



Improved Allegro to Pro/E Bidirectional Data Exchange

**CDN Live! Silicon Valley 2007
Session 7.2**

**Michael Wilson, Dell Inc.
Andreas Kulik, PTC**



Agenda

Introduction

Data Representation

Translation Process

IDF 3.0 Limitations

New Object Model

Conclusion

Future Translators

Brief History of IDF

IDF (Intermediate Data Format)

IDF 2.0, 1993

Specification consists of two files. A board outline file and a component placement file. Files contained basic information about the shape of the board, the position and size of plated and non-plated circular holes on the board, and the position and basic shape of the components on the board.

IDF 3.0, 1996

Added support for manufacturing panels with the same features as the board specification. Also added support for more keep-out and outline types, placement areas, and owner properties. The two formats are very similar in structure and content. This is the current specification that is used as a data exchange basis by Cadence and PTC, Pro/Engineer.

IDF 4.0, 1998 (Never adopted)

File format brought the ability to interchange much more detailed information. IDF 4.0 files contain information about board features such as land patterns, pads, vias and copper filled areas which aren't supported by IDF 2.0 and 3.0. The IDF 4.0 format also allows entities to be defined in much more detail than the previous versions of the specification.

Data Representation

IDF 3.0 Exchange Format

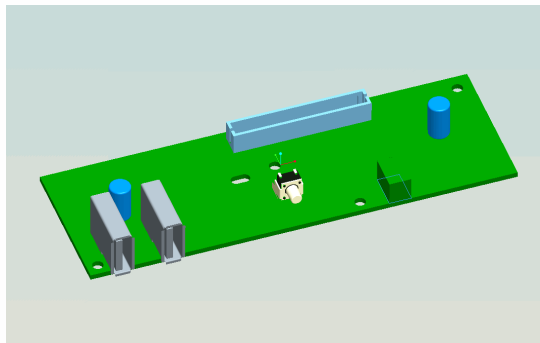
Board Outline - Including Cutouts

Holes

Keepout Areas

Keepin Areas

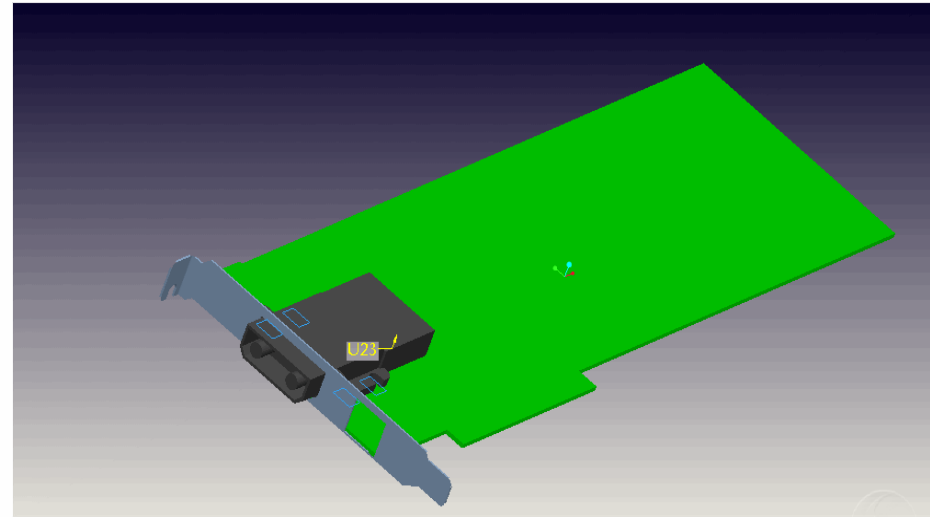
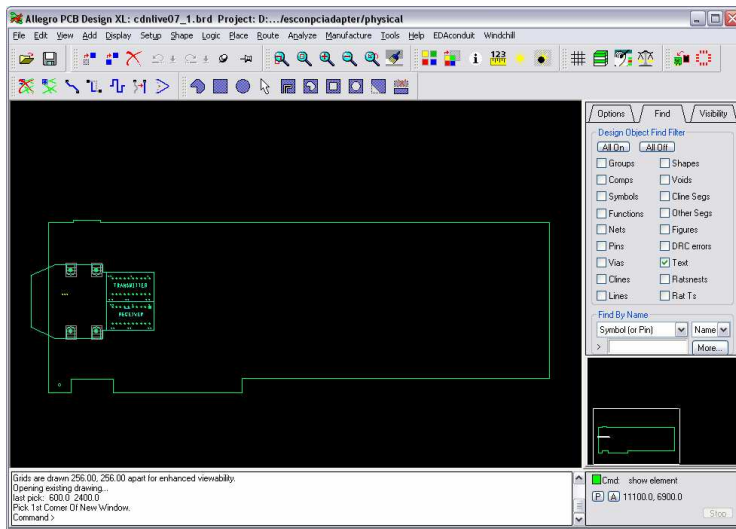
Component Placement



```
.HEADER
BOARD_FILE          3.0  "Allegro 15.51"  2007/08/17.15:46:05  1
uf887a00.brd  THOU
.END_HEADER
.BOARD_OUTLINE  ECAD
  62.00
0   4820.00   -130.00     0.000
0   4820.00   1533.00     0.000
0   -130.00   1533.00     0.000
0   -130.00   -130.00     0.000
0   4820.00   -130.00     0.000
1   1876.00    835.00     0.000
1   1876.00    715.00    180.000
1   1976.00    715.00     0.000
1   1976.00    835.00    180.000
1   1876.00    835.00     0.000
.END_BOARD_OUTLINE
.PLACE_KEEPOUT  ECAD
TOP      283.00
0   3332.00    270.00     0.000
0   3332.00   -130.00     0.000
0   3589.00   -130.00     0.000
0   3589.00    270.00     0.000
0   3332.00    270.00     0.000
.END_PLACE_KEEPOUT
.DRILLED_HOLES
  130.00    2351.00     850.00  NPTH  NOREFDES  DR130  UNOWNED
  125.00         0.00         0.00  NPTH  NOREFDES  DR125  UNOWNED
  125.00    4645.00    1355.00  NPTH  NOREFDES  DR125  UNOWNED
  125.00    3040.00         0.00  NPTH  NOREFDES  DR125  UNOWNED
.END_DRILLED_HOLES
.PLACEMENT
2RAD268D492H098P  JJ185  C15
  495.00    625.00         0.00    90.000  TOP  PLACED
2RAD268D492H098P  JJ185  C14
  4220.00    710.00         0.00    90.000  TOP  PLACED
CON_USB_4P_RTFF_SHLDT_H_SLT  W6307  USB_FNT_B
  205.00    236.00         0.00         0.000  TOP  PLACED
CON_USB_4P_RTFF_SHLDT_H_SLT  W6307  USB_FNT_A
  791.00    236.00         0.00         0.000  TOP  PLACED
CON_HDR_2X20_SHR_2MMK21  U6602  FNT_PNL
  3500.00    1350.00         0.00    180.000  TOP  PLACED
SW_2P_PB_RA2  U6603  PWR_SW
  2254.00    370.00         0.00         0.000  TOP  PLACED
.END_PLACEMENT
```

Translation Process

The IDF data exchange process can originate in either Pro/Engineer or Allegro.



Best practice:

MCAD creates Board Outline and data is imported into Allegro

ECAD places parts and exports data to MCAD.

