CONFIGURATION AND USE

SAP ECTR INTERFACE TO PTC CREO

English 2021 - 01

Contact:	www.riess.de
Version:	2021 - 01
Validity:	SAP ECTR interface to PTC Creo ≥ 1.1.29
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SAP Engineering Control Center interface to PTC CREO (short: **SAP ECTR interface to PTC Creo**) is developed by .riess engineering. For information on .riess engineering and **SAP ECTR interface to PTC Creo**, please refer to our website.

Visit us at: http://www.riess.de

SAP Community Network

Join the **SAP Community Network**! It provides valuable information and discussion forums on SAP ECTR.

https://wiki.scn.sap.com/wiki/display/PLM/FAQ+-+SAP+ECTR

Legend

Symbols

The symbols in the margin have the following meaning:

User actions		Text classification		Text classification (highlighted)	
Symbol	**Meaning**	**Symbol**	**Meaning**	**Symbol**	**Meaning**
	Instruction		Note		Note
	Example		For further reference		Link
	Setting		Warning		Warning
	Tip		Prerequisite		Prerequisite
	Recommended		SAP note		Forbidden
	Not recommended		Error / symptom		Error / symptom
			Cause / question		Cause / question
			Solution / answer		Solution / answer
			Configuration / parameter		
			Validity		
			Comment / hint		
			Note (with placeholder)		

Formatting

In this text, the following formatting is used:

Format	Meaning
bold	Commands and buttons in procedures are formatted in bold. Important information in the text is highlighted in bold.
italics	References to other parts of the text and hyperlinks are formatted in italics.
`Courier`	Filenames, folders, and paths are formatted in Courier font.
`Courier`	Options, parameters, and their values are formatted in Courier font.
*	An asterisk is used to mark the default value of an option or parameter.
`[ ]`	Square brackets mean that no option or parameter value has been set.
`<user input>`	<user input> is formatted in Courier and enclosed in angle brackets.
`%path%`	Placeholders for paths are printed with the % sign and using Courier.

Default values

Options and parameters for which no explicit values are specified use the default value.

Inhaltsverzeichnis

This section informs you about the objective and the intended audience of this manual.

1 Objective and intended audience of this manual

1.1 Target

This manual has been designed to provide you with all information required to work with **SAP ECTR interface to PTC Creo** and to configure **SAP ECTR interface to PTC Creo**.

SAP ENGINEERING CONTROL CENTER documentation

For information on how to use and administer SAP ENGINEERING CONTROL CENTER (short: SAP ECTR), please refer to the SAP ECTR documentation.

Select **Hilfe ▶ Dokumentation anzeigen (Help ▶ Display Documentation)** to display the documentation directly from within SAP ECTR.

1.2 Intended audience

This manual is intended for administrators and users of **SAP ECTR interface to PTC Creo**.

1.2.1 Administrators

This manual is intended for all persons responsible for the installation, administration, and maintenance of **SAP ECTR interface to PTC Creo**.

The responsible persons should have the following knowledge related to PTC Creo Parametric and SAP:

- Administering the PTC Creo Parametric CAD system
  - Installing PTC Creo Parametric
  - Configuring PTC Creo Parametric (incl. `config.pro`, `parametric.bat`)
- Administration of SAP ECTR
  - Installing SAP ECTR
  - Configuring SAP ECTR (incl. `dtype.xml`, `default.txt`)
  - Understanding the concept and application of SAP ECTR
- Administering SAP systems, including:
  - Importing SAP notes / transports
  - Using SAP Download Center
  - Knowledge of the SAP Document Management System (documents, workstation applications)
  - Performing customizing (including transactions: `SPRO`, `CSADMIN`, `CDESK_CUS` [**ECC**] and `CAD_SRV_CUS` [**S4/Hana**], respectively)
  - Creating and transferring customization transports
  - Implementing, enabling and disabling `BAdI`
  - Importing `Business Configuration Set`

1.2.2 Users

The How to use the menu and commands section of this manual is intended for administrators and users of **SAP ECTR interface to PTC Creo**.

Users should have the following knowledge related to PTC Creo Parametric and SAP:

- Using PTC Creo Parametric
  - Knowledge and understanding of the dependencies and references between CAD objects
- SAP PLM concepts and applications
  - Understanding and using SAP ENGINEERING CONTROL CENTER
  - Understanding and using the SAP Document Management System
  - Creating and editing documents
  - Creating and editing material master records
  - Creating and editing bills of material
  - Creating and editing change master records

*The Introduction contains general information and basics related to **SAP ECTR interface to PTC Creo**.*

2 Introduction

SAP ECTR interface to PTC Creo integrates PTC Creo Parametric with das SAP ENGINEERING CONTROL CENTER. You can launch **SAP ECTR interface to PTC Creo** together with PTC Creo Parametric from within SAP ECTR.

2.1 SAP ECTR interface to PTC Creo TOOLKIT tool

SAP ECTR interface to PTC Creo is registered and embedded into PTC Creo Parametric as a TOOLKIT tool (auxiliary application).

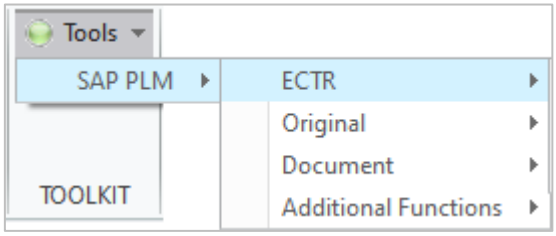


*Figure: **SAP ECTR INTERFACE TO PTC Creo** TOOLKIT tool*

SAP ECTR interface to PTC Creo allows direct access to SAP ECTR functions from within the CAD session.

Among the functions provided by **SAP ECTR interface to PTC Creo** are the following:

- Open SAP ECTR
- Create originals including document info record
- Save originals to SAP and load from SAP
- Import files to SAP
- Refresh CAD parameters (attributes) using SAP field data and vice versa
- View and edit documents

2.2 Data model

The data model used to map CAD data from PTC Creo Parametric to SAP PLM uses the following objects, among others:

- Documents and object types
- Document structures and sort terms.

2.2.1 Documents and object types

Any file that is created with PTC Creo Parametric and is to be managed in SAP PLM will require a document info record. The CAD file will be linked to the document info record. CAD files with an existing document info record are referred to as “originals” or “CAD originals”. Documents consist of a document info record and an original.

PTC Creo Parametric allows the creation of different types of CAD originals. Depending on the CAD type, a corresponding entry in the **Objekttyp (Object Type)** (`Reserve Field 4`) will be added to the document info record of a CAD original. This entry is called an “object type”.

Object type

The object type consists of two parts.

- Application type
- Application role

The application type and application role are specified as part of the `DType` definition. In the object type, the application type and application role are separated by a colon, e.g., `PRO:P`.

Application type

The application type depends on the CAD system used. The application type for PTC Creo Parametric is `PRO`.

Application role – Object types

The application role is created based on the type of the CAD original. Further information is then added depending on the function of the original.

CAD type	Extension	Application role
Sketch	`*.sec`	`S`
Part	`*.prt`	`P`
Assembly	`*.asm`	`A`
Manufacturing	`*.asm (*.mfg)`	`M`
Drawing	`*.drw`	`D`
Format	`*.frm`	`F`
Report	`*.rpt`	`R`
Diagram	`*.dgm`	`T`
Notebook	`*.lay`	`L`
User-Defined Feature	`*.gph`	`H`

Table: Object types of originals created with PTC Creo Parametric – Type

Application role – Object type additions

Depending on its type and function the application role may contain multiple flags. The following additions are possible:

Function	Application role
Family table – generic file	G
Family table – instance	I
Skeleton	S
Standard part / library part	N

Table: Object type additions

Object types with additions

The following table shows a selection of possible object types for parts:

Type	Function	Object type
Part	Design part	PRO:P
Part	Skeleton in assembly	PRO:PS
Part	Family table – generic file	PRO:PG
Part	Family table – instance	PRO:PI
Part	Design part – standard part / library part	PRO:PN
Part	Family table – generic file – standard part / library part	PRO:PGN
Part	Family table – instance – standard part / library part	PRO:PGN

Table: Selection of possible object types

2.2.2 Document structures and sort terms

Relationships within CAD structures, e.g., between an assembly and its components, are defined using document structures (document BOMs).

The relationships between documents in the document structure and the document of the BOM header are defined at the BOM position in the **sort term** in the `SORTF` field.

Sort term

The sort term consists of two parts:

- Application type
- Reference type

In the sort term, the application type and reference type are separated by a colon, e.g., `PRO:C`.

Application type

The application type depends on the CAD system used. The application type for PTC Creo Parametric is `PRO`.

2.2.2.1 Sort terms

Sort terms

The creation of the reference type is based on the type of the referenced object. By default, **SAP ECTR interface to PTC Creo** distinguishes the following sort terms:

Reference type	Subtype	Sort term
Assembly	Component	`PRO:C`
	Suppressed components	`PRO:CS`
Reference	General reference	`PRO:Y`
Family table	Instance generic file	`PRO:I`
	Instance of a generic file	`PRO:I`
Drawing	Represented Modell	`PRO:DC`
	Additional represented model	`PRO:C`
	Drawing format used	`PRO:C`
Notebook	Controlling model notebook	`PRO:L`

Table: Reference types – Standard

2.2.2.2 Sort terms in CAD Desktop compatibility mode

Sort terms in CAD Desktop compatibility mode

The creation of the reference type is based on the type of the reference target. In CAD Desktop compatibility mode, **SAP ECTR interface to PTC Creo** distinguishes the following sort terms:

Reference type	Subtype	Sort term
Assembly	Component	`PRO:C`
	Suppressed components	`PRO:CS`
	Work set	`PRO:CV`
Reference	Automatically updated reference	`PRO:CE`
	Manually updatable reference	`PRO:CY`
	Referenced interchange assembly	`PRO:CA`
Family table	Instance generic file	`PRO:I`
	Instance of a generic file	`PRO:I`
Drawing	Represented Modell	`PRO:DM`
	Additional represented model	`PRO:D`
	Drawing format used	`PRO:F`
Notebook	Controlling model notebook	`PRO:L`

Table: Reference types – Standard

Special case: Sort terms for mergings and inheritances

References in features that were created from mergings and inheritances may optionally be interpreted as components (`PRO:CN`).

