

Best Practices: Working with IDF

**Product Version 23.1
October 2023**

© 2023 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

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

Allegro PCB Editor contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission. All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR52.227-7013 et seq. or its successor.

Contents

<u>Best Practices: Working with IDF</u>	1
<u>Introduction</u>	1
<u>The MCAD/ECAD Flow</u>	2
<u>Cadence IDF 3.0 Translator</u>	6
<u>ECAD Features in Pro/ENGINEER</u>	15
<u>Limitations and Tips</u>	22
<u>Conclusion</u>	26

A

<u>Pro/Engineer Configuration Settings</u>	29
<u>Configuration Variables</u>	29
<u>Configuration Parameters</u>	31
<u>Example</u>	33

Working with IDF

Best Practices: Working with IDF

Introduction

Today's printed circuit boards require tighter integration between electrical engineering and mechanical engineering. This is a result of increasing component density and mechanical complexity, as well as an increasing number of mechanical constraints such as PCB outline, holes, slots, cutouts, keepouts, interconnects, heatsinks, and so on.

In the past, most ECAD engineers created board outlines from paper drawings that were created by MCAD engineers. Now, this error-prone and time-consuming task can be simplified and automated using the Cadence IDF 3.0 translator.

For additional information, see [Intermediate Data Format](#) in the *Allegro PCB and Package User Guide: Transferring Logic*.

This paper describes the process of transferring data between Allegro PCB Editor and PTC Pro/ENGINEER™ using the Cadence IDF 3.0 translator. It also:

- Describes the basic flow of importing and exporting data, including using front-end tools such as Allegro Design Entry HDL.
- Explains how information is interpreted by Allegro PCB Editor.
- Shows how Allegro PCB Editor and Pro/ENGINEER detects changes.

Acknowledgements

This Best Practices paper is based on the paper, “*Electromechanical Design: Integrating Mechanical Design and PCB Layout with Cadence’s IDF 3.0-Based Translator*” written by Andreas Kulik and Andy Roemer of LTX Corporation of Westwood, Massachusetts. Cadence Design Systems is grateful to the authors and LTX Corporation for providing us with this information.

The Best Practices paper also draws information from the following sources:

- IDF 3.0 Specification – www.intermedius.com
- Pro/ENGINEER Online Documentation – www.ptc.com
- Cadence Research and Development; Marketing

The MCAD/ECAD Flow

The enhanced IDF interface, which transfers data between MCAD and ECAD, is powerful, much like the Packager-XL interface from Cadence, used to transfer information between Allegro Design Entry HDL and Allegro PCB Editor.

With the enhanced IDF interface, you can transfer entire mechanical assemblies. It allows bi-directional data transfer between mechanical and electrical engineering also known as electromechanical design.

To transfer data between ECAD and MCAD, a new flow is introduced. Key elements of this flow are reference designators and coordinate systems of components or parts. Table 1-1 describes the primary and more traditional MCAD/ECAD flow.

Table 1-1 MCAD/ECAD Flow

Step	MCAD	ECAD
1.	Creates a printed circuit board and its associated assembly including mounting holes and ECAD areas, places connectors and other mechanically critical objects at the assembly, and assigns reference designators to electrical components aligning with the electrical design.	
2.	Exports to an IDF 3.0 file.	
3.		Imports the schematic or netlist into Allegro PCB Editor.
4.		Imports the IDF file using the <code>idf in</code> command, which results in the placement of critical components with matching reference designators.
5.		Creates the stackup and places remaining components. May also start routing.
6.		Exports PCB Layout to an IDF 3.0 file, using the <code>idf out</code> command.

Working with IDF

Best Practices: Working with IDF

Table 1-1 MCAD/ECAD Flow

Step	MCAD	ECAD
7.	Appends the new IDF file to the original assembly, which includes new components and holes added by ECAD. Component changes are automatically moved or conflicts can be reported allowing the mechanical engineer to accept or reject changes.	

You can repeat the steps in Table 1-1, because the IDF interface in Allegro PCB Editor is driven by ownership and reference designators. Only the objects that have reference designators in both MCAD and ECAD are tracked with respect to changes in orientation and location.

To successfully transfer component placement information, the MCAD engineer must identify a coordinate system. In Pro/ENGINEER, two coordinate systems are available:

- Board Coordinate System
- Component/Part Coordinate System

The mechanical engineer uses the `config.pro` file in the Pro/ENGINEER environment to specify a name for both coordinate systems, for example `ECAD_BOARD_CSYS` or `ECAD_COMP_CSYS` (see Table A-1 for additional information). If you do not specify a coordinate system, Pro/ENGINEER uses the name `ECAD_DEFAULT`.

Table 1-2 shows a modified MCAD/ECAD flow that you might consider if you are having problems synchronizing part numbers and reference designators.

Table 1-2 Modified MCAD/ECAD Flow

Step	MCAD	ECAD
1.	Creates a printed circuit board, including mounting holes, mechanical parts and known areas such as route keepins, and place keepouts.	
2.	Exports the design to an IDF 3.0 file.	

Working with IDF

Best Practices: Working with IDF

Table 1-2 Modified MCAD/ECAD Flow

Step	MCAD	ECAD
3.		Imports the schematic or netlist into Allegro PCB Editor.
4.		Imports the IDF file.
5.		Places all the components using Quick Place; exports the file.
6.		
7.	Imports the IDF file into the board assembly. This results in all parts being aligned around the board edges. The mechanical engineer now can replace the part blocks with existing Pro/ENGINEER models or just place the critical components.	
8.		Analyzes the board and assists the mechanical engineer in placing mechanical critical components.
9.	Places critical components.	
10.	Exports the IDF file.	
11.		Imports the IDF file; places and routes, simulates and so on.
12.	Imports the IDF file with all components placed.	
13.		Exports the IDF file when placement is final.
14.	Performs final validation.	

Figures [1-1](#) and [1-2](#) show the example flow. Note that this deviation of the suggested flow may result in additional work for the mechanical engineer if the resulting blocks of components have to be replaced with Pro/ENGINEER models and parts. In this case, it is recommended that you use the `ecad_hint.map` file to ensure proper part reference and translation. The coordinate systems must match between the Allegro PCB Editor footprint and the part model in Pro/ENGINEER. See [Component Placement](#) on page 10 for additional information.

Working with IDF

Best Practices: Working with IDF

Figure 1-1 Allegro PCB Editor Quickplace Results in Pro/ENGINEER

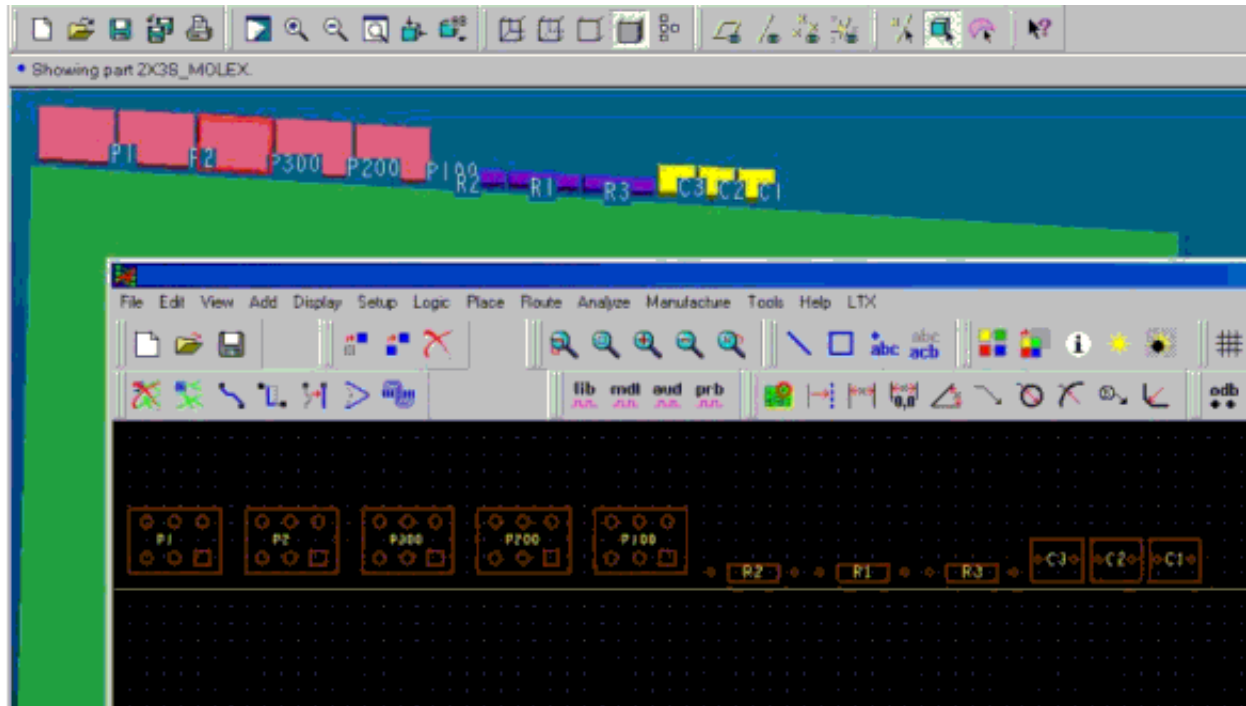
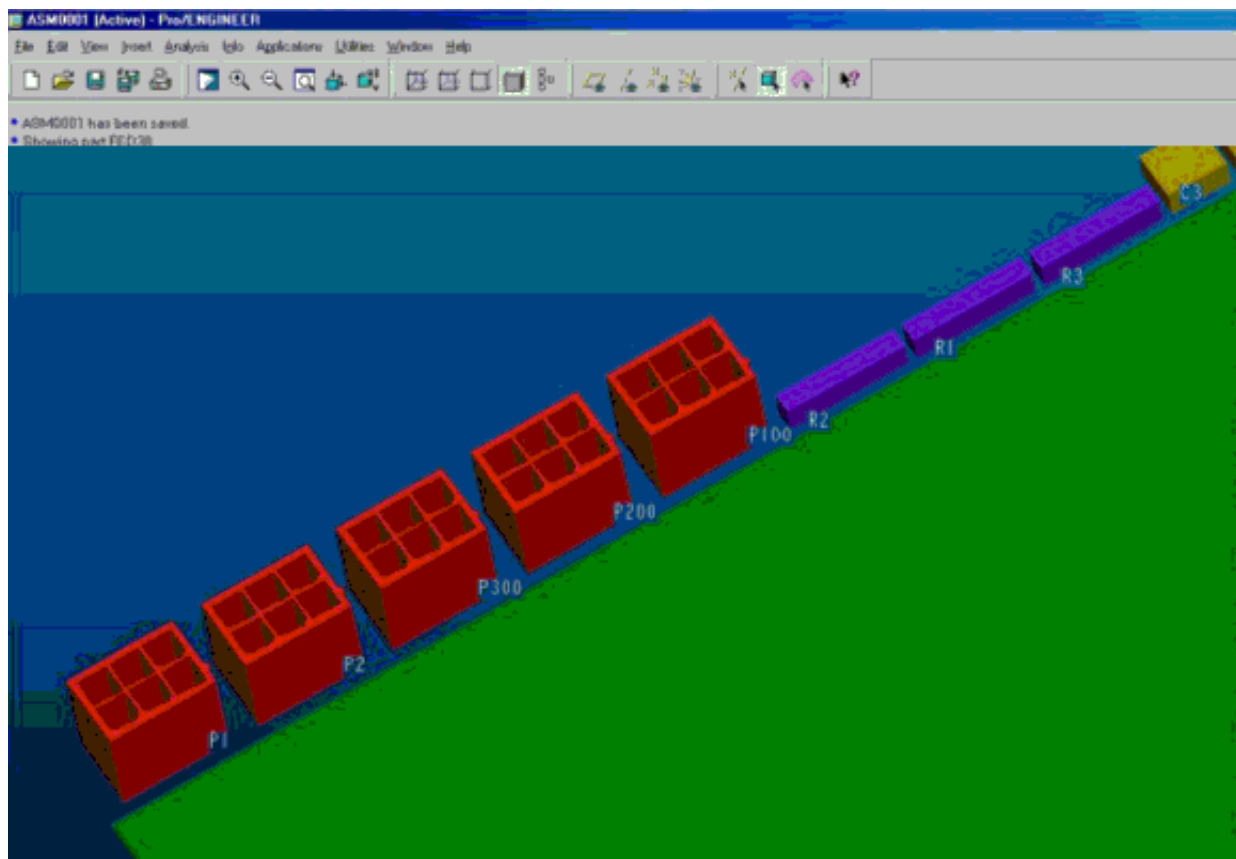