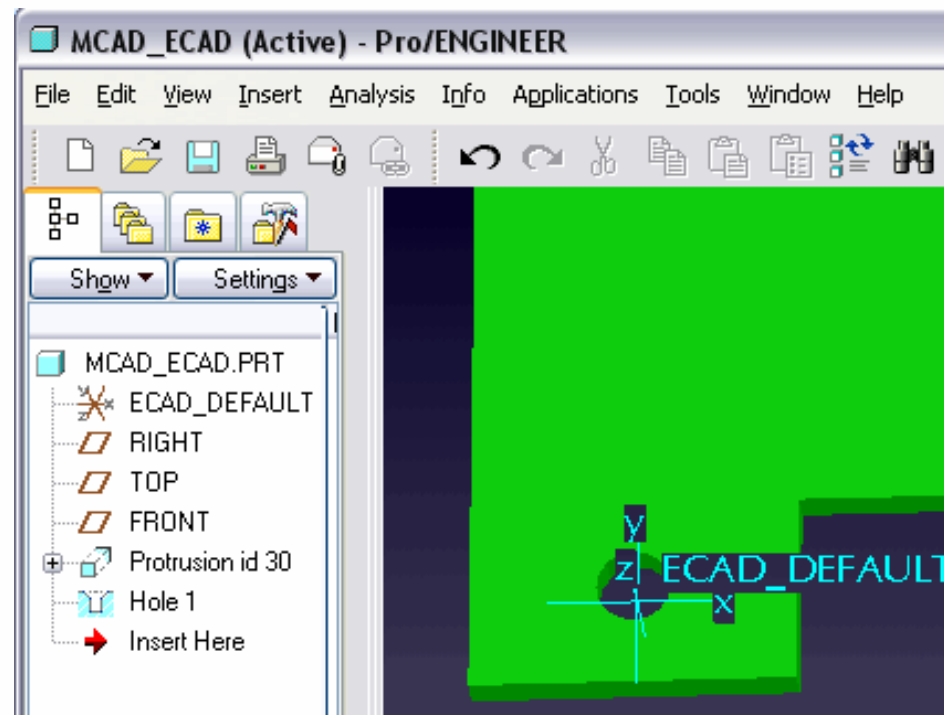
Problem:

Keep-out areas associated with the Allegro symbols will be disassociated from the symbols during import in Pro/Engineer.

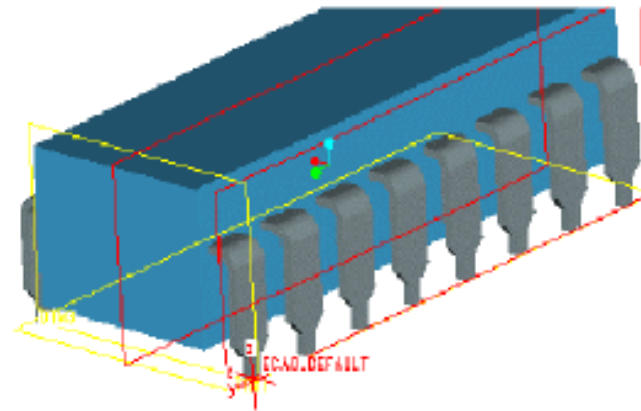
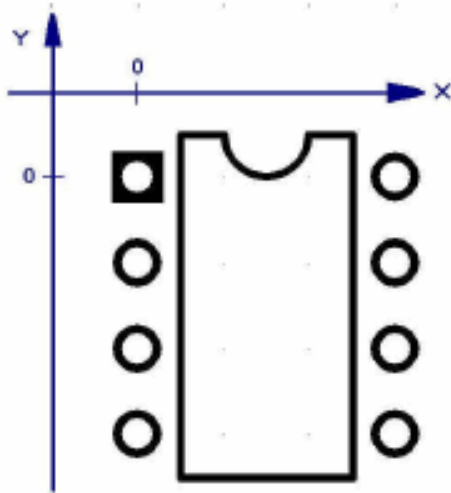
Translation Process

Coordinate systems are used to identify and to calculate location coordinates in the IDF file.

The coordinate system in Pro/Engineer for the board outline is defined by the ECAD_DEFAULT coordinate system. The import process from Allegro to Pro/Engineer for the board outline is dependent on the 0,0 location in Allegro as a reference point to the ECAD_DEFAULT coordinate system in Pro/Engineer.



Translation Process - Parts

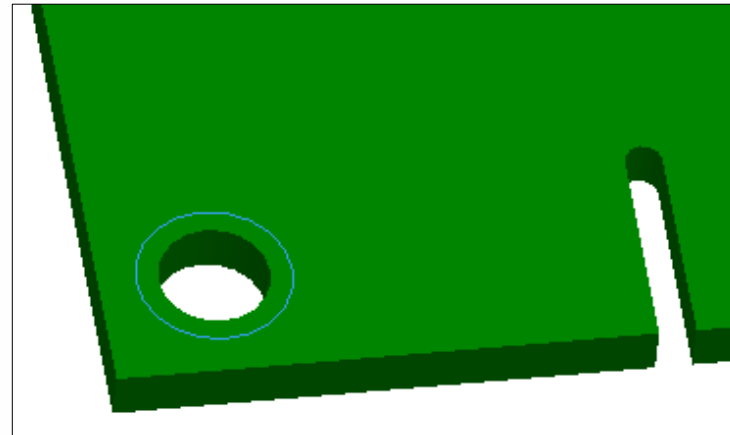
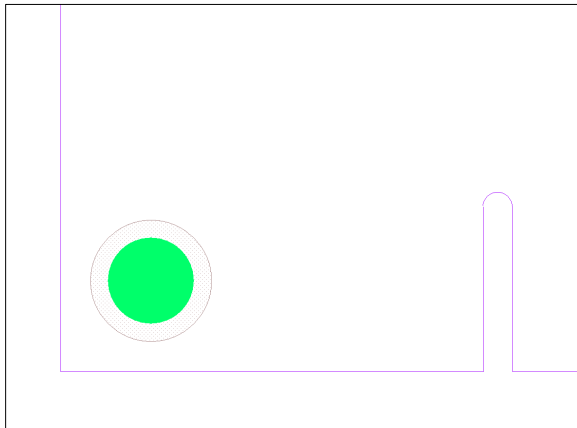


During the import into Pro/Engineer, placement coordinates for parts that are created in Pro/Engineer are calculated based on the ECAD_DEFAULT coordinate system on the Pro/Engineer part referenced to the ECAD_DEFAULT coordinate system on the PCB part. For extruded parts the symbol origin defined in Allegro is used as a reference to the ECAD_DEFAULT coordinate system on the PCB part.

The coordinate systems for the components in each application must match in order for the import/export exchange to be accurate.

Translation Process - Holes

Representation of holes in the IDF transfer can be confusing. Holes in Allegro are represented by symbols. In Pro/Engineer they are features or cuts in the PCB part. The best practice is to model holes in Allegro and use ECAD Parameters in Pro/Engineer to influence the definitions in the exported IDF file.



When exporting holes from Allegro PCB Editor, depending on the value of the IDF_OWNER property, the export results will differ. The Allegro PCB Editor will export ECAD owned or UNOWNED holes as an entry in the .DRILLED_HOLES section of the IDF file as well as an entry in the .PLACEMENT section.

If the IDF_OWNER property on a hole symbol in Allegro is set to MCAD, the hole will only appear in the .DRILLED_HOLE section of the IDF file exported from Allegro. The hole will be imported in Pro/Engineer as a 3D cut through the PCB.

