Database Policies and Procedures

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Preface

Database Policies and Procedures is for CAD/CAM managers and system administrators who use PTC software. This guide has recommended policies and procedures to help you maintain your databases.

Related Documents

The following documents may be helpful as you use *Database Policies and Procedures*:

- *Managing CADDS 5*
- *Explicit Modeling User Guide and Menu Reference*
- *Installing CADDS 5*

Book Conventions

The following table illustrates and explains conventions used in writing about CADDS applications.

Convention	Example	Explanation
Menu selections and options	List Section option, Specify Layer field	Indicates a selection you must make from a menu or property sheet or a text field that you must fill in.
User-selected graphic location	X, d_1 or P1	Marks a location or entity selection in graphic examples.

User input in CADDS text fields and on any command line	`cvaec.hd.data.param`
	
	`tar -xvf /dev/rst0`
		Enter the text in a CADDS text field or on any command line.

System output	`Binary transfer complete.`	Indicates system responses in the CADDS text window or on any command line.

Convention	Example	Explanation
Variable in user input	`tar -cvf /dev/rst0` filename	Replace the variable with an appropriate substitute; for example, replace filename with an actual file name.
Variable in text	tagname	Indicates a variable that requires an appropriate substitute when used in a real operation; for example, replace tagname with an actual tag name.
CADDS commands and modifiers	INSERT LINE TANTO	Shows CADDS commands and modifiers as they appear in the command line interface.
Text string	`"SRFGROUPA"` or `'SRFGROUPA'`	Shows text strings. You must enclose text string with single or double quotation marks.
Integer	n	Supply an integer for the *n.*
Real number	x	Supply a real number for the *x.*
#	`# mkdir /cdrom`	Indicates the root (superuser) prompt on command lines.
%	`% rlogin` remote_system_name `-l root`	Indicates the C shell prompt on command lines.
$	`$ rlogin` remote_system_name `-l root`	Indicates the Bourne shell prompt on command lines.

Window Managers and the User Interface

According to the window manager that you use, the look and feel of the user interface in CADDS can change. Refer to the following table:

Look and Feel of User Interface Elements

User Interface Element	Common Desktop Environment (CDE) on Solaris and HP	Window Manager Other Than CDE on Solaris, HP, and Windows
Option button	ON — Round, filled in the center OFF — Round, empty	ON — Diamond, filled OFF — Diamond, empty
Toggle key	ON — Square with a check mark OFF — Square, empty	ON — Square, filled OFF — Square, empty

Online User Documentation

Online documentation for each book is provided in HTML if the documentation CD-ROM is installed. You can view the online documentation in the following ways:

- From an HTML browser

- From the Information Access button on the CADDS desktop or the Local Data Manager (LDM)

Please note: The LDM is valid only for standalone CADDS.

You can also view the online documentation directly from the CD-ROM without installing it.

From an HTML Browser:

1. Navigate to the directory where the documents are installed. For example,

   `/usr/apl/cadds/data/html/htmldoc/` (UNIX)

   Drive:`\usr\apl\cadds\data\html\htmldoc\` (Windows)

2. Click `mainmenu.html`. A list of available CADDS documentation appears.

3. Click the book title you want to view.

From the Information Access Button on the CADDS Desktop or LDM:

1. Start CADDS.

2. Choose Information Access, the i button, in the top-left corner of the CADDS desktop or the LDM.

3. Choose DOCUMENTATION. A list of available CADDS documentation appears.

4. Click the book title you want to view.

From the Documentation CD-ROM:

1. Mount the documentation CD-ROM.

2. Point your browser to:

   CDROM_mount_point/`htmldoc/mainmenu.html` (UNIX)

   CDROM_Drive:`\htmldoc\mainmenu.html` (Windows)

Online Command Help

You can view the online command help directly from the CADDS desktop in the following ways:

• From the Information Access button on the CADDS desktop or the LDM

• From the command line

From the Information Access Button on the CADDS Desktop or LDM:

1. Start CADDS.

2. Choose Information Access, the i button, in the top-left corner of the CADDS desktop or the LDM.

3. Choose COMMAND HELP. The Command Help property sheet opens displaying a list of verb-noun combinations of commands.

From the Command Line: Type the exclamation mark (!) to display online documentation before typing the verb-noun combination as follows:

```
#01#!INSERT LINE
```

Printing Documentation

A PDF (Portable Document Format) file is included on the CD-ROM for each online book. See the first page of each online book for the document number referenced in the PDF file name. Check with your system administrator if you need more information.

