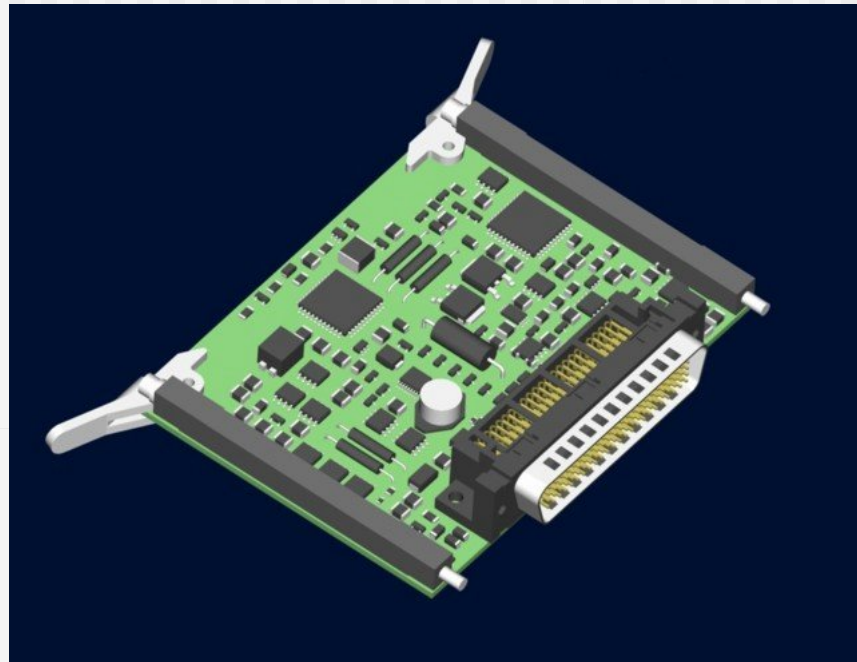


Creating a 3D PCB in Pro/Engineer

If you have any questions or comments please contact:

Keith Richman
Simplified Solutions Inc
keith@simplifiedsolutioninc.com
www.simplifiedsolutionsinc.com
(410)420-7853



Basic steps to create your 3D PCB in Pro/Engineer

1. Gather the ECAD Data and locate specifications for each component on the PCB.
2. Create or identify a matching 3D Pro/Engineer model for every ECAD component.
3. Link the ECAD Data for each component to its corresponding 3D model.

Files Required to create your 3D PCB

- IDF (Intermediate Data) Files in version 2.0 or 3.0 generated in your ECAD tool.
- 3D component models in Pro/Engineer. Each component must have the identical coordinate system location and orientation when compared to the ECAD Geometry.
- An **ecad_hint.map** file which links ECAD geometry names to 3D Pro/Engineer .prt and .asm files.

The IDF Files (.emn and.emp files)

- EMN File Contains:
 - PCB Outline
 - Component Location
 - Component Orientation
 - Hole Information
 - Keep In and Keep Out Regions

- EMP File Contains:
 - ECAD outlines for every component
 - ECAD component height information

Typical .emn file component entry

cap0805	34567890	D8			
17.145	33.655	0.0	90.0	BOTTOM	PLACED

- **cap0805** is the ecad_name (package type recommended)
- **34567890** is the ecad_alt_name (unique part number)*
- **D8** is the reference designator
- **17.145**, **33.655**, and **0.0** represent the location with respect to the PCB's (0,0) point X, Y, Z directions.
- **90.0** represents the orientation of the component.
- **BOTTOM** specifies the side of the PCB.
- **PLACED** specifies component will be added to 3D PCB.

***HINT:** To maximize reuse of Mechanical Models, set up your ECAD Design Software to output a package type as the ecad_name and either an internal corporate part number or a vendor part number as the ecad_alt_name. **Do not** set up your ECAD IDF output to use **identical** ecad and ecad_alt names.