Figure 1-2 Allegro PCB Editor Quickplace Results in Pro/ENGINEER Using the ecad_hint.map File



Cadence IDF 3.0 Translator

Figures [1-3](#) and [1-4](#) show the new graphical interfaces of the Cadence IDF translator.

Allegro PCB Editor performs an enhanced syntax check while processing the specified IDF file. Also, these improvements have been made:

- The import of NOTES and OTHER_OUTLINE has been added.
- The import of components has been improved and introduces a new flow.
- The following new subclasses have been added under the BOARD_GEOMETRY class to accommodate notes and other outline features:
 - NOTES => subclass IDF_NOTES

- ❑ OTHER_OUTLINE => subclass OTHER_OUTLINE_TOP or OTHER_OUTLINE_BOT

Figure 1-3 IDF In Dialog Box

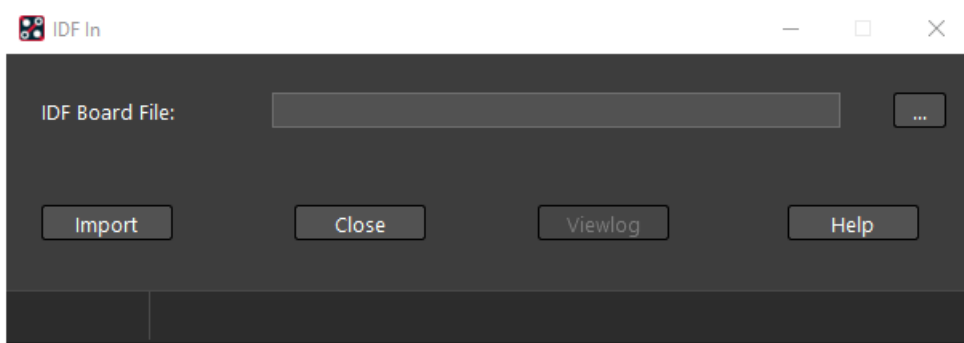
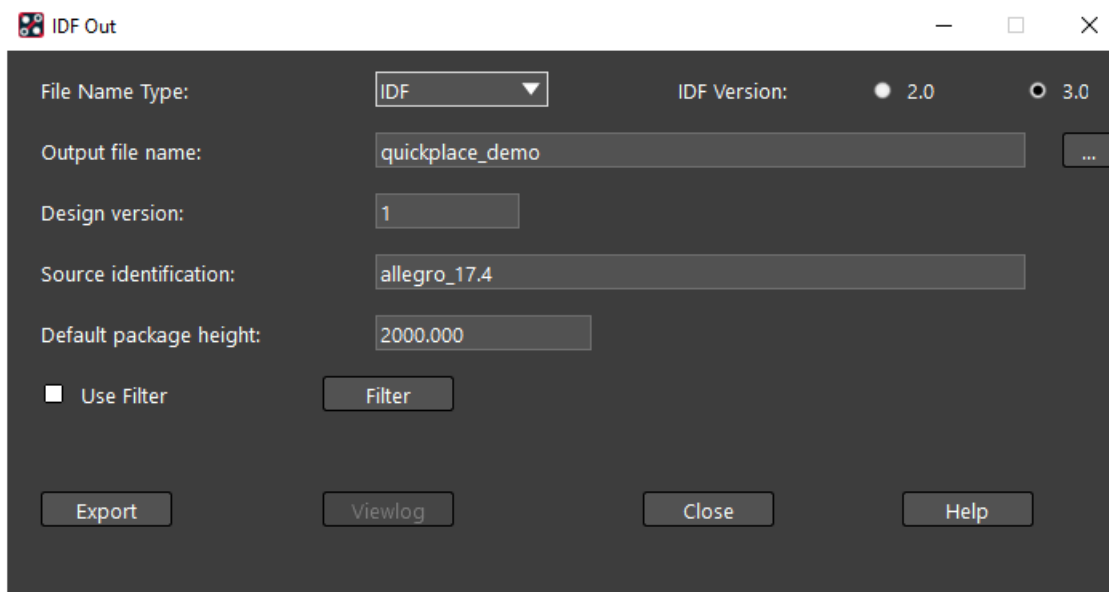


Figure 1-4 IDF Out Dialog Box



Rules for IDF Import to Allegro PCB Editor

Since the IDF 3.0 Specification introduces object ownership (MCAD, ECAD and UNOWNED), the IDF translator follows a revised rule set for importing IDF data into Allegro PCB Editor. Table [1-3](#) shows the action taken based on object ownerships.

Working with IDF

Best Practices: Working with IDF

Table 1-3 IDF-In Owner Action Map

If the IDF Object Owner Is...	And Allegro PCB Editor...	Then, Allegro PCB Editor...
MCAD	Has no existing object because this is a new design	Imports the object and sets IDF_OWNER=MCAD.
ECAD	Has no existing object because this is a new design	Imports the object and sets IDF_OWNER=ECAD.
UNOWNED	Has no existing object because this is a new design	Imports the object and sets IDF_OWNER=MCAD.
MCAD	Shows the object ownership as MCAD	Deletes the existing PCB Editor, APD object. It imports the object and sets IDF_OWNER=MCAD.
MCAD	Shows the object ownership as ECAD	Rejects the IDF object and reports the rejection in the log file.
MCAD	Shows the object without an IDF_OWNER property	Deletes the PCB Editor, APD object. It then imports the object and sets IDF_OWNER=MCAD.
ECAD	Shows the object ownership as MCAD	Deletes the existing PCB Editor, APD object. It imports the object and sets IDF_OWNER=ECAD.
ECAD	Shows the object ownership as ECAD	Rejects the IDF object and reports the rejection in the log file.
ECAD	Shows the object without an IDF_OWNER property	Imports the object and sets IDF_OWNER=ECAD.

Working with IDF

Best Practices: Working with IDF

Table 1-3 IDF-In Owner Action Map

If the IDF Object Owner Is...	And Allegro PCB Editor...	Then, Allegro PCB Editor...
UNOWNED	Shows the object ownership as MCAD	Deletes the existing PCB Editor, APD object. It then imports the object and sets IDF_OWNER=MCAD.
UNOWNED	Shows the object ownership as ECAD	Rejects the IDF object and reports the rejection in the log file.
UNOWNED	Shows the object without an IDF_OWNER property	Deletes the PCB Editor, APD object. It then imports the object and sets IDF_OWNER=MCAD.

Note:

1. The IDF import program never alters an Allegro PCB Editor object with IDF_OWNER=ECAD.
2. The rules in Table 1-3 can be directly applied to any IDF-related object that has a unique identifier, or unique instance, including:
 - ❑ BOARD_OUTLINE (only one allowed per design)
 - ❑ PANEL_OUTLINE (only one allowed per design; only recognized by IDF Version 3.0).
 - ❑ ROUTE_OUTLINE (only one allowed; layers always set to ALL)
 - ❑ PLACE_OUTLINE (only one allowed; layers always set to ALL; height property does not apply in Allegro PCB Editor)
 - ❑ PLACE_REGION (identified by name)
 - ❑ PLACEMENT (identified by reference designator)
3. If the object is an IDF PLACEMENT object, the effective action is to move the symbol, based on ownership relations shown in Table 1-3.
4. For IDF-related objects that do not have a unique identifier or unique instance, additional rules are applied. Such objects are VIA_KEEPOUT, ROUTE_KEEPOUT and PLACE_KEEPOUT.
 - ❑ Keepout areas without an IDF_OWNER attribute are not deleted.

Working with IDF

Best Practices: Working with IDF

- ❑ IDF keepouts that exactly match Allegro PCB Editor keepouts are ignored.
- ❑ IDF keepouts that do not match existing Allegro PCB Editor keepouts are imported as is. This may create overlapping keepouts.

5. Because Allegro allows the user to add geometry and text to BOARD GEOMETRY/OUTLINE, when an IDF file is imported, all items on BOARD GEOMETRY/OUTLINE tagged with IDF_OWNER = MCAD will be deleted. Any other items remain. This occurs regardless of the ownership of the incoming board outline in the IDF file. If any items exist on BOARD GEOMETRY/OUTLINE after the MCAD items are deleted, the following message appears in the log file:

```
WARNING(SPMHXL-48): There is existing geometry on Board Geometry / Outline.
This geometry may conflict with the IDF data.
Review the board outline in Allegro after idf_in is complete.
This conservative approach is necessary because of the relaxed rules about
defining a board outline in Allegro. User interaction is needed to correctly
resolve the conflict.
```

Component Placement

The MCAD/ECAD flow indicates that you import a netlist into the Allegro PCB Editor database prior to importing an IDF file. The database contains components with reference designators. As a result, you can take advantage of the reference designator field in the PLACEMENT section of the IDF file. For example, an Allegro PCB Editor database contains a connector with the reference designator P100 referencing the `conn100p_2r_50mm` symbol in the netlist. The component itself is not placed.

The statements in the PLACEMENT section of an IDF file (shown below) and the rules the enhanced IDF translator applies place the connector P100 at the location specified, using the footprint referenced in the database.

```
.PLACEMENT
500-9999-00  CONN100    P100
3300.00000  3500.00000  0.00000  270.00000  TOP    MCAD
600-0000-00  CONN50      P200
3300.00000  1500.00000  0.00000  270.00000  TOP    MCAD
.END_PLACEMENT
```

P200, however, is placed using the string "500-9999-00" as the name of the footprint. You can control the package name field by changing the value of the ECAD_NAME parameter on the part instance in Pro/ENGINEER. ECAD_REF_DES identifies the reference designators, P100 and P200, in the example above. Coordinate systems (Figures 1-5 and 1-6) play a key role in making this flow successful.

Figure 1-5 Coordinate Systems in Allegro PCB Editor

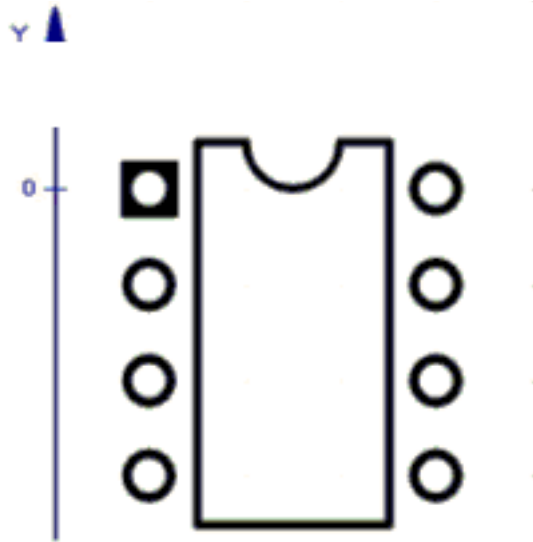
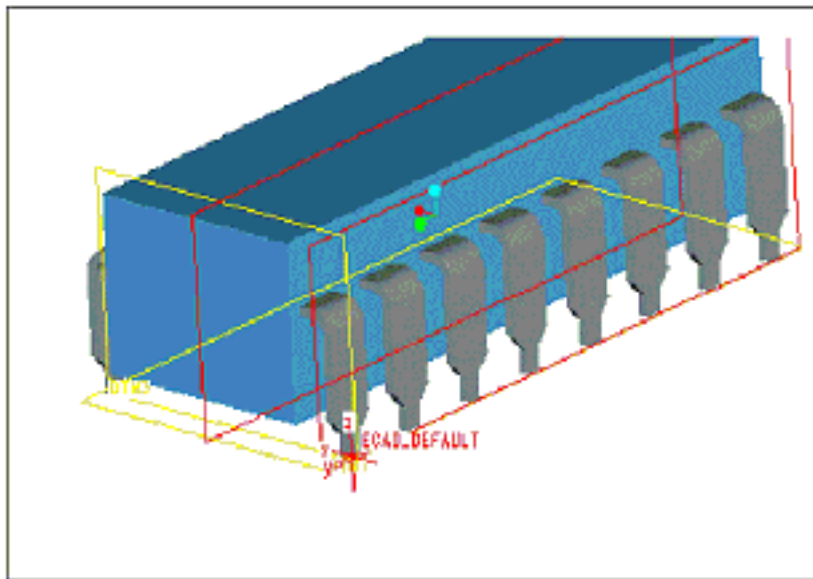


Figure 1-6 Coordinate System in Pro/ENGINEER



The IDF translator performs the following checks when importing components:

Working with IDF

Best Practices: Working with IDF

- If the reference designator matches a component instance in the Allegro PCB Editor database and the component is not placed, IDF places it using the footprint specified in the database. If the component is placed, IDF moves it if the ownership is set to MCAD.
- If the reference designator is not part of the database, IDF imports the component as a symbol using the package name specified in the IDF file, provided that such a symbol can be found in the location specified by the `PSMPATH` variable.

Holes

The following example shows an IDF 3.0 drilled hole record, where each hole definition follows this syntax:

Field 1



	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							

IDF 3.0 reserves VIA, PIN, MTG, TOOL, and OTHER. For backward compatibility Cadence reserves TARGET. The following naming formula is used to create hole symbols in Allegro PCB Editor.

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

Holes belonging to reference designators (see example P1 above) are not imported into Allegro PCB Editor.

If the hole type is not PIN, MTG, TOOL, or VIA, the naming formula is ignored and the value of the hole type field is used. The above example, therefore, results in these symbols in a Allegro PCB Editor database:

`NPIN400 PVIA400 NMTG400 PTOOL400 150C125`

If a symbol with this name is already defined, then it is used; otherwise a new symbol is defined. This symbol contains a single instance (a pin) of a padstack with the same name and

Working with IDF

Best Practices: Working with IDF

is placed at the drilled hole's x,y location. As you can see, there are three methods of exporting drill holes from Pro/ENGINEER to Allegro PCB Editor:

- Modify the value of the hole TYPE in Pro/ENGINEER to use an existing padstack name
- Build a Allegro PCB Editor library with full-featured padstacks using the naming formula above
- Replace all hole symbols with existing library symbols after importing the IDF file

In Pro/ENGINEER, the following ECAD parameters are used to change the values of the fields 4, 6 and 7 in the syntax listed in the beginning of this section:

- ECAD_HOLE_TYPE = PIN, VIA, MTG, TOOL, other
- ECAD_PLATING_STYLE =NPTH, PTH
- ECAD_HOLE_OWNER = MCAD, ECAD, UNOWNED

Pro/ENGINEER ECAD Area to Allegro PCB Editor Object Mapping

Table 1-4 shows the object mapping from Pro/ENGINEER to Allegro PCB Editor.

Table 1-4 Pro/ENGINEER/Allegro PCB Editor Object Mapping

Pro/ENGINEER 2001 Object Name	IDF Object Name	Allegro PCB Editor Object Name
Board outline depending on the coordinate system	BOARD_OUTLINE	Class: BOARD_GEOMETRY Subclass: OUTLINE Voids/cutouts allowed.
PANEL_OUTLINE	PANEL_OUTLINE	Class: BOARD GEOMETRY Subclass: PANEL_OUTLINE. If this subclass does not exist, idf_in creates it automatically. If an outline exists on BOARD GEOMETRY / PANEL_OUTLINE, idf_out exports it.

Working with IDF

Best Practices: Working with IDF

Table 1-4 Pro/ENGINEER/Allegro PCB Editor Object Mapping

Pro/ENGINEER 2001 Object Name	IDF Object Name	Allegro PCB Editor Object Name
A part to represent a volume. In this part, set ECAD_PART_TYPE to be OTHER_OUTLINE. You can also set ECAD_PART_NAME to a name that you want for the OTHER_OUTLINE section in the exported IDF file.	OTHER_OUTLINE	Class: BOARD_GEOMETRY Subclasses: OTHER_OUTLINE_TOP OTHER_OUTLINE_BOT
ROUTE_KEEPPIN	ROUTE_OUTLINE	Class: ROUTE_KEEPPIN Subclass: ALL Only one ROUTE_KEEPPIN allowed. No voids.
PLACE_KEEPPIN	PLACE_OUTLINE	Class: PACKAGE_KEEPPIN Subclass: ALL Only one PLACE_KEEPPIN allowed.
ROUTE_KEEPOUT	ROUTE_KEEPOUT	Class: ROUTE_KEEPOUT Subclass: TOP/BOTTOM/ALL Multiple/overlay; no voids.
VIA_KEEPOUT	VIA_KEEPOUT	Class: VIA_KEEPOUT Subclass: TOP/BOTTOM/ALL Multiple/overlay; no voids.

Working with IDF

Best Practices: Working with IDF

Table 1-4 Pro/ENGINEER/Allegro PCB Editor Object Mapping

Pro/ENGINEER 2001 Object Name	IDF Object Name	Allegro PCB Editor Object Name
PLACE_KEEPOUT	PLACE_KEEPOUT	Class: PACKAGE_KEEPOUT Subclass: TOP/BOTTOM/ ALL Multiple/overlay; no voids.
PLACE_REGION	PLACE_REGION	Class: BOARD_GEOMETRY Subclass: TOP_ROOM BOTTOM_ROOM BOTH_ROOMS
Hole features	DRILLED_HOLES	Special handling on import.
Notes	NOTES	Class: BOARD_GEOMETRY Subclass: IDF_NOTES Note: You need to review text size, the x, y location, and alignment for any conflicts.
Components/parts	PLACEMENT	Placed based on reference designator and footprint or just footprint.

ECAD Features in Pro/ENGINEER

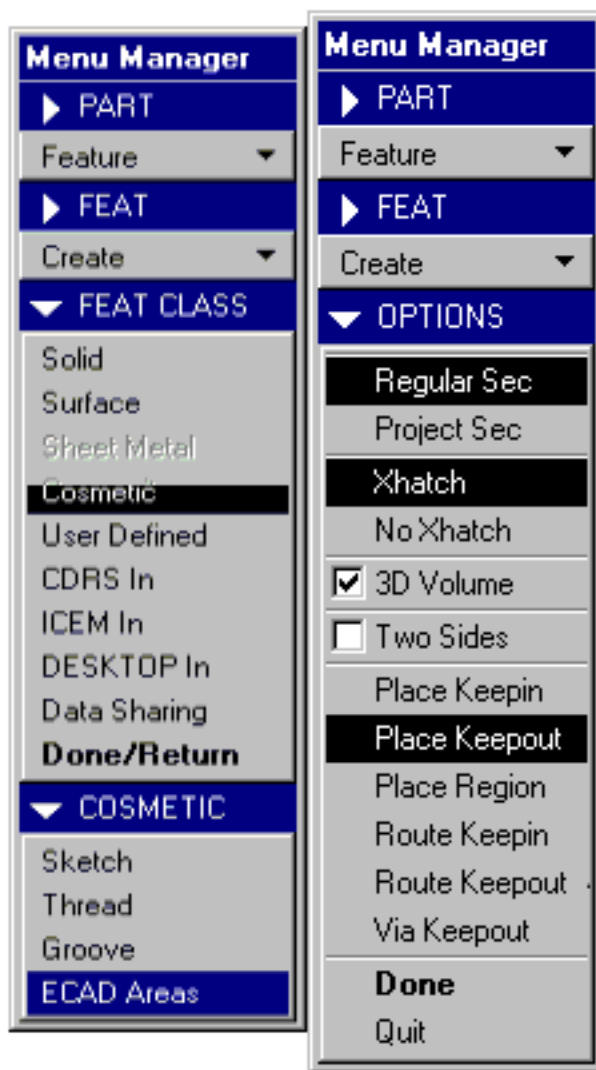
Creating Geometry

Create PCB geometry as you would in any other model.

Creating Areas

In general, you should list areas as the last features in the model tree. Figure 1-7 shows the current (Pro/ENGINEER 2001) menu items used to create ECAD areas.

Figure 1-7 Menu Items for Creating Place Keepout Features



It is recommended that you create individual layers for each ECAD area, for example:

- PLACE_KEEPOUT_TOP or PLACE_KEEPOUT_PRIMARY_SIDE
- PLACE_KEEPOUT_BOT or PLACE_KEEPOUT_SECONDARY_SIDE
- PLACE_KEEPOUT_ALL and so on

Since the creation of ECAD areas in Pro/ENGINEER is similar for each individual area type, this section focuses only on place keepout areas. Note that Allegro PCB Editor does not allow for multiple place keepsins and route keepsins. Tables [1-4](#) and [A-2](#) contain additional

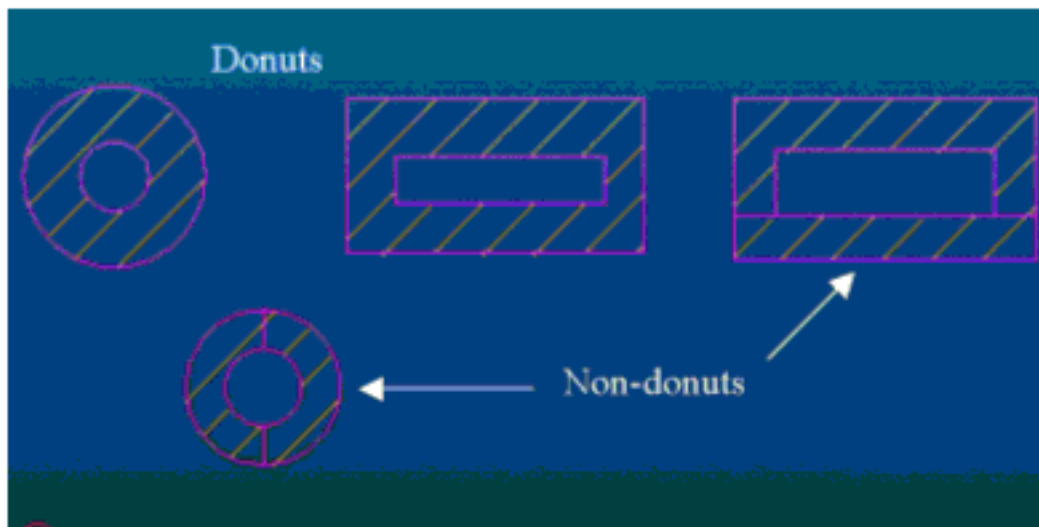
information on how to control the output fields of the resulting IDF file, and how ECAD areas translate to Allegro PCB Editor.

You should create all place keepout areas as three-dimensional features and assign them a non-zero height. This lets you use the edges of previous place keepout features in creating future keepout areas. Route keepout and via keepout areas are two-dimensional; you cannot use them as references to future keepout areas.

Note: The secondary-side height starts from the secondary side, not the primary side. Input the specified value; do not factor in the thickness of the board.

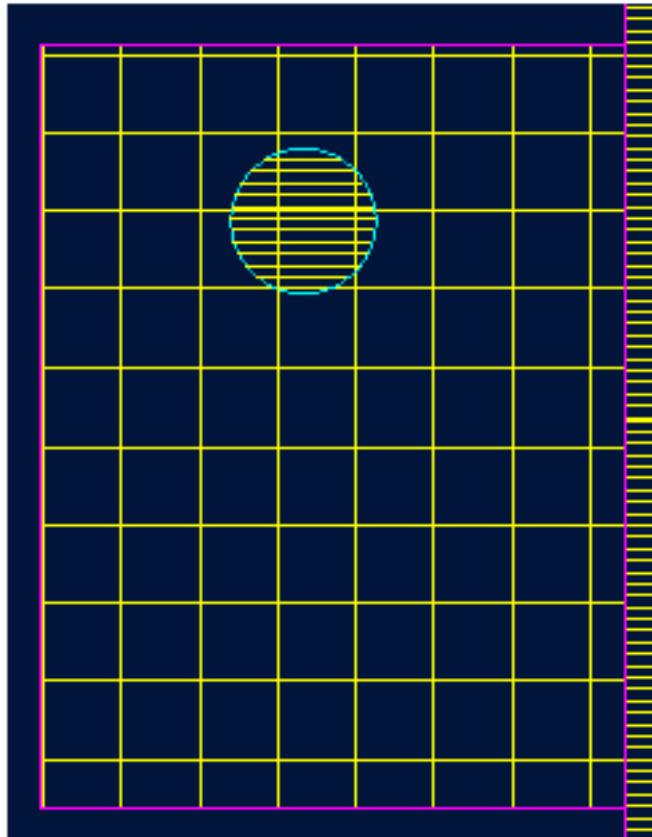
Furthermore, areas containing voids are not allowed, due to restrictions in Allegro PCB Editor. As shown in Figure 1-8, you can work around this by creating multiple separate areas, though this may defeat the design intent of the keepout areas.

Figure 1-8 Donuts and Workarounds



Another possible workaround is to create overlapping areas. When creating overlapping ECAD areas, Allegro PCB Editor understands the most stringent keepout area code of the overlapped areas. So if you have a "no component, no trace" area surrounding a mounting hole, and then a "primary side .500/secondary side .250" area surrounding that area, Allegro PCB Editor does not place components inside the smaller hole. This helps avoid the donut problem.

Figure 1-9 Overlapping ECAD Areas



The circular keepout area in Figure 1-9 represents a "no component, no trace through all" restriction. The rectangular keepout area allows trace and components up to .250" tall, except through the more restrictive circular region. It is acceptable for the ECAD areas to overlap in this fashion.

When using a separate part for creating an ECAD area (for example, using a stiffener to create a place keepout to limit component and trace placement), all features can be created using "offset edge" within the assembly with a minimum value of .050 to create the geometry. After successfully creating the geometry in the assembly, you should switch to the part file and delete all assembly references. When dealing with complex geometry, delete only a few assembly references at a time and constrain the geometry to the part.

As a typical best practice procedure, make sure that all dimensions and constraints are strong. Also, make sure you assign the correct keepout area height after completing all keepout areas.

Creating Chamfers

Chamfers made in corners (perpendicular to the primary side of the board) translate as typical geometry when exported to ECAD formats. Chamfers that are in any other direction do not translate cleanly. You must suppress this geometry before exporting to ECAD so that Allegro PCB Editor can read the entire exterior profile of the board. See Figures [1-10](#) and [1-11](#).

Figure [1-10](#) shows two types of chamfers in this model. Notched chamfers are shown in the upper corners, and a slanted chamfer for finger connectors at the bottom edge, shown as the horizontal line .25" from the bottom.

Figure 1-10 Chamfers

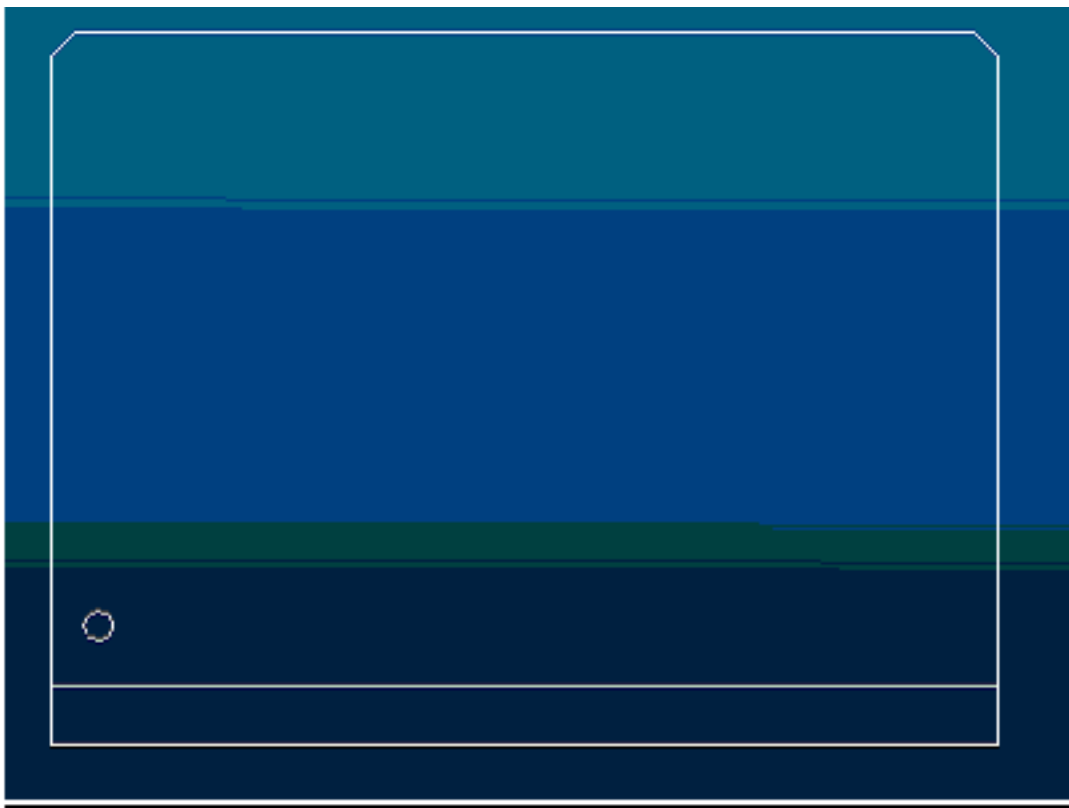


Figure [1-11](#) shows the imported ECAD file. The notched edges translated correctly, but the edge chamfer along the bottom was cut back to the edge where the chamfer returned to the original board height. The chamfer is ignored during the import process.

Figure 1-11 Chamfers – Translation Issue



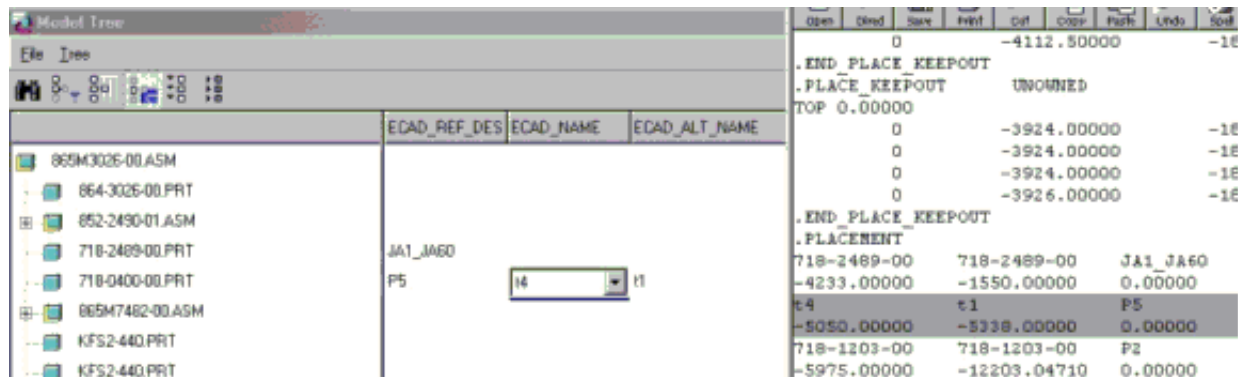
Assembling Components

You can add components to assemblies and export using IDF. To assemble components such as connectors, stiffeners, electrical components, and heat sinks on a board, you must create a coordinate system (csys) on the board that has the correct orientation and location for the component (see Figure [1-5](#)). Then, you assemble the component to the appropriate csys. After the component is assembled, you need to assign a reference designator. Choose *SETUP – NAME* from the setup menu, choose the component from the model tree or screen, and then enter the appropriate reference designator. You also can use the Model Tree window and display the ECAD parameters that you want to change (see Figure [1-12](#)).

Working with IDF

Best Practices: Working with IDF

Figure 1-12 Pro/ENGINEER ECAD Parameters



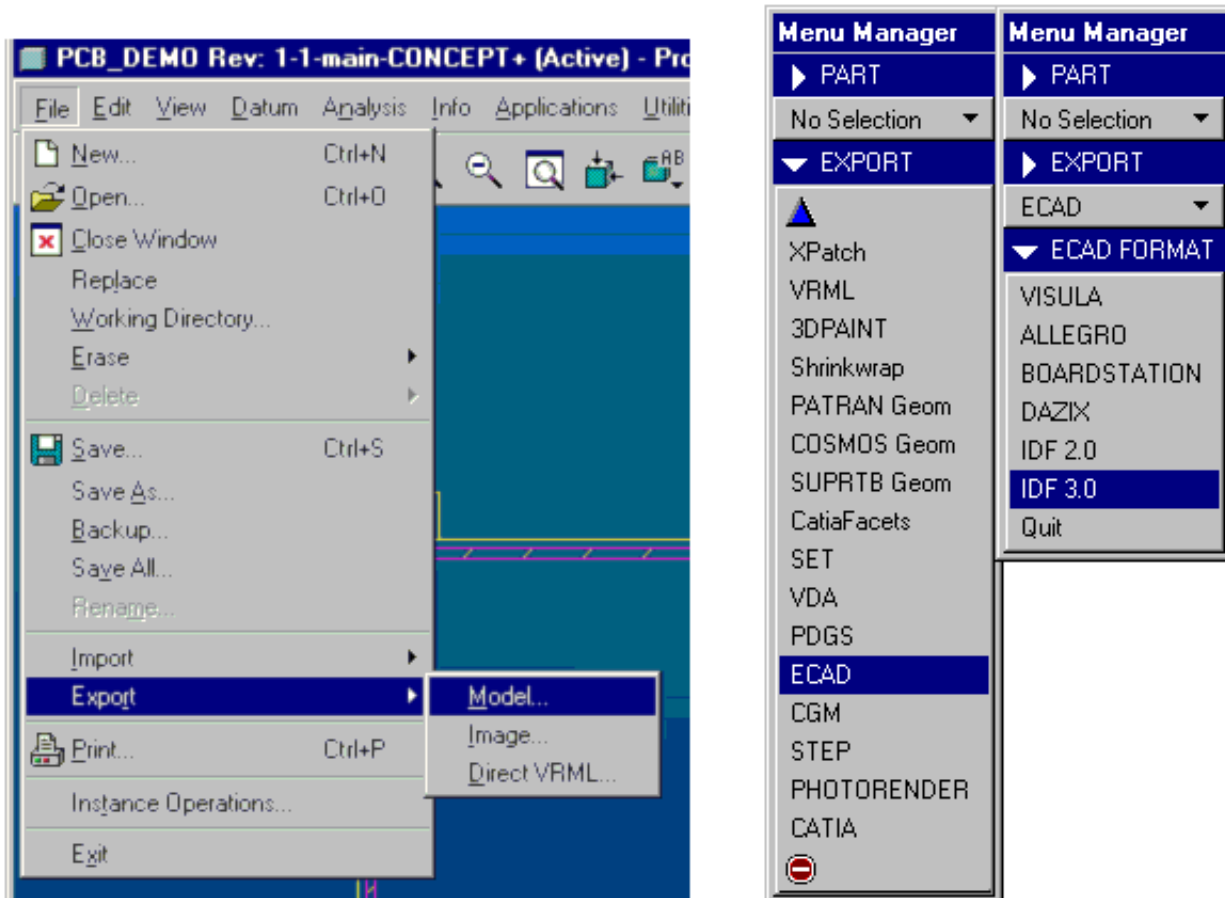
Exporting Files

Before exporting the board file, you must create an ECAD_DEFAULT csys. Make sure that the csys is located at the 0,0 of the board. Start the positive Z-axis on the bottom surface of the board and point through the board. Point the positive X-axis to the right and the positive Y-axis up, using the right-hand rule.

The ECAD group receives completed PCB outlines electronically, when you export the database in Pro/ENGINEER. The menu items are *File – Export – Model...ECAD – IDF 3.0* or *File – Save As*.

When exporting a board, choose *Board* from the menu, and choose the ECAD_DEFAULT csys, if prompted. The output file has the .emn extension.

Figure 1-13 Current Pro/ENGINEER 2001 Menu Items for Exporting ECAD Files



Limitations and Tips

This section describes the limitations when importing data and tips for use in importing data.

Holes

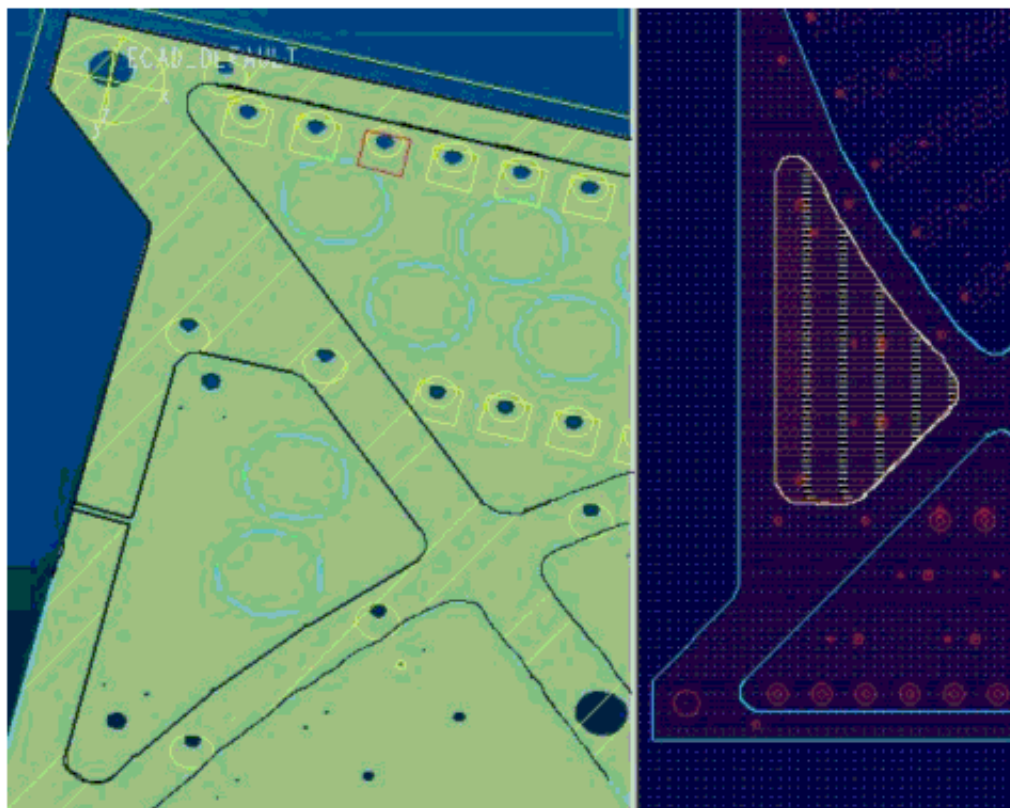
The import of drilled holes into Allegro PCB Editor follows a specific naming convention, which is based on the field records of the DRILLED_HOLES section of the IDF file (see [Holes](#) on page 12). It is recommended that you create a library of hole symbols, which follows this naming convention, since it eliminates the step of replacing or deleting hole symbols after importing IDF data.

In Pro/ENGINEER, you can change the hole type using the ECAD_HOLE_TYPE parameter. Note that when you export the mechanical database, the value of this parameter is truncated to 10 characters plus an additional "M" if the string is longer than 9 characters.

Areas

According to the IDF Specification, the first and last coordinates of an area must be the same (see the BOARD OUTLINE section in [Example](#) on page 33). Figure 1-14 illustrates how an inappropriate accuracy can cause false areas. Although the IDF translator in Allegro PCB Editor verifies that the first and last coordinate of an area match, the mechanical engineer should be aware of this minor issue. If IDF detects such violations, the translator logs an error containing the line number in the IDF file that marks the beginning of the area definition, and continues with the next item in the IDF file.

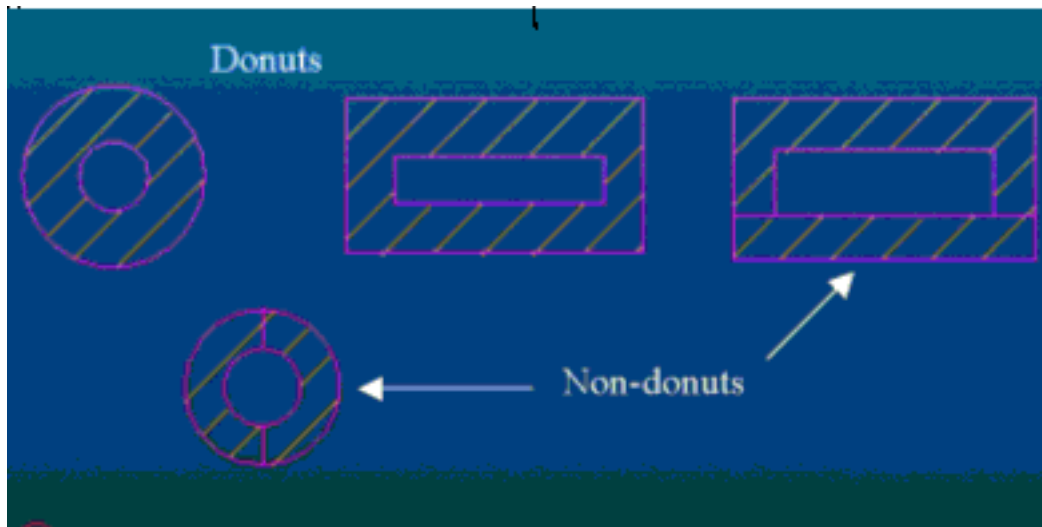
Figure 1-14 Accuracy in Pro/ENGINEER



As indicated in Table 1-4, most of the ECAD areas related to the IDF flow are not allowed to contain voids. This relates to the Allegro PCB Editor implementation of the respective shapes.

Figure 1-15 shows how you can work around this issue to successfully implement areas containing voids.

Figure 1-15 Areas Containing Voids

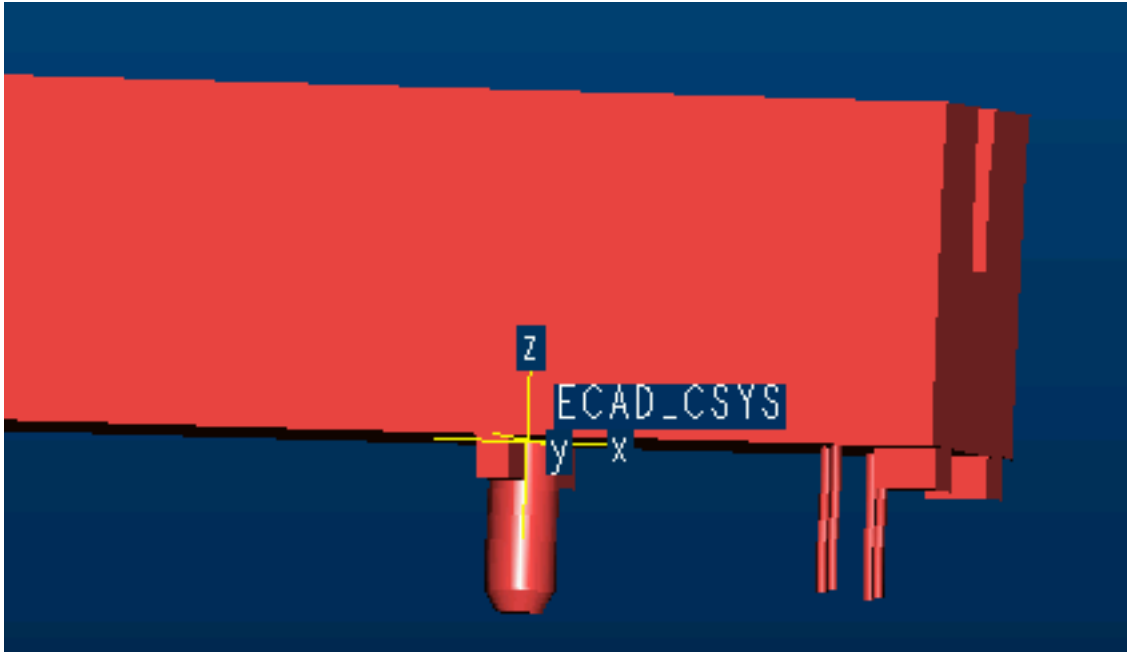


Also, when importing IDF files from Allegro PCB Editor to Pro/ENGINEER, be aware that ECAD areas such as PLACE_KEEPOUT are not imported into the database in Pro/ENGINEER.

Components

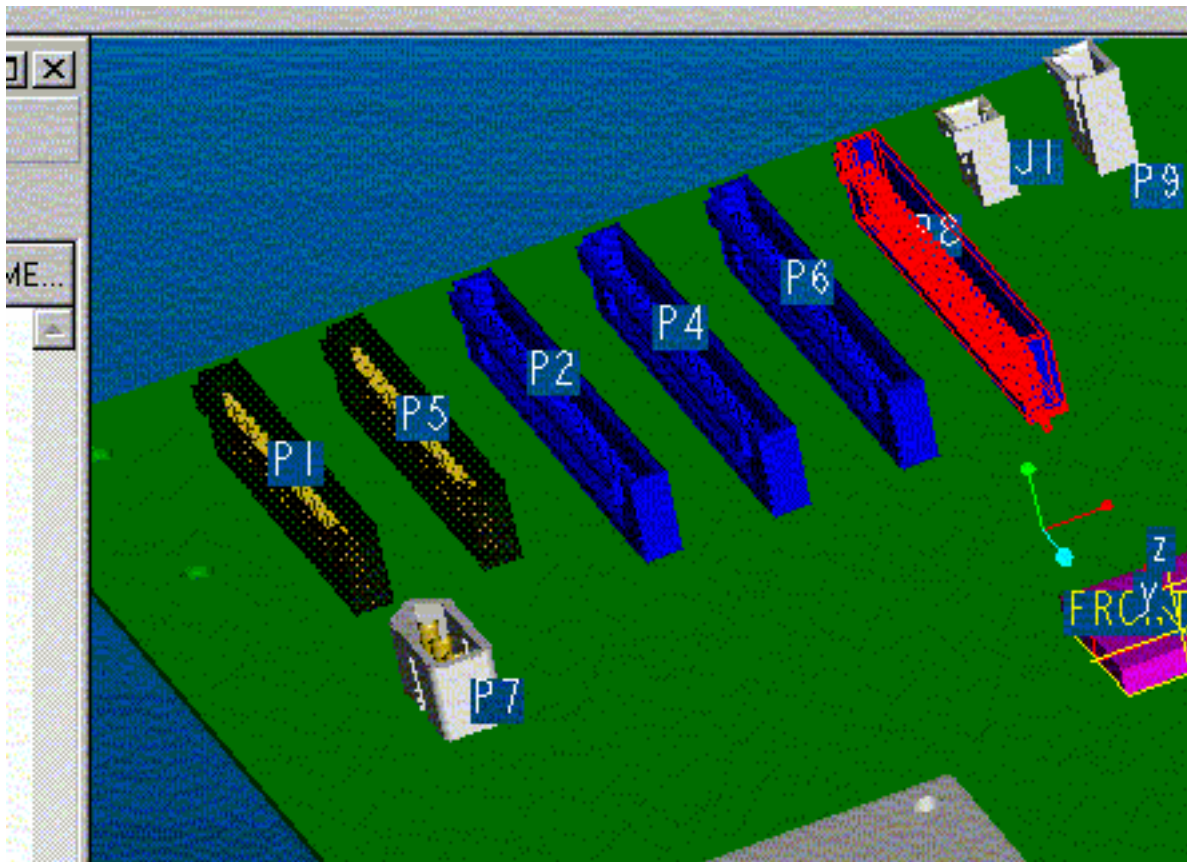
Coordinate Systems in Pro/ENGINEER are the key to successful transfer of component placement between ECAD and MCAD systems. Since this transfer takes place between a three-dimensional and a two-dimensional environment, you need to clearly specify the Primary (top) and Secondary (bottom) sides of the board and locate both the Component Coordinate System and the Board Coordinate System at the bottom surface. The location of the component coordinate (Figure 1-16) shows a Pro/ENGINEER part model with the Component Coordinate System ECAD_CSYS at the wrong location, inducing a mounting offset, which is set back to 0.0 while importing data into Allegro PCB Editor. An IDF export from Allegro PCB Editor to Pro/ENGINEER reports that this component has changed its location.

