Designator
ECAD_OWNER	ECAD, MCAD, UNOWNED	Shows the owner of the board or other object.
ECAD_PART_TYPE	Panel, Board, Electrical, Mechanical, other outline	Shows component type
ECAD_PART_NAME	any	Shows the other outline name
ECAD_PLACEMENT_STATUS	Fixed	

Drawing objects

Object	Allegro Class/Subclass	Description
BOARD_OUTLINE	BOARD GEOMETRY/OUTLINE	Board outline and cut outs
ROUTE_KEEPIN	ROUTE KEEPIN/ALL	Only one allowed. No voids
PLACE_KEEPIN	PACKAGE KEEPIN/ALL	Only one Allowed. No voids
ROUTE_KEEPOUT	ROUTE KEEPOUT/ALL ROUTE KEEPOUT/TOP ROUTE KEEPOUT/BOTTOM	Multiple allowed, no voids.
PLACE_KEEPOUT	PACKAGE KEEPOUT/ALL PACKAGE KEEPOUT/TOP PACKAGE KEEPOUT/BOTTOM	Multiple allowed, no voids. Height restrictions may be applied.
VIA_KEEPOUT	VIA KEEPOUT/ALL VIA KEEPOUT/TOP VIA KEEPOUT/BOTTOM	Multiple allowed, no voids
PLACE_REGION	BOARD GEOMETRY/TOP_ROOM BOARD GEOMETRY/BOTTOM_ROOM BOARD GEOMETRY/BOTH_ROOMS	Multiple allowed, no voids, identified by a given name.