Typical emp file entry

```
.ELECTRICAL
cap0805      34567890      THOU  59.0
0    -92.0    63.0    0
0    -92.0   -63.0    0
0     92.0   -63.0    0
0     92.0    63.0    0
0    -92.0    63.0    0
.END_
ELECTRICAL
```

- The EMP file contains information describing each component's 2D shape and height. This information gets disregarded when the Pro/ECAD User replaces this data with real 3D models

The ecad_hint.map file

The ecad_hint.map file "links" components in your ECAD Tool to 3D Pro/Engineer Models

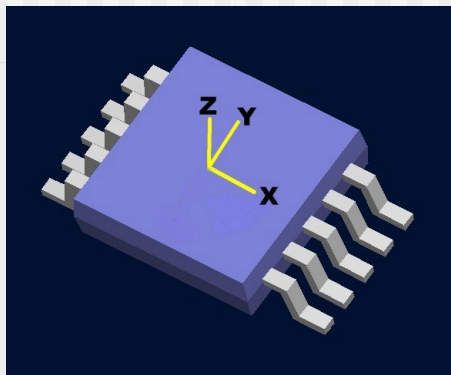
cap0805	34567890	D8			
17.145	33.655	0.0	90.0	BOTTOM	PLACED

When an IDF (emn) File is imported into Pro/Engineer, Pro/E cross-references the ecad_name and ecad_alt name from the emn file against the ecad_hint.map. If it a match is found, Pro/Engineer will replace "on the fly" geometry with a real 3D Pro/Engineer part or assembly. In this case, **c0805h100**

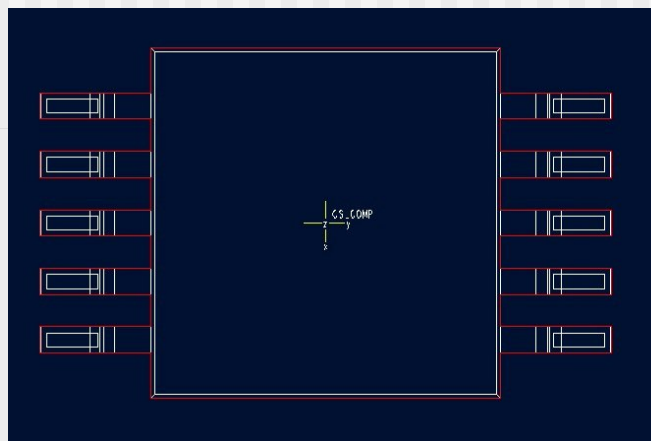
```
map_objects_by_name ->
ecad_name "cap0805"
ecad_alt_name "34567890"
ecad_type ""
mcad_name "c0805h100"
mcad_type part
end
```

Matching 3D components to existing ECAD Geometries

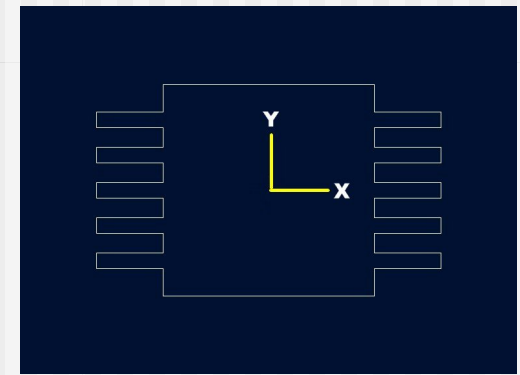
- The coordinate system in all 3D Models must have the identical orientation and be in the identical location when compared to the matching ECAD Geometry.
- The Simplified Solutions Library of Components follows IPC-7351 with respect to component orientation. Pin 1 is in the upper left hand corner. For surface mount components, the coordinate system is through the geometric center of the component, while for thru-hole parts, the coordinate system is through the center of pin 1.