2.2.3 Icons for originals created with PTC Creo Parametric

In SAP ECTR, certain document-related information is displayed in an icon format. In addition to general icons that apply to all documents, special icons exist for the PTC Creo Parametric object types. Depending on the document status, the background of object type icons may have a specific color.

Additional information on the object created with PTC Creo Parametric is displayed in an overlay on top of the object type icon.

Example

The following figure shows the object type icon of the object created with PTC Creo Parametric (first icon from the left). The icons at the other positions are general icons.

Figure: Document icons

Configuring Icons

In the file `dtype.xml` you can specify the icons to be displayed and their order of display for each `DType`. The status-dependent background color for the object type icons is defined in the file `default.txt`.

More information

For further details on the icons and overlays in SAP ECTR, please refer to the SAP ECTR documentation.

2.2.3.1 Object type icons for PTC Creo Parametric

The object type icon is specified using the XML attribute `application_role` in the `DType` definition.

Object type icons

Object type icons for PTC Creo Parametric:

Icon	Meaning
	**Sketch**
	**Part**
	**Assembly**
	**Manufacturing**
	**Drawing**
	**Format**
	**Report**
	**Diagram**
	**Notebook**
	**User-Defined Feature**

Table: Object type icons in SAP ECTR for PTC Creo Parametric

2.2.3.2 Object type overlays for PTC Creo Parametric

Object type overlays – supplemental object type information

The following overlays provide supplemental information on the object type:

Overlay	Meaning
	Family table – generic file
	Family table – instance

Table: Object type overlays – supplemental object type information

Object type overlays – flags

These are overlays that display additional information:

Overlay	Meaning
	Object has `Dirty Flag`
	Object is obsolete

Table: Object type overlays – flags

2.3 Launch SAP ECTR interface to PTC Creo

PTC Creo Parametric is started from within SAP ECTR. The CAD session will then be connected to SAP ECTR.

Each time you start PTC Creo Parametric, **SAP ECTR interface to PTC Creo** is registered as a TOOLKIT tool and embedded into the application.

Starting PTC Creo Parametric from SAP ECTR

Start PTC Creo Parametric from within SAP ECTR.

1) Start SAP ECTR.
2) Select **System ▶ Applikation starten ▶ PTC Creo starten (System ▶ Launch Application ▶ Launch PTC Creo)**.

Result: This launches PTC Creo Parametric and embeds **SAP ECTR interface to PTC Creo** as a TOOLKIT tool. The session is connected to SAP ECTR.

SAP ECTR interface to PTC Creo user interface language

The user interface language of **SAP ECTR interface to PTC Creo** is determined (automatically) as follows:

- PTC Creo Parametric will open with the user interface language defined in the the language variable, e.g., `set LANG=english`, of the PTC Creo Parametric start file, e.g., `%PTC_CREO_PARAMETRIC_INSTDIR%\Parametric\bin\Parametric.bat`.
- If no language variable has been set or if it is invalid, PTC Creo Parametric will start with the current operating system language.
- **SAP ECTR interface to PTC Creo** will be integrated with the same language as used by PTC Creo Parametric.
- If this user interface language is not supported by **SAP ECTR interface to PTC Creo** yet, **SAP ECTR interface to PTC Creo** will be started in `English`.

*This chapter contains information on the system requirements for **SAP ECTR interface to PTC Creo**. In addition, it describes how to install and configure **SAP ECTR interface to PTC Creo**.*

3 System requirements and installation

SAP ECTR interface to PTC Creo can only be installed successfully if the system requirements are met. The installer must be downloaded from the **SAP Software Download Center** first. **SAP ECTR interface to PTC Creo** can then be installed using the `SAP Front-End Installer`.

Update installations are equally performed through the `SAP Front-End Installer`.

3.1 Notes in the SAP Support Portal

The following important notes are available in the **SAP Support Portal**:

Installation and patches

2112629 **SAP ECTR interface to PTC Creo**:
Installation and patches

Extensions

2992107 SAP ECTR interface to PTC Creo:

.riess enhancements for the support of the SAP ECTR interface to PTC Creo

Licensing for SAP ECTR interface to PTC Creo

2107448 **SAP ECTR interface to PTC Creo**:
Licensing for SAP ECTR interface to PTC Creo

Support process

2107391 **SAP ECTR interface to PTC Creo**:
Support process for .riess engineering gmbh for SAP ECTR interface to PTC Creo

Release strategy for SAP ECTR

2049249 SAP ECTR:
Release strategy for ABAP Add-on ECTR

Improvement requests

2307014: SAP ECTR:
Improvement request for SAP ECTR

Product availability matrix

For information on the currently available software, please refer to the **ECTR INTERFACE TO CREO 1.x** and **SAP ECTR 5.x** entries in the **Product Availability Matrix**.

https://support.sap.com/release-upgrade-maintenance/pam.html

3.2 System requirements

The following requirements for the systems / components listed must be met in order to use **SAP ECTR interface to PTC Creo**:

System	Prerequisite
SAP ECTR	See note 2112629
PTC Creo Parametric	See note 2112629
Hardware	See PTC Hardware Notes

*Table: System requirements for **SAP ECTR interface to PTC Creo***

SAP ECTR

In order to use SAP ECTR further system requirements must be met, in particular with respect to the SAP system. For further details, please refer to the SAP ECTR documentation.

3.3 Downloading the software

SAP ECTR interface to PTC Creo is available for download from the SAP Software Download Center.

Prerequisite

The software is only available as a download for customers with a valid contract or trial contract. Without any contract, you will not have access to the software in the SAP Software Download Center.

Access permissions for SAP Software Download Center

To download the software, you must be logged in as an **S user** who has the permission for software download.

Procedure

Download **SAP ECTR interface to PTC Creo** from the SAP Software Download Center.

1) Log in to the SAP Software Download Center:
   https://support.sap.com/swdc
   or
   https://launchpad.support.sap.com/#/softwarecenter
2) Using the **By Alphabetical Index (A-Z)** search mode, select the search letter **E** and search for `ECTR INTERFACE TO CREO`.
3) In the results list, select the current software version.
4) Click the **Support Packages and Patches** icon.
5) Click the **Comprised Software Component Versions** icon.
6) Download the most recent patch from the SAP Software Download Center.

Result: **SAP ECTR interface to PTC Creo** is now available on your workstation.

3.4 Installation

SAP ECTR interface to PTC Creo is installed using the `SAP Front-End Installer`. During the installation, the installer creates the installation directory and various subdirectories.

Note 2992107

⚠ Make sure that transport was implemented in your SAP system for the SAP note 2992107.

Extensions

2992107 SAP ECTR interface to PTC Creo:

.riess enhancements for the support of the SAP ECTR interface to PTC Creo

3.4.1 SAP Front-End Installer

Prerequisite

⚠ In order to install **SAP ECTR interface to PTC Creo**, SAP ECTR must already have been installed.

SAP Front End Installer

The `SAP Front-End-Installer` of **SAP ECTR interface to PTC Creo** auto-detects the installed SAP ECTR version. The `SAP Front-End-Installer` dialog displays the version for which **SAP ECTR interface to PTC Creo** is installed in a green font.

Procedure

Install **SAP ECTR interface to PTC Creo**.

1) Unzip the `Download Object`previously downloaded from SAP Software Download Center to a folder on your workstation.
2) Run the `EctrCreo<xx>P<y>.exe` program.
3) Follow the instructions in the SAP Front-End Installer.
4) Exit the SAP Front-End Installer.

Result: **SAP ECTR interface to PTC Creo** is installed to the `%PLM_INSTDIR%\applications\pro` folder of the SAP ECTR install directory.

Registrierung

When **SAP ECTR interface to PTC Creo** is launched for the first time from SAP ECTR, it will be automatically registered as a TOOLKIT tool.

3.4.2 Installation directory

The installation directory `%PLM_INSTDIR%\applications\pro` of **SAP ECTR Interface to PTC Creo** contains the following folders:

- `addons`
- `appdata`
- `basis`
- `customize`
- `documentation`
- `template-customize`
- `template-templates`
- `templates`

3.4.2.1 “addons” folder

The `addons` folder contains add-ons to **SAP ECTR Interface to PTC Creo**.

3.4.2.2 “appdata” folder

The `appdata` folder contains various components of the SAP ECTR interface to PTC Creo application (Software, `Preferences` and `Properties`).

3.4.2.3 “basis” folder

The `basis` folder contains files of SAP ECTR that are required for **SAP ECTR interface to PTC Creo**.

3.4.2.4 “customize” folder

The `customize` folder contains the SAP ECTR configuration files that are required for using **SAP ECTR interface to PTC Creo**, such as the file `dtype.xml`.

Editing files in the “customize folder” %PLM_INSTDIR%\applications\pro

Edit the files in the `customize` folder in order to perform a user-specific and CAD system-specific configuration of SAP ECTR and SAP ECTR interface to PTC Creo. For further details, please refer to the SAP ECTR documentation.

Update

Please note that, if the `customize` folder already contains files, they will **not** be overwritten when you update **SAP ECTR interface to PTC Creo** using the SAP Front-End Installer.

3.4.2.5 “documentation” folder

The `documentation` folder contains the **SAP ECTR interface to PTC Creo** documentation.

3.4.2.6 “template-customize” folder

The `template-customize` folder contains templates for the SAP ECTR configuration files that are required for using **SAP ECTR interface to PTC Creo**, such as the file `dtype.xml`.

The `template-customize` folder of **SAP ECTR interface to PTC Creo** contains the following subfolders:

- `bc_sets`
- `config`
- `creo`
- `license`
- `scripts`

Initial installation

When you install **SAP ECTR interface to PTC Creo** for the first time on your workstation, the entire content of the `template-customize` folder is copied to the `customize` folder.

3.4.2.7 “templates” folder

The `templates` folder contains the template files (start files) that will be used as templates (`seedfiles`) when creating new originals.

Using company-specific start files

Copy your own company-specific start files to the `templates` directory. Set the paths to your start files in your company-specific `config.pro` configuration file. Add the file names of your start files to the file `dtype.xml`.

Update

Please note: If the `templates` folder already contains any files, these files will **not** be overwritten when you update **SAP ECTR interface to PTC Creo** using the SAP Front-End Installer.

3.4.2.8 “template-templates” folder

The `template-templates` folder contains the templates for template files (start files) that will be used as templates when creating new originals.

