

Pro/ECAD


Benefits, Techniques and Best Practices Tutorial



PTC™
Shaping Innovation

Table of Contents:

- 1) [Objective](#)
- 2) [Overview](#)
- 3) [Metric](#)
- 4) [Tutorial](#)
- 5) [Key Vocabulary](#)
- 6) [Tutorial Evaluation](#)

Title: Pro/ECAD Techniques, Benefits & Best Practices	
Date: 12/05/01	
PTC-MSS Services	

Objective:

At the end of this tutorial, you will be able to:

- Repeat the steps necessary to ensure a smooth exchange of data between the Electrical and Mechanical department.
- Formulate an internal process for managing changes between the Electrical Engineer & the Mechanical Engineer.
- Improve product quality.
- Promote a concurrent design environment w/ out fear of design ramifications downstream and from design issues w/in your team members.

Overview:

In the typical design process, the Mechanical Designer defines the board shape, specifies important “keep in” and “keep out” areas, and places critical components such as connectors, switches, displays, and LED’s using Pro/Engineer. This information is exported via an IDF file to the PCB designer to use as the basis for the board layout in the PCB layout system. After placing the remaining components, the fully placed board assembly is passed back through the IDF file to the Mechanical Designer to make sure the board assembly fits into the final product package. Multiple iterations of this basic flow typically occur during the product design phase.

Why is it beneficial to the Electrical Engineer?

The electrical engineer can communicate design requirements effortlessly to the Mechanical Engineer. This information includes, hole placement, pin hole placement, keep in/keep out areas, and board size.

Why is it beneficial to the Mechanical Engineer?

Mechanical Engineers can specify mechanical requirements and transmit them directly to an electrical engineer’s PCB layout program. Most all PCB layout programs have an export capability called an IDF file. This is similar to exporting an IGES file. The difference is that exact component information (placement and size) is contained within the file. This information is transmitted effortlessly to the Mechanical Engineer. With other translation methods, such as IGES, data is often lost in the conversion.

What are the mechanical aspects that are critical to the Mechanical Engineer?

- 1) Interference Checking- Accurate assemblies are created and can be joined with other assemblies to check interference between mating parts.
- 2) Mass Properties can be calculated automatically by using a library of ECAD component part files that contain accurate mass property information.
- 3) Static, dynamic, thermal analysis can be performed.

Metric:

The following example illustrates the immediate impact on utilizing Pro/ECAD to automate Printed Wiring Assembly (PWA) design information exchange between the PCB design group and the mechanical engineering group.

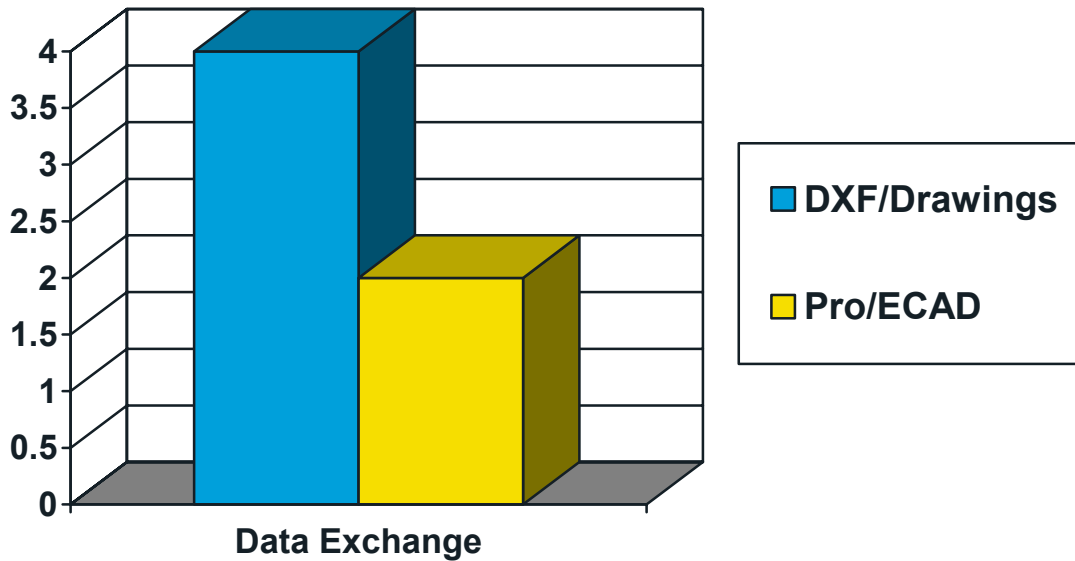
This is done to:


1. **Increase accuracy**
2. **Enhance efficiency**

And removing need of interface drawing creation:

1. **Translates in 1-2 man-days saving per PWA assembly**
2. **Immediate communication of design changes to Mechanical Engineering upon import of PCB design data**
3. **Eliminates need for change documents**

The previous practice of exchanging drawings and DXF data resulted in longer effort and inaccurate results as exact component placements were difficult to establish.



Title: Pro/ECAD Techniques, Benefits & Best Practices	
Date: 12/05/01	
	PTC-MSS Services

Tutorial:

What is the best way to use Pro/ECAD?

You can import parts from an ECAD database in two ways:

1. Let the Pro/ECAD translator automatically create basic 3-D extrusions of the part outlines as they are read in, or
2. Use a map file to reference a library of ECAD part files you have set up beforehand.

If you use the automatic method, a separate .prt file for each component is automatically created and added to the session. These files can be saved and customized in Pro/E as necessary to more accurately show the true shape of the component. After they are customized, you can use the import map file (ecad_hint.map) to substitute them for the automatically generated parts if you run the import process again.

If you already have custom part files for ECAD components (ECAD Library), you can set up the map file in advance to substitute them for the automatically generated parts. Using this method you get a more accurate representation of the components. You will see examples below showing with a library and with out.

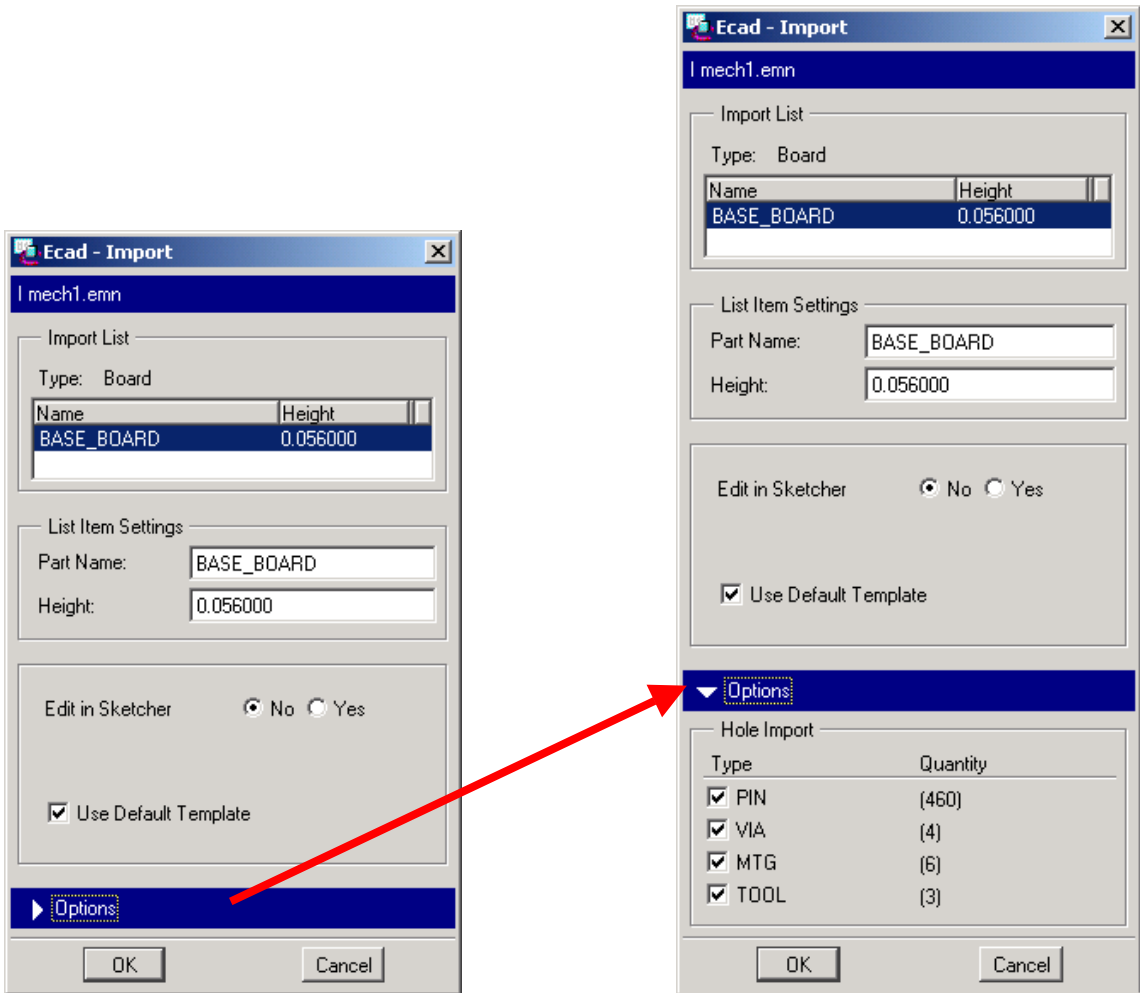
A. How do I use existing PCB files with Pro/ECAD?