Translation Process - Holes

An IDF 3.0 drilled hole record contains the following fields:

```
| DIA | X | Y | PLATING | ASSOC | TYPE | OWNER |
DRILLED_HOLES.
40.00000 2000.00000 2000.00000 NPTH BOARD PIN MCAD
40.00000 2165.00000 2000.00000 PTH BOARD VIA MCAD
40.00000 2165.00000 2330.00000 NPTH BOARD MTG MCAD
40.00000 2000.00000 2330.00000 PTH BOARD TOOL MCAD
125.00000 9750.00000 250.00000 NPTH BOARD 150C125 MCAD
60.00000 750.00000 250.00000 NPTH P1 OTHER MCAD
60.00000 850.00000 250.00000 NPTH P1 OTHER MCAD
.END_DRILLED_HOLES
```

The following naming formula is used to create hole symbols in Allegro during the IDF import:

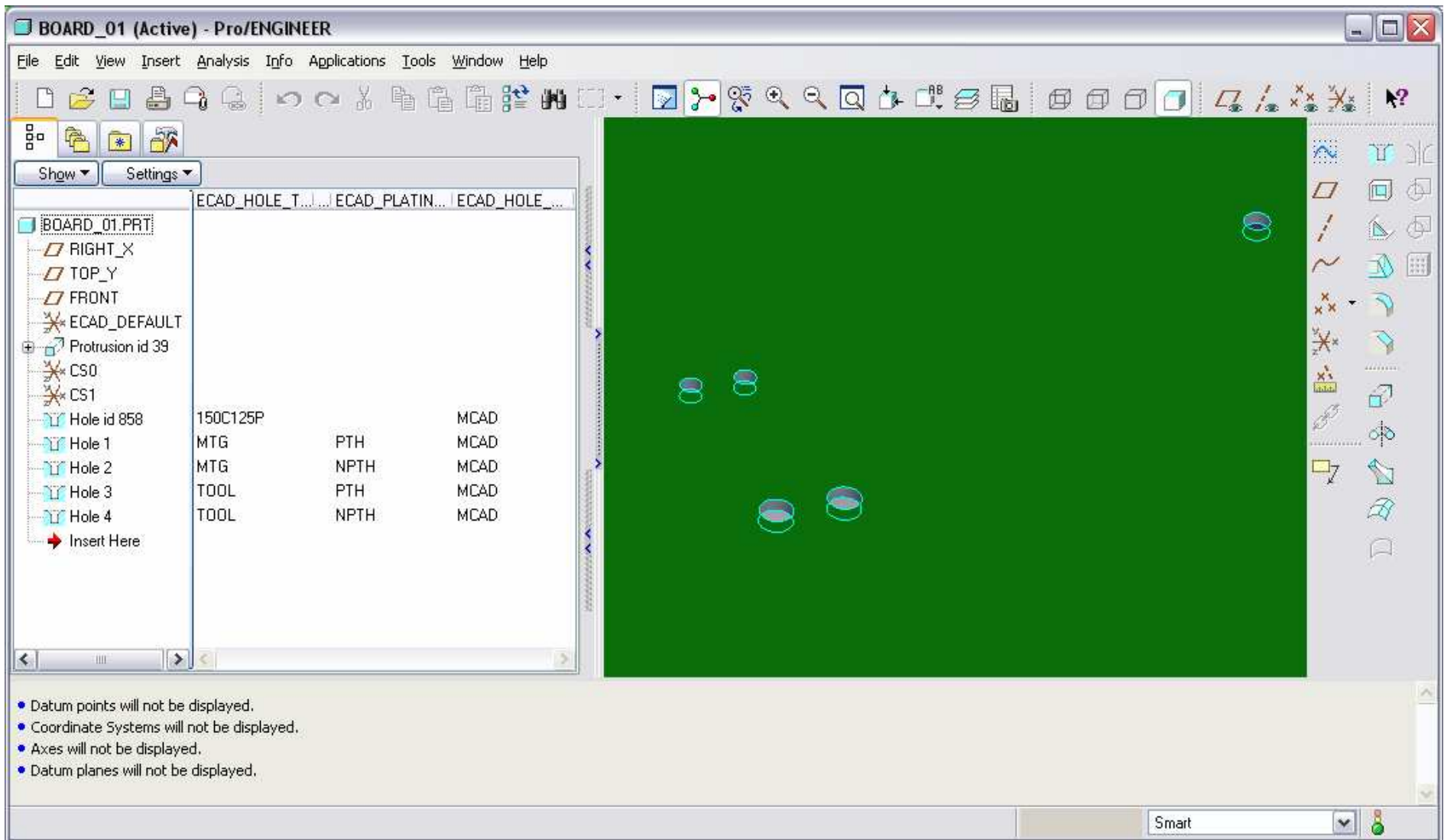
<(N)on-plated or (P)lated> + <TYPE> + <diameter in mil * 10>

The example above will result in the following symbols and pad stacks in the Allegro PCB database as a result of the IDF import:

NPIN400, PVIA400, NMTG400, PTOOL400, 150C125

Allegro symbols will be placed for holes with reference designators.

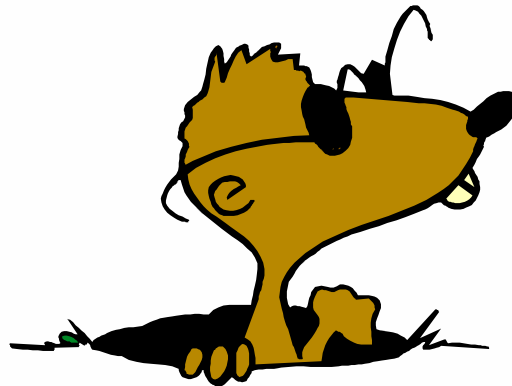
Translation Process - Holes



Translation Process - Holes

Three solutions for hole definition:

1. Modify the value of the hole TYPE in Pro/Engineer to use an existing Allegro symbol name.
2. Build an Allegro library with full-featured padstacks using the naming convention of the IDF 3.0 specification.
3. Manually replace all hole symbols with existing library symbols after importing the IDF file into Allegro.



Current IDF 3.0 Limitations

The current IDF 3.0 based interface is not robust enough to translate Cadence Allegro® 2D data to Pro/Engineer® 3D data without extensive knowledge of the interface and outside configuration files.

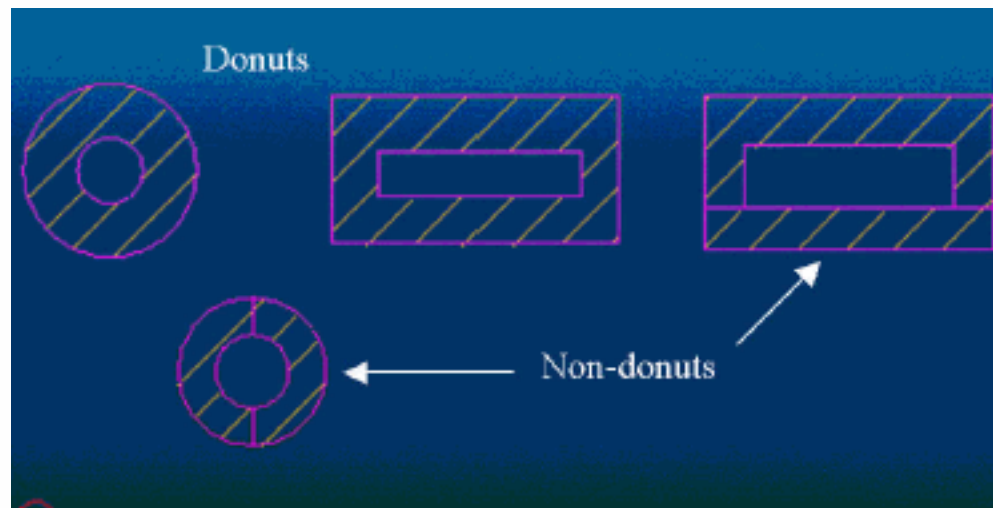
Limitations in the specification and limitations of the IDF implementation in Allegro and Pro/Engineer make the current exchange of data difficult without extensive knowledge of the IDF 3.0 specification and the specific implementation of the specification in the tools.

Current IDF 3.0 Limitations

Keep-out areas and PLACE_BOUND_TOP shapes can only be drawn as closed polygons.

Time consuming manual edits have to be made to create voids and cavities in keep-out areas.