4/19/2008



Revision 1.1



8

Steps to create your PCB using Simplified Solutions' Products

1. Generate your IDF files using your ECAD tool.
2. Import your .emn file to our online [hint.map tool](#).
3. While in our hint.map tool, for each line item, choose a 3D replacement model from our online library.
4. Identify components that are not in our library. For these items, either submit a component request on our website or build the 3D component on your own. Fill in the component name in the hint.map tool.
5. Generate your ecad_hint.map file.
6. Add the proper setting to your config.pro file.
7. Import your PCB into Pro/Engineer.

The hint.map tool

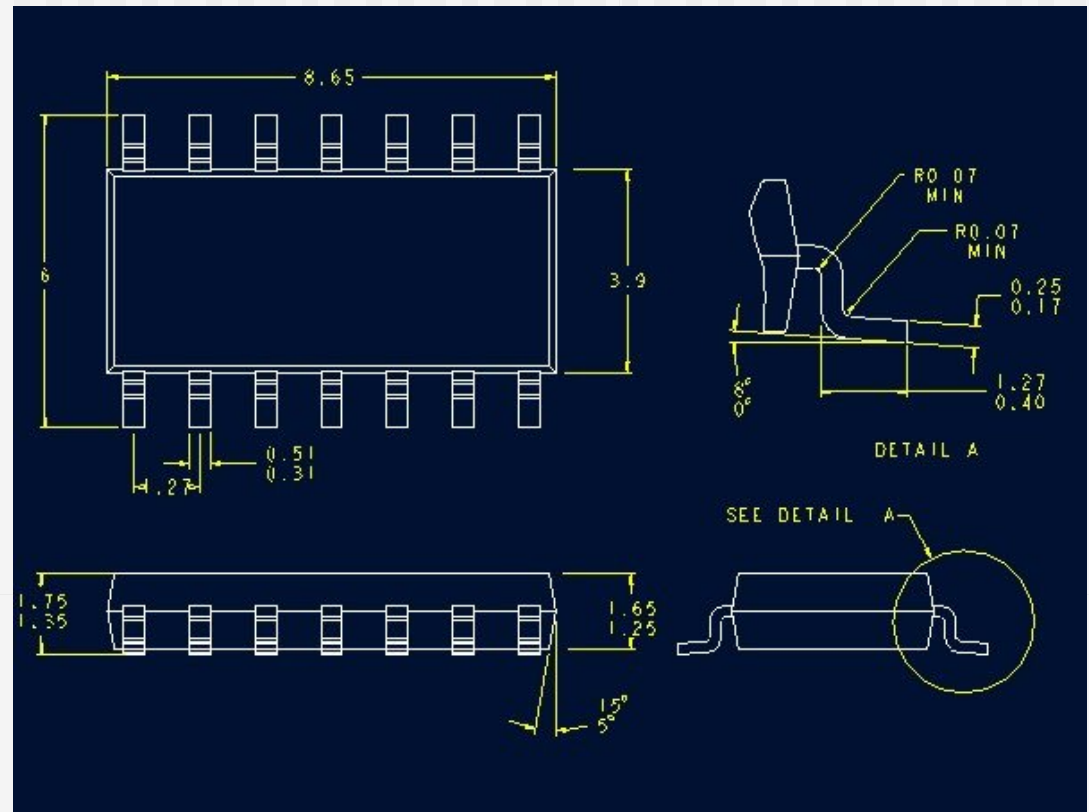
- Upload your first emn file
- The tool will generate a table with a line item for each unique combination of the "ecad_name" and "ecad_alt_name". Quantity will be listed as well.
- If working with multiple emn files, continue uploading your remaining files.
- If you have an existing ecad_hint.map, upload it as well.
- Choose replacement 3D models for each item by either:
 1. Clicking the magnifying glass and searching our library.
 2. Typing in a Pro/Engineer model name.
- Generate and download your ecad_hint.map file
- Download the listed components that you have chosen from our library.