STEP 1: Create a New Model w/ the .emn file:

File > Open > To create a new model > Change your type to ***ECAD IDF (*.emn)***

Select an .emn file and provide a name for the board.


The following dialog box will then appear:



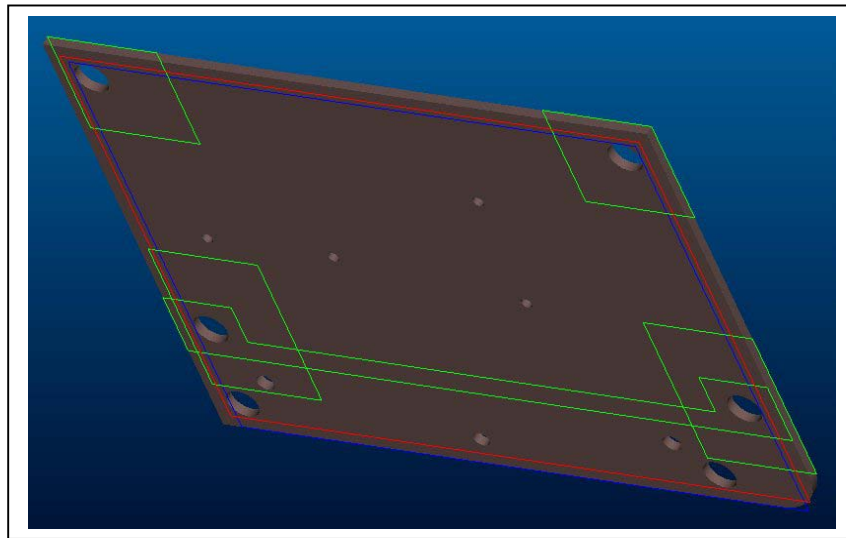
Select **Options** near the bottom of the screen:

Under the options tab you will see a listing of holes that will be imported. You may want to shut the holes off depending on the number. Some times you will see a number (say for this example 460). This represents the pin holes in the PCB board. Pro/Engineer will generate a hole for every type listed. To reduce regeneration times turn on only the holes that are needed.

Uncheck the first option to filter out the 460 PIN holes since they are not needed for our design.