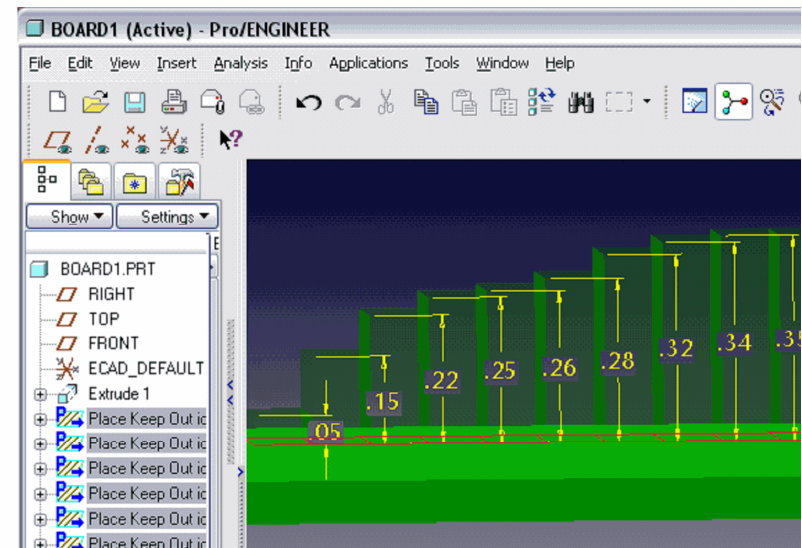
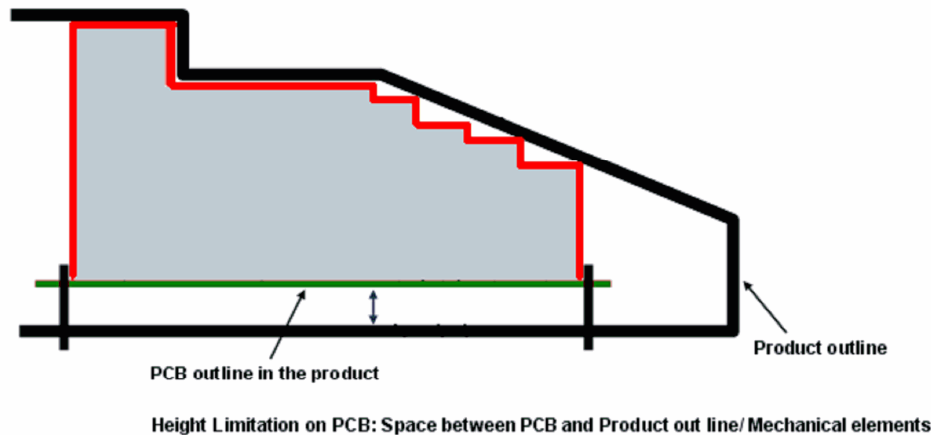
This limitation is due to the fact that Allegro PCB Editor does not allow voids in keep-out areas.



Current IDF 3.0 Limitations

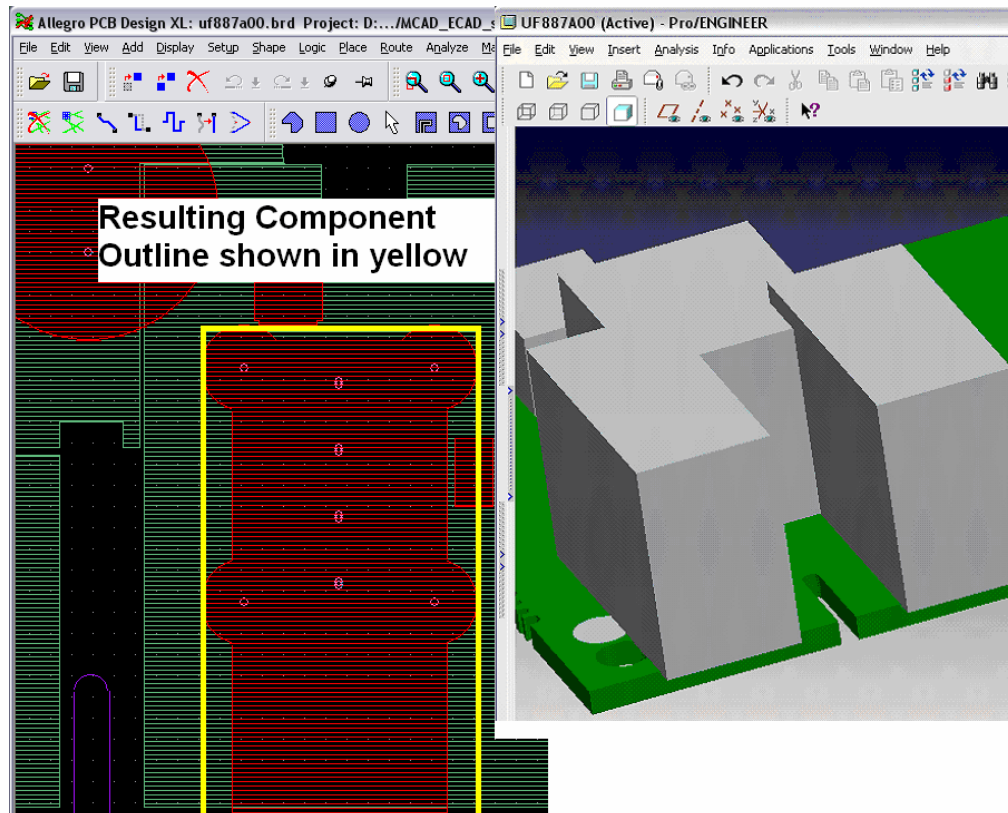
Individual keep-out areas are required that represent different heights. The ability to use stacked keep-out areas or define different heights in keep-out areas is necessary in most designs.

The image below illustrates how the housing of a component defines different height restrictions. These areas have to be created as either overlapping or stacked placement keep-out areas in Allegro and/or Pro/Engineer.



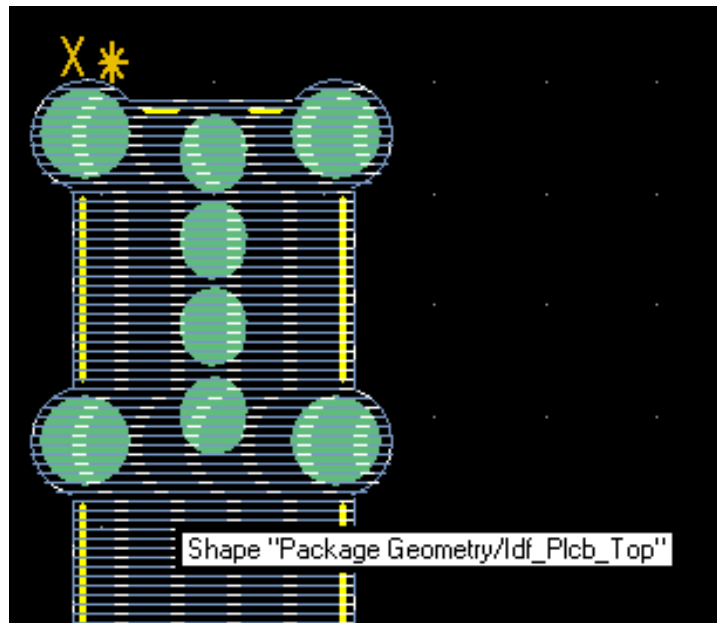
Current IDF 3.0 Limitations

- Currently only the PLACE_BOUND_TOP subclass can have a height that is transferred to Pro/Engineer and extruded as a 3D solid due to IDF 3.0 limitations.
- If a PLACE_BOUND_BOTTOM shape is used, the PLACE_BOUND_TOP and PLACE_BOUND_BOTTOM shapes are merged into one shape in the Allegro IDF export file. The merged shape is used as the basis for the extruded component in Pro/Engineer.