You must have Acrobat Reader installed to view and print PDF files.

The default documentation directories are:

- `/usr/apl/cadds/data/html/pdf/`**doc_number**`.pdf` (UNIX)

- **CDROM_Drive:**`\usr\apl\cadds\data\html\pdf\`**doc_number.**`pdf` (Windows)

Resources and Services

For resources and services to help you with PTC (Parametric Technology Corporation) software products, see the *PTC Customer Service Guide*. It includes instructions for using the World Wide Web or fax transmissions for customer support.

Documentation Comments

PTC welcomes your suggestions and comments. You can send feedback electronically to `doc-webhelp@ptc.com`.

Chapter 1 — Maintaining a Sound Database

This chapter discusses the `validate_db` diagnostic and maintenance tool and other useful commands and utilities.

- Managing Parts
- Using validate_db
- Managing Files
- Explicit CHECK DBASE
- Repairing and Reformatting Parts
- Preventive Measures
- Troubleshooting

Managing Parts

The validate_db utility provided by PTC and the ckCAD utility provided by CADtools automatically validate parts when you activate or file either a part or an assembly. Automatic validation of a part ensures that you do not activate or file corrupt parts.

Please note: In case of an assembly, `validate_db` validates only adrawings or model parts.

Setting the Environment Variables

To use the ckCAD utility for automatic database validation, set the value of the `DB_CHECKING` environment variable to the path of the `ckCAD` executable file. If you have not set the `DB_CHECKING` environment variable, validation of the database is disabled.

To use the validate_db utility for automatic database validation when you activate a part, set the value of the `DB_CHECKING_ACT` environment variable to the path of the `validate_db` executable file.

To use the validate_db utility for automatic database validation when you file a part, set the value of the `DB_CHECKING_FILE` environment variable to the path of the `validate_db` executable file.

Please note: If you have set the `DB_CHECKING` and the `DB_CHECKING_ACT` and `DB_CHECKING_FILE` environment variables, CADDS checks if you have set the `DB_CHECKER` environment variable. If you have set the `DB_CHECKER` environment variable, the ckCAD utility is used for automatic database validation. If you have not set the `DB_CHECKER` environment variable, the validate_db utility is used for automatic database validation.

Results of Validation

Appropriate status messages are displayed and the selected database checking tool returns the status of validation. The status messages and actions are as listed below:

Clean:
The database is validated to be CLEAN and can be activated or filed. If the part is a model of a component of an assembly, the component can be viewed.

Modified:
The database is validated and a copy of it is modified and saved. If the process is activation, you will have the option to either overwrite the existing part or save it under a new name. If you choose to save the part under a new name, the new part will be activated.

Please note: In case of an adrawing of an assembly, only overwrite option is available. If you are working on a model, you can save the file with a new name, but only the current one will be activated. However, a message "use CHANGE COMPONENT" is displayed to attach the new model to the component.

If you are attempting to view the component of an assembly, the following message is displayed.

```
validate and correct the model before viewing
```

While filing, you can overwrite the part being filed with the validated and modified copy.

Fatal:
If this status is returned the part is not activated.

In case this message appears when the part is being filed, you will have the option to exit without filing, or continue in the active session.

If you attempt to view the component, it will not be viewed if the fatal message appears.

Limitations

The database checking during ACTIVATE and VIEW component commands are done when either of the following conditions are true.

• The database has a last filed revision level older than the active CADDS revision regardless of the validation condition

• The database has no setting to signify that the part has been validated

Using validate_db

`validate_db` is a database diagnostic and maintenance tool with two major functions, check the integrity of the primary files that comprise a part, and examine and report the status of various data within a part. `validate_db` is very fast because it uses the object-oriented database structure at the OS level. It performs its tasks faster than the duplicate commands in CADDS. The `validate_db` command acts on the part's `_pd` file.

Please note: If you want to activate a part and the Parametric database is corrupt for the part, the part is not activated. The database validation tool reports an appropriate fatal error and exits out of CADDS.