The Simplified Solutions Library

A dimensioned print similar to the one to the right is viewable for all components

All Components are in the following formats:

- Pro/Engineer .prt
- STEP
- IGES
- EMN Component Outline (for import to ECAD packages)



Config.pro Settings

- Add the following settings to your config.pro prior to importing your IDF emn file into Pro/Engineer.

ecad_comp_csys_def_name cs_ecad_comp

ecad_mapping_file “Choose the location of your mapping file”

Importing your PCB into Pro/E

- Verify that the appropriate config.pro settings are active in your session.
- **Import the PCB into Pro/Engineer** by using the following commands:

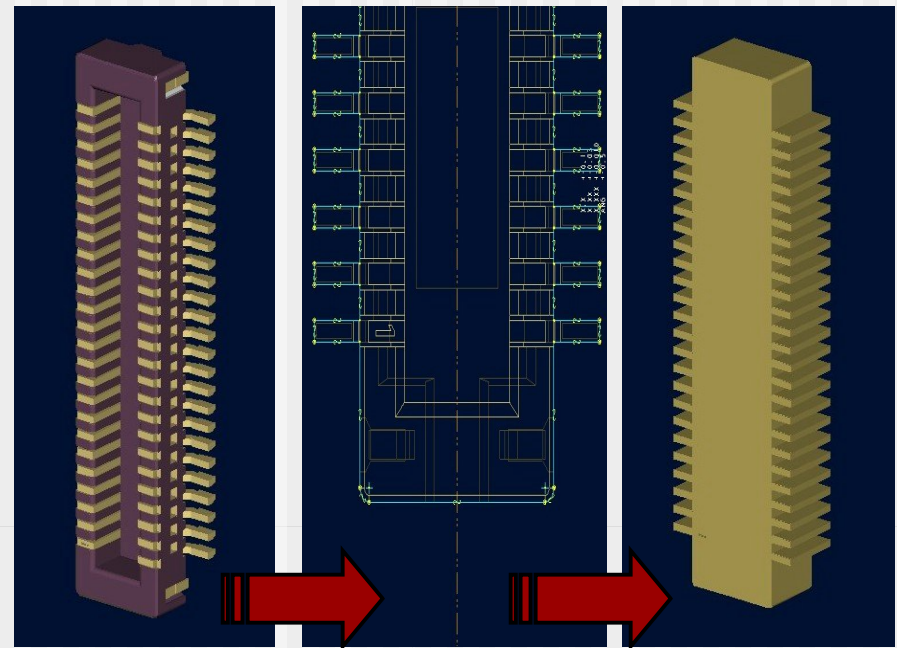
File>Open>Choose the proper EMN File>Open>

Choose Assembly Button and enter Assembly Name>

OK>Choose the proper EMP File>Open

Generating ECAD Component Outlines

- Create a protrusion using the surface that contacts the PCB as the sketching plane
- Choose all external edges of the component as seen from the top view.
- Extrude the protrusion to the top surface of the component
- Choose File, Save A Copy, select ECAD IDF (*.emn), and save the file.
- Import the EMN file into your ECAD software. Note: The EMN file will not export if there is not a consistent cross section at every Z height.



Suggested Design Process for PCB Development:

- The mechanical engineer creates the PCB board outline in Pro/ENGINEER and exports the file as an IDF EMN file.
- The PCB designer imports the PCB outline EMN file into the electrical design tool, places critical mechanical parts outside the PCB profile, and exports an EMN file.
- The mechanical engineer imports the EMN file back into Pro/ENGINEER, moves incorrectly placed components to the proper locations, and exports an EMN file.
- The PCB designer imports the EMN file, completes circuit layouts, etc., and exports an EMN file.
- The mechanical team imports the EMN file and performs analyses, including interference checks.
- The mechanical team and PCB designer go through an iterative process, checking and moving components until the PCB is ready for release.