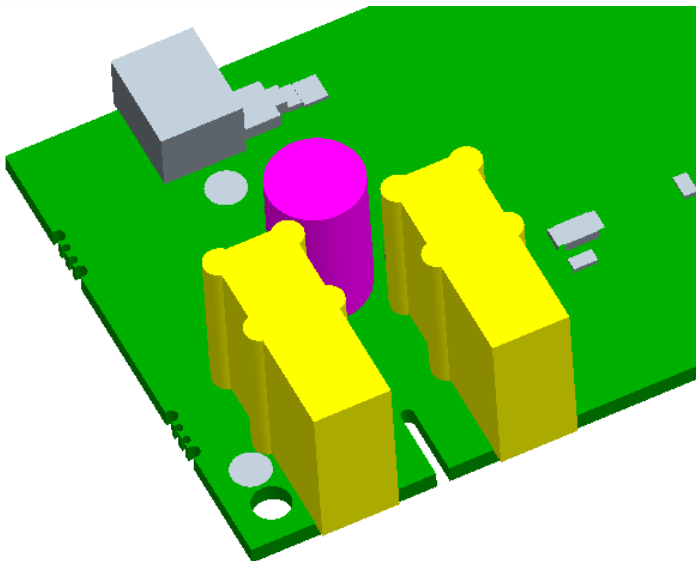
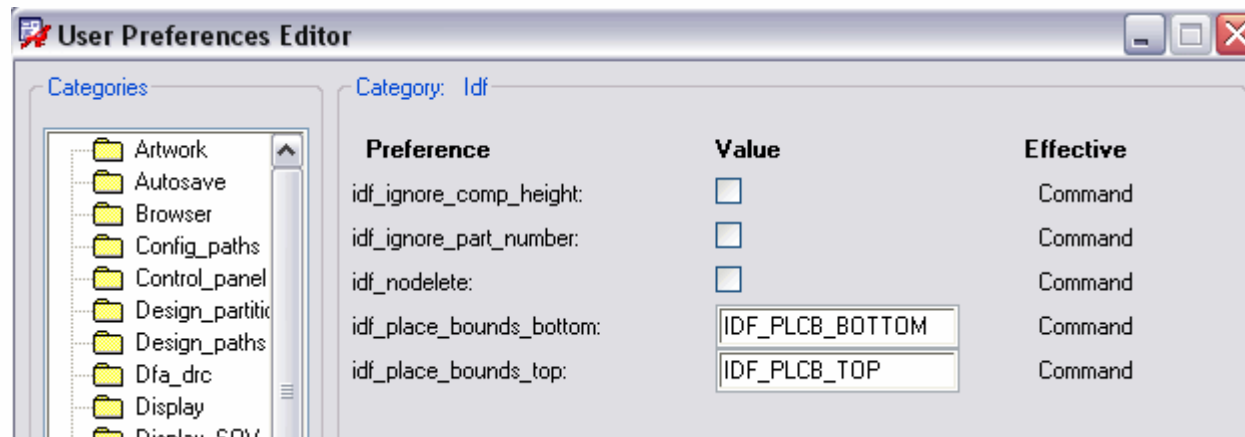
Current IDF 3.0 Limitations

- Use the PLACE_BOUND_TOP and PLACE_BOUND_BOTTOM subclasses to draw the boundaries as you normally would in Allegro.
- Allegro allows the user to specify which subclass to use for PLACE_BOUND_TOP and PLACE_BOUND_BOTTOM.
- Define a layer class for the top and bottom component boundaries. In this example we use:
 - IDF_PLCB_BOTTOM and IDF_PLCB_TOP.
- Draw the top component boundaries on the user defined layer subclass, IDF_PLCB_TOP, for the top IDF output. **Do not use** the IDF_PLCB_BOTTOM subclass.



Current IDF 3.0 Limitations

The subclass names for the IDF export can be set in the User Preferences in Allegro. The user defined bottom subclass `IDF_PLCB_BOTTOM` will have nothing on it. This will override anything that may be on the `PLACE_BOUND_BOTTOM` subclass.



The result in Pro/Engineer will be an extrusion represented by the combination of outline on the `IDF_PLCB_TOP` and the `PACKAGE_HEIGHT_MAX` assigned to the `PLACE_BOUND_TOP` area in Allegro.

Model every part in Pro/Engineer and Allegro

Pros:

A complete MCAD-ECAD library that ensures exact data exchange.

Cons:

The creation and maintenance of models is labor intensive and error prone.

Part representations must be synchronized either manually or by using a revision control or PDM system.

Current IDF 3.0 Limitations - Symbol and Part Modeling

Don't model any parts. Use only the part extrusion.

Pros:

No library maintenance symbol cross reference necessary.
All part extrusions come from PLACE_BOUND_TOP,
COMPONENT_HEIGHT_MAX in Allegro footprints.

Cons:

All part extrusions come from PLACE_BOUND_TOP,
COMPONENT_HEIGHT_MAX in Allegro footprints.
Model accuracy depends completely on the PLACE_BOUND_TOP
definition.

Current IDF 3.0 Limitations - Symbol and Part Modeling

Model critical parts in Pro/Engineer. Non-critical parts extruded.

Pros:

Only Critical parts are modeled in Pro/Engineer and Allegro for minimum overhead.

Non-critical part extrusions are defined from PLACE_BOUND_TOP subclass and the COMPONENT_HEIGHT_MAX attached to the PLACE_BOUND_TOP subclass.

Cons:

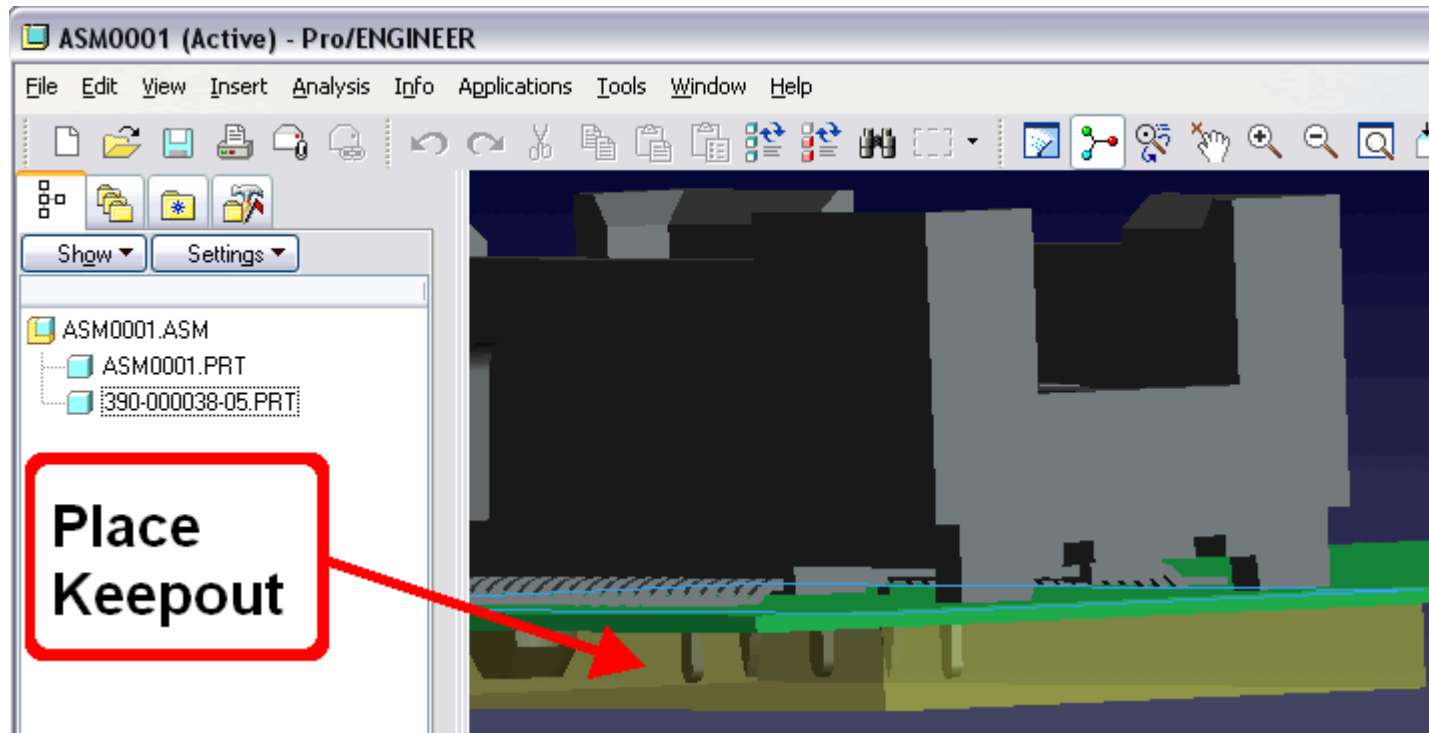
Extrusion height dependent on definitions from Allegro footprints or pre-defined height.