Initial installation

When you install **SAP ECTR interface to PTC Creo** for the first time on your workstation, the entire content of the `template-templates` folder is copied to the `templates` folder.

3.5 Installing updates

Any updates to **SAP ECTR interface to PTC Creo** are installed using the `SAP Front-End Installer`. During the installation, the content of the installation directory and most subdirectories will be overwritten.

“templates” and “customize” folders

⚠ Please note that the content of the `templates` and `customize` folders will **not** be overwritten when updating **SAP ECTR interface to PTC Creo**.

The template files for PTC Creo Parametric (start files) in the `templates` folder that are used as templates (`seedfiles`) when creating new originals will be kept.

The configuration files in the `customize` folder will also be kept.

Editing configuration files

If an update of **SAP ECTR interface to PTC Creo**contains new configuration options, you can edit the configuration files in the `customize` folder to add the new options.

Comparing configuration files

After each update check if new configuration options are available. Compare the configuration files in the `%PLM_INSTDIR%\applications\pro\template-customize\config` folder with your own configuration files in the `%PLM_INSTDIR%\applications\pro\customize\config` folder. The files `default.txt` and `mapkeys.txt` in particular may contain important changes or enhancements.

Configuration dialog – Save

When opening the configuration dialog **SAP ECTR interface to PTC Creo**, the **Save** button may be active immediately. The **Save** button is always active after changes were applied to the configuration. If this is the case, new configuration options were added by the update but have not yet been saved. The new configuration options are set to their respective default values.

3.6 Licensing

SAP ECTR interface to PTC Creo requires a valid license check file in the directory `%PLM_INSTDIR%\applications\pro\customize\license`.

License check file for production systems

In order to use **SAP ECTR interface to PTC Creo** on your production system, you need a license check file. If more users than indicated in the license check file work with **SAP ECTR interface to PTC Creo**, a corresponding message is displayed. This license check file allows you to use **SAP ECTR interface to PTC Creo** also on non-production systems (development, test, QA, etc.).

Temporary license check file for non-production systems

In order to use **SAP ECTR interface to PTC Creo** on your non-production systems, you need a temporary license check file.

Licensing for SAP ECTR interface to PTC Creo

2107448 **SAP ECTR interface to PTC Creo**:
Lizenzierung für SAP ECTR interface to PTC Creo (Licensing for SAP ECTR interface to PTC Creo)

Deploying your configuration through the SAP system

If the configurations for SAP ECTR and **SAP ECTR Interface to PTC Creo** are deployed through the SAP system, the license check file needs to be located in the path defined by the preference `PLM_SAPCONFIG_BASEDIR`.

Deployment of the configuration via the SAP system is configured in the file `%PLM_INSTDIR%\customize\config\plm_initialize.bat`.

- `set PLM_USE_SAPCONFIG=TRUE`
- `set PLM_SAPCONFIG_BASEDIR=<Path To Configuration Dir>`

If no path is specified through `PLM_SAPCONFIG_BASEDIR`, SAP ECTR will use the following path:

`C:\Users\<User>\AppData\Local\SAP\ECTR\conf\<PLM_INSTDIR>\<SytemID>_<Client>_<SAPUser>\applications\pro\customize\license`

This chapter explains how to configure SAP ENGINEERING CONTROL CENTER INTERFACE PTC CREO.

4 Configuration

SAP ECTR interface to PTC Creo must be configured in the context of other systems interacting with it. When configuring SAP ECTR interface to PTC Creo, keep in mind that the configurations of one system may have an impact on one or several other systems of the overall ecosystem. Optionally, configurations may be deployed in a central manner through the SAP system.

4.1 Configurable systems

In order to integrate PTC Creo Parametric with **SAP ECTR interface to PTC Creo** and SAP ECTR, the following systems need to be configured:

- SAP
- SAP ECTR
- PTC Creo Parametric
- SAP ECTR interface to PTC Creo

As an alternative, you can use the pre-configured configuration files that were been copied to the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` during the installation of **SAP ECTR interface to PTC Creo**.

Prerequisite

⚠ Before starting to configure **SAP ECTR interface to PTC Creo** make sure that SAP ECTR has been configured properly, as described in the SAP ECTR documentation.

4.2 Deploying configurations through the SAP system

Deploying your configuration through the SAP system

⚠ The paths to config files contained in this manual generally refer to the installation directory of an SAP ECTR client on the CAD workstation (`%PLM_INSTDIR%`). If you deploy configurations through the SAP system, SAP ECTR and **SAP ECTR interface to PTC Creo** will use the configurations stored in the directories used for the deployment of configurations. In this case, replace the path to `%PLM_INSTDIR%` with the path to `%PLM_SAPCONFIG_BASEDIR%`.

Make sure that these directories contain your current configurations.

Configuring the deployment of configurations through the SAP system

🛈 Deployment of the configuration via the SAP system is configured in the file `%PLM_INSTDIR%\customize\config\plm_initialize.bat`. Use the following preferences:

- `set PLM_USE_SAPCONFIG=TRUE`
- `set PLM_SAPCONFIG_BASEDIR=<Path To Configuration Dir>`

If no path is specified through `PLM_SAPCONFIG_BASEDIR`, SAP ECTR will use the following path:

`C:\Users\<User>\AppData\Local\SAP\ECTR\conf\<PLM_INSTDIR>\<SytemID>_<Client>_<SAPUser>\...`

More information

🛈 For detailed information on deploying your configuration through the SAP system, please refer to the SAP Community Wiki:

https://wiki.scn.sap.com/wiki/display/PLM/How+to+configure+the+distribution+of+SAP+ECTR+configurations+via+SAP+system

4.3 Configuration files

4.3.1 Read and write permissions

Configuring **SAP ECTR interface to PTC Creo** requires read and write permissions on the files in the directory `%PLM_INSTDIR%\applications\pro\customize\config`.

4.3.2 Configuration files

The functionality of **SAP ECTR interface to PTC Creo** is configured in the following file:

- `SAPECTRInterface.prefs`

Additionally, the SAP ECTR functions required to use **SAP ECTR interface to PTC Creo** are configured in the following files:

- `dtype.xml`
- `default.txt`
- `menu.guidef`
- `menu_macros.txt`
- `standard_icons.txt`
- `attributes-from-sap.xml`
- `attributes-to-sap.xml`
- `load_scenarios.xml`
- `mapkeys.txt`
- `sheet_format.txt`
- `pro_bom_base_unit_of_measure.txt`
- `example_pro_pdf_export.dop`

More information

For information on how to use the SAP ECTR configuration files, please refer to the SAP ECTR documentation.

Editing configuration files

Use a suitable editor, e.g., `Notepad++`, to edit the additional configuration files.

4.3.3 Settings file SAPECTRInterface.prefs

The `SAPECTRInterface.prefs` file contains static settings that will not be changed by **SAP ECTR interface to PTC Creo**.

Location of the file SAPECTRInterface.prefs

`%PLM_INSTDIR%\applications\pro\customize\config\SAPECTRInterface.prefs`

Editing the file `SAPECTRInterface.prefs`

Do NOT edit the file `SAPECTRInterface.prefs` with an external editor. Use the **SAP ECTR interface to PTC Creo** configuration dialog instead. The configuration dialog also serves to save your settings. The settings will be written to the file `SAPECTRInterface.prefs` upon saving.

Other system settings

Other settings are stored in the file `%APPDATA%\Roaming\.riess\SAPECTRInterface\SAPECTRInterface.prefs`. This file contains dynamic settings that are made automatically by **SAP ECTR interface to PTC Creo**. These settings may not be edited manually.

4.3.4 Using the pre-configured configuration files

When **SAP ECTR interface to PTC Creo** is installed for the first time, the preconfigured configuration files from the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` are copied to the directory `%PLM_INSTDIR%\applications\pro\customize\config`.

The settings defined in the various configuration files are coordinated. The SAP objects specified in the configuration files, e.g., for workstation applications, match the specifications in the SAP ECTR documentation.

The following objects must be created in the SAP system unless they are added through the `Business Configuration Set`.

- Workstation applications
- Document types

Using configuration templates

The pre-configured configuration files help you set up a working environment for **SAP ECTR interface to PTC Creo** quickly. You can then successively customize this environment to suit your specific requirements.

Business Configuration Sets

Parts of the required configuration in the SAP system may also be added through the `Business Configuration Set`. The SAP ECTR backend installation already contains a suitable `Business Configuration Set` for **SAP ECTR interface to PTC Creo**: `/DSCAG/ECTR_PRO`.

The `Business Configuration Set` in the directory `%PLM_INSTDIR%\applications\pro\template-customize\bc_sets` may be used as an alternative.

4.3.5 Deploying configuration files

Make sure that all CAD workstations run the same software versions and use the same configuration files, e.g., by copying the SAP ECTR installation directory from a central reference workstation to all other workstations.

The SAP ECTR and **SAP ECTR interface to PTC Creo** configuration files may also be stored in SAP as documents and deployed to CAD workstations through SAP.

Deploying the SAP ECTR configuration through the SAP system

For information on deploying your configuration, please refer to the SAP Community Wiki: https://wiki.scn.sap.com/wiki/display/PLM/Best+Practices+regarding+Configuration+and+Installation under How to configure the distribution of SAP ECTR configurations via SAP system

Storage location of configuration files

Note that configuration files deployed through SAP are stored in the directory `%PLM_SAPCONFIG_BASEDIR%\%PLM_INSTID%\%PLM_SYSTEM%_%PLM_CLIENT%_%PLM_USER`. If configuration files are deployed through SAP, the configuration files in the SAP ECTR installation directory will not be read and used.

4.4 Configuring the SAP System

SAP ECTR interface to PTC Creo and SAP ECTR store CAD files in the SAP Document Management System. For this purpose, workstation applications and document types including a status network and a classification are required.

The required document types and status networks depend on the customer-specific design process mapping concept.

Document types and workstation applications are required to define the `DType`.

Detailed information

For details on how to configure the SAP system, please refer to the SAP ECTR documentation in the directory `%PLM_INSTDIR%\documentation\Manuals`.

4.4.1 Workstation application for CAD originals

In addition to the workstation application mentioned in the SAP ECTR documentation, define the `PRO` workstation application.