Use `validate_db`,

- Before activating an archived part and before filing it to ensure that the part has not been corrupted

- Before activating an archived or migrated part to establish a baseline of part validity before the beginning of the CADDS session

- On a part containing parametric solids after executing EXTRACT PART on the part

- Before or after you use a converter or data exchange product (for example, IGES, DXF, CIMIO, or SET)

- On a part that does not yield predictable outcomes (for example, a command that returns "unknown error", makes CADDS exit prematurely, or causes a bus error).

For more details about `validate_db`, see Chapter 2, "Maintaining Files" in *Managing CADDS 5*.

Managing Files

Maintaining Files

When maintaining files, the following actions are recommended. They prevent other users from accidentally changing your library parts and rendering them inaccessible to you.

- Use the same operating system and platform from start to finish during each work session.

- Periodically archive all part data against possible loss. Part data comprises the part file (`_pd` for an Explicit part; `_fd` for a Parametric part) and all other files needed to design, detail, test, and analyze or manufacture the part.

- Rename and document as a replacement, any archived assembly, Nfigure, Pfigure, or Sfigure modified by an engineering change order (ECO).

Archiving Files

When archiving files:

- Follow all the recommendations of the manufacturer of the disk or tape.

- Use disks which are a more reliable medium than tape.

- Use a tape cleaning machine every 6 to 12 months to increase the life of archival tape. If current software does not support the format or density of archival tape, copy the tape to the supported format or density.

Follow these steps to archive a part:

1. Use `validate_db` and its `file part` command. Do not save files that need regeneration, for example, `_nfig` or `_pict` files.

2. Archive all part and part-related files (library parts, `nfig _pd`, `sfig _pd`, `CVMAC`, and other files) together using `validate_db`'s ARCHIVE DESIGN command.

3. Use the operating system level `compress` command to save storage space.

4. Use the command `tar cvfb /dev/rst0 126` **filelist** to set a block size of 126 for fast transfer to tape.

5. Verify that the designated files have been transferred to the archival medium and are sound.

   One way to do this is to use the command `tar -cvf` **filename** followed by the command `tar -tvf` **filename**.

6. Store the archival medium safely and record its location.

Retrieving Files

When retrieving files:

- Put archived part files for an ECO (especially libraries) into a search path that is different from the current development path.

- When an ECO requires that you not only revise part files but also convert them, use vendor-recommended procedures. CDS4000 (CGOS) customers should use the procedures and practices discussed in the *CADDS 4X Converter User Guide.*

- Use the `uncompress` command on all files that you earlier compressed.

Transferring Files

When transferring files, the following actions are recommended:

- When working on parts on different platforms or revisions, keep dependent files (figures) equally current on both platforms. Then you can use the `ftp` command or tar files to transfer a file (`_pd` or `_fd`) between platforms.

Please note: PTC only warranties the forward migration of parts.

- Prepare nested figures on the platform where they are used and remember that commands like REGENERATE GRAPHICS and CHANGE EXTENTS LEAST may behave differently near round-off points when operating on different platforms.

Explicit CHECK DBASE

Use CHECK DBASE, a CADDS database tool on a CADDS part for regular maintenance or when you have a problem with the part.

Please note: PACK DBASE and SORT DBASE functionalities are now supported by CHECK DBASE. By default CHECK DBASE performs the same function as the command PACK DBASE ALL. If a CAM database file exists, then by default CHECK DBASE performs the same function as PACK DBASE. PACK DBASE and SORT DBASE have been disabled in CADDS.

CHECK DBASE within the parametric environment only removes orphaned entities from the parametric database (_fd). The savings on disk can be gained only when part filing has been completed successfully.

CHECK DBASE

Use this command to verify the active part. It should be used either as a regular database/part maintenance routine and whenever a problem is encountered. This command uses `validate_db` (see *Managing CADDS 5* for additional information) or `ckcad` (Third-Party database checker), depending on the option set in the environment variable DB_CHECKER. Set this environment variable in the '.caddsrc-local' file. You can set DB_CHECKER to "the path of the ckcad executable and license information" to use `ckcad`. The default is `validate_db` when DB_CHECKER is not set.

The output from `validate_db` or `ckcad` appears in a Report Window for your analysis. You can overwrite the active part by entering `ok` at the appropriate prompt. If you choose to bypass this functionality the current active part and its disk version are not updated.

Please note: When the option is set to `ckcad`, you must make sure that the `ckcad` executable is available, since this is not supplied as a part of CADDS.

The CHECK DBASE command executes `validate_db` or `ckcad` with the `-p` option by default. The `-p` option performs a function similar to the action performed by PACK DBASE ALL, whereby all holes (deleted entities) in the database are reclaimed with miptrs reordered. In the event of a CAM database file stored on disk in the active parts directory, `validate_db` or `ckcad` executes without the `-p` option, thus not invalidating CAM databases.

Please note: Ignore system messages appearing in the command window during execution of the command when an active part has parts which are viewed in.

Additional checks, not found in `validate_db` or `ckcad`, are done on the part by CHECK DBASE. For example, if a bad hybrid entity is encountered in the explicit modeling database, the entity is deleted.

Repairing and Reformatting Parts

Repairing Parts

Use the following commands to discover and repair parts problems while working in CADDS. Check the online command files and the *Explicit Modeling User Guide and Menu Reference* for specific information about each command.

• CHANGE EXTENTS LEAST UPDATE

• DELETE ENTITY ODRAW

• CHECK DBASE

Reformatting Parts

Explicit parts can be single-precision or double-precision, while parametric parts are exclusively double-precision.

A single-precision part represents numbers with fewer places reserved right-of-the-decimal than does a double-precision part.

If you want to activate a single-precision part for reference or to utilize its geometry for double-precision modeling operations, you must first reformat that part.

Executing REFORMAT PART does the following:

• Appends zeros to single-precision numbers. While appending zeros does not give those numbers double-precision accuracy, it does ensure that all numbers within the part are in double-precision format.

• Assures that the results of subsequent operations have double-precision accuracy.

Preventive Measures

Version Compatibility

You cannot use CADDS Release 10 or earlier to activate a part that is created using CADDS Release 11 or later.

Release 11 onward, CADDS checks for backward revision status, that is, 12 to 11, 13 to 11, and so on. However it is still the users responsibility not to take Release 11 or forward parts back to Release 10 or earlier releases.

CADDS 5 15.0 onward, CADDS checks for the backward compatibility status of figures. You cannot insert a figure into a CADDS part, if the figure is created using a later release of CADDS. For example, if you are using CADDS 5 15.0, you cannot insert figures created in CADDS 5 Release 16.

Maintaining Swap Space

Ensure that the system has enough swap and temporary space before you start a CADDS session or file a part. For lack of enough space, the system may fail and unsound data may be written. *Installing CADDS 5* contains information to help you access sufficient swap space.

Adhere to these guidelines to maintain adequate free swap space:

- Do not use the /tmp directory to store temporary files. Rebooting clears /tmp, making crash recovery impossible.

- Do not move a part to a previous revision of CADDS since that part may not be backward compatible. (All parts are upward compatible.)

- Do not tamper with "live" disk partitions while applications are using them.

- Do not allow "live" disk partitions to become more than 85% full.

Testing Software

Develop tools for CADDS that are based on your product process and the way in which you use these products. Use `validate_db` as part of your quality assurance process.

If you have a new feature, test it. If you have a revised interface, test it. If you have a problem or failure when you are testing, report the situation to PTC.

Please note: You can give your tests to PTC for use in quality assurance.

Troubleshooting

Follow these steps after a generic fatal message:

1. File the part under a new name.

2. Exit the part and quit.

3. Execute `validate_db` on the newly named part if the fatal message was provoked by a creating or modifying entity command.

Please note: If step 3 is unsuccessful, exit the part and/or CADDS and report the problem to PTC.

4. Activate the newly named part.

Handling Software Problems

Chapter 2

This chapter provides information on PTC's software problem reporting process.

- Problem Reports

Problem Reports

Problem reports are classified in the following categories.

- You have a service contract with PTC

- You do not have a service contract with PTC

If You Have a Service Contract with PTC

From Locations Within North America

1. Search the knowledge database at `http://www.ptc.com/cs/search.htm`. for existing calls similar to your call. Use specific keywords to find related issues and workarounds. If you have access to the Internet but do not have a Web account, call the Technical Support number given in step 3 to open an account.

2. Review the events that led to your concern. Check the related documentation to ensure correct use of software commands and system features. Try the commands again in a simple test case. If the error recurs, report it as a problem.

3. Submit the issue online via the PRO/CALL LOGGER at `http://www.ptc.com/support/index.html`. Alternatively, call the Technical Support Center at 1-800-506-2657.

4. You must have the following information available:
   - Config ID
   - Contact name and telephone number
   - Product being used (for instance, CADDS 5)
   - Command or keywords with which there is concern (for example, INSERT LINE)
   - CADDS 5 software revision (for instance, Release 10)

5. A software tracking number is assigned to your issue.

   All issue tracking is based on your license file number. Therefore, ensure that the license file number used in your Software Performance Record (SPR) is correct.

6. Your call is forwarded to the Technical Support Center. A member of the technical support team discusses your problem, performs a preliminary analysis, and provides information about similar problems and available solutions. When your call is returned, have the following information available:

   • Syntax of command used

   • Results from the system

   • Tasks or station where the situation occurred

7. After the engineer recreates the situation and verifies that the software concern represents a valid deficiency, an SPR number is generated and forwarded to R&D. If an engineer is unable to recreate the problem, the issue is closed with your agreement, or you are asked to provide additional data as mentioned in step 1.

From Locations Outside North America

If you are located outside North America, report the software issue by calling the local PTC office. Customers worldwide can track their calls using the PRO/CALL TRACKER and a Web account that can be opened online or by calling the response center at 800-506-2657.

If You Do Not Have a Service Contract with PTC

Call your service provider for assistance or your local PTC service office for billable services.

Chapter 3 Updating Pre-CADDS 5 15.0 Parts for New Explicit SWR Environment

This chapter explains the procedures for updating pre-CADDS 5 15.0 parts for new Explicit Single Window Rendering (SWR) environment.

- Introduction
- The update_gr Utility
- The update_fig Utility
- Manually Updating Draw Mode Figures

Introduction

CADDS stores the drawing and shading information related to a part in separate external files. To view your pre-CADDS 5 15.0 parts in the new Explicit SWR environment you must update these files. Use the following procedure to update these files:

1. Use the `update_fig` command to update multiple library figures. A detailed description of the `update_fig` command is available in "The update_fig Utility" on page 3-5.

2. Use the `update_gr` command to update multiple parts and assemblies. A detailed description of the `update_gr` command is available in "The update_gr Utility" on page 3-3.

The update_gr Utility

To view the components in CAMU and to shade the viewed components, it is necessary that the part database of the components have the updated drawing and shading data for the components.

The `update_gr` script is used for updating the drawing and shading information related to the components of an assembly so that they can be displayed in the CAMU environment. The `update_gr` script automatically starts CADDS, activates each component in the assembly in the part mode, and files it in the Explicit environment.

The `update_gr` script is available in the following location:
`/usr/apl/cadds/scripts/update_gr`

When you try to view the pre-CADDS 5 15.0 parts in the new Explicit SWR environment without updating them using the `update_gr` script, one of the following messages is displayed.

* `Warning: Graphic file does not exist or may be corrupt for the following part =USERS.JOHNDOE.PARTS.PART1`

    `Temporary graphics data is being generated for the current session. Activate and file the part to create and store the graphics data.`

* `Unable to open drawing for specified part.`

    This could mean the drawing file does not exist.

`Update_gr` is a batch tool that you can run on all the parts in a specified directory or all the components of a specified assembly.

Please note: Using the `update_gr` script you can update only those parts whose names are less than 60 characters in size.

Syntax

```
update_gr [-assembly] [-cgos] [-check] [-draw name] [-path xxx]
[-noshade] [ -smoothness n ] [directory]
```

Options

* `-ass[embly]`

    Checks if the specified directory contains a single part or an assembly. If it contains a single part, the part is updated. If it contains an assembly, all the components of the assembly are updated.

- `-cgo[s]`

  Considers that the specified directory is of CGOS style and not UNIX style.

- `-che[ck]`

  Does not update drawing and shading files. Only reports to the standard output, the names of the parts and the missing files that contain the shading information.

- `-dra[w] name`

  For each part, the script updates only the drawing named `name`. If you do not specify the draw option, the script updates all available drawings. If there are no suitable drawings to be updated, then the script creates a default drawing named `UPDATE-GR`.

- `-pat[hsuffix] xxx`

  Specifies that CVPATHxxx is used instead of CVPATH

Please note: Use this option carefully, since it modifies the value of the size that you have already set.

- `-nosha[de]`

  Does not create tessellated data in the `_gr` file when you save the part.

Please note: Use this option carefully because you can view the model in any of the shade or HLR rendering modes in the assembly environment only if the `_gr` file contains tessellated data.

- `smo[othness] n`

  Specifies the smoothness of the shaded picture. You can specify any value between zero and 1.0. The default value is 0.3.

- `directory`

  Specifies the directory that contains the parts or assemblies to be updated. You can specify the path to more than one directory by separating them with spaces.

Environment Variables

- `KEEPTMP`

  Set this variable to 1, to save to the temporary directory, the log files generated while executing the `update_gr` script.

- `TMPDIR`

  Specifies the directory to be used as the temporary directory. The `update_gr` script uses this directory to store the working files and the results.

The update_fig Utility

The `update_fig` script is used for updating the figures in the pre-CADDS 5 15.0 parts, so that they can be displayed in the new Explicit SWR environment.

Please note: The `update_fig` script updates subfigures and nodal figures but not draw mode figures. You must update the draw mode figures manually. For details see Manually Updating Draw Mode Figures.

The `update_fig` script also reports:

* The list of model mode figures in the given directory

* The list of draw-mode figures

* The list of model mode figures, which could not be updated due to errors. These errors could be due to the presence of lock files in the part or due to other part activation errors. You must try activating such parts to know the exact nature of the error.

The `update_fig` script is available in the following location: `/usr/apl/cadds/scripts/update_fig`

`Update_fig` is a batch tool that can be run on all the parts in a specified directory.

Syntax

```
update_fig [-cgo] [-che] [-sfig] [-nfig] [-help] [directory]
```

Options

* `-cgo[s]`

  Considers that the specified directory is of CGOS style and not UNIX style.

* `-che[ck]`

  Does not update the figures. Reports the names of the figures that the `update_fig` script can update, and the names of the figures than you should update manually by activating and filing them in the new Explicit SWR environment.

* `-help`

  Displays the instructions to use the `update_fig` script

- `-sfig`

  Updates only the sub figures.

- `-nfig`

  Updates only the nodal figures.

Please note: If you execute the `update_fig` command without specifying either the `-sfig` option or the `-nfig` option, then both subfigures and nodal figures are updated.

- `directory`

  Specifies the directory that contains the parts or assemblies to be updated. You can specify the path to more than one directory by separating them using empty spaces. You can also specify the absolute path or the relative path of the directory.

  In some cases, after you update the library figures using the `update_fig` script, if you try to insert them into the active part, you may get the following message:

  ```
  Subfigure's parent part had a different name when the
  subfigure was prepared than it does at this time.
  ```

  ```
  The subfigure is being inserted under the name you
  requested
  ```

  To avoid this, ensure that if you are using a relative path for the figure in CADDS, you must provide the same relative path for the `update_fig` script also.

  For example, if a figure is located in the directory `/usr/apl/cadds/data/cvpd/stru/el`, and if you use the figure name `cvpd.stru.el` in CADDS, then you must update this figure by running the `update_fig` script from the `/usr/apl/cadds/data` directory, and using the following commands:

  `update_fig -nfig cvpd` (to update all figures in `cvpd` directory)

  `update_fig -nfig cvpd/stru/el` (to update only this figure)

Environment Variables

- `KEEPTMP`

  Set this variable to 1, to save to the temporary directory, the log files generated while executing the `update_fig` script.

- `TMPDIR`

  Specifies the directory to be used as the temporary directory. The `update_fig` script uses this directory to store the working files and the results.

Manually Updating Draw Mode Figures

Since the `update_fig` script does not update the draw mode figures, you must manually update the draw mode figures in your pre-CADDS 5 15.0 parts, so that you can view them in the new Explicit SWR environment.

Please note: You can use the `update_fig` script with the `-check` modifier to get a list of draw mode figures in a directory.

To update the draw mode figures manually, follow the steps below:

1. Activate the part containing the draw mode figures to be updated.

   `ACTIVATE PART pname`

2. If the part has multiple drawings, activate the drawing from which the draw mode figure should be created.

   `ACTIVATE DRAWING dname`

   `SELECT MODE DRAW`

3. If the drawing has only one view, and creating nfig then run the following command.

   `FILE PART NFIG DRAW dname`

4. If the drawing has only one view, and creating sfig then run the following command.

   `FILE PART  SFIG DRAW dname`

5. If drawing has multiple views, then specify the view name also.

   For nfig:

   `FILE PART  NFIG DRAW dname VIEW vname`

   For sfig:

   `FILE PART  SFIG DRAW dname VIEW vname`

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