Non-critical part height extrusions come only from PLACE_BOUND_TOP subclass in Allegro footprints. Extrusions default to a defined setting when the data is created from Allegro if there is not a COMPONENT_HEIGHT_MAX defined in Allegro. Place boundaries in Allegro do not always represent the exact part outline.

This method seems to be the best balance of part representation and maintenance.

Current IDF 3.0 Limitations

Package keep-out areas assigned at the board level in Allegro can be used to represent bottom side height restricted areas.

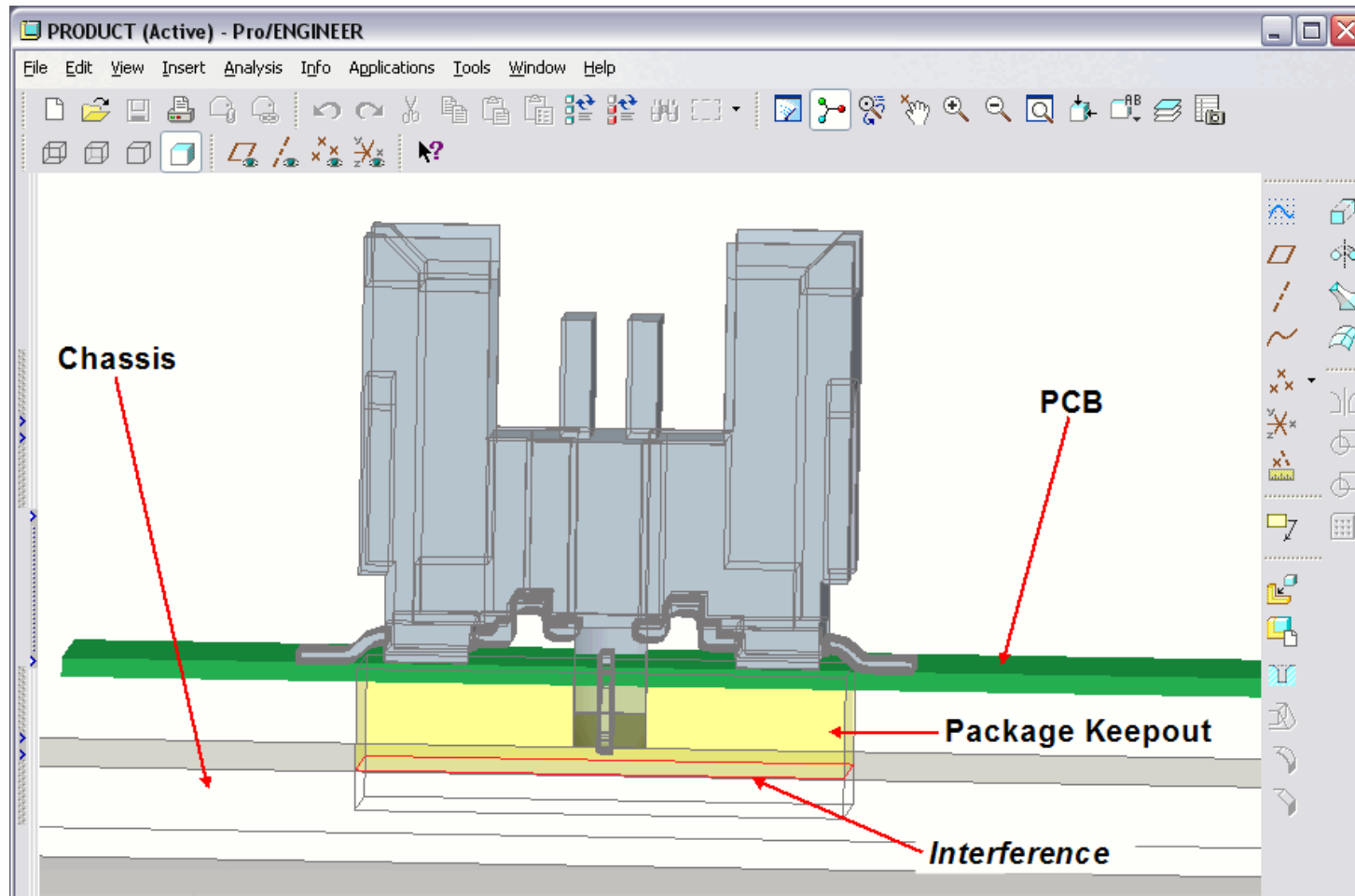


The keep-out area in the above example represents the length of the bottom side pins of the connector plus some tolerance.

The problem with this method is the non-associability of PLACE KEEPOUT areas and the component they belong to.

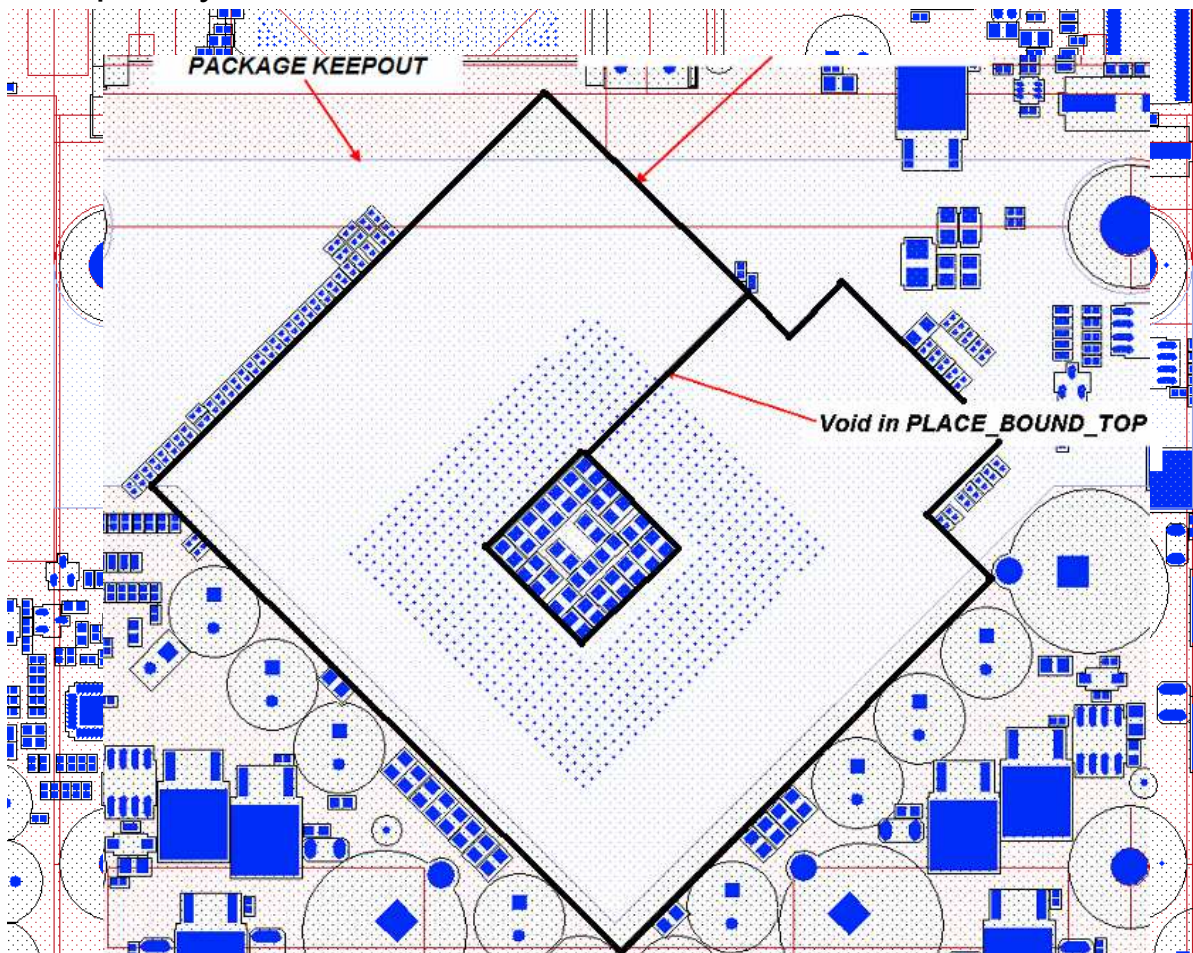
Current IDF 3.0 Limitations

This example shows how an inference analysis can reveal a board assembled too close to the housing of the product using a PACKAGE KEEPOUT area.