Figure 1-16 Component CSYS at Wrong Location



Pro/ENGINEER is also limited in importing or updating component information, where components are placed or arranged using a pattern (see Figure 1-17).

Figure 1-17 Patterns in Pro/ENGINEER



Conclusion

Importing mechanical design information into Allegro PCB Editor can be a challenging task. Currently, the most popular method is the use of DXF files. Comparing IDF and DXF translators, you will find that both have advantages and disadvantages. Knowing the limitations helps to determine which format works best when importing the data you want.

IDF allows for a quick and accurate import of mechanical data such as components, outlines, and keepout or keepin areas. To import component placement, the mechanical engineer should assign reference designators. The coordinate systems of both MCAD parts and the Allegro PCB Editor footprint must match. Although it appears that there is a lot of work to complete prior to using the IDF flow, once these requirements are in place, this flow decreases the amount of time it takes to transfer information between MCAD and ECAD from weeks to minutes.

Figure 1-18 shows a large printed circuit board with a complex connector placement. The approximate dimensions are 20 x 27 inches. It also contains complex keepout figures on the top and bottom surface (see the blue shapes in Figure 1-19). Importing this type of mechanical data is not unusual, and in the past was done in several weeks including changes in placement.

Figure 1-18 Pro/ENGINEER Assembly

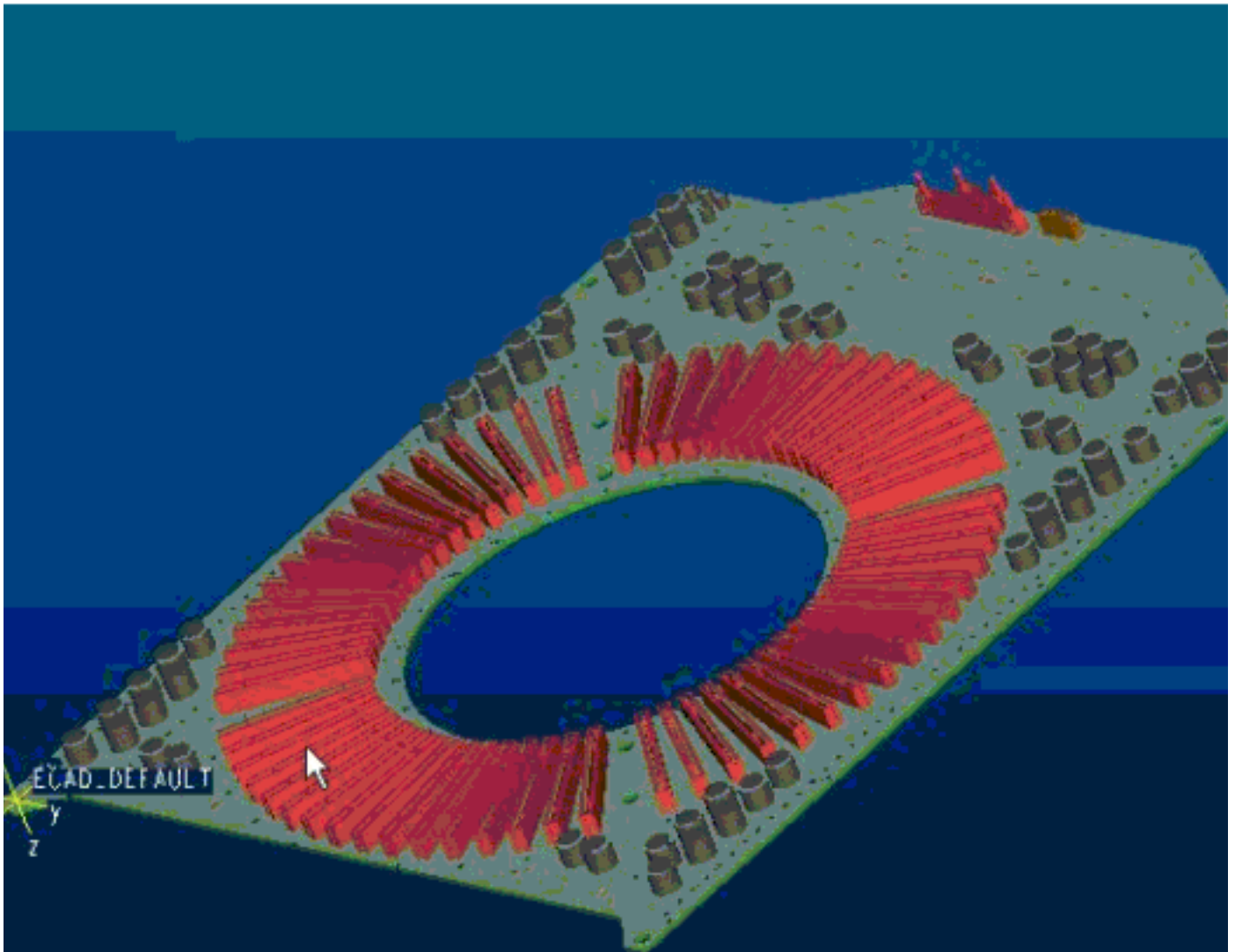
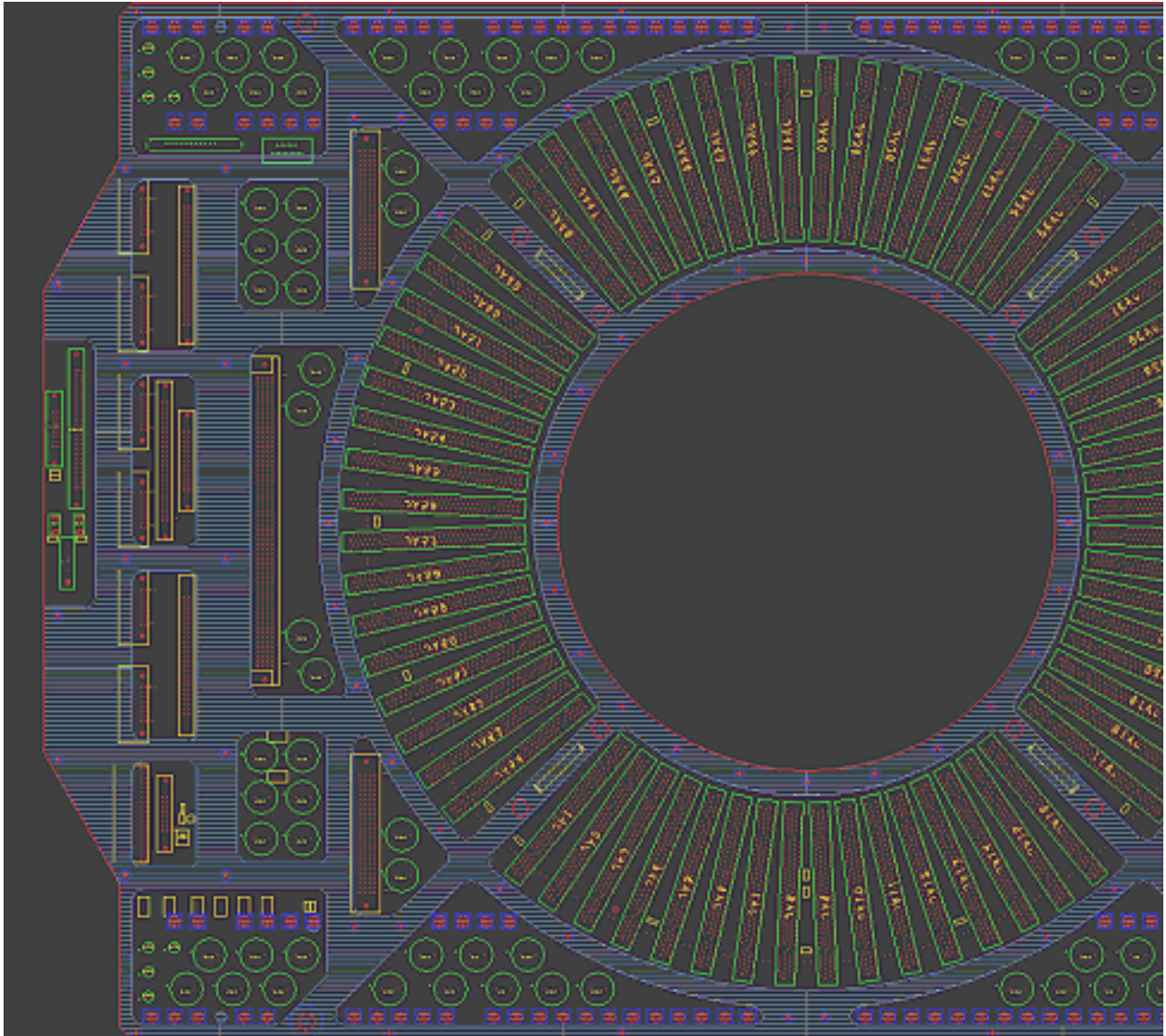


Figure 1-19 Sample Allegro PCB Editor Board



Using the enhanced IDF 3.0 translator in Allegro PCB Editor, data transfer is complete in less than 5 minutes (PIII 900MHz, 512MB RAM, 133MHz front side bus) and includes the placement of all connectors and capacitors. At this point, the suggested flow is not truly bi-directional. Importing data back into Pro/ENGINEER is not as easy as it seems, especially if ECAD areas created in Allegro PCB Editor are not imported. Most issues have been reported to Parametric Technology Corporation (PTC).

