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Pro/ECAD™
Help Topic Collection

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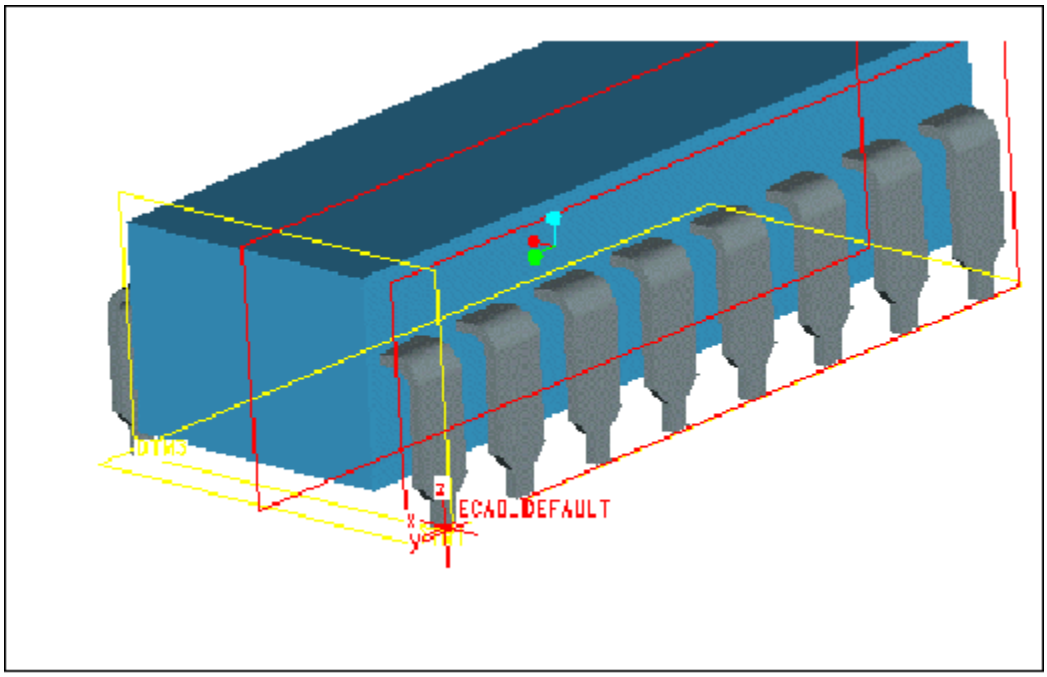
Pro/ECAD

Using Pro/ECAD

About Pro/ECAD in the Design Process

Pro/ECAD lets you import data for printed circuit boards and PCB components into a Pro/ENGINEER assembly. After importing, you can view the board and parts in a 3D context, for example, the PCB within the metal case that contains it.



Alternatively, the board outline dimensions can be established in Pro/ENGINEER, based on the case assembly, and exported for parts placement and routing in the ECAD package.

With Pro/ECAD, you can edit certain physical aspects of the board, for example, component positions, keepout areas or mounting holes, and export the edited board back to the original ECAD package.

You can also:

- Use custom-designed component models, based on part outlines imported from your ECAD library, that represent the 3D shape of components more accurately.
- Use Pro/ENGINEER Assembly commands (or information from the ECAD package) to pre-place certain critical components.

Note: Pro/ENGINEER does not import trace or connectivity data from the ECAD package. It deals with the three-dimensional shape and placement of components only.

Suggested ECAD-MCAD Design Sequence

To best integrate Pro/ENGINEER with the ECAD design process, use a sequence like this:

1. **Create 3D parametric models of components**—Import each component outline that you will use from the ECAD system library into Pro/ENGINEER, and use it as a basis to create a more detailed 3D model as a part file. When you import the board outline and placement information into an assembly, you can substitute these parts for those that would automatically be created on import.
2. **Parametrically design the board outline**—(complete with cuts, mounting holes, and keepin and keepout areas) as a part file. You can also define keepin or keepout areas for export to the PCB designer.
3. **Create an assembly and place the board part into it**—Using the 3D component models, place parts whose placement should be determined by heat or physical obstruction considerations.
4. **Export the assembly as board outline and component placement information**—The keepin and keepout outlines are also exported with this information. For export, select one of the available ECAD formats.

When the PCB designer has completed placement, you can import all component placement information into Pro/ENGINEER to check for interference between components and other physical obstructions in the assembly.

Use Pro/ENGINEER to make any necessary component placement corrections. Export the board back to the PCB designer to demonstrate the new placement.

Finally, perform a final mechanical analysis of the completed board in Pro/ENGINEER.

Note: The transfer of electrical routing information on the board is not supported by the IDF standard at this time.

About Setting Defaults for Faster Import

Click **Tools** > **Options** and use the **Options** dialog box to set defaults for the import process. When you set these defaults, you are not prompted for values each time a part is imported.

Component height—Ideally, the ECAD system specifies component height in its part information. If it does not, then when you import components through Pro/ECAD, you can use the `ecad_default_comp_height` configuration option to assign a default height to components with a zero height value in the interface file. The value for the option should be the value of the default height and its corresponding units.

Coordinate system names—When the board and components are imported, a csys is created at the origin of each part that is created. Use the `ecad_board_csys_def_name` and `ecad_comp_csys_def_name` configuration options to assign a default csys name to each object type.

Default part and assembly templates—During import, you are prompted to choose a template for the imported part or assembly. You can specify a default

template using the `template_ecadpart` and `template_ecadasm` configuration options.

`ecad_hint.map` **file default location**—You can use the `ecad_mapping_file <path>` configuration option to set a default location for the `ecad_hint.map` file.

Import parts to automatically created layers—When the Preferences option `ecad_comp_layer_map` is set to `YES`, the import routine creates a new layer for each component name and places each newly created reference designator on the appropriate layer.

About Fixing Import Problems in Sketcher

When you import parts created in an ECAD (2D) system into Pro/ENGINEER, they may contain bad contours caused by open or self-intersecting loops.

Select **Sketcher Fix** in the **Import ECAD Board File** dialog box to correct the import file.

If the ECAD import process is unable to create a contour for a board outline, the following message appears:

```
Do you want to correct <file name> part geometry with Sketcher? [Yes]
```

You can either abort the import or enter the Sketcher mode to fix the problem. In the Sketcher mode, you can see the imported profile geometry. The sketcher regeneration process ensures correct creation (regenerated or nonregenerated section).

You can create dimensions on the section. All tools in Sketcher are available to correct any problem, for example, to trim corners.

After you have corrected the problems in Sketcher, Pro/ENGINEER continues the import and automatic creation of the feature. The following is written to the `ecad.log` file:

```
If you plan to import this component again, you should export it and
then insert the corrected outline as a replacement for the problem
outline in the import files.
```

Note: If the outline is corrected on import, it is not automatically corrected in the ecad library file. Export the corrected component separately and use the outline to update the `.emp` file to avoid having to correct the same profile repeatedly.

Tip: The ECAD Log File

Pro/ECAD interface routines write commentaries, warnings, and error messages to a file named `ecad.log` in your current working directory. Read this file each time you use the Pro/ECAD interface.

Tip: The Z-Axis Offset in IDF

When you use Pro/ECAD to import components into Pro/ENGINEER using IDF 2.0, the system places components with a z-axis offset of 0.0. After import, you can modify the z-axis offset of the component's placement.

In IDF 3.0, you can define an offset.

Tip: Parametric Dimensions in Pro/ECAD

The dimensions of parts created in ECAD and imported by Pro/ENGINEER are not parametric, and you cannot modify them. However, you can go into the Sketcher mode and add dimensions by redefining the part base geometry feature. To maintain maximum flexibility during the design process, you can create your board outlines and components in Pro/ENGINEER.

After you export parts created in Pro/ENGINEER through Pro/ECAD, you lose parametric associativity permanently. If you import them back into Pro/ENGINEER later, the outline is not updated.

Model Parameters

The following table lists the model parameters along with additional information to determine when and how you can use them.

Parameter	Optional Value	Valid Format	Description
ECAD_OWNER	ECAD MCAD UNOWNED	IDF 3.0	Shows the owner of board or other outline.
ECAD_PART_TYPE	Panel Board Electrical Mechanical Other Outline	IDF 2.0 and IDF 3.0	Shows the component type.
ECAD_PART_NAME	any	IDF 2.0 and 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_THETA_JC			
ECAD_POWER			

Feature Parameters

The following table lists the feature parameters along with additional information to determine when and how you can use them.

Note: All hole parameters are automatically created during import only if you import the holes as features. Set the `ecad_import_holes_as_features` configuration option to `yes` to import holes as features.

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 and 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 area.

ROUTING_LAYERS	TOP BOTTOM BOTH, ALL	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 IDF standard.)
ROUTING_LAYERS	TOP BOTTOM BOTH, ALL INNER	IDF 3.0	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 IDF standard.)
COMPONENT_GROUP_NAME	Any	IDF 2.0 and IDF 3.0	Allow adding or keeping a name for place region ECAD area feature.
ECAD_ASSOCIATED_PART	BOARD NOREFDES	IDF 2.0	In the ECAD system, this is the part to which the Pro/ENGINEER hole feature relates.
ECAD_ASSOCIATED_PART	BOARD NOREFDES PANEL Reference Designator	IDF 3.0	In the ECAD system, this is the part to which the Pro/ENGINEER hole feature relates.

Component Feature Parameters

This table lists the feature parameters in Assembly mode for components, along with additional information to determine when and how you can use them.

Parameter	Optional Value	Valid Format	Description
ECAD_NAME	Any	ALL	The ECAD system name of the component (becomes the name of the Pro/ENGINEER .prt file.)
ECAD_ALT_NAME	Any	ALL except Allegro	The ECAD system alternative name for the component.
ECAD_REF_DES	Reference Designator	Any	The ECAD reference designator for the component.
ECAD_PART_TYPE			
ECAD_PART_NAME			
ECAD_PLACEMENT_STATUS	FIXED PLACED UNPLACED	IDF 2.0	Shows the placement type of the component.
ECAD_PLACEMENT_STATUS	PLACED UNPLACED ECAD MCAD	IDF 3.0	Shows the placement type of the component.

To Perform a Clearance-Interference Check

Click **Analysis** > **Model Analysis** and use the **Model Analysis** dialog box to run a Volume Interference check to detect whether a Pro/ECAD component interferes with or falls entirely within a closed quilt representing a Pro/ECAD keepin or keepout area.

Configuring Pro/ECAD

About Configuring ECAD

You can set environment options and other global settings by specifying the required settings in a configuration file. To set configuration options, use the **Options** dialog box.

This help module contains a list of configuration options in alphabetical order. Each option contains the following information:

- Configuration option name.
- Default and available variables or values. The default values are in italics.
- Brief description and notes describing the configuration option.

To Set ECAD Configuration Options

1. Click **Tools** > **Options**. The **Options** dialog box opens.
2. Click the **Show only options loaded from file** check box to see currently loaded configuration options or clear to see all configuration options.
3. Select the configuration option from the list or type the configuration option name in the **Option** box.
4. In the **Value** box, type or select a value.

   Note: The default value is followed by an asterisk (*).
5. Click **Add / Change**. The configuration option and its value appear in the list. A green status icon confirms the change.
6. When you finish configuring, click **Apply** or **OK**.

dazix_default_placement_unit

micron, mm, thou

micron—Specifies the units to be used for data imported in Dazix files.

dazix_export_mounthole

yes, *no*

yes—Causes the MOUNTHOLE section of a Dazix file to be processed as a mount hole.

no—Causes the MOUNTHOLE section to be processed as a cut.

dazix_z_translation

yes, no

yes—Passes the objects in the .edn files through z translation.

ecad_area_default_import

cosm_area, 3d_volume

cosm_area—Treats imported ECAD areas as cosmetic area features.

3d_volume—Imports ECAD areas with z-heights as a 3D 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, you are prompted 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, you are prompted for a name.

ecad_comp_layer_map

yes, no

yes—Allows layer mapping for the ECAD component's import into Assembly.

ecad_create_hint_add

yes, no

Assists in the creation of the ecad_hint.map file.

yes—If required, automatically renames components, each time a library of component outlines is imported into Pro/ENGINEER. This does not create an ecad_hint.map file. It controls the creation of ecad_hint.add file.

ecad_default_comp_height

Sets the default value and units for an ECAD component that is 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 units of the current component are used.

ecad_default_comp_place_status

placed, unplaced, fixed, mcad, ecad

placed—Sets default component placement status for export (parameter setting overrides the default).

ecad_exp_both_two_areas

yes, *no*

no—Supports the export of ECAD areas with different "Above Board" and "Below Board" conditions.

yes—Enables you to export both-sided keepin or keepout ECAD areas as two individual areas (top and bottom).

ecad_export_holes_as_cuts

yes, *no*

no—Exports Pro/ENGINEER holes as cuts to ECAD systems.

ecad_import_holes_as_features

yes, *no*

no—Imports sections specified as DRILLED_HOLE as through-all holes. Boards created with Pro/ENGINEER drilled holes export with a 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_import_relative_accuracy

0.0012

Specifies the relative accuracy value to be used during import of ECAD files.

ecad_mapping_file

Specifies that the ecad_hint.map file will be used for ECAD operations.

ecad_missing_component_status

keep_missing, delete_missing

keep_missing—Missing components are kept by default.

delete_missing—Missing components are deleted by default.

Set up the default status for missing components using the **Component Investigate** dialog box. Using this dialog box, you can keep missing components by default or delete missing components by default.

ecad_pin_hole_import

no, yes

Controls the default setting for the hole filter of type PIN for IDF 3.0 files.

yes—PIN holes within in the IDF 3.0 file will be imported.

no—PIN holes within the IDF 3.0 file will not be imported.

ecad_via_hole_import

yes, no

Controls the default setting for the hole filter of type VIA for IDF 3.0 files.

yes—VIA holes defined in the IDF 3.0 file are imported.

no—VIA holes defined in the IDF 3.0 file are not imported.

ecad_other_outl_csys_def_name

Specifies the default coordinate system name for the OTHER_OUTLINE section of an IDF 2.0 ECAD component being imported. If you set this variable, you are prompted for a name.

mentor_ver_2_0

yes, no

yes—Exports the file in IDF 2.0 format.

no—Exports the file in IDF 1.0 format unless you select IDF 2.0 from the ECAD FORMAT menu.

template_ecadasm

Specifies that the model will be used as the default ECAD assembly template.

template_ecadpart

Specifies that the model will be used as the default ECAD part template.

Importing ECAD Databases

About Importing ECAD Databases

Board Assembly Files and Library Files

The import routine requires two reference files, the board file and a library file. The board file describes the board outline, thickness, any keepin or keepout areas, and

the placement positions of all the parts. The library file contains the footprint and height information for all parts referenced in the board file.

In the IDF format, the board file has an `.emn` extension while the library file uses the `.emp` extension. Proprietary formats from other vendors use their own extensions. During the import, you are prompted for the location of both these files.

Note: When importing a panel assembly, specify the board file (IDF `*.emn`) instead of `*.emp` when prompted for the library file.

Automatic or Partial Assembly on Import

You can import the whole database, including the board, all placed components, and keepin or keepout outlines in an automated sequence that produces a completed board assembly. Alternatively, you can choose to import the board and parts separately and assemble them in the normal way. If you choose the automated sequence, you are prompted for any part library files or default dimensions needed during the sequence.

Automatic Part Creation

When the parts list and placement information are read in, Pro/ECAD uses the 2D part outlines it detects to extrude each part into a 3D shape. During the import, if no defaults are specified in your `config.pro` file, you are prompted for a height or for a coordinate system name for the part being created. Alternatively, you can reference a library of custom-made parts that you have prepared in Pro/ENGINEER that show more detail than a simple extrusion.

As each part is created in 3D and added to the assembly, it is saved as a separate `.prt` file in the working directory. The original component name, for example, `SN74ALS133N`, becomes the file name. One `.prt` file is created for each reference designator. All reference designators appear in the Model Tree.

Custom Part Substitution

As an alternative to letting Pro/ENGINEER create the parts automatically, you can create a library of more accurately-shaped custom parts and reference it when you import the PCB. You use an ASCII map file named `ecad_hint.map` to reference the custom parts as substitutes for the automatically created ones.

Automatic Layer Assignment

When the preferences option, `ecad_comp_layer_map`, is set to `yes`, the import routine creates a new layer for each component name and places each newly created reference designator on the appropriate layer.

If you are referencing an `ecad_hint.map` file, you can add lines to the file to direct layer creation and part-layer assignment.

Additional Supported Objects for Import-Export

In addition to the board and electrical components, Pro/ECAD also imports and exports the following objects in accordance with the IDF 3.0 specification:

Panel Outline—The manufacturing step-and-repeat panel and boards mounted on it. The panel outline is described in an optional separate panel file which can reference one or more board assemblies described in separate files. Any component placed on the panel itself is referenced in a library file.

Other Outline—Extruded shapes of nonelectrical parts that would not have a reference designator, such as a heat sink.

Holes—Drilled holes

Keepin and Keepout areas—Separate outlines can show where to place or prohibit parts, routing, or vias.

For complete information on the IDF import file conventions, see the Intermediate Format Specification Version 3.0.

Using Automatic Layer Assignment

When the `ecad_comp_layer_map` preferences option is set to `yes`, the import routine creates a new layer for each component name and places each newly created reference designator on the appropriate layer.

If you are referencing an `ecad_hint.map` file, you can add lines to the file to direct layer creation and part-layer assignment.

New layers are named after the component name, or for `.map` file references, the mapped name. The syntax is `ECAD_<comp name or mapped name>`.

If no `ecad_hint.map` file exists, Pro/ECAD automatically creates a separate layer for each component type that is imported and places the component or components on that layer. Components with the same name are placed on the same layer.

Automatic Layer Assignment Example

For a `*.emn` file containing the following placement record:

```
 PLACEMENT
CC1206-1 151K41000S C1
1200.00000 1550.00000 90.00000 TOP PLACED
CC1206-2 151K31001S C2
1200.00000 550.00000 90.00000 TOP PLACED
CC1206 151K41000S CC1
1900.00000 1000.00000 90.00000 BOTTOM PLACED
RC1206 161F21050S RR1
1200.00000 1000.00000 270.00000 BOTTOM PLACED
SO14 104F50074S U1
1400.00000 1400.00000 0.00000 TOP PLACED
SO14 104C50000S U2
```

```
1400.00000 400.00000 0.00000 TOP PLACED
SO14 104C50004S UU1
1700.00000 900.00000 0.00000 BOTTOM PLACED
.END_PLACEMENT
```

This layer configuration is produced:

Layer Name	Components on this layer
ECAD_CC1206-1	CC1206-1
ECAD_CC1206-2	CC1206-2
ECAD_CC1206	CC1206
ECAD_RC1206R	C1206
ECAD_SO14	SO14 (u1, u2, uu1)

Using the ecad_hint.map File

Using the `ecad_hint.map` file, you can specify an overriding mapping (over the automatic mapping) to produce a new `ECAD_<mapped name>` layer. The component is placed in that layer.

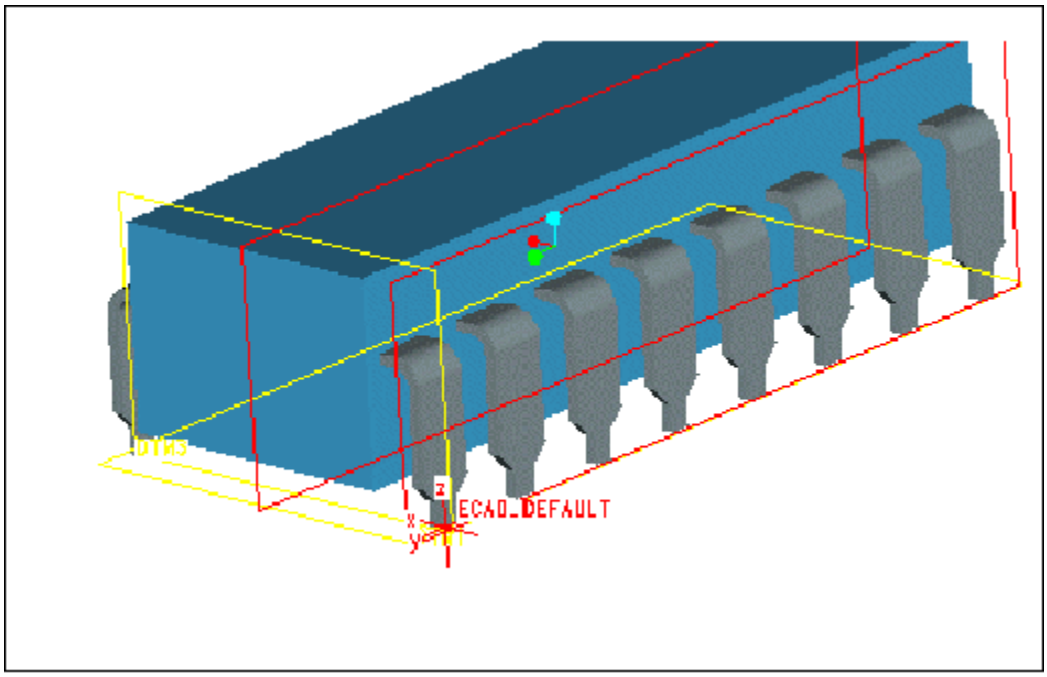
```
map_objects_by_name->
ECAD_NAME "eesmb"
ECAD_ALT_NAME "DM7442N"
ECAD_TYPE ""
MCAD_NAME "EESMB_DM7442N"
MCAD_TYPE "part"
MCAD_LAYER "<ANY>"
END
```

Using Coordinate Systems

If you create your own component models, you must provide each one with a coordinate system that represents the part origin to the ECAD package, and that includes the height dimension along the z-axis. If you are creating the component as a custom part in Pro/ENGINEER, start the part by placing the (`0,0,0`) of the coordinate system at the intersection of the three default datum planes.

As Pro/ECAD places components by locating component coordinate systems in relation to the board coordinate system, the coordinate system orientation you use for placement on the ECAD outline should match that of the custom Pro/ENGINEER part. If the replacement is an assembly, an assembly coordinate system must be present. Incorrect orientation results in incorrect placement of the replacement component.

The illustration below shows a custom part, created in Pro/ENGINEER, representing a DIP16 with the csys ECAD_DEFAULT located at the part origin (usually pin 1).



In IDF 3.0, the components can be placed using the **Mate Offset** assembly option. For all other formats, the system mates the components to the board.

You can create a section plane for the component outline as a cross section, or select a planar surface for the component outline. The section plane is used as the footprint of the extents of the part and must be parallel to the xy-plane of the placement coordinate system. You are prompted to identify the section plane as part of the export procedure.

Orient the component so that the z-axis of its chosen coordinate system points in the direction normal to the surface of the board, away from the surface:

- If you place a component on the top surface of the board, the direction of the z-axis of the component's coordinate system points in the same direction as the z-axis of the board's coordinate system.
- If you place a component on the bottom surface of the board, its z-axis points in the opposite direction from the board's z-axis.

You should know how the default component coordinate systems on the parts in your ECAD package are positioned.

Top and Bottom Side

Pro/ENGINEER uses the coordinate system to distinguish between top and bottom surfaces:

- The z-axis of the coordinate system of the board should be normal to the top and bottom planes.
- The positive z-axis should point from the bottom plane to the top plane.

You can select or create any coordinate system where the z-axis is normal to the top and bottom planes and the origin is located on either the top or the bottom surface of the board part.

Note:

- Pro/ENGINEER does not support left-handed coordinate systems. If used, component placement will not be as expected.
- As the direction of the z-axis is normal to the top and bottom, and also points toward the top, the bottom and top can be different sides of the same surface.
- When importing or exporting information that depends on the identification of the top and bottom surfaces of the board, the system checks to see if two parallel planes are normal to the z-axis. If so, it designates these the top and bottom based on the direction of the z-axis.
- If more than two planes are normal to the z-axis, the system prompts you to select a top and a bottom. The bottom surface can be a surface or datum. The top can be:
    - A surface or datum (if importing)
    - A surface or datum with a cross-section on it (if exporting)

Naming Default Coordinate Systems

During import or export, when Pro/ECAD needs a coordinate system, it first searches for a default coordinate system. Pro/ECAD automatically looks for a coordinate system with the default name, `ECAD_DEFAULT`. Pro/ECAD prompts you if it cannot find one, or if you have not specified another default name.

To specify a default name of your own, set these configuration options:

- `ecad_board_csys_def_name`—Establishes a default name for the board's coordinate system.
- `ecad_comp_csys_def_name`—Establishes a default name for the coordinate systems that components use.

If a default coordinate system exists, Pro/ENGINEER uses it automatically, without requesting any additional information about coordinate systems. If you use the default coordinate system name for all components, you can assemble all necessary components to the board without specifying any additional component placement information.

The part's outline is imported as an extruded section to create a Pro/ENGINEER part when you either place the component for the first time, or when you import it separately as a new part. The default coordinate system is located on the plane of the extruded section, and the z-axis is parallel to the direction of the extrusion.

To Import a PCB or Panel Assembly

To import the whole assembly from either a board or a panel file:

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select the ECAD board or panel file to import (IDF `*.emn`).
3. Click **Open**. The **Import New Model** dialog box opens.
4. Under Type, select **Assembly**. The **Include ECAD Import Dialog** check box is available for selection.
5. If you want to use the **ECAD – Import** dialog box, select the **Include ECAD Import Dialog** check box and click **OK**. The **ECAD – Import** dialog box opens.
6. Click **Yes** if you want to edit the board in Sketcher.
7. The **Use Default Template** check box is selected by default. Clear the **Use Default Template** check box to select from available templates. A new assembly is created with the board imported as the first part. If you reference a panel file, the panel is imported as the first part.

   If any components in the assembly are missing, Pro/ENGINEER prompts you to select a library file (IDF `*.emp` file). (For a panel assembly, specify the board file (IDF `*.emn`) instead of the `*.emp`).

   If the board data contains drilled hole information, under **Options**, the **Hole Import** list displays the hole information. By default, all hole types, except `PIN` are selected.
8. Modify hole import selection as required.
9. Click **OK**. The **Ecad Import** dialog box opens.
10. Click **OK**. The PCB components are imported and assembled in the panel.

To Import a Board Outline

Use this procedure to import a board, with holes and keepin or keepout areas, but minus the components, as a Pro/ENGINEER part. You must have an output file from one of the supported PCB design systems, or an IDF file of your PCB layout ready to read into Pro/ENGINEER.

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select the ECAD package from the **Type** list.
3. Select the ECAD panel file that you want to import from the list of available files.
4. Click **Open**. The **Import New Model** dialog box opens. Under **Type**, the **Part** check box is selected by default.
5. Accept the default name or type a name for the new part in the **Name** box.
6. Click **OK**. The **ECAD - Import** dialog box opens.

The import list shows only one part type, that is, the board, though the import file contains data of the board and all parts. Import the board as a part first and create an assembly using the board as the base part. Import the components into the assembly.

7. Click **Yes** if you want to edit the board in Sketcher.
8. The **Use Default Template** check box is selected by default. Clear the **Use Default Template** check box to select from available templates. A new assembly is created with the board imported as the first part. If you reference a panel file, the panel is imported as the first part.

   If any components in the assembly are missing, Pro/ENGINEER prompts you to select a library file (IDF *.emp file). (For a panel assembly, specify the board file (IDF *.emn) instead of the *.emp).

   If you are using a format that complies with the IDF 3.0 standard and the board data contains drilled hole information, under **Options**, the **Hole Import** list displays the holes information. By default, all hole types, except PIN are selected.
9. Modify the hole import selection as required.

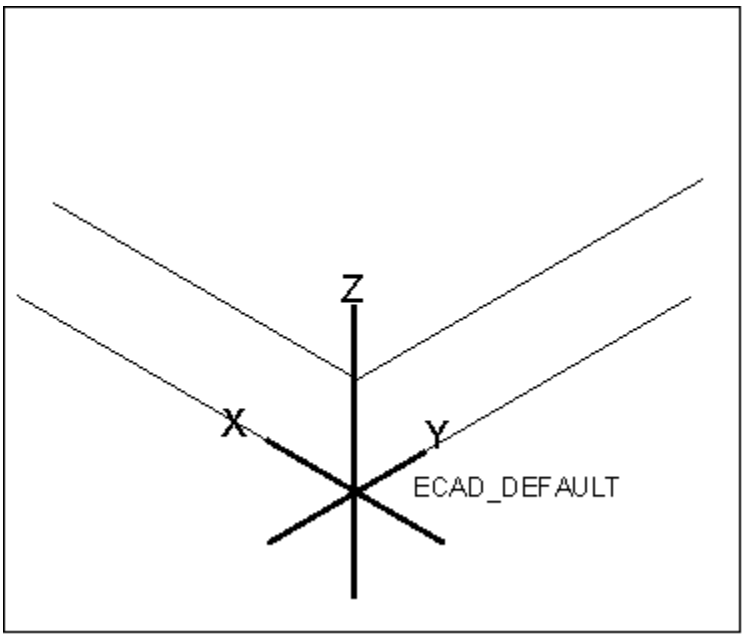
   Note: Only IDF 3.0 supports hole filter.
10. Click **OK**. The data is imported and converted to a Pro/ENGINEER part.
    - A coordinate system is created at the ECAD_DEFAULT origin.
    - The z-axis direction is bottom to top.

    Note:

    Use the ecad_board_csys_def_name configuration option to specify a default name for the board csys.

    Create an assembly with this part. Use **File** > **Open** to add the parts from the same input file.

The illustration below shows the ECAD_DEFAULT csys with (0,0,0) at the bottom of the board. The z-positive direction is bottom to top.



Note: An ECAD import, unless performed through **Sketcher Fix**, produces sections without dimensions (nonregenerated sections). Therefore, the dashboard is not available when you redefine extrusions that are created during such an import. Instead, the redefinition menu from Pro/ENGINEER Release 2001 opens. The dashboard is available only after you regenerate the section.

To Import a Panel Outline

Note: Only IDF 3.0 supports panel import.

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select the ECAD package from the **Type** list.
3. Select the ECAD panel file that you want to import from the list of available files
4. Click **Open**. The **Import New Model** dialog box opens. Under **Type**, the **Part** check box is selected by default.
5. Accept the default name or type a name for the new part in the **Name** box.
6. Click **OK**. The **ECAD - Import** dialog box opens. The import list shows only one part, that is, the panel.
7. Click **Yes** if you want to edit the board in Sketcher.
8. The **Use Default Template** check box is selected by default. Clear the **Use Default Template** check box to select from available templates. A new assembly is created with the board imported as the first part. If you reference a panel file, the panel is imported as the first part.

If any components in the assembly are missing, Pro/ENGINEER prompts you to select a library file (IDF *.emp file). (For a panel assembly, specify the board file (IDF *.emn) instead of the *.emp).

If you are using a format that complies with the IDF 3.0 standard and the board data contains drilled hole information, under **Options**, the **Hole Import** list displays the holes information. By default, all hole types, except PIN are selected.

9. Modify the hole import selection as required.

   Note: Only IDF 3.0 supports hole filter.

10. Click **OK**. The panel is imported and converted to a Pro/ENGINEER part.

    - A coordinate system is created at the ECAD_DEFAULT origin.
    - The z-axis direction is bottom to top.

    Note: Use the ecad_board_csys_def_name configuration option to specify a default name for the board coordinate system of the board.

To Import Components as Parts

Use this procedure to create a .prt file for a single component or to bring .prt files for all components into session.

1. Click **File** > **Open**. The **File Open** dialog box opens.

   - Select an ECAD Lib package from the **Type** list, click the ECAD file that you want to import from the list of available files, and click **OK**.
   - A new part file opens. The **Import ECAD Library File** dialog box also opens.

2. Use the dialog box to select all components, or a single component. If you select a single component, you can select **Sketcher Fix** to open the part outline in Sketcher. You must know the exact name of the part as it appears in the import file to specify it in the **Component Name** box. Otherwise, the part is automatically extruded into 3D.

3. Click **OK**.

   - If you selected a single component, it is displayed as a .prt file.
   - If you selected **All Components**, the .prt files are stored in session, but are not stored on disk or displayed. Use **File > Open** and select **In Session** to select the file or files to work on.
   - A coordinate system named ECAD_DEFAULT is created at the part origin. The z-axis direction is bottom to top.

     Note: Use the ecad_comp_csys_def_name configuration option to specify a default name for the board csys.

To Import Components Into an Assembly

Use this procedure to import ECAD components after the PCB outline has been imported or created and opened as the base part of an assembly.

1. In an assembly file, click **Insert** > **Shared Data** > **From File**. The **Open** dialog box opens.
2. Click the ECAD package from the **Type** list, and choose the ECAD file that you want to import from the list of available files. (IDF = `*.emp`)
3. Click **Open** to open the new file. Choose from the following check list from the **IMPORT TYPE** menu:
   - **Components**—Imports all component outlines.
   - **Placement**—Imports component placement information.
   - **Other Outline**—(ECAD IDF only) Selects the `.OTHER_OUTLINE` sections to import from the IDF file. ("Other outlines" are volumes representing non-electrical items that would not have reference designators.) If you select this option, a separate menu opens, enabling you to choose the `.OTHER_OUTLINE` sections you want to import.
   - **Investig Plc**—(ECAD IDF only) Imports placements in investigate mode. This option lets you selectively accept or reject the placements of new or changed components. You must select **Placement** to use this option.
4. Use **GET COORD S** or **SEL COORD S** to create or select a coordinate system if you did not specify one for the board in your configuration file. You should not use **Default** to create a coordinate system, because that coordinate system might not comply with restrictions that pertain to the top and bottom board surfaces, or have the correct orientation.
5. The **Open** dialog box opens with the library format for the ECAD type you are importing in the **Type** box. Choose the ECAD Lib file that you want to import from the list of available files.
6. Click **Open** to open the file and start the import process.
7. If the file you import contains component placement records, use **GET COORD S** or **SEL COORD S** to create or select a coordinate system for each component, unless you specified a default component coordinate system in the configuration file.

   Pro/ENGINEER imports the components. If it also imported placement information, the system automatically places the components on the board.

   Saving the assembly also saves component placement information when both the component outlines and the placement information are present. If only one is present, you must retrieve each component separately and save it later when you retrieve the assembly.

To Substitute Custom Pro/ENGINEER Parts

To import using custom Pro/ENGINEER parts, place the name of the replacement Pro/ENGINEER component in the `mcad_name` section of the `ecad_hint.map` file.

- If changes are necessary during import, the system specifies any additions in a file named `ecad_hint.add`.
- Specify both `ECAD_NAME` and `ECAD_ALT_NAME` when replacing components with `ecad_hint.map`.

To Import a Non-Electrical Volume

An other outline is an extruded footprint that represents a solid part, such as a heatsink, that would not have a reference designator. It is used only in the IDF 2.0 and 3.0 transfer formats.

To import an other outline volume to an assembly:

1. Click **Insert** > **Shared Data** > **From File**. The **Open** dialog box opens.
2. Select the `.emn` file or file with Other Outline information and click **Open**.
3. In the **Menu Manager**, select **Other Outline**.
4. Click **Done**. You are prompted to select a coordinate system. When you specify a coordinate system, the other outline volume is assembled to the board.

To Specify Parts as Electrical or Mechanical

1. In a part file, click **Tools** > **Parameters**. The **Parameters** dialog box opens.
2. Click  and create a string parameter, `ECAD_PART_TYPE`, with a value of either `ELECTRICAL` (electrical component) or `MECHANICAL` (mechanical component).

If no parameter is defined, the part is mechanical by default.

To Show or Hide Reference Designators

You can show or hide reference designators using the **Environment** dialog box.

1. Click **Tools** > **Environment**. The **Environment** dialog box opens.
2. Check the **Reference Designators** box to display the reference designators in the model.
3. Click **OK**.

To Assign or Rename a Reference Designator

Use this procedure to assign a reference designator to components that you design and position with Pro/ENGINEER.

1. In **Assembly** mode, click **Edit** > **Setup** > **Name** > **Component**.
2. Select the component whose name you want to change.

3. At the prompt, type a new name.
4. Click the check box next to the prompt. The name of the reference designator is changed to the one you specify.

You can also assign or rename a reference designator using the Model Tree.

1. Click **Settings** > **Tree Columns**. The **Model Tree Columns** dialog box opens.
2. In the **Type** box, select **ECAD Params.**
3. Click **ECAD_REF_DES** from the list of available columns, then **>>** to move the selected column to the **Displayed** list.
4. Click **OK**. The reference designators appear in a column associated with each part in the Model Tree. Assign or rename the reference designator of a part by clicking the cell adjacent to the part.

When the system exports the component from the assembly, it passes on the new name to the receiving ECAD system.

Note:

- If you do not specify a reference designator for a new component that was placed in Pro/ENGINEER, and you use IDF to export the assembly, the neutral file uses the NONAME tag as the reference designator for that component. When you import the same neutral file back into Pro/ENGINEER, it does not recognize the component as the original. Instead, it uses the information in the neutral file to place a new component in the assembly. As you have not specified one, the reference designator for this new component is not present.
- You can rename a reference designator in **Assembly** mode but you cannot remove the text note entirely.

To Manipulate a Reference Designator

You can manipulate reference designators in:

- The **Assembly** mode
- The **Drawing** mode

In the Assembly Mode

1. Open the **Note** dialog box by doing one of the following:
    - From the **ASSEMBLY** menu, click **Edit** > **Setup** > **Notes** > **Modify**.
    - Right-click a reference designator note in the Model Tree and then click **Properties** from the shortcut menu.
    - Select a reference designator in the graphics screen, click the right mouse button and then click **Properties** from the shortcut menu.
2. Under **Name**, you can view the reference designator name.
3. Under **Style**, choose the text style.

4. Under **Placement**, click **Move** to change the location of the reference designator.
5. Click **OK**.

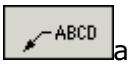
Note: You cannot modify the text of the reference designator.

You can also use the model tree to manipulate reference designators:

1. Click **Settings** > **Tree Filters**. The **Model Tree Items** dialog box appears.
2. Check **Notes** and click **OK**. Reference designators are displayed in the model tree just as 3D notes but with a different icon for identification.
3. Select a reference designator note in the Model Tree.
4. Right-click and select one of the following options and make the necessary changes:
    - **Add Link**
    - **Move**
    - **Erase**
    - **Text Style**
    - **Properties**
    - **Info**

    Refer to Pro/LEGACY for more information on these options. The **Add Link**, **Move**, **Erase**, **Text Style**, **Properties**, and **Info** options are also available when you select a reference designator in the graphics window and right-click.

In the Drawing Mode

1. Click **View** > **Show and Erase** > 

and then select the ECAD object, to display the reference designators of the object in the drawing.
2. Select a reference designator note.
3. Right-click and select one of the following and make the necessary changes:
    - **Erase**
    - **Edit Attachment**
    - **Move Item to View**
    - **Insert Jog**
    - **Arrow Style**
    - **Save Note**
    - **Move Special**
    - **Properties**

- **Add Hyperlink**

Refer to Pro/LEGACY for more information on these options.

Note: Insert Jog, **Arrow Style** and **Edit Attachment** are available on the menu if a leader is added to the reference designator note.

To Import Using Investigate Mode

In Assembly Mode

You can use the Investigate mode to examine final component placement before you complete the import procedure. In the investigate mode, you can accept or reject the placement of an existing component that is modified into your assembly. You can select **Investig Plc** only if you have selected **Placement** and only if you have selected ECAD IDF as your import format.

To start the investigate placement mode, click **Investig Plc** from the **IMPORT TYPE** menu during the import procedure. If you have selected this option, the investigate placement mode is activated during import.

In the investigate placement mode, component placement in the imported file is compared with its previous placement. Components with reference designators that did not exist in the assembly earlier are treated as new components. Each of these new or changed components is imported and displayed in Pro/ENGINEER as a working component, and the **Component Investigate** dialog box opens.

Note: The system compares two components for placement changes only if they have the same reference designator. Therefore, if a reference designator has changed, any change in its placement is not detected, and the component is treated as if it were a new component in the imported file.

The **Component Investigate** dialog box lists the reference designators of all components that are new, changed, or missing in the assembly along with the Pro/ENGINEER model name and its status. The status of each component is one of the following:

- **Replace**—The placement of this component is replaced with the new component placement from the file being imported.
- **Keep**—The placement of this component overrides any new component placement in the file being imported.
- **Accept New**—This new component is placed in the imported assembly.
- **Reject New**—This new component is not placed in the imported assembly.
- **Keep Missing**—This component is missing in the imported file, but is kept in the assembly.
- **Delete Missing**—This component is missing in the imported file and is deleted in the assembly.

Status Color Descriptions

- **Replace** status appears in blue showing the new position.
- **Accept New** status appears in magenta.
- **Keep Missing** components appear in green.

By default, all changed components are replaced and all new components are accepted.

The `ecad_missing_component_status` configuration option enables you to keep missing components or delete missing components by default, in the **Component Investigate** dialog box. Use the `keep_missing` option to keep missing components by default.

Filter the components displayed in the **Component Investigate** dialog box to display only new components, changed components, or both, by selecting from the list at the bottom of the dialog box. The default is to display both new and changed components. If you select a component in the list, it is highlighted in the assembly.

To change the status of a component, select the component in the dialog box and then click **Change Status** to switch between **Replace** and **Keep** for changed components or **Accept New** and **Reject New** for new components.

To preview any changes you make to the new or modified components, click **Preview**. The new or modified components change according to the specifications in the dialog box.

When you finish making changes to the status of the component placements, click **OK** to complete the import.

In Part Mode

1. Click **Insert** > **Shared Data** > **From File**. The **Open** dialog box opens.
2. Click the required ECAD package from the **Type** list, and select the ECAD file that you want to import from the list of available files. The **Ecad - Import** dialog box opens.

   Note: The **Ecad - Import** dialog box opens only if you have inserted shared data from a file.
3. Click **Yes** to activate the **Investigate Geometry** option in the **Ecad - Import** dialog box.
4. Click **OK**. The **Board Geometry Investigate** dialog box opens. The feature identified for the geometry update is displayed under **Feature Type** and its status is displayed under **Status**.
5. In the **Filter** drop-down box, select one of the following:

   All—To select the entire part

   Part Section—To select board outlines

   Area Features—To select keepin, keepout areas

Depending on your selection in the **Filter** drop-down list, the relevant feature is displayed under **Feature Type**.

6. To change the status of a feature type, select the feature type in the dialog box and click **Change Status** to switch between **Replace** and **Keep.** Click **OK** to complete the import.

Configuration File Options

The following configuration options apply specifically to Pro/ECAD. The default settings are listed first.

dazix_default_placement_unit	(mm, thou, micron)
dazix_export_mounthole	(Y/N)
dazix_z_translation	(Y/N)
ecad_area_default_import	(cosm_area, 3d_volume)
ecad_board_csys_def_name	(ecad_default, name)
ecad_comp_csys_def_name	(ecad_default, name)
ecad_create_hint_add	(yes, no)
ecad_default_comp_height	(value unit) - default depends on geometry
ecad_export_holes_as_cuts	(yes, no)
ecad_exp_both_two_areas	(yes, no)
ecad_import_holes_as_features	(yes, no)
ecad_import_relative_accuracy	(0.0012)
ecad_pin_hole_import	(no, yes)
ecad_via_hole_import	(yes, no)
ecad_mapping_file	(path for ecad_hint.map)
ecad_missing_component_status	(keep_missing, delete_missing)
ecad_other_outl_csys_def_name	(ecad_default, name)
Mentor_ver_2_0	(yes, no)

Using Keepin and Keepout Areas

About Keepin and Keepout Areas

ECAD keepin and keepout areas specify where you can or cannot place components to avoid interfering with other components or electrical routing. In Pro/ENGINEER, create keepin and keepout areas as cosmetic features of a board part. While creating cosmetic features, you can select edges of other ECAD areas. An ECAD area can be used in Sketcher to create entities from edges or datum curves or to create draft entities at an offset from edges by clicking  or  , respectively. Pro/ECAD adds the necessary commands to the **COSMETIC** and **PROJ SECT** menus in the part mode.

In Pro/ENGINEER, keepin and keepout areas can have a closed 3D volume, represented by a quilt. Use these quilts to perform a clearance and interference check to determine whether an ECAD component violates the keepin and keepout areas, even if the component is entirely within the keepout volume.

The `ecad_exp_both_two_areas` configuration option enables you to export an ECAD area with different heights for above and below board conditions. Use the `YES` option to export both-sided keepin and keepout ECAD areas as two individual areas (top and bottom).

The following table lists the Pro/ENGINEER cosmetic area feature types and the corresponding IDF region names.

PRO/ENGINEER ECAD AREA	IDF ECAD AREA
Place Keepin	Placement Outline Section
Place Keepout	Placement Keepout Section
Place Region	Placement Group Area Section
Route Keepin	Route Outline Section
Route Keepout	Route Keepout Section
Via Keepout	Via Keepout Section

Remember the following characteristics of keepin and keepout areas:

- As with other sketched cosmetic features, you need not dimension or regenerate keepin and keepout areas.
- Nondimensioned, nonregenerated areas are always nonparametric.
- Use **Name** in the **PART SETUP** menu to name keepin and keepout area features.
- You can add parameters to the keepin and keepout area features such as `board_side` or `routing_layers`.

- When you export keepin areas from Pro/ENGINEER through Pro/ECAD, the value of their height exported to the interface file is `0.0`. Manually edit the interface file or use the ECAD application to change this to a nonzero value if necessary.
- You can display keepin and keepout areas as 3D volumes, provided the value of their height value is greater than `0.0` by redefining the feature attributes and checking or clearing the **3D Volume** option.
  Use the `ecad_area_default_import` configuration option to define how imported ECAD areas are treated, that is, as outlines or volumes.

To Add a Keepin or Keepout Area

1. In a part, click **Insert** > **Cosmetic** > **ECAD Area**. The **OPTIONS** menu appears. The following commands are available:

   Regular Sec—Use a regular sketching plane for placement of the region.

   Project Sec—Use a section projection on a selected surface for placement of the region.

   Xhatch—Mesh the created region.

   No Xhatch—Leave the created region unmeshed.

   3D Volume—Determine whether the area appears with a 3D quilt. Area outline and keepin or keepout height define volume.

   Two Sides—Place the area on top and bottom.

2. In the same menu, define the type of area: (**3D Volume** and **Two Sides** are available where applicable.)

   Place Keepin—Creates a placement keepin region. You can specify the area height above and below the board while creating this region. The default height is zero. **Place Keepin** is not available for the `Using_the_Dazix_Format` interface.

   Place Keepout—Creates a placement keepout area. You can specify the area height above and below the board while creating this region. The default height is zero. **Place Keepout** is not available for the Visula interface.

   Place Region—Creates a placement region. Available only with IDF.

   Route Keepin—Creates a routing keepin region where routing actions are permitted. Available only with IDF.

   Route Keepout—Visual way to indicate that you are not to route in this area. This option is available only with IDF.

   Via Keepout—Visual way to indicate that you are not to create vias in this area. Available only with IDF.

3. Select a sketching plane and a reference plane. Sketch the area.

4. Exit **Sketcher**. If you created an area requiring a height dimension, you are prompted for it now. A red arrow shows the direction of the height you are

specifying. If you selected **Two Sides**, you are prompted for both, one after the other. The red arrow shows the direction of each height value.

To Assign a Group Name to an ECAD Area

Add the `Component_Group_Name` feature parameter to a keepin area to specify a component group name. This parameter corresponds to the Component Group Name parameter for IDF. This feature is automatically created when you import a model with a placement region group name.

Exporting Data to ECAD

About Exporting ECAD Data

You can export board, part outline, and placement data from the part level or the assembly level. You can use any of the available translator formats or the IDF format.

Each component to be exported needs a coordinate system to identify the part origin, and a section plane selected to represent the footprint. When you export individual parts, you are prompted for each. If you assigned a default value to `ecad_comp_csys_def_name` in the `config.pro` file, and named the origin csys the same value, you are not prompted for the csys.

When you export from assembly mode, a dialog box handles the csys selection, component name and reference designator values, and lets you allow or not allow selected items from the export.

To Export Board Outline and Component Assembly Data

You can export a board outline (including any holes) and component placement information from Pro/ENGINEER through Pro/ECAD.

1. In the assembly to be exported, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select the type of the ECAD package to which you want to export information, specify a new file name., if required,and click **OK**.
3. If you have selected IDF (`.emn`) type, then in the **Menu Manager**, select the **ECAD FORMAT** for your export. The **ECAD Assembly Export** dialog box opens. The board and all the components are listed with an export status (`yes` or `no`), and a csys to be used as an origin. Use the dialog box to disallow any objects from the export, or to add or edit coordinate systems.

   Note: If parts do not have coordinate systems, they are not included in the export.
4. Click **OK**. Pro/ENGINEER performs the export. The `ecad.log` file contains warnings or error messages, if any.

To Export Board or Component Parts

In Part mode, you can export the outline of the current board (including any holes) or component. If the part is a board, you can also export its keepin and keepout areas. To be exported, a part must be active. You can export a component outline only when you are in Part mode.

1. In the part to be exported, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select the type of ECAD package to which you want to export information, specify a new file name, if required, and click **OK**.
3. If you have selected the IDF (`.emn`) type, then in the **Menu Manager**, select the **ECAD FORMAT** (board or panel) for your export. You are prompted for a coordinate system to be used as the board or part origin.
4. Identify the csys in the **Menu Manager**.
5. If you have selected the IDF (.emp) type to export a component, you are prompted to export a section plane to represent its outline. Pro/ENGINEER prompts you to specify a cross-section or a planar surface to export as the outline. Click one of the following from the **PLANE/XSECT** or **ECAD PLANE** menu:
   - **Plane**—The **ECAD PLANE** menu appears with **Default**, **Select**, and **Quit** for selecting a planar surface of the component.
   - **XSection**—The **XSEC NAMES** menu appears. Select the name of a previously created cross-section.

   When you select the plane, Pro/ENGINEER performs the export.

To Export a Panel File in Assembly Mode

You can export a panel file from Pro/ENGINEER through Pro/ECAD. The assembly is exported as a panel file if the first component of the assembly has the ECAD_PART_TYPE model parameter set to PANEL.

1. In the assembly mode, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select ECAD IDF (*.emn) for the type of ECAD export and click **OK**.
3. In the **Menu Manager**, click IDF 3.0 in the **ECAD FORMAT** menu. The ECAD Assembly Export dialog box opens.
4. Select the export status and coordinate system to be used.
5. Click **OK**. Pro/ENGINEER performs the export.

To Export a Panel File in Parts Mode

You can also export a panel file from Pro/ENGINEER through Pro/ECAD in the part mode.

1. In the part mode, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Change the type to ECAD IDF (*.emn), specify the required name for the output file, and click **OK**.
3. In the **ECAD FORMAT** menu in the **Menu Manager**, select IDF 3.0.
4. Select **Panel** in the **OBJ TYPE** menu.
5. Select the coordinate system for export from the **SEL COORD S** menu in the **Menu Manager**. Pro/ENGINEER performs the export.

Tip: Placing Subassemblies as Components for Export

When used correctly, the `ecad_hint.map` file can place a subassembly as a component just as you can place a part as a component. You cannot export component outlines of assemblies. If you place subassemblies as components, the following restrictions apply:

- If the current directory or session contains a part with the same name as the subassembly, the system uses the part rather than the subassembly.
- You can use a subassembly only if it contains an assembly coordinate system with the appropriate datum planes. This assembly coordinate system must be built explicitly on three assembly datum planes and have an orientation recognized by Pro/ECAD.

Controlling Import - Export with the ecad_hint.map File

About the ecad_hint.map File

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

- Substituting custom-made Pro/ENGINEER parts for automatically extruded parts on import.
- Substituting a custom-made Pro/ENGINEER MCAD assembly for an automatically extruded part, that is, an ECAD component, 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/ENGINEER.
- Changing an ECAD other outline string for import. If, for example, the ECAD reference designator uses characters that are illegal in Pro/ENGINEER.

Pro/ECAD searches the working directory for ecad_hint.map and references it every time an import occurs. The file is ignored if it is empty or has no relevant information.

You can use the ECAD_MAPPING_FILE <path> configuration option to set a default location for the ecad_hint.map file. If you set a path with this configuration option, the working directory is not searched.

To Create the ecad_hint.map File

1. Run an automatic import of your ECAD part list into an assembly. This creates a log file called ecad_hint.add, that includes entries for each part type in the part list.
2. Edit this file, if required, substituting values as necessary, and save it in the working directory as ecad_hint.map.
3. If you have custom .prt files prepared for substitution, delete the automatically generated parts from the assembly and reimport the ECAD part file using the newly edited ecad_hint.map file.

Using the ecad_hint.add File

When Pro/ECAD creates a component .prt file automatically during import, it:

- Combines the ecad_name and ecad_alt_name as an mcad_name for the part file, and
- Records the creation of the component in a text file called ecad_hint.add.

Do not confuse this with the ecad_hint.map file. However, you can use the .add file as a template for a .map file.

In the ecad_hint.add file, each part receives the following entry (examples shown as values):

```
Map_objects_by_name ->

ECAD_NAME "CSTCS"
ECAD_ALT_NAME "N7414N"
ECAD_TYPE " "
MCAD_NAME "CSTCS_N7414N"
MCAD_TYPE "part"
END
```

Component names that include a dollar sign ($), slash (/), period (.), or percent sign (%) are not valid. Pro/ENGINEER uses the ecad_hint.add file to change each of these characters to an underscore (_). If this adjustment causes the file names to be identical, new file names are created. To use the translated names, edit the ecad_hint.add file and append it to the ecad_hint.map file.

If the ecad_hint.map file contains names that are the same except for a symbol that Pro/ENGINEER does not use, Pro/ENGINEER creates a new part name and notes this in the log file. For example, ecad_name Aaa$ becomes mcad_name AAA_ and ecad_name Aaa/ becomes mcad_name AAA_1.

When you do not use the ecad_hint.map file to determine the correspondence between ECAD part names and Pro/ENGINEER part names, Pro/ENGINEER interprets the part names of components in all uppercase letters. For example, Pro/ENGINEER searches for a component named AaaA in an ECAD system as AAAA.

If you do not want the system to create an ecad_hint.add file, set the ecad_create_hint_add configuration file option to no.

Using the ECAD_ALT_NAME

Pro/ECAD parts may have the same values for ecad_name and different values for ecad_alt_name. For example, the resulting .add file from an import may have the following entries:

Resistors entry 1	Resistors entry 2
Map_objects_by_name->	map_objects_by_name->
ECAD_NAME "Resistors"	ECAD_NAME "Resistors"
ECAD_ALT_NAME "R5K"	ECAD_ALT_NAME "R2K"
ECAD_TYPE ""	ECAD_TYPE ""
MCAD_NAME "RESISTORS_R5K"	MCAD_NAME "RESISTORS_R2K"
MCAD_TYPE "part"	MCAD_TYPE "part"
End	End

In this case, if you have different models for a R5K and a R2K resistor, (for example, they may be of different colors) you can specify the individual .prt names in the ecad_alt_name field. For example, if the Pro/ENGINEER part files are called resistor1_5k.prt and resistor1_2k.prt, use the entries below:

Resistors entry 1	Resistors entry 2
Map_objects_by_name->	map_objects_by_name->
ECAD_NAME "Resistors"	ECAD_NAME "Resistors"
ECAD_ALT_NAME "R5K"	ECAD_ALT_NAME "R2K"
ECAD_TYPE ""	ECAD_TYPE ""
MCAD_NAME "resistor1_5k"	MCAD_NAME "resistor1_2k "
MCAD_TYPE "part"	MCAD_TYPE "part"
End	End

Map File Standard Conventions and Examples

In the ecad_hint.map file:

- The # character is the comment character.

- Object and value fields are separated by a space.
- Spaces are permitted in value strings if the string is surrounded by quotation marks.
- Wildcard (*) is valid for "all", for example:

```
mcad_in_ignore ->
ecad_name "resistor"
ecad_alt_name "*"
ecad_type "part"
ref_des "*"
end
```

- Each section begins with the purpose, followed by "->".
- Each section ends with "end". For example:

```
map_objects_by_name->

ECAD_NAME "GEN_DIP"
ECAD_ALT_NAME "GEN_DIP"
ECAD_TYPE ""
MCAD_NAME "PROE_GEN_DIP"
MCAD_TYPE "part"
end
```

- If you include the string ecad_create_hint_add N in the config.pro file, the .add file is not created.

To Reference Custom Parts During ECAD Import

1. In a text editor, open the ecad_hint.map file.
2. Place the name of the replacement Pro/ENGINEER component in the mcad_name section of the part's entry. The first line of each entry should be map_objects_by_name->. For example, if the Pro/ENGINEER part file you want to reference is called resistor1.prt, specify resistor1.
3. Repeat for each .prt file that you want to reference. When you run the import again, your custom parts are substituted for the automatically generated ones.

Note: Set the value of the parameter MCAD_TYPE to "ASM" if you want to substitute an MCAD assembly for an automatically extruded part.

To Control Automatic Layer Assignment

Using the ecad_hint.map file, you can specify an overriding mapping (over the automatic mapping) to produce a new ECAD_<mapped name> layer. The component is placed in that layer:

```
map_objects_by_name->
ECAD_NAME "eesmb"
ECAD_ALT_NAME "DM7442N"
ECAD_TYPE ""
MCAD_NAME "EESMB_DM7442N"
MCAD_TYPE "part"
MCAD_LAYER "<ANY>
```

Note: The ecad_comp_layer_map preferences option must be set to yes.

To Change the Other Outline String for Import

Use an entry such as the following in the ecad_hint.map file to replace an .OTHER_OUTLINE name with an MCAD name for import purposes:

```
map_other_outline_name->

ecad_board_name "board"
ecad_other_outline ".OTHER_OUTLINE"
mcad_part_name "heatsink"

END
```

To Change the Reference Designator String for Import

Use an entry such as the following in the ecad_hint.map file if a reference designator contains illegal characters. You can replace the reference designator with another string for import.

```
reference_designator->

ECAD_REF_DESIG "R/1"
MCAD_REF_DESIG "R1"
END
```

To Exclude Parts From Import or Export

Use an entry such as the following in the ecad_hint.map file to exclude a part from import:

```
mcad_in_ignore ->

ecad_name "resistor"
ecad_alt_name "resistor_5k"
ecad_type "part"
ref_des "*"

END
```

To exclude a part from export, use an entry such as the following in the ecad_hint.map file:

```
mcad_out_ignore ->

mcad_name "HEAT_SINK"
mcad_type part
ref_des "HS1"

END
```

Intermediate Data Format (IDF)

About the Intermediate Data Format (IDF)

The Intermediate Data Format (IDF) is a data exchange format for printed wire assemblies. IDF is available in versions 2.0 and 3.0. Pro/ECAD supports the import and export of both versions of IDF file format.

The IDF 2.0 specification does not support z-axis offset component placement information. When you export information from Pro/ENGINEER to the IDF 2.0 format, all z-axis offset information is lost.