Current IDF 3.0 Limitations

- In the example below the PLACE_BOUND_TOP area had to be created with a small void, the black line across the PLACE_BOUND_TOP outline, to allow the Package KEEPOUT area Max Height to be less than the PLACE_BOUND_TOP Max Height. A Min and Max height that describes the component areas more completely is needed.



Current IDF 3.0 Limitations

- A method is needed to describe areas that may contain one part type or multiple part types in the PCB database.
- Assignment of rooms at the schematic level can be used to define parts that are allowed in the room.
- An easier approach would be to be able to assign the parts to a room in the PCB database. If the room definition could have a height attribute, there could be an interference check at the board level without having to assign a duplicate component keep-out area.
- Additionally there is a need to be able to constrain overlapping rooms or keep-out areas where the height and component type restrictions may apply.
- *Primary area is .3 max height. An area in the middle is .1 max height.* Symbols to be placed in the .1 area need to be limited by height and symbol type or reference designator.
- For example; only capacitors and resistors less than .1 can be placed inside the area.

Entity Representation

- The current IDF 3.0 implementation in Allegro PCB Editor uses reference designators to associate parts between the PCB and the mechanical model.

A better approach would be to represent each entity with an object reference assigned during the intermediate file creation.

- The IDF 4.0 data model is based on a hierarchy of assemblies, parts, and features. Assemblies are constructed from instances of parts and other assemblies. Parts are constructed from features.
- Features define the geometric shape and other physical characteristics of parts, and convey functional information as well. IDF 4.0 also provides figures, footprints, and sub-layouts to represent associations among instances and features that are useful to maintain during the design of the PCA.
- This example may not be the ultimate answer to the data model. It is used here as an example of an object oriented hierarchy that would work better than the current IDF 3.0 data model.

So ... What Now?



So... What Now?

What do we need in an Electrical/Mechanical Interface?

Easy to use

- Match entities in both mechanical and electrical packages as closely as possible using extrusions.
- Simplify the association of part representations between systems.

Better representation of entities

- Entities not tied to Reference Designators. Use object references.
- XML or similar structure so that entity definition can be extendible.
- More extensive model attributes and properties exposed in Pro/Engineer.
- Ability to add multiple attributes to restricted areas for easier specification of areas.
- Min/Max Height of components from PCB database.

Ability to use the data in multiple disciplines

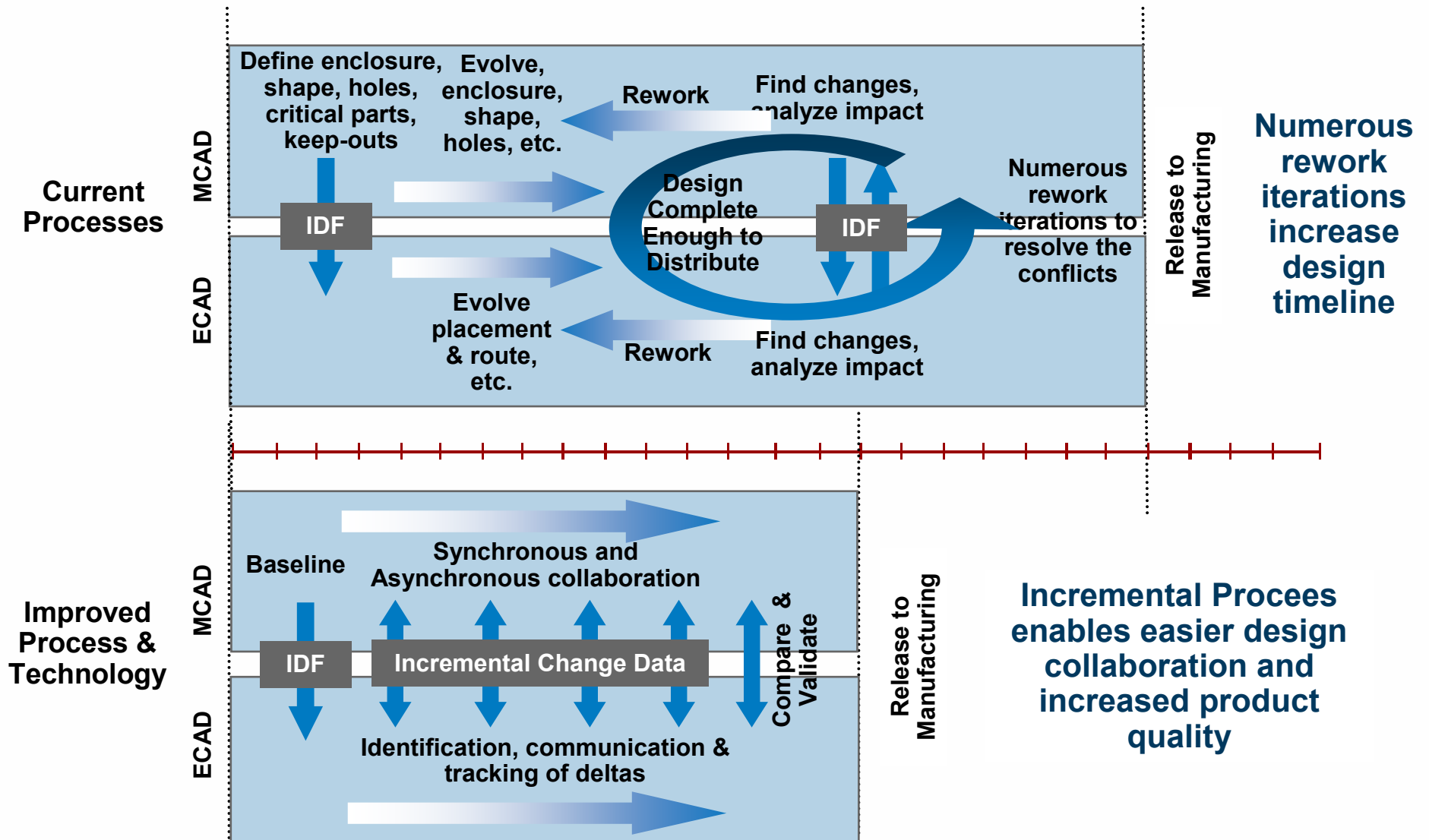
- Mechanical, Thermal, Electrical, Signal Integrity
- Based on what users need... Not a specification...