Creating the PRO workstation application

Create the `PRO` workstation application for CAD originals in the SAP Document Management System.

1) Use the `DC30` transaction to open the **Define workstation application** view.
2) Click the **New Entries** icon.
3) Define the following settings for the `PRO` workstation application:

Workstation application	
WS application	`PRO`
Description	`Creo Parametric`
File format	`*.*`
☑ **Start Authorization**	
☑ **You cannot rename temporary files**	

Table: Settings for the PRO workstation application

4) Save your settings.

Result: The `PRO` workstation application has been crated and can be used for defining the `DType`.

You cannot rename temporary files

Make sure to enable the [☑] **You cannot rename temporary files** checkbox.

Workstation application for additional originals

If you create additional originals, e.g., for preview thumbnail images, when saving CAD originals in SAP, the corresponding workstation applications must be available in your SAP system.

4.4.2 Document types for CAD originals

Define the following document types for Originale aus PTC Creo Parametric:

Document type	File type	Extension
PPR	Part	*.prt
PAS	Assembly	*asm
PDR	Drawing	*.drw
PFR	Frame	*.frm
PMG	Manufacturing	*.asm/[*.mfg]
PUF	User-Defined Feature	*.rpt
PXY	All Other	*.sec/*.lay/*.cem/*.rpt/*.dgm/*.mrk

Table: Document types for Originale aus PTC Creo Parametric

Using other document types

If you need other document types, you can create them. Specify these document types by editing the corresponding DType in the XML attribute document_type in the file dtype.xml.

Defining document types

How to define document types for Originale aus PTC Creo Parametric:

1) Use the DC10 transaction to open the **Define document types** view.
2) Click the **New Entries** icon.
3) Define a document type for CAD originals by editing the input fields.
4) Save your settings.
5) Click the  icon.
6) In the dialog structure, click the new document type.
7) In the dialog structure, double-click **Define document status**.
8) Define the status network for the new document type.
9) Save your settings.

Result: The document type has been defined and can now be used for CAD originals and for defining the DType.

Document type attributes

Make sure to define the attributes of the document type as follows:

Defining document types	
Use KPro	
☐ **Status Change**	
☐ **Version assign.**	
Def. Def. WS appl	PRO
Dis. WS applic.	PRO

Table: Defining document types

The other attributes can be defined in line with the customer-specific requirements.

Defining other document types

- First, define a document type including its status network. Then, copy the document type.

Document types and statuses

- Use numerical names for your document types and statuses. The document type and status descriptions can be maintained in multiple languages and edited at any time.

4.5 Configuring SAP ECTR

In order to use **SAP ECTR interface to PTC Creo**, SAP ECTR needs to be configured as well. Settings made in the following configuration files in the directory `%PLM_INSTDIR%\applications\pro\customize\config` will have an influence on the behavior of SAP ECTR interface to PTC Creo:

- `dtype.xml`
- `default.txt`
- `menu_macros.txt`
- `attributes-from-sap.xml`
- `attributes-to-sap.xml`
- `SAPECTRInterface.prefs`

Detailed information

For details on how to configure SAP ECTR, please refer to the SAP ECTR documentation.

Defining the DType for PTC Creo Parametric files

Create a separate `DType` for each file type. Use a separate `seedfile` for each subtype. Please note that additional `DType` definitions will be required for family tables (generic files and instances).

Types in PTC Creo Parametric

Among others, PTC Creo Parametric defines the following types:

- `Skizze` (Sketch)
- `Teil` (Part)
- `Baugruppe` (Assembly)
- `Fertigung` (Manufacturing)
- `Zeichnung` (Drawing)
- `Format` (Format)
- `Notizbuch` (Notebook)

Examples

Sample configuration files can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config`.

The file `%PLM_INSTDIR%\applications\pro\template-customize\config\dtype.xml` contains sample `DType` definitions.

For further reference

For further information on how to configure SAP ECTR, please refer to the **SAP Community Network** using the following links:

- https://wiki.scn.sap.com/wiki/display/PLM/CAD+Integration+in+PLM
- https://wiki.scn.sap.com/wiki/display/PLM/FAQ+-+SAP+ECTR

4.6 Configuring PTC Creo Parametric

SAP ECTR interface to PTC Creo requires the following options and values to be set in the `config.pro` file:

Option	Value
`allow_create_pdm_param`	`yes`
`enable_tree_indep`	`yes`
`file_open_default_folder`	`working_directory`
`let_proe_rename_pdm_objects`	`yes`
`load_ui_customization_run_dir`	`yes`
`override_store_back`	`yes`
`remember_replaced_components`	`no`
`retrieve_data_sharing_ref_parts`	`ignore_missing`
`retrieve_instance_dependencies`	`instance_and_generic_deps`
`retrieve_merge_ref_parts`	`ignore_missing`
`save_instance_accelerator`	`none`
`save_object_in_current`	`no`
`save_objects`	`changed_and_specified`
`sel_insts_on_comp_retrieval`	`no`
`tk_enable_ribbon_custom_save`	`yes`
`trail_dir`	`<Path>` e.g., `C:\Temp`
`visible_message_lines`	`<Number>` e.g., `4`

Table: Required options and values in the config.pro file

Purpose of the user-specific config.pro file

During the installation of **SAP ECTR interface to PTC Creo**, a file named `config.pro` is copied to the **SAP ECTR interface to PTC Creo** installation directory. This file contains the settings required for **SAP ECTR interface to PTC Creo**. Add your user-specific settings to this file.

If you do not save your `config.pro` file to the SAP ECTR interface to PTC Creo installation directory, make sure that your user-defined `config.pro` file contains the required options and is read when PTC Creo Parametric is launched.

In this case, delete the `config.pro` file from the **SAP ECTR interface to PTC Creo** installation directory.

4.7 Starting SAP ECTR interface to PTC Creo

SAP ECTR interface to PTC Creo is started together with PTC Creo Parametric from within SAP ECTR. When **SAP ECTR interface to PTC Creo** is started for the first time, the configuration dialog appears.

Procedure

To start PTC Creo Parametric for the first time from SAP ECTR:

1) Start SAP ECTR.
2) Select **System** ▶ **Launch Application** ▶ **Launch PTC Creo**.
3) In the **PTC Creo Start File** field, enter the path to the PTC Creo Parametric start file, e.g., `C:\Program Files\PTC\Creo 4.0\M010\Parametric\bin\parametric.bat`.
4) In the **Execute in** field, enter the path to the PTC Creo Parametric startup directory, e.g., `C:\SAP PLM start`.
5) Click the **Start** icon.

Result: PTC Creo Parametric has been started. **SAP ECTR interface to PTC Creo** has been embedded as a TOOLKIT tool. The configuration dialog is displayed.

4.8 Displaying SAP ECTR interface to PTC Creo

The graphical user interface (GUI) of SAP ECTR interface to PTC Creo is preconfigured to integrate with PTC Creo Parametric at various locations.

The **SAP PLM** menu is displayed:

- in the **TOOLKIT** group on the **Tools** tab of the ribbon
- in the right-click menu of the Model Tree
- on the quick access toolbar
- Context menu of the SAP ECTR application structure

Optionally, the **SAP PLM** menu can be displayed at the following user-defined locations:

- on a user-defined tab of the ribbon
- on the General tab of the ribbon

The ribbon and quick access toolbar can be configured using the standard methods available in PTC Creo Parametric.

The right-click menu can be configured in the **SAP ECTR interface to PTC Creo** configuration dialog.

Right-click menu of the Model Tree

Instead of using the preconfigured **SAP PLM** right-click menu in the Model Tree you may also extend the **Shortcut Menus** of PTC Creo Parametric by adding a **User Defined Section**.

4.8.1 Configuring the ribbon

By default, **SAP ECTR interface to PTC Creo Parametric** is delivered with a preconfigured tab on the ribbon. The **SAP PLM** tab may be configured differently based on the CAD mode.

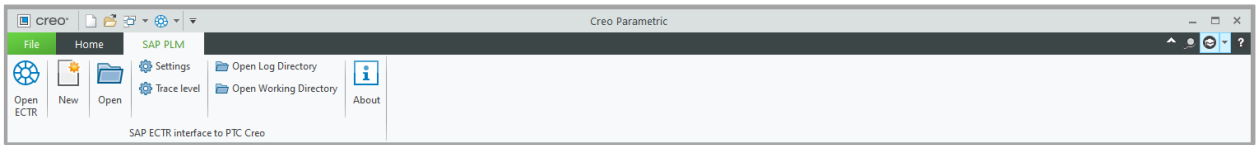


Figure: SAP PLM tab in Stand By mode

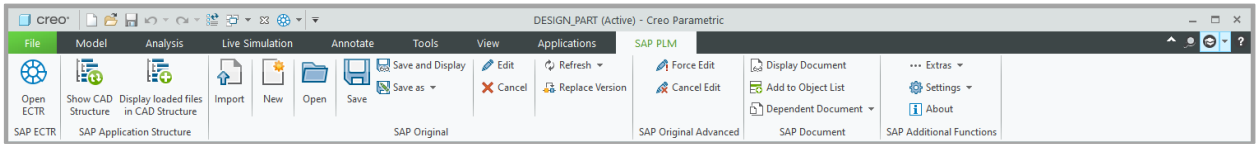


Figure: SAP PLM tab in Design Part mode

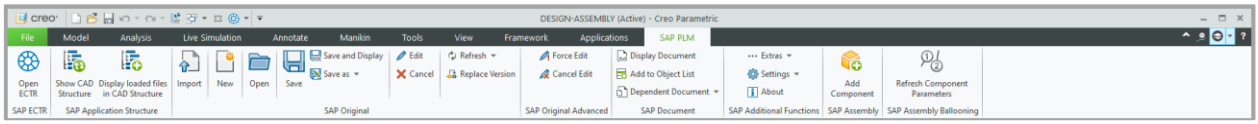


Figure: SAP PLM tab in Design Assembly mode

The configuration of the **SAP PLM** tab is saved in the following files:

- `toolkitribbon.rbn`
  The file `%PLM_INSTDIR%\applications\pro\appdata\lib\text\toolkitribbon.rbn` contains the order of groups, buttons and overlays (cascades) for each application as well as the icons used.
- `ribbonui.txt`
  The file `%PLM_INSTDIR%\applications\pro\appdata\lib\text\<language>ribbonui.txt` contains the language-specific names of groups and overlays (cascades).