Title: Pro/ECAD Techniques, Benefits & Best Practices	
Date: 12/05/01	
PTC-MSS Services	

The imported PCB board with holes and keep out/keep in areas is shown below:



Step 2: Create a new assembly

File > **New** > **Assembly** > (give it a name)

Component > **Assemble** > (In Session) > Select the PCB board part

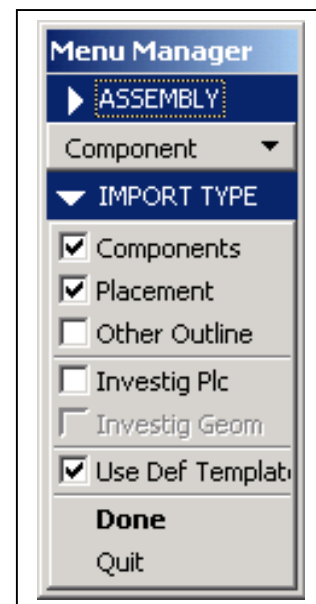
(Select the default constraint to assemble automatically.)

Step 3: Add the components from the .emn file.

Insert > **Data From File...** > Change your type to **ECAD IDF (*.emn)** and select the same .emn file > **Open**

The following options will appear in the menu:

- **Components:** Import component library
- **Placement:** Import component placement
- **Other Outline:** Import other outline(s) which are volumes representing non-electrical items that would not have reference designators
- **Investig Plc:** Imports component placements in investigate mode. This option lets you selectively accept or reject the placements of new or changed components
- **Investig Geom:** Investigate component's geometry changes
- **Use Def Template:** Use default template for new Components

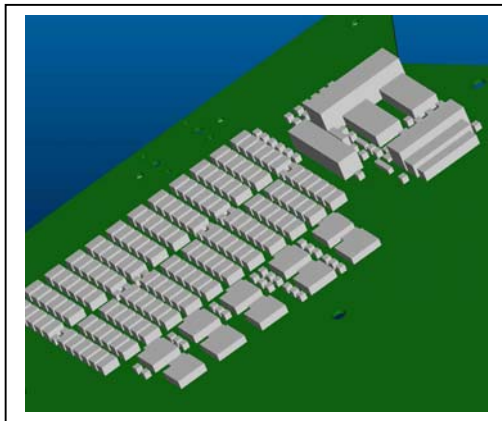


Accept the default options and select **Done**.

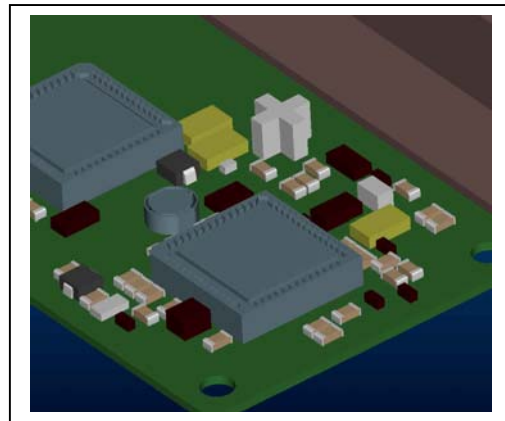
The file will import all the components creating new part files. View the model tree to verify this.

Below is the imported assembly:

1) w/out ECAD library



2) w/ ECAD library



B. How do I use start the board design in Pro/E?

STEP 1: Create an ECAD Area in the model:

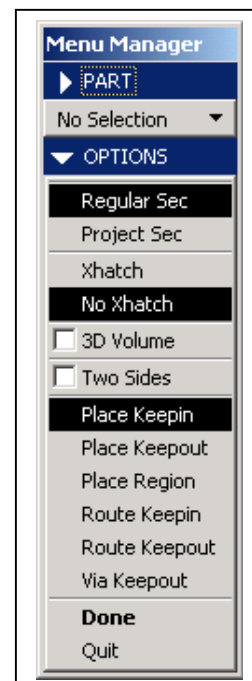
W/ a board already modeled,

Insert > Cosmetic > ECAD Area...

