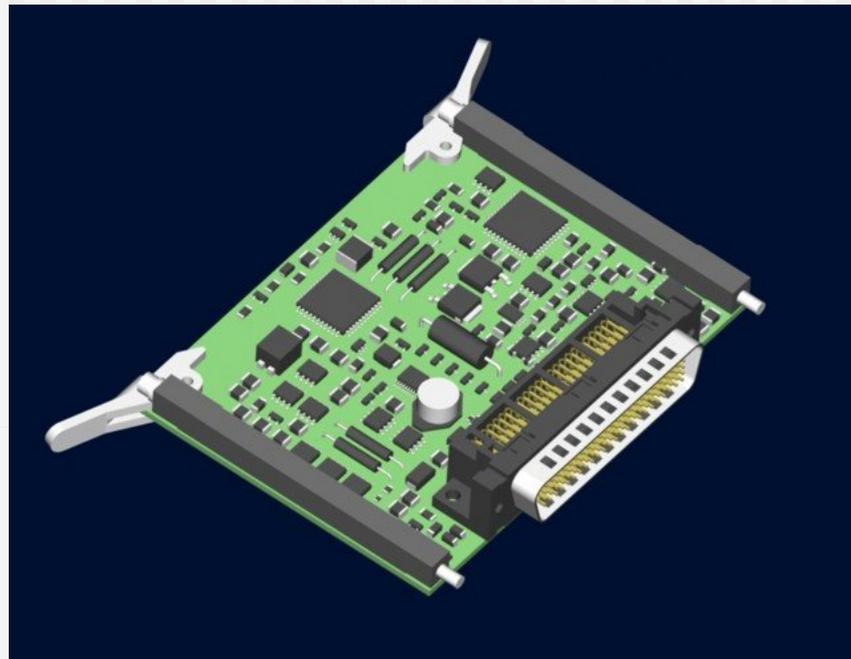# Creating a 3D PCB in Pro/Engineer

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.
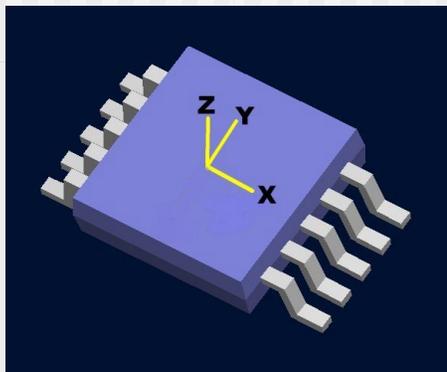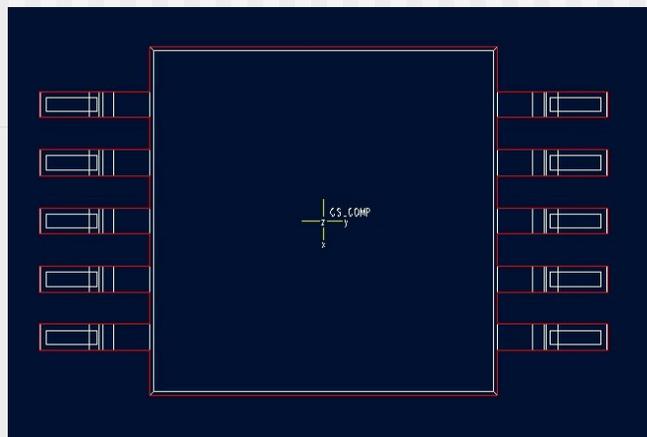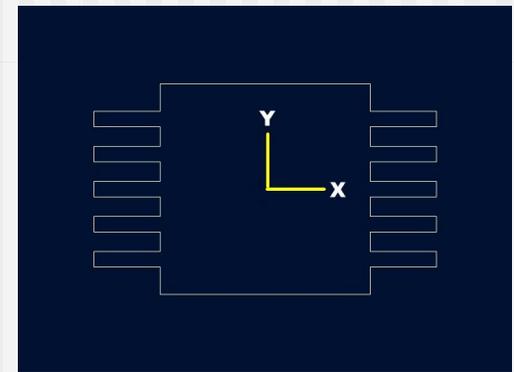
Steps to create your PCB using Simplified Solutions' Products

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.
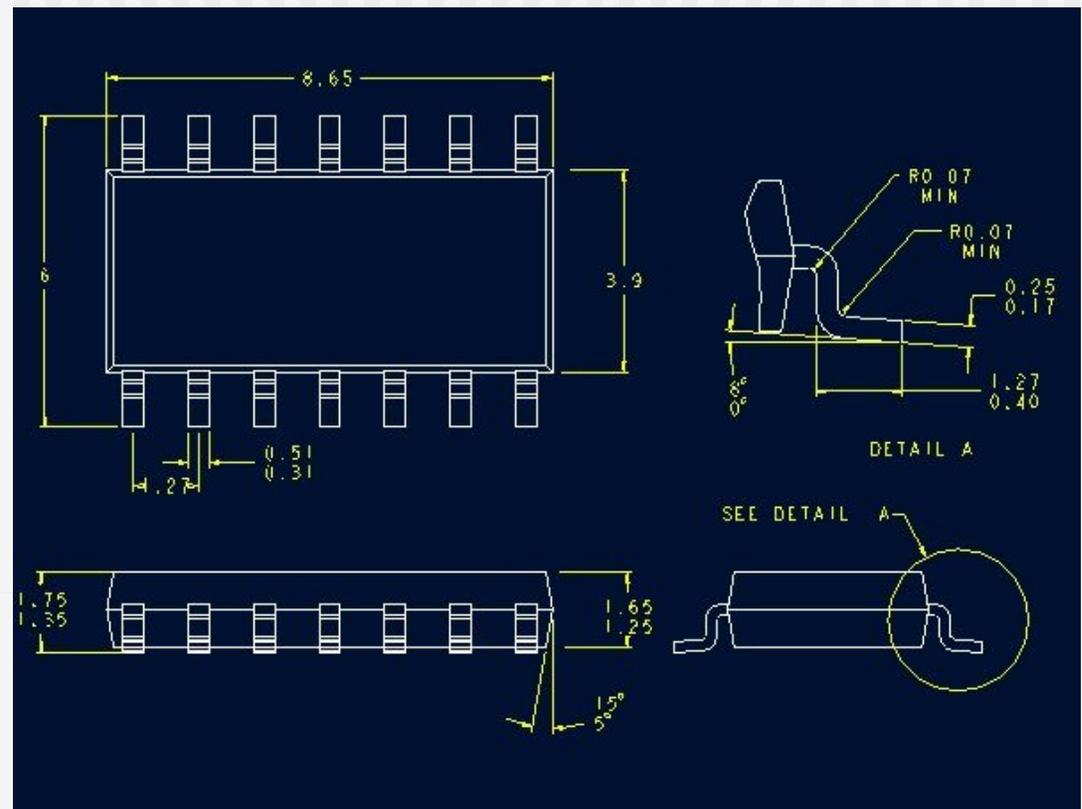7. Import your PCB into Pro/Engineer.

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.
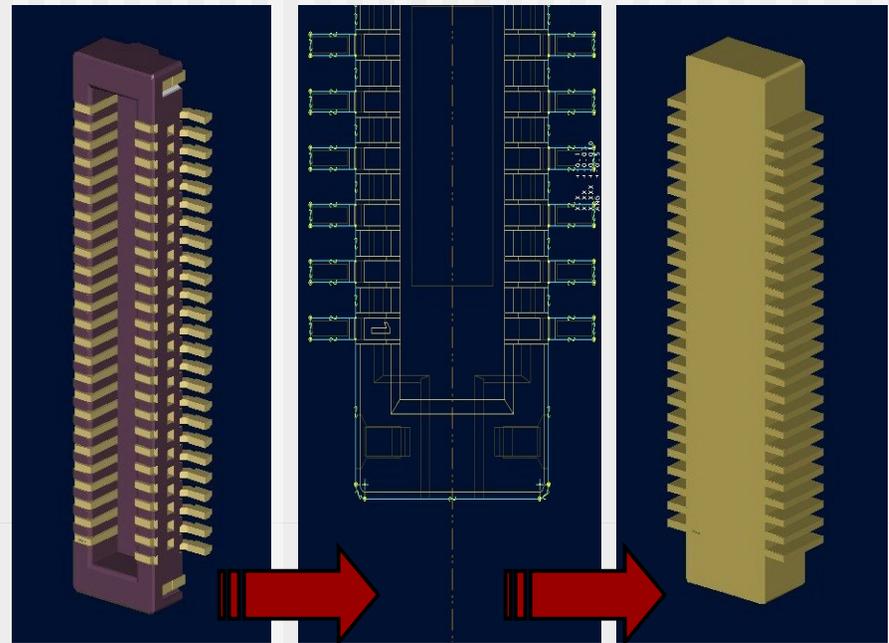
Suggested Design Process for PCB Development:

- 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.