You can customize the ribbon to suit your specific requirements. The available options for customization of the ribbon depend on your version of PTC Creo Parametric. For instance, PTC Creo Parametric 4.0 and above provide the **Common** tab.

config.pro file / tk_enable_ribbon_custom_save configuration option

⚠ In the `config.pro` file, set the `tk_enable_ribbon_custom_save` option to `yes`. This will allow you to save the specific ribbon configuration for each application in the file `toolkitribbon.rbn`.

4.8.1.1 Customizing the ribbon

You can customize the **SAP PLM** tab of the PTC Creo Parametric ribbon for each application to suit your individual requirements.

- Start PTC Creo from within SAP ECTR using `English` as the language setting.
- Enable the appropriate [☑] checkboxes to define the modes in which a group of TOOLKIT commands should be displayed.

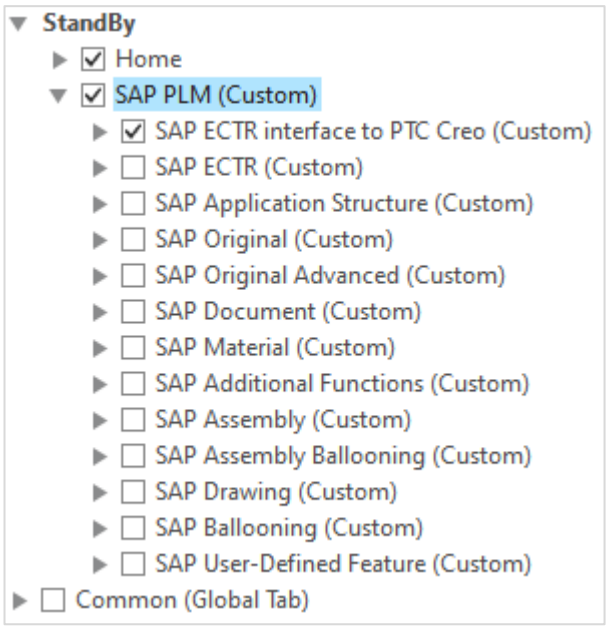


Figure: SAP PLM tab in Stand By mode

- Add TOOLKIT commands to the groups or remove commands from the groups that are not required.

SAP PLM tab

⚠ The design of the preconfigured **SAP PLM** tab ensures that any changes to the tab, its groups and overlays within the groups are reflected in all PTC modes.

Example: User-Defined Feature group

For instance, if the commands of the **SAP User-Defined Features** group appear as inactive because they are not used, you may hide that group from display.

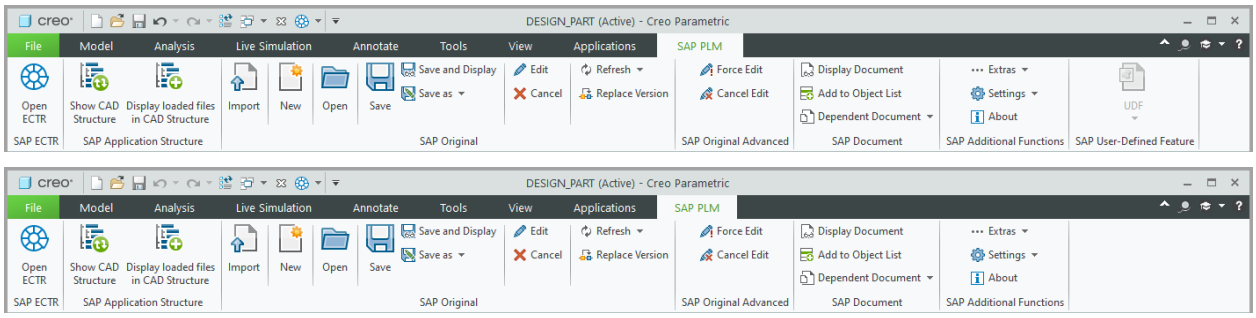


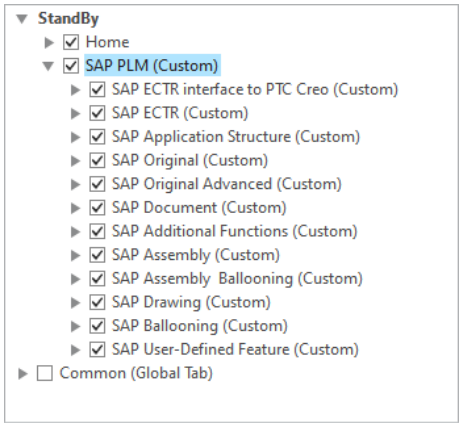
Figure: SAP PLM tab with / without the SAP User-Defined Features group

4.8.1.2 Creating a customer-specific ribbon

You can create a customer-specific **SAP PLM** tab of the PTC Creo Parametric ribbon for each application from scratch to suit your individual requirements.

Always perform the steps in the order indicated below:

- Delete the existing file `%PLM_INSTDIR%\applications\pro\appdata\lib\text\toolkitribbon.rbn`.
- Start PTC Creo Parametric from within SAP ECTR using `English` as the language setting.
- Configure the ribbon using `English` as the language setting.
- In a first step, create an empty **SAP PLM** tab, i.e., without any content, for each required mode of PTC Creo Parametric (Drawing, Parts, Assemblies, etc.).
- In **StandBy** mode, create groups. These will be used for grouping the SAP commands.



- Add TOOLKIT commands from the **SAP PLM** menu to the groups.
- Group the newly added commands into overlays (cascades) as required.
- Use large buttons for your favorite commands.
- Use icons when creating overlays.
- Define the modes in which a group of TOOLKIT commands should be displayed.
- Save the configuration to the file `toolkitribbon.rbn`.
- Translate the names of the tabs, groups, and overlays in the localization file `ribbonui.txt`.

Creating a user-defined tab

Only add commands from the **SAP PLM** menu to user-defined tabs. This ensures that the commands from the user-defined tab will apply to the same scope as the commands from the **SAP PLM** menu on the **Tools** tab.

4.8.1.3 Editing the ribbon

Using English as the source language

If you would like the ribbon customizations to apply to other languages and translate the names of tabs and groups, select `English` as the user interface language. Start PTC Creo Parametric using `English` as the language setting and create the tabs and groups in `English`. Then translate the names of tabs and groups in the localization file `ribbonui.txt`.

Creating tabs

How to create the **SAP PLM** tab:

1) Start PTC Creo Parametric using the `English` language setting.
2) From the right-click menu of the ribbon, select **Customize the Ribbon**.
3) Create a new tab for each required mode.
4) Rename the tabs for each mode, e.g., to **SAP PLM**.
5) Confirm the **Propagate Customization** dialog with **Yes**.
6) Use the arrow keys to move the tabs to the desired locations.
7) Click the **OK** icon.

Result: The **SAP PLM** tab is displayed in the ribbon.

Creating groups

How to create the groups of the SAP PLM tab:

1) Start PTC Creo Parametric using the `English` language setting.
2) From the right-click menu of the ribbon, select **Customize the Ribbon**.
3) Enter a mode, e.g., **StandBy**, and create the groups according to your requirements.
4) Rename the groups as desired.
5) Use the arrow keys to move the tabs and groups to the desired location.
6) Click the **OK** icon.

Result: The tab is displayed in the ribbon.

Existing groups

Be careful to avoid creating any groups on the SAP PLM tab that already exist on other tabs. If the **Propagate Customization** dialog is displayed when you rename a newly created group, this group already exists on a different tab in a different context.

Unique group name

Add a prefix or suffix to your group name and make sure the group name is unique, e.g., **SAP Document** instead of **Document**.

Adding TOOLKIT commands

How to add commands to the TOOLKIT group of the tab:

1) From the right-click menu of the ribbon, select **Customize the Ribbon**.
2) Click the desired group in the main tab.
3) From the **Category** list box, select the **TOOLKIT Commands** option.
4) Select the desired command.
5) Use the arrow keys to move each TOOLKIT command to the desired location.
6) Click the **OK** icon.

Result: The TOOLKIT commands are displayed in the groups.

Selecting TOOLKIT commands

Make sure to add only TOOLKIT commands from the **SAP PLM** menu to the ribbon and not from the right-click menu of the Model Tree in **SAP PLM**. For commands with identical names, always use the topmost command from the selection list.

Recognizing TOOLKIT commands from the right-click menu of the ribbon

If you accidentally select TOOLKIT commands from the **SAP PLM** right-click menu of the Model Tree to be displayed in the ribbon, these TOOLKIT commands will be inactive and cannot be used unless an object is marked in the Model Tree. Replace these commands in the ribbon.

Disabled commands

If you disable the display of commands of the SAP PLM menu in the **SAP ECTR interface to PTC Creo** configuration, they will not be removed from the ribbon. Disabled commands are displayed in gray. Disabled commands should be removed from the ribbon.

Using large buttons

How to use large buttons for your favorite TOOLKIT commands in a group:

1) From the right-click menu of the ribbon, select **Customize the Ribbon**.
2) Expand the desired group on the tab.
3) Select the desired TOOLKIT command.
4) Click the **Modify** icon.
5) Select ⊙ **Large Button**.
6) Click the **OK** icon.

Result: Your favorite TOOLKIT commands are displayed with large buttons.

Moving commands to the overflow menu

Do the following to move rarely used TOOLKIT commands to a group’s overflow menu:

1) Move the mouse pointer to the desired command.
2) From the right-click menu, select **Move to Overflow**.
3) Click the **OK** icon.

Result: The rarely used TOOLKIT commands are now located in the group's overflow menu.

Cascading commands

The TOOLKIT commands in a group menu can be cascaded as overlays.

1) From the right-click menu of the ribbon, select **Customize the Ribbon**.
2) Expand the desired group.
3) Click the location where you want to create a new cascade.
4) Click **Add New Cascade**.
5) Rename the cascade.
6) Use the arrow keys to move the cascade to the desired location.
7) Use the arrow keys to move the TOOLKIT commands to the desired location in the cascade.
8) If required, use a `<Seperator>` to group the TOOLKIT commands into subsets.
9) Click the **OK** icon.

Result: The TOOLKIT commands are now grouped in cascades and structured by separators.

Using icons for cascades

Icons from **SAP ECTR interface to PTC Creo** resources may be placed in front of cascades.