The following menu will appear:

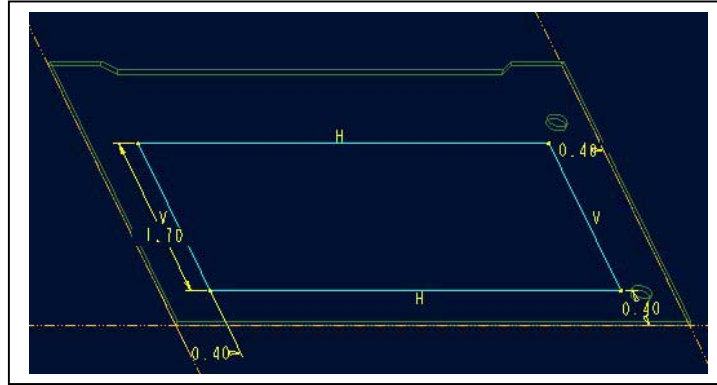
- **Regular Sec:** Feature will regular sketching plane
- **Project Sec:** Feature will use projection of section on selected surface.
- **Xhatch:** The created feature will be meshed
- **No Xhatch:** The created feature will not be meshed
- **3D Volume:** The created feature will be 3D quilt
- **Two Sides:** Create two areas w/ the same section on both sides of the board.

Refer to Key Vocabulary section for remaining definitions.

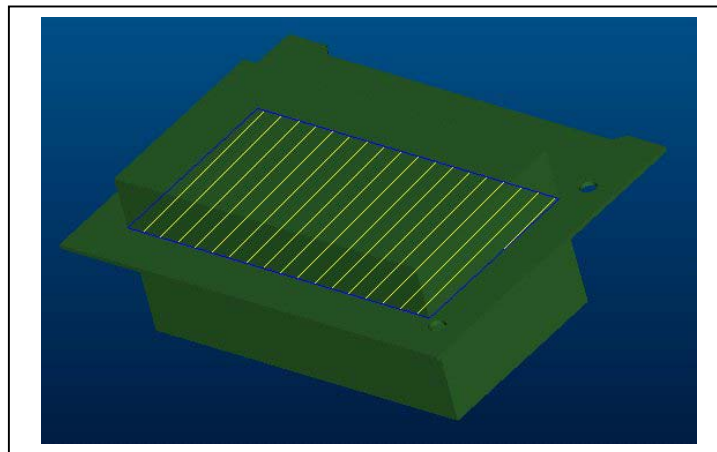


Let's select *Xhatch* > *3D Volume* and select any surface for the feature to be created on.

Sketch your area:



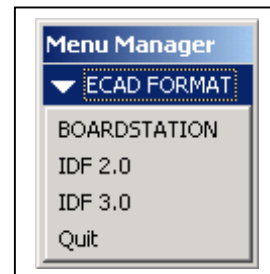
And enter the height values to get the following result:




STEP 2: Export the board outline and component information:

File > Save A Copy

Ensure that the type is set to *ECAD IDF (*.emn)* and select from the following options:




Now you are ready to import the file into your ECAD package.

Title: Pro/ECAD Techniques, Benefits & Best Practices	
Date: 12/05/01	
	PTC-MSS Services

Workflow That Enables The Best Use Of Pro/ECAD

- 1) The Mechanical Engineer creates a PCB board inside an assembly that mounts to their existing hardware.
- 2) Mounting holes are created.
- 3) Keepin/Keepout areas are created.
- 4) Connectors (from the ECAD library) are placed on the PCB board.
- 5) File is exported to an IDF 3.0 file.
- 6) The IDF file is read into the PCB layout program.
- 7) Components are added observing the keep in and keep out areas on the board.
- 8) The file is exported out of the PCB layout program back to an IDF 3.0 file.
- 9) The Mechanical Engineer opens up their existing PCB board and appends the new IDF file. New components are added and existing components are automatically moved because of the PCB layout engineer's design change.

This is a single iteration, but this can continue until the board is fully designed. Interference checks can be run and thermal analysis is one step away because of a fully designed PCB board.

Title: Pro/ECAD Techniques, Benefits & Best Practices	
Date: 12/05/01	
PTC-MSS Services	

Key Vocabulary for Pro/ECAD:

ECAD: Electrical Computer Aided Design


ecad_hint.map file: *an ASCII file you use to control the following functions in the ECAD import-export process:*

- *Substituting custom made Pro/E parts for automatically extruded parts on import.*
- *Allowing or disallowing specified parts on import.*
- *Allowing or disallowing specified parts on export.*
- *Changing an ECAD reference designator to a different string for import (If for example the ECAD reference designator uses characters that are illegal in Pro/E).*
- *Changing an ECAD 'other outline' string for import (If for example the ECAD reference designator uses characters that are illegal in Pro/E).*

Pro/ECAD searches the working directory for ecad_hint.map and references it every time import occurs. If the file is empty or has no relevant information it is ignored. You can use the configuration option ECAD_MAPPING_FILE <path> to set a default location for the ecad_hint.map file. If you set a path with this config option, the working directory is not searched.