The Future



Improved Electro-Mechanical Co-Design Process

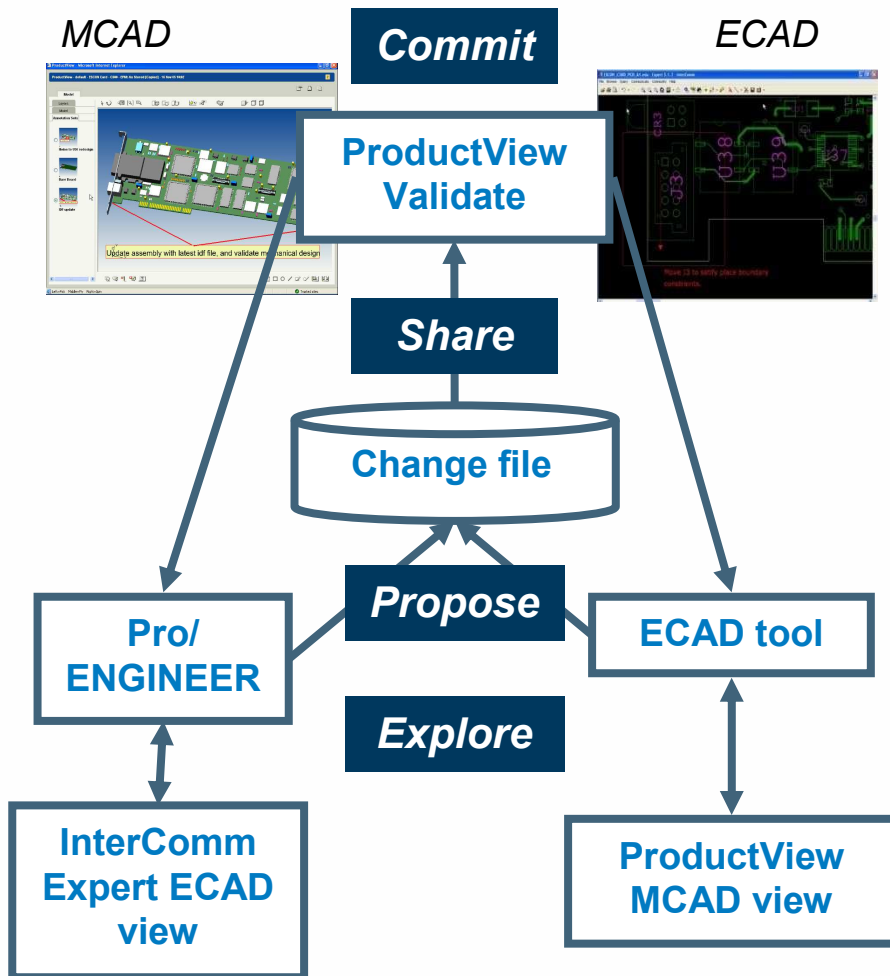
MCAD-ECAD Collaboration



How does it work?

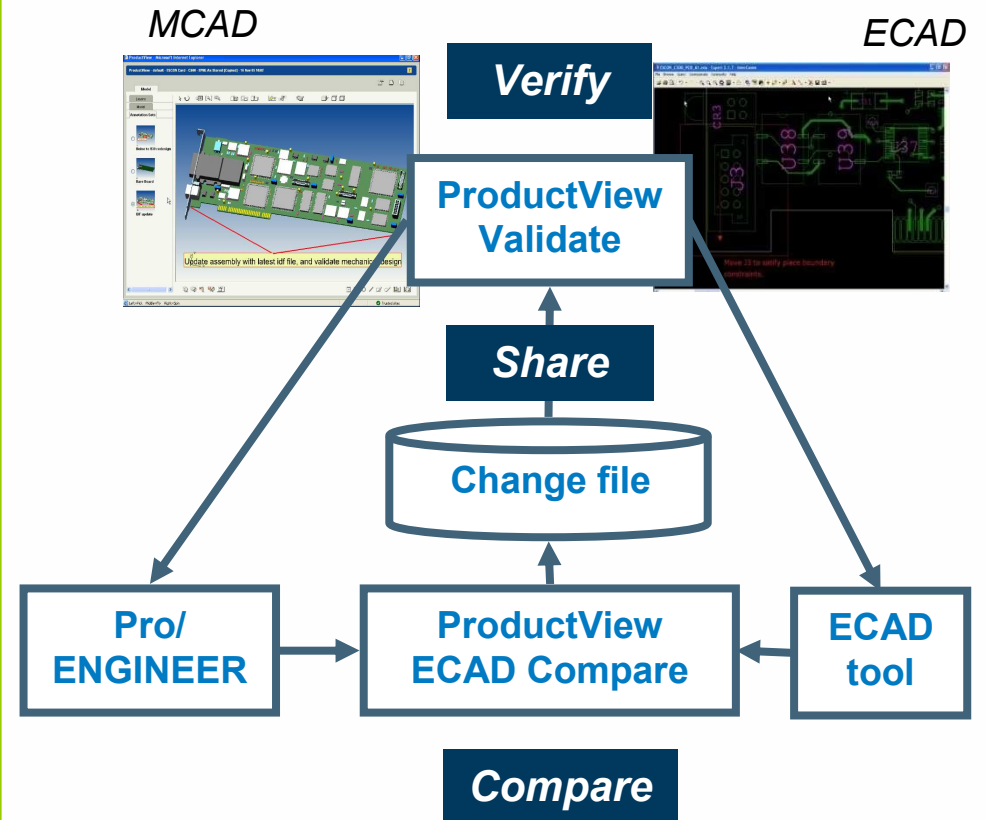
Propose Incremental Change

“How about this theoretical change?”



Verify Changes

“What actually did change?”



Key Technologies

IDX File

- XML structures based on the EDMD schema from ProSTEP (open industry standard)
- Incremental changes data only, no design meta data
- Can be used as an IDF replacement for initial board/component transfer from MCAD to ECAD
- Each change description contains:
 - A unique change ID
 - Reference to objects involved by unique IDs
 - All change data including ownership
 - Change status: proposed, accepted, rejected - with traceability (user, date, etc)
- Can be looked at as a tool-agnostic format for “intelligent” annotations/markups
- Will be understood by Windchill

ProductView Validate

- A normalized way to “see” a change without the context of design data
- Same look and feel for all tools
- Displays content of the IDX file in a tabular form
- Cross-Highlights changes to the authoring or visualization tools (sand-box)
- Allows to declare a new change status (propose > reject/accept)
- Will become part of Windchill

*Replacing full design swaps
with exchange of
incremental changes*

PV Validate – Managing Change

Validation Manager 3.0

File Edit Tools Help

Transactions

- All (15)
 - Added (2)
 - Component (1)
 - Net (1)
 - Removed (1)
 - Variant (6)
 - Component (6)
 - Variant - Geometric (4)
 - Top (6)
 - Sublayer Name (4)

Selected Transactions (Filtered 6 of 9)

Filter: Modified By Find Clear

Type	Object	State	Revise State	Trans ID
Variant Component	U100	Reject	Accept	TA1010
Variant Component	C25	Difference	Accept	TA1011
Unique Component	A12	Difference	Accept	TA1012
Added Component	C2	Difference		TA1100
Added Net	clock<12>	Accept		TA1502
Removed Component	R2002	Exclude		TA1412

Transaction Details

Attribute / Property	Current	Baseline
Design Name:	cell_phone_ra3	cell_phone_ra2
Object ID:	U100	U100
Reference Designator:	U100	U100
Part Name:	SOIC16	SOIC16
Device Type:	MAX232	MAX232
Package name:	SOIC16	SOIC16
Layer/Sheet:	TOP	TOP
X Location:	4225	4065
Y Location:	3600	3064
Rotation (Deg):	90	0
Pin Count:	16	16
COMP_NO_SWAP:	020-006-004	020-006-004

History	State	Date / Time
Original State	Difference	1/17/2007 1:11 AM
pawel	Reject	1/19/2007 2:04 PM

Notes

TID:U100 – 1/19/2007 – 2:04 PM – pawel
 U100 is rotated 90 degrees. This has caused it to move into an area that will cause mechanical interference.

Active Notes: U100
 U100 is rotated 90 degrees. I talked with the mechanical engineer and he was able to remove the mechanical interference. Therefore U100 can stay rotated as is.

Status:

We are not alone!

IDX file, a cooperative effort

- An open industry standard
- Based on XML schema
- Spans MCAD and ECAD definitions
- It does work, we are using it!

Who are we working with?

- ProSTEP is the sponsor
- PTC is an active partner
- Mentor Graphics is an active partner
- Cadence is investigating
- Several EOMs and software vendors are participating as well

PROSTEP

SIEMENS VDO

Mentor
Graphics

Continental

DELPHI

em engineering
methods AG

LIFE CYCLE
ENGINEERS

PID Tec.



CONNECT: IDEAS

CDNLive! 2007 Silicon Valley