1) From the right-click menu of the ribbon, select **Customize the Ribbon**.
2) Expand the desired group.
3) Select the desired cascade.
4) Click the **Modify** icon.
5) Click **Choose Image Name**.
6) In the **Choose Image Name** dialog, mark the line **Creo ECTR Integration**.
7) Click **OK**.
8) In the **Choose Image Name** dialog, enter the name of the icon without the file extension.
9) Click **OK**.

Result: The icon is displayed in front of the cascade.

Icons included in SAP ECTR interface to PTC Creo resources

The directory `%PLM_INSTDIR%\applications\pro\appdata\lib\text\usascii\resource` contains icons that may be used for cascades. Select the desired icon file name without the file extension and without the `_large` suffix.

4.8.1.4 Using the General tab

Starting with PTC Creo Parametric 4.0, commands from the **SAP PLM** menu may be used as custom commands in the **Common** tab of the ribbon.

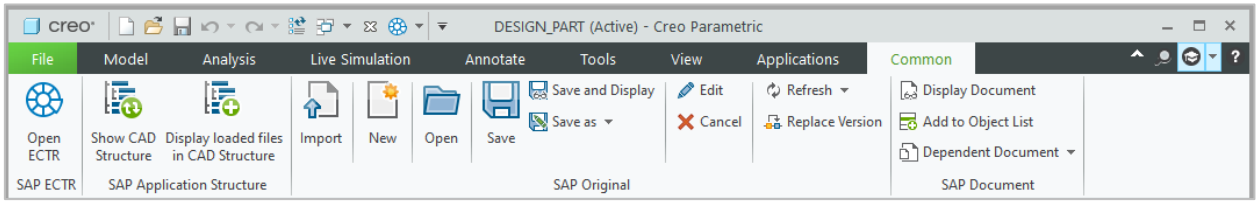


Figure: Commands from the SAP PLM menu in the Common tab

4.8.2 Configuring the quick access toolbar

SAP ECTR interface to PTC Creo is delivered with a preconfigured overlay (cascade) on the quick access toolbar. The **SAP ECTR interface to PTC Creo** cascade is configurable.

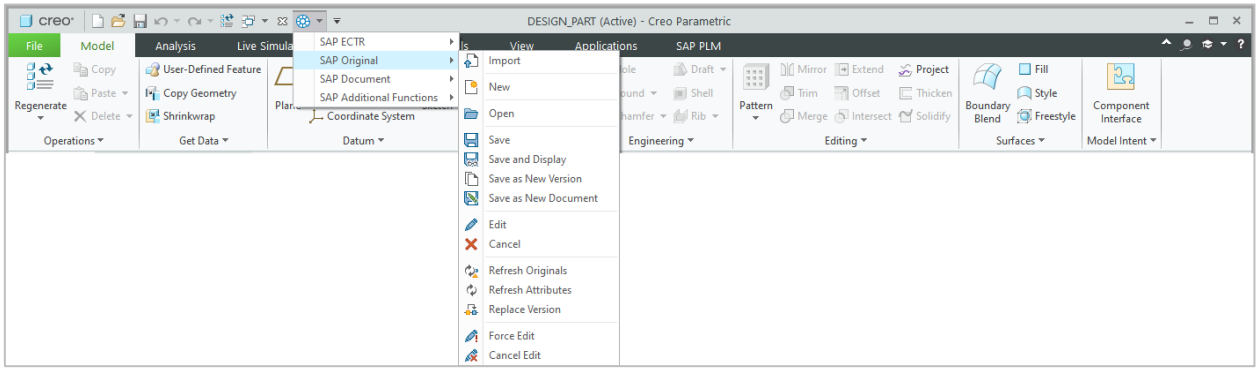


*Figure: **SAP ECTR interface to PTC Creo** on the quick access toolbar*

Configuring the quick access toolbar

How to configure the **SAP ECTR interface to PTC Creo** cascade:

1) Start PTC Creo Parametric using the `English` language setting.
2) From the right-click menu of the ribbon, select **Customize Quick Access Toolbar**.
3) Configure the toolbar using the same functions that you used in the ribbon configuration.
4) Click the **OK** icon.

Result: The **SAP ECTR interface to PTC Creo** cascade is displayed in the quick access toolbar.

4.8.3 Exporting and loading settings

The customization file `toolkitribbon.rbn` only contains application-specific configuration settings related to the ribbon and the quick access toolbar.

When you start a TOOLKIT tool for which a `toolkitribbonui.rbn` file exists, the functions configured in the customization file `toolkitribbonui.rbn` are added to the ribbon.

Exporting the application-specific customization file toolkitribbonui.rbn

Do the following to export the ribbon customizations:

1) Launch PTC Creo Parametric together with **SAP ECTR interface to PTC Creo**.
2) From the right-click menu of the ribbon, select **Customize the Ribbon** or **Customize Quick Access Toolbar**.
3) Click **Export**.
4) Select **Save the Auxiliary Application User Interface**.

Result: The application-specific customization file `toolkitribbonui.rbn` has been saved to the `text` folder of the helper application, e.g., `%PLM_INSTDIR%\applications\pro\appdata\lib\text`.

Loading the application-specific customization file toolkitribbonui.rbn

How to load the application-specific customization file `toolkitribbonui.rbn`:

If an application-specific customization file `toolkitribbonui.rbn` exists in the `text` folder of the TOOLKIT tool, the application-specific file `toolkitribbonui.rbn` will be loaded when the TOOLKIT tool is launched. The application-specific customization file `toolkitribbonui.rbn` adds new features to the existing PTC Creo Parametric ribbon.

4.8.4 Translating the ribbon

The commands added to the ribbon tabs and groups are displayed automatically in the language selected for the user interface.

The names of tabs, groups, and cascades, however, need to be translated **manually**. These names are translated in the localization file `ribbonui.txt`.

The localization file ribbonui.txt

The localization file `ribbonui.txt` must be stored in the appropriate language directory of the TOOLKIT application, e.g., `%PLM_INSTDIR%\applications\pro\appdata\lib\text\german`.

English as original language

Please note that you cannot change the names of tabs and groups in the `English` source language by means of the localization file `ribbonui.txt`. If you need to change them, first define the tabs and groups in the `English` source language and then create the language-specific file `ribbonui.txt`.

Creating the file ribbonui.txt

How to create a `ribbonui.txt` file for each desired language:

1) Open a text editor.
2) Create a file named `ribbonui.txt`.
3) Edit the file `ribbonui.txt` by adding the following 4 lines for each label (i.e., tab and group name):

Line 1:
character, followed by the label name as defined in the file `toolkitribbonui.rbn`.

Line 2:
Name of the label as defined in the file `toolkitribbonui.rbn`.

Line 3:
Desired translation of the label.

Line 4:
Blank line (no content).

4) Save the file `ribbonui.txt` in the appropriate language directory, e.g, inside the `%PLM_INSTDIR%\applications\pro\appdata\lib\text\german` folder.

When you start PTC Creo Parametric, the tabs and group names will be displayed in the desired language.

Sample localization file ribbonui.txt

A localization file `ribbonui.txt` containing the German translations of a file `toolkitribbonui.rbn` created for the `English` language might look like this:

```
#SAP PLM¶
SAP PLM¶
SAP PLM¶
¶
#SAP Original¶
SAP Original¶
SAP Original¶
¶
#SAP Additional Functions¶
SAP Additional Functions¶
SAP Zusätzliche Funktionen¶
¶
```

UTF-8 encoding

⚠ If you set your editor to use `UTF` encoding, make sure to save the localization file `ribbonui.txt` with the `UTF-8 (with BOM)` encoding.

Blank lines

⚠ Make sure to add blank lines as separators between the labels in the localization file `ribbonui.txt`.

Also do not forget to add a blank line at the end of the localization file `ribbonui.txt`.

4.9 Configuring SAP ECTR interface to PTC Creo

A configuration dialog is available for the configuration of **SAP ECTR interface to PTC Creo**.
Any settings applied in the configuration dialog will be stored in the file `SAPECTRInterface.prefs`.

You can only fully benefit from the scope of functionality of **SAP ECTR interface to PTC Creo** if the entire system has been configured accordingly. This usually implies editing various configuration files of the systems involved.

The following functions can be configured:

- Startup behaviour and CAD environment
- Menus and dialogs
- Application structure
- Loaing, opening, and editing of originals
- Saving and importing originals
- Importing originals
- Document structures and material BOMs
- Refreshing attributes and initializing parameters
- Managing drawing formats in SAP PLM
- Directories
- Using PTC Creo ModelCHECK for saving to SAP PLM
- Default limitations
- SmartContainers
- Settings for family tables
- Ballooning functions
- User-Defined Features (UDFs)
- External libraries
- Macros
- Cloning
- PTC Creo Unite Technology
- CAD Desktop compatibility mode
- Icons depending on their status
- Preferences

4.9.1 Opening configuration dialog

You can open the configuration dialog from the **SAP PLM (SAP PLM)** menu or using the `configure.exe` application.

In this dialog, the available configuration options are grouped by topics and displayed on separate pages.

The configuration dialog consists of the following pages:

- **Start**
- **Menu**
- **Dialogues**
- **Save**
- **Import**
- **Parameters**
- **Directories**
- **Ballooning**
- **ModelCHECK**
- **Advanced**

Other settings

⚠ Please note that settings in the SAP ECTR configuration files are required in addition to the settings in the configuration dialog for **SAP ECTR interface to PTC Creo** in order to fully benefit from the features of **SAP ECTR interface to PTC Creo**.

The file configure.exe in the installation directory

ⓘ The `configure.exe` application is stored under the installation directory in the following path:

`%PLM_INSTDIR%\applications\pro\appdata\configure.exe`.

Permissions

⚠ Write permissions on the directory `C:\Users\<Username>\AppData\Roaming\.riess\SAPECTRInterface` are needed.
SAP ECTR interface to PTC Creo will not start without write permissions.

Write permissions on the directory `%PLM_INSTDIR%\applications\pro\customize\config` are needed.
SAP ECTR interface to PTC Creo cannot be configured without write permissions.

Opening the configuration dialog from within SAP ECTR interface to PTC Creo

How to open the configuration dialog from the **SAP PLM** menu:

1) Click **SAP PLM** ▶ **Additional Functions** ▶ **Settings** ▶ **Settings**.
2) Configure **SAP ECTR interface to PTC Creo**.
3) Save your settings.

Result: The configuration dialog is open and the settings have been saved.

Alternative: Opening the configuration dialog with configure.exe

How to open the configuration dialog with `configure.exe`:

1) In Windows Explorer, navigate to the directory `%PLM_INSTDIR%\applications\pro\data`.
2) Start `configure.exe`.
3) Configure **SAP ECTR interface to PTC Creo**.
4) Save your settings.

Result: The configuration dialog is open and the settings have been saved.

Closing the configuration dialog

Close the configuration dialog by clicking  .

4.9.2 Configuring SAP PLM, startup behavior and CAD environment

You can influence the startup behavior of **SAP ECTR interface to PTC Creo** by:

- Starting PTC Parametric and **SAP ECTR interface to PTC Creo** inside a CAD environment
- Starting **SAP ECTR interface to PTC Creo** with a defined user interface language
- Starting **SAP ECTR interface to PTC Creo** within different CAD environments
- Integrating **SAP ECTR interface to PTC Creo** into CAD environments started with external launchers
- Starting **SAP ECTR interface to PTC Creo** as the last auxiliary application
- Setting the trace level

Required entries

The `%PLM_INSTDIR%\applications\pro\customize\config\SAPECTRInterface.prefs` file must contain the following entries:

- **PTC Creo Start-Datei (PTC Creo Start File)**
  `de.riess.proeectr.wildFireExe =<Path\PTC_CREO_PARAMETRIC_START_FILE>`
- **Ausführen in (Execute in)**
  `de.riess.proeectr.proeExecFolder =<Path\PTC_CREO_PARAMETRIC_START_DIR>`

4.9.2.1 Starting PTC Parametric and SAP ECTR interface to PTC Creo inside a CAD environment

On the **Start** page you can specify the files and paths that are required to start **SAP ECTR interface to PTC Creo**.

In order for **SAP ECTR interface to PTC Creo** to be embedded and started as a TOOLKIT tool in PTC Creo Parametric, all the required parameters need to be set on the **Start** page.

If you start PTC Creo Parametric for the first time from SAP ECTR and **SAP ECTR interface to PTC Creo** has not been configured completely on the **System** page, the configuration dialog will open automatically.

To start PTC Creo Parametric from within SAP ECTR, use the option **System ▶ Launch Application ▶ Launch PTC Creo**.

In the **PTC Creo Start File** field, specify the path to the start file to be used for starting PTC Creo Parametric.

Example

`%PTC_CREO_PARAMETRIC_INSTDIR%\Parametric\bin\parametric.bat`

In the **Execute in** field, specify the path to the directory you would like to use as the start directory for PTC Creo Parametric.

Example

`C:\Data\CAD_Start_Directory`

Specifying the start file for PTC Creo Parametric

In the **PTC Creo Start File** field, specify the part to the start file for PTC Creo Parametric.

1) Click inside the **PTC Creo Start File** field.
2) Specify the full path to the start file including its file name or use **Browse** or specify the appropriate environment variable.
3) Save your settings.

Result: **System ▶ Launch Application ▶ Launch PTC Creo** will start PTC Creo Parametric using the specified start file.

Path to the start file

Depending on the location of the start file, **SAP ECTR interface to PTC Creo** determines other paths, e.g., to start the corresponding version of **SAP ECTR interface to PTC Creo** that matches the version of PTC Creo Parametric.

If you intend to use a start file other than `%PTC_CREO_PARAMETRIC_INSTDIR%\Parametric\bin\parametric.bat`, it must be located in the directory `%PTC_CREO_PARAMETRIC_INSTDIR%\parametric\bin`.

Using an environment variable

Instead of entering the path, you can specify an environment variable that contains the path to the startup directory of PTC Creo Parametric. If you use environment variables, it is not possible to start PTC Creo Parametric directly from within the configuration dialog. Instead, do the following to start PTC Creo Parametric from within SAP ECTR:

Example

`%PTC_CREO_PARAMETRIC_START_FILE%`

Using uppercase letters

Please note that the environment variable must always be written in UPPERCASE letters.

Example

```
Set PTC_CREO_PARAMETRIC_START_FILE=
%PTC_CREO_PARAMETRIC_INSTDIR%\Parametric\bin\parametric.bat
```

Specifying the start file for PTC Creo Parametric

In the **Execute in** field, specify the path to the start file for PTC Creo Parametric.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Start** page.
3) Click inside the **Execute in** field.
4) Specify the full path to the start file including its file name or use **Browse** or specify the appropriate environment variable.
5) Save your settings.

Result: **System ▶ Launch Application ▶ Launch PTC Creo** will start PTC Creo Parametric using the specified startup directory.

Using an environment variable

Instead of the path, you can specify an environment variable that contains the path to the startup directory. If you use environment variables, it is not possible to start PTC Creo Parametric directly from within the configuration dialog. Instead, do the following to start PTC Creo Parametric from within SAP ECTR:

Example

```
%PTC_CREO_PARAMETRIC_EXECUTE_IN%
```

Using uppercase letters

Please note that the environment variable must always be written in UPPERCASE letters.

Example

```
Set PTC_CREO_PARAMETRIC_EXECUTE_IN=C:\Data\CAD_Start_Directory
```

Saving config.pro in the startup directory

Make sure to save your user-defined configuration files for PTC Creo Parametric, e.g., `config.pro`, in the startup directory.

4.9.2.2 Starting SAP ECTR interface to PTC Creo with a defined user interface language

The user interface language of **SAP ECTR interface to PTC Creo** is determined **automatically** as follows:

- PTC Creo Parametric will start with the user interface language defined in the start file of PTC Creo Parametric, e.g., `%PTC_CREO_PARAMETRIC_INSTDIR%\Parametric\bin\Parametric.bat`.
- The language with which PTC Creo Parametric will be started is set through the `LANG` language variable, e.g., `set LANG=english`.
- If no language variable has been set or if it is invalid, PTC Creo Parametric will start with the current operating system language.
- **SAP ECTR interface to PTC Creo** will be integrated with the same language as used by PTC Creo Parametric.
- If this user interface language is currently not supported by **SAP ECTR interface to PTC Creo**, **SAP ECTR interface to PTC Creo** will be started with `English` as the user interface language.

4.9.2.3 Starting SAP ECTR interface to PTC Creo within different CAD environments

Due to varying requirements or customers, it might be necessary to work with PTC Creo Parametric in different CAD environments. **SAP ECTR interface to PTC Creo**, together with SAP ECTR, supports you in switching between these environments.

When starting PTC Creo Parametric, you can select a defined CAD environment. Using a batch file allows you to explicitly specify the start file `parametric.bat` of PTC Creo Parametric and the startup directory (Execute in) to be used. For example, PTC Creo Parametric reads the `config.pro` file stored in the startup directory and starts up using the configuration defined in that file. Using a batch file, you can also set other environment variables that are specific to the CAD environment and copy additional files to the startup directory.

When opening an original, **SAP ECTR interface to PTC Creo** checks, together with SAP ECTR, whether the active CAD environment matches the one specified for the original. The CAD environment is derived automatically from specific environment properties.

CAD environment properties

The various CAD environments can be distinguished based on the following CAD environment properties:

- **CAD Version**
- **Date Code**
- **Environment**

You can create a separate CAD startup directory for each CAD environment. PTC Creo Parametric will be executed from this startup directory and will use the configuration files stored in that directory, among others.

For each CAD environment, a corresponding batch file must be available that will be executed before PTC Creo Parametric is launched. The batch files must contain environment variables indicating the path to the corresponding PTC Creo Parametric start file and startup directory.

Additionally, you can use a batch file to copy environment-specific PTC Creo Parametric configuration files, such as the `config.pro` file, to the startup directory.

If different CAD environments require different `seedfile`s, you need to check in the corresponding `seedfile` to SAP PLM. The `DType` definition needs be adapted accordingly.

Typical use cases for different CAD environments

Possible use cases may include:

- Launching PTC Creo Parametric from within SAP ECTR using a selected environment.
- Launching PTC Creo Parametric by opening an original in SAP ECTR.
- PTC Creo Parametric has already been launched and an original is being opened from SAP ECTR.

Launching PTC Creo Parametric from within SAP ECTR using a selected environment

If you launch PTC Creo Parametric from SAP ECTR using **Launch PTC Creo**, a selection dialog displays. In the selection dialog, you can select the desired CAD environment from a list of available CAD environments. With **Launch PTC Creo**, PTC Creo Parametric will then be launched in the selected environment. SAP ECTR will then assign the corresponding CAD environment properties to all CAD originals you create or modify in the selected CAD environment or to the associated documents.

- **CAD Version**
- **Date Code**
- **Environment**

Launching PTC Creo Parametric by opening an original from SAP ECTR

If you launch PTC Creo Parametric by opening an original from SAP ECTR, a selection dialog displays. The selection dialog displays and defaults to the CAD environment derived from the CAD environment properties ([☑] check box). You can use **Launch PTC Creo** to launch PTC Creo Parametric in the default CAD environment. If you want to open the original in a different CAD environment, select it in the selection dialog ([☑] checkbox).

No CAD environment specified

If no CAD environment has been associated with the original, there is no default CAD environment in the selection dialog.

Opening an original from SAP ECTR while PTC Creo Parametric is already running

If you open an original from SAP ECTR with PTC Creo Parametric already being started in a selected CAD environment, SAP ECTR compares the CAD environment properties with the running CAD environment. If the CAD environment matches these properties, the original will be opened in PTC Creo Parametric. If the CAD environment does **not** match these properties, a dialog will display. There, the user can select whether to cancel opening the original or to open the original in the currently running CAD environment.

No CAD environment specified

If no particular CAD environment is associated with the original, the original will be started in the currently running environment.

Prerequisite

In order to work with multiple CAD environments, make sure that the following prerequisites are met:

Configuration of SAP ECTR

- The preference `plm.check.environment.PRO = true` must be set in the file `default.txt`.

Configuration of SAP ECTR interface to PTC Creo

- The environment variables set in the batch files must be specified in the **SAP ECTR interface to PTC Creo** settings.
- The CAD environments must be defined and enabled.

CAD startup directories

- The CAD startup directories for each CAD environment must exist.
- The PTC Creo Parametric configuration files specific to each CAD environment must exist in the CAD startup directories.

Batch files

- The directory
  `%PLM_INSTDIR%\applications\pro\customize\scripts\batches\prepare_environment`
  must contain the batch files specific to that CAD environment.
- The batch files must contain environment variables indicating the path to the corresponding Startdatei von PTC Creo Parametric start file and startup directory.

Optionally: DType and seedfile

- For each CAD environment, you can define an environment-specific `DType` and `seedfile`.

Setting the SAP ECTR preference for the environment check

Set the preference `plm.check.environment.PRO` in the file `default.txt`.

1) Use a text editor, e.g., Notepad++, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.
2) Search for `plm.check.environment.PRO`.
3) Set `plm.check.environment.PRO = true`.

Result: The environment check for originals created with PTC Creo Parametric is enabled in SAP ECTR.

Defining and enabling the PTC Creo Parametric start environment

Only use the preference `plm.check.environment.PRO` if you have enabled the [☑] **Define and activate PTC Creo Parametric start environments** checkbox in the **SAP ECTR interface to PTC Creo** configuration dialog.

Defining and enabling CAD environments in SAP ECTR interface to PTC Creo

Do the following to configure and enable the CAD environments:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Start** page.
3) Click inside the **PTC Creo Start File** field.
4) Specify the required environment variable, e.g., `%PTC_CREO_PARAMETRIC_START_FILE%`.
5) Click inside the **Execute in** field.
6) Specify the required environment variable, e.g., `%PTC_CREO_PARAMETRIC_EXECUTE_IN%`.
7) Enable the [☑] **Define and activate PTC Creo Parametric start environments** checkbox.
8) Click the **New** icon.
9) Specify all properties for the environment (from left to right):
   CAD Version | Date Code | Environment
10) Click **OK**.
11) Repeat these steps for other CAD environments.
12) Click **Save** to save your settings.

Result: The CAD environments are now properly defined and enabled.

plm.check.environment.PRO

If you enable the [☑] **Define and activate PTC Creo Parametric start environments** check box, the preference `plm.check.environment.PRO = true` must be set in the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.
If you do not enable the [☑] checkbox, the preference `plm.check.environment.PRO = false` must be set.

Date Code

When entering the date code, make sure to specify it correctly, e.g.: `M010` or `6.0.3.0`.

Environment

Make sure to enter a meaningful name for the environment.

Batch files

If you have entered data in all of the **CAD Version | Date Code | Environment** fields, a suggested file name will be displayed in the **Batch File** field. Confirm the suggested filename. Alternatively, you can select a different filename or path. In this case, click the ... button.

Creating CAD startup directories

Do the following to create the CAD startup directories:

1) Create a separate CAD startup directory for each CAD environment.
2) Copy all PTC Creo Parametric configuration files relevant to the respective environment to these directories.

Result: The CAD startup directories have been created and **SAP ECTR interface to PTC Creo** can be run from there.

CAD working directories

The following figure shows an example of how the CAD startup directories could be organized:

```
C:\CAD Environment
      |→ Creo6_English
      |→ Creo6_German
```

Creating batch files

Do the following to create a separate batch file for each CAD environment:

1) Open a text editor, e.g., `Notepad++`.
2) Set environment variables that contain the following paths:
   - Path to the CAD start file
   - Path to the CAD startup directory
3) Add other entries to the batch file, depending on the further CAD environment requirements.
4) Store the file as a batch file to the following directory:
   `%PLM_INSTDIR%\applications\pro\customize\scripts\batches\prepare_environment`.

Result: Once you select a specific environment, the associated batch file will be executed. The PTC Creo Parametric start file indicated in the batch file is executed from the specified CAD startup directory.

Environment variables

Use the same environment variables as those specified in the **SAP ECTR interface to PTC Creo** configuration dialog, e.g.:

- **PTC Creo start file**
  `Set PTC_CREO_PARAMETRIC_START_FILE=`
  `C:\Program Files\PTC\Creo 6.0.3.0\Parametric\bin\parametric_en.bat`
- **Execute in**
  `Set PTC_CREO_PARAMETRIC_EXECUTE_IN=`
  `C:\CAD Environment\Creo6_English`

Encoding

When saving your batch files, make sure to use the correct encoding. In your editor, e.g., Notepad++, set the encoding to `ANSI`.

Quotes

When specifying the paths, **do not** use quotes (double inverted commas)!

Filenames

Use the filenames suggested in the **SAP ECTR interface to PTC Creo** configuration dialog for the respective environment, e.g.:

- `Creo 6.0_6.0.3.0_english.bat`
- `Creo 6.0_6.0.3.0_german.bat`

Checking in the seed file to SAP PLM

Do the following to create an environment-specific `seedfile`:

1) Start PTC Creo Parametric in the desired CAD environment.
2) Create a new `seedfile`.
3) Check in the `seedfile` to SAP PLM.
4) Release the `seedfile`.

Result: A document has been created for the new, released `seedfile`. The `seedfile` can be added to the `DType` definition.

Adding the seed file to the DType definition

Do the following to add the `seedfile` to the `DType` definition:

1) Edit the file `%PLM_INSTDIR%\applications\pro\customize\config\dtype.xml`.
2) Add the `seedfile` to the `document_create` XML element of the `DType` definition.

Result: The `seedfile` can now be used. When creating a new original, **SAP ECTR interface to PTC Creo**, together with SAP ECTR, checks whether the active CAD environment matches the one specified in the `seedfile`.

seedfile in the DType definition

Add the `seedfile` to the `DType` definition as shown in the following example:

```
<seedfile
      default="yes"
      type="template"
      workstation_application="PRO"
      docnumber="1000004711"
      doctype="PPR"
      docpart="000"
      filename_pattern=".+(?&lt;!(\.var|_gp))\.prt$"
      component_reference="no"
      appl_enwironment="english"
      appl_release="Creo 4.0"
      appl_release_ext="M010"
      >
      <description language="EN" text="Solid"/>
      <description language="DE" text="Volumenkörper"/>
</seedfile>
```

Make sure that the environment attributes match the configuration of **SAP ECTR interface to PTC Creo**:

Environment attributes	SAP ECTR interface to PTC Creo
`appl_environment`	**Environment**
`appl_release`	**CAD Version**
`appl_release_extern`	**Date Code**

Table: Environment attributes and matching configuration options

4.9.2.4 Integrating SAP ECTR interface to PTC Creo into CAD environments started with external launchers

According to the default configuration of **SAP ECTR interface to PTC Creo**, PTC Creo Parametric is started from within SAP ECTR. During startup of PTC Creo Parametric, **SAP ECTR interface to PTC Creo** is registered and embedded into PTC Creo Parametric as a TOOLKIT tool (auxiliary application).

If you use applications and startup tools to control your CAD environment, you will be able to change the startup behaviour of **SAP ECTR interface to PTC Creo**. **SAP ECTR interface to PTC Creo** will then be embedded into a previously started CAD environment.

To do so, set the following environment variables:

- `PLM_RIESS_ENABLE_EXTERNAL_START_AND_DELAY_TIME_FOR_CAD_SYSTEM`
- `PLM_RIESS_EXTERNAL_START_MAX_TRIES`

Batch files

Make sure to use batch files for setting environment variables.

PLM_RIESS_ENABLE_EXTERNAL_START_AND_DELAY_TIME_FOR_CAD_SYSTEM

The `PLM_RIESS_ENABLE_EXTERNAL_START_AND_DELAY_TIME_FOR_CAD_SYSTEM` environment variable modifies the startup behavior of **SAP ECTR interface to PTC Creo**. This way, **SAP ECTR interface to PTC Creo** can be integrated into an existing PTC Creo Parametric session that has already been started. The value of this environment variable indicates the startup delay in seconds. Once the startup delay has expired, **SAP ECTR interface to PTC Creo** is started and embedded into the running PTC Creo Parametric session.

PLM_RIESS_EXTERNAL_START_MAX_TRIES

There will be three attempts to embed **SAP ECTR interface to PTC Creo** into the externally started PTC Creo Parametric session before the waiting time specified by `PLM_RIESS_ENABLE_EXTERNAL_START_AND_DELAY_TIME_FOR_CAD_SYSTEM` expires. The `PLM_RIESS_EXTERNAL_START_MAX_TRIES` environment variable may be used to specify the number of attempts to be made before the waiting time expires.

Creating batch files

Do the following to create a separate batch file for each CAD environment:

1) Open a text editor, e.g., `Notepad++`.
2) Set the `PLM_RIESS_ENABLE_EXTERNAL_START_AND_DELAY_TIME_FOR_CAD_SYSTEM` environment variable
   and, if required, the `PLM_RIESS_EXTERNAL_START_MAX_TRIES` environment variable.
3) Set environment variables that contain the following paths:
   - Path to the CAD start file
   - Path to the CAD startup directory
4) Add other entries to the batch file, depending on the CAD environment requirements.
5) Store the file as a batch file to the following directory:
   %`PLM_INSTDIR%\applications\pro\customize\scripts\batches\prepare_environment`.

Result: Once you select a specific environment, the associated batch file will be executed. The PTC Creo Parametric start file indicated in the batch file is executed from the specified CAD startup directory.

Waiting time

Specify a waiting time (in seconds) that is slightly longer than the startup time of your CAD system, e.g.:
`set PLM_RIESS_ENABLE_EXTERNAL_START_AND_DELAY_TIME_FOR_CAD_SYSTEM=10`

Environment variables

Use the same environment variables as those specified in the **SAP ECTR interface to PTC Creo** configuration dialog, e.g.:

- **PTC Creo Start File**
  `Set PTC_CREO_PARAMETRIC_START_FILE=`
  `C:\Program Files\PTC\Creo 4.0\M010\Parametric\bin\parametric_en.bat`
- **Execute in**
  `Set PTC_CREO_PARAMETRIC_EXECUTE_IN=`
  `C:\CAD Environment\ Creo4_M010_English`

4.9.2.5 Starting SAP ECTR interface to PTC Creo as the last auxiliary application

In the event that auxiliary applications other than **SAP ECTR interface to PTC Creo** require embedding into PTC Creo Parametric, for technical reasons they must be completely started before starting up **SAP ECTR interface to PTC Creo**.

SAP ECTR interface to PTC Creo can be configured to be started only after the startup of all auxiliary applications specified in the file `creotk.dat` is complete.

SAP ECTR interface