Pro/Engineer Configuration Settings

This appendix contains:

- Pro/ENGINEER configuration variables
- Pro/ENGINEER configuration parameters
- Example file

Configuration Variables

Table A-1 describes the Pro/ENGINEER variables.

Table A-1 Pro/ENGINEER Variables

Pro/ENGINEER Environment Variable	Description
ecad_area_default_import	Defines how imported ECAD areas are treated. <code>Cosm_area</code> treats imported ECAD areas as cosmetic area features; <code>3d_volume</code> imports ECAD areas with z-heights as a three-dimensional, enclosed quilt.
ecad_board_csys_def_name	Specifies the default coordinate system name added to an ECAD board being imported. If you do not set this variable, the system prompts you for a name.
ecad_comp_csys_def_name	Specifies the default coordinate system name added to an ECAD component being imported. If you do not set this variable, the system prompts you for a name.
ecad_comp_layer_map	Allows layer mapping for the ECAD component's import into assembly.

Working with IDF

Pro/Engineer Configuration Settings

Table A-1 Pro/ENGINEER Variables

Pro/ENGINEER Environment Variable	Description
ecad_create_hint_add	Helps create an <code>ecad_hint.map</code> file. Yes automatically renames components, if necessary, each time the library of component outlines is imported to Pro/ENGINEER. This does not create an <code>ecad_hint.map</code> file. It controls whether the <code>ecad_hint.add</code> file is created.
ecad_default_comp_height	Sets the default value and units for an ECAD component being imported. Units can be: inch, mil (1E-3 inches), thou (1E-6 inches), cm, mm, micron (1E-6 meters), dsu (1E-8 meters). If not set, the system uses the current component's 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 Pro/ENGINEER holes as cuts to ECAD systems.
ecad_exp_both_two_areas	Supports the export of ECAD areas with different "Above Board" and "Below Board" conditions. Yes enables you to export both sided keepin and keep-out ECAD areas as two individual areas (top and bottom).
ecad_import_holes_as_features	Imports sections specified as DRILLED_HOLE as through-all holes. Boards created with Pro/ENGINEER drilled holes export with the default value of NPTH for the ECAD_HOLE_TYPE parameter. Create this feature parameter if a value of PTH is needed (IDF 2.0/3.0).
ecad_mapping_file	Specifies the <code>ecad_hint.map</code> file that will be used for ECAD operations.
ecad_other_outl_csyes_def_name	Specifies the default coordinate system name for the OTHER_OUTLINE section of an IDF 2.0 ECAD component being imported. If you do set this variable, the system prompts you for a name.
ecad_panel_csyes_def_name	Specifies the default coordinate system name added to an ECAD panel being imported. If you do not set this variable, the system prompts you for a name.

Working with IDF

Pro/Engineer Configuration Settings

Table A-1 Pro/ENGINEER Variables

Pro/ENGINEER Environment Variable	Description
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

Table [A-2](#) describes the Pro/ENGINEER ECAD parameters.

Table A-2 Pro/ENGINEER Feature Parameters

Parameter	Optional Value	Valid Format	Description
ECAD_OWNER	ECAD, MCAD, UNOWNED	IDF 3.0 only	Owner of ECAD area.
ECAD_HOLE_TYPE	PTH, NPTH, PIN, VIA, MTG, TOOL, OTHER	IDF 2.0 IDF 3.0	The type of hole feature. If the hole is not imported as a feature, the parameter is not imported.
ECAD_PLATING_STYLE	PTH, NPTH	IDF 3.0 only	Plated through hole, non-plated through hole.
ECAD_HOLE_OWNER	ECAD, MCAD, UNOWNED	IDF 3.0 only	Owner of the hole feature.
BOARD_SIDE	TOP, BOTTOM, BOTH	ALL (according to the areas)	The side associated with the place region, place keepin, or place keepout ECAD areas.

Working with IDF

Pro/Engineer Configuration Settings

Table A-2 Pro/ENGINEER Feature Parameters

Parameter	Optional Value	Valid Format	Description
ROUTING_LAYERS	TOP, BOTTOM, BOTH	IDF 2.0	Shows which ECAD board routing layers in the electrical system are included in a Pro/ENGINEER routing area ECAD feature. No actual routing on the board is supported for transfer by the IDF standard.
	ALL, TOP, BOTTOM, BOTH, ALL, INNER	IDF 3.0	
COMPONENT_GROUP_NAME	ANY	IDF 2.0 IDF 3.0	Allows you to add or keep a name for the place region ECAD area feature.
ECAD_ASSOCIATED_PART	BOARD NOREFDES BOARD NOREFDES PANEL Reference Designator	IDF 2.0 IDF 3.0	In the ECAD system, this is the part to which the Pro/ENGINEER hole feature relates.
ECAD_NAME	ANY	ALL	The ECAD system name of the component which becomes the name of the Pro/ENGINEER .prt file.
ECAD_ALT_NAME	ANY	ALL but PCB Editor	The ECAD system alternative name for the component.
ECAD_REF_DES	Reference Designator	Any	The ECAD reference designator for the component.

Working with IDF

Pro/Engineer Configuration Settings

Table A-2 Pro/ENGINEER Feature Parameters

Parameter	Optional Value	Valid Format	Description
ECAD_OWNER	ECAD, MCAD, UNOWNED	IDF 3.0	Shows the owner of the board or other outline.
ECAD_PART_TYPE	Panel Board Electrical Mechanic Other outline	IDF 2.0 IDF 3.0	Shows the component type.
ECAD_PART_NAME	any	IDF 2.0 IDF 3.0	Shows the other outline name.
ECAD_CAPACITANCE	any	IDF 3.0 only	Capacitance value.
ECAD_RESISTANCE			Resistance value.
ECAD_TOLERANCE			Tolerance value.
ECAD_THET_JC			
ECAD_POWER			
ECAD_PLACEMENT_STATUS	fixed	IDF 3.0 only	

Example

The following is an example to use when importing an IDF file.

```
.HEADER
BOARD_FILE    3.0    "ICU 2003"    2003/01/02.11:40:32    25.0
basic_idf.brd    THOU
.END_HEADER
.BOARD_OUTLINE
MCAD
62.50
0    4350.00    -3100.00    0.00
0    5100.00    -2350.00    90.00
0    5100.00    5950.00    0.00
```

Working with IDF

Pro/Engineer Configuration Settings

```

0      4350.00   6700.00   90.00
0      3550.00   6700.00    0.00
0      2800.00   5950.00   90.00
0      2800.00  -2350.00    0.00
0      3550.00  -3100.00   90.00
0      4350.00  -3100.00    0.00
1     -4900.00   8700.00    0.00
1     -4900.00  -5100.00    0.00
1     -4400.00  -5600.00   90.00
1     12700.00  -5600.00    0.00
1     13200.00  -5100.00   90.00
1     13200.00   8700.00    0.00
1     12700.00   9200.00   90.00
1     -4400.00   9200.00    0.00
1     -4900.00   8700.00   90.00
.END_BOARD_OUTLINE
.ROUTE_OUTLINEECAD
ALL
1     -4400.00  -5200.00    0.00
1     -4400.00   8800.00    0.00
1     12800.00   8800.00    0.00
1     12800.00  -5200.00    0.00
1     -4400.00  -5200.00    0.00
.END_ROUTE_OUTLINE
.PLACE_OUTLINEUNOWNED
BOTH
1     -4000.00  -4900.00    0.00
1     -4000.00   8500.00    0.00
1     12400.00   8500.00    0.00
1     12400.00  -4900.00    0.00
1     -4000.00  -4900.00    0.00
.END_PLACE_OUTLINE
.ROUTE_KEEPOUTUNOWNED
ALL
1      2400.00  -3400.00    0.00
1      2400.00   6900.00    0.00
1      5600.00   6900.00    0.00
1      5600.00  -3400.00    0.00
1      2400.00  -3400.00    0.00
.END_ROUTE_KEEPOUT
.VIA_KEEPOUTUNOWNED

```

Working with IDF Pro/Engineer Configuration Settings

```

1      7200.00  -4400.00      0.00
1      7200.00  -2400.00      0.00
1     12000.00  -2400.00      0.00
1     12000.00  -4400.00      0.00
1      7200.00  -4400.00      0.00
.END_VIA_KEEPOUT
.PLACE_KEEPOUTUNOWNED
BOTH
1      8700.00    700.00      0.00
1      8700.00   3700.00      0.00
1     12000.00   3700.00      0.00
1     12000.00    700.00      0.00
1      8700.00    700.00      0.00
.END_PLACE_KEEPOUT
.PLACE_REGIONUNOWNED
TOP CPU
1      6700.00   4100.00      0.00
1      6700.00   8000.00      0.00
1     12000.00   8000.00      0.00
1     12000.00   4100.00      0.00
1      6700.00   4100.00      0.00
.END_PLACE_REGION
.DRILLED_HOLES
    36.00  -1700.00   2200.00PTHU1  PINUNOWNED
    36.00  -1400.00   1800.00PTHU1  PINUNOWNED
    36.00  -1400.00   1700.00PTHBOARD  PINUNOWNED
    36.00  -1400.00   1600.00PTHBOARD  VIAUNOWNED
   125.00      0.00    6000.00  PTH    BOARD  MTGUNOWNED
   125.00      0.00    5000.00  NPTH    BOARD  TOOL    UNOWNED
   125.00  -1000.00    5000.00  PTH    BOARD  user    UNOWNED
.END_DRILLED_HOLES
.NOTES
    200.0011700.00 200.00  5500.00 "Demonstrating the translation to"
    200.0011500.00 200.00  2000.00 "and from the IDF format."
    200.0011300.00 200.00  2400.00 "These notes should appear on"
    200.0011100.00 200.00  2000.00 "Board Geometry/IDF_Notes"
.END_NOTES
.PLACEMENT
SOIC20WDEVICE?U1
    -2500.00   2000.00      0.00      0.00TOPPLACED
DIP14_3DEVICE?U2

```

Working with IDF

Pro/Engineer Configuration Settings

-1700.00 2200.00 0.00 0.00TOPPLACED
.END_PLACEMENT