IDF 3.0 does support z-offset for component placement.

About IDF File Sections

Pro/ENGINEER supports all sections of the IDF 2.0 Board File Format, and all sections of the IDF 2.0 Library File Format. All IDF 2.0 functionality supported in IDF 3.0 continues to be supported.

IDF files default to the following extensions:

- .emn—For a neutral file of the board outline and component placement
- .emp—For a profile file that contains component outlines

When you import or export with the IDF format, Pro/ENGINEER supports the Library File fields .ELECTRICAL and .MECHANICAL. Use these parameters to indicate whether an imported or exported component is electrical or mechanical. Use them in the .emp (export profile) file to set the type of component.

The system imports parts from the .OTHER_OUTLINE sections of an IDF file with default datum planes and coordinate systems at their origin. The system also imports each of them with two part parameters. The first parameter, ECAD_PART_TYPE, has the value .OTHER_OUTLINE. The second parameter, ECAD_PART_NAME, has a value equal to the name of the .OTHER_OUTLINE section where it originated.

Sometimes when you import a part, a part with the same name already exists in the current directory or session. In such cases, the part that already exists is used rather than the part specified in the IDF file.

If you prefer, you can specify another name for the system to use during import. To do this, add the lines to the ecad_hint.map file. For example, <board_name> is the

name of the board part being imported, <.OTHER_OUTLINE <name>> is the name in the .OTHER_OUTLINE section of the IDF file, and <desired_name> is the name you want to use. Use this method to specify an alternative name for the .OTHER_OUTLINE part or, if the <desired_name> is the same as the name of a part in the current directory or session, specify another part to use instead of the part from the IDF file.

All .OTHER_OUTLINE parts that Pro/ENGINEER imports keep their corresponding coordinate system and name. Specify the name of the coordinate system in the .OTHER_OUTLINE section with the ecad_other_outl_csys_def_name configuration option. If you do not set this option, the system assumes that the name of the coordinate system is ECAD_DEFAULT.

The system uses the units for the ECAD system board from which this part was imported as the units for the imported part.

To export parts that you create in Pro/ENGINEER as an .OTHER_OUTLINE section, create a part parameter called ECAD_PART_TYPE, with a value of .OTHER_OUTLINE. To specify the name of the .OTHER_OUTLINE section you create when you export this part, create a second part parameter called ECAD_PART_NAME. Assign it a value equal to the name you want for the .OTHER_OUTLINE section. If you do not specify this parameter, the system specifies the name of the part being exported as the name of the .OTHER_OUTLINE section.

Exported components must have two surfaces (TOP and BOTTOM) parallel to the TOP and BOTTOM of the board in the assembly.

To import all sections specified as .DRILLED_HOLE with either the PTH (Plated Holes) or the NPTH (Non-Plated Holes) attribute in the IDF file as Through-All Hole features, set the ecad_import_holes_as_features configuration option to YES.

In IDF 2.0, the system stores all imported holes with the ECAD_HOLE_TYPE feature parameter. The value of this parameter is either PTH (plated through hole) or NPTH, (non-plated through hole) depending on the value of the .DRILLED_HOLE section in the IDF file.

IDF 3.0 uses an ECAD_PLATING_STYLE parameter similar to the ECAD_HOLE_TYPE parameter

The system exports boards with drilled holes, created by Pro/ENGINEER, with a default value of NPTH for the ECAD_HOLE_TYPE feature parameter. Create this feature parameter to specify a value of PTH.

Supported ECAD File Formats

About Supported ECAD File Formats

Pro/ECAD supports the ECAD systems shown below for import to and export from Pro/ECAD. Additionally, you can use the IDF format, either 2.0 or 3.0.

Using_the_Visula_Format—Uses the Visula ECAD/MCAD Neutral Interface format to interface with Visula version 6.0 software from Zuken-Redac Systems.

Using_the_Allegro_Format—Uses the Allegro Interface format to interface with Allegro software version 5.1 and later from Cadence Design Systems. Cadence recommends that all users use IDF 2.0.

Using_the_Boardstation_Format—Uses versions 1.0 and 2.0 of Intermediate Data Format (IDF) from Mentor to interface with Mentor Graphics Board Station software. The `mentor_ver_2_0` configuration option controls whether Pro/ENGINEER data is exported in version 1.0 or version 2.0 format.

Using_the_Dazix_Format—Supports the interface with:

- CDX Classic software (formerly Dazix CDX Classic)
- **AT Designer**—Intergraph Corporation v 12 through v12.3, (formerly PCB Engineer). V12.4 and later use the IDF 2.0 format.

IDF2.0—Intermediate Data Format (IDF) specifications for version 2.0.

IDF3.0—Intermediate Data Format (IDF) specifications for version 3.0.

IDF files default to the following extensions:

- `.emn`—For a neutral file of the board outline and component placement
- `.emp`—For a profile file that contains component outlines

About Using the Allegro Format

Note: Cadence and PTC recommend that you use the IDF format for data exchange, not the older direct format.

Allegro files have one of the following extensions:

- `.mdb`—For board outline files
- `.mdc`—For component placement files
- `.mdf`—For footprint files, such as the ones in component outline libraries

Allegro cannot read the outlines of components. In part mode, you can only export boards to Allegro.

Allegro represents mounting holes and other such features on a board as components. Therefore, component placement files contain their placement information.

To process these features properly, Pro/ENGINEER requires the diameter of the holes, even though Allegro does not write this information to these interface files. To work around this, include a special record in the `ecad_hint.map` file. This record is called `map_hole_component` and is formatted as follows:

```
map_hole_component ->

ecad_name "hole_name"
ecad_type ""
unit "unit_name"
diameter "diam_value"

End
```

For example:

```
map_hole_component ->
ecad_name "MH125"
ecad_type ""
unit "mil"
diameter "125"

End
```

To support the diameter of the holes, write the following units in the `ecad_hint.map` file:

```
mm
in
mils
```

All other Pro/ENGINEER units are converted as follows:

```
cm converted to mm
m converted to mm
ft converted to mils
microns converted to mils
```

For example, a Pro/ENGINEER part with a hole diameter of 1.5 inches can have an Allegro `ecad_hint.map` mapping file written as follows:

```
map_hole_component ->

ecad_name "kuku"
ecad_type ""
unit "ft"
diameter "0.125"

End
```

or

```
map_hole_component ->

ecad_name "kuku"
ecad_type ""
unit "cm"
diameter "3.81"

End
```

or

```
map_hole_component ->

ecad_name "kuku"
ecad_type ""
unit "inch"
diameter "1.5"

End
```

When Pro/ENGINEER exports Pro/ECAD models to Allegro, it places a warning in the ECAD export log file if the export model contains area types that Allegro does not support. Allegro ignores the unsupported area types.

Note: The Allegro interface format does not specify height information for placement keepin and keepout areas; Even if Pro/ENGINEER contains this information, Pro/ENGINEER cannot export it to Allegro.

About Using the Boardstation Format

Boardstation files use the following extensions:

- `.emn`—For a neutral file of the board outline and component placement. Pro/ECAD neither imports nor exports information other than board outline and component placement. When Pro/ENGINEER imports information, it looks for files with the `.emn` extension. However, Boardstation can export these files with the `.lib` or `.brd` extensions instead. When Pro/ENGINEER exports the file, it uses the `.emn` extension.
- `.emp`—For a profile file that contains component outlines. When Pro/ENGINEER imports information, it looks for files with the `.emp` extension. However, Boardstation can export these files with the `.pro` extensions instead. When Pro/ENGINEER exports, it uses the `.emp` extension.

Pro/ENGINEER accepts exported information from Mentor Graphics that uses the 3D Mechanical interface. See your Mentor Graphics documentation for more information.

About Using the Dazix Format

Dazix files use the following extensions:

- `.edn`—For a neutral file of the board outline and component placement. Dazix refers to this as a core file.
- `.edp`—For a profile file that contains component outlines. Dazix refers to this as a library file.

Dazix component names are mixed-case, prefixed with Z. Pro/ENGINEER converts these names to all uppercase. Dazix does not recognize uppercase names. Thus, to effect the transfer of component data, Dazix automatically creates an `ecad_hint.map` file to map between the two.

Note: The Dazix interface format does not specify placement keepin areas or height information for placement keepout areas. Even if Pro/ENGINEER contains this information, Pro/ENGINEER cannot export it to Dazix.

About Using the Visula Format

All Visula files use the extension `.evs`.

The Visula interface format does not specify package keepout areas. Even if you create such an area in Pro/ENGINEER, Pro/ENGINEER does not export it. The IDF 2.0 format, however, does specify keepout areas.

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