Sample excerpt:

```
#
# A template for ecad_hint.map
#
map_objects_by_name->
ECAD_NAME "CSTCS1"
ECAD_ALT_NAME "N7414N"
ECAD_TYPE ""
MCAD_NAME "CSTCS1_PN-"
MCAD_TYPE "part"
END
#
map_objects_by_name->
ECAD_NAME "GF-1"
ECAD_ALT_NAME "SC1A17.53-ND"
ECAD_TYPE ""
MCAD_NAME "GF-1_PN-"
MCAD_TYPE "part"
END
#
map_objects_by_name->
ECAD_NAME "J1_10"
ECAD_ALT_NAME "A2099-ND"
ECAD_TYPE ""
MCAD. . .
.
.
.
```

Title: Pro/ECAD Techniques, Benefits & Best Practices	
Date: 12/05/01	
PTC-MSS Services	

IDF: Intermediate Data Format

PCB: Printed Circuit Board.

***Place Keepin:* region of board within which components are placed.**

***Place Keepout:* region of board where components cannot be placed.**

***Place Region:* region of board to place similar components within.**

***Pro/ECAD:* an interface tool that is available with every Pro/Engineer Foundation seat. This functionality enables communication between the Electrical Engineers creating PCB layouts and Mechanical Engineers creating packaging for the electrical components.**


This technology enables 3D models of the PCB board, components, and connectors to be created and placed automatically for the Mechanical Engineer. It also allows communication of keep in/keep out areas, board size, and hole placement between both Mechanical and Electrical Engineers.

***Pro/ENGINEER:* the leading 3D product development solution, enabling designers and engineers in 31,000 companies to bring superior products to market faster. Spanning the entire product development process, from creative concept through detailed product definition to serviceability, Pro/ENGINEER delivers measurable value to companies of all sizes and across all industries.**

***Route Keepin:* region of board where routing is allowed.**

***Route Keepout:* region of board where routing is not allowed.**

***Via Keepout:* region of board within which vias are not allowed.**

Title: Pro/ECAD Techniques, Benefits & Best Practices	
Date: 12/05/01	
PTC-MSS Services	

Tutorial Evaluation:

Name:						
Title:	<input type="checkbox"/> Engineer <input type="checkbox"/> Designer <input type="checkbox"/> Draftsmen <input type="checkbox"/> Mfg. Engr. <input type="checkbox"/> Tech. Pubs. <input type="checkbox"/> Analyst					
PTC Products Used:	<input type="checkbox"/> Foundation <input type="checkbox"/> Advanced Assembly Extension <input type="checkbox"/> Advanced Surface Extension <input type="checkbox"/> Behavioral Modeling <input type="checkbox"/> Intralink <input type="checkbox"/> Modelcheck <input type="checkbox"/> All					
Time using Pro/E:	<input type="checkbox"/> 0-6 months <input type="checkbox"/> 6-12 months <input type="checkbox"/> 1-2 years <input type="checkbox"/> 2-5 years <input type="checkbox"/> 5+ years					
1 – Strongly Disagree 3 – Agree 5 – Strongly Agree						
1. This tutorial content met my expectations:	1	2	3	4	5
2. The exercise was easy to understand:	1	2	3	4	5
3. This tutorial will help me on current projects:	1	2	3	4	5
4. These techniques make Pro/E a more effective tool:	1	2	3	4	5
5. These techniques will increase my speed using Pro/E:	1	2	3	4	5
What concepts/techniques learned from this tutorial will you apply on the job?						
1)						
2)						
3)						
What would you like to see as a future tutorial at your company?						
1)						
2)						
3)						
What can be done to improve these tutorials for your company?						
1)						
2)						
3)						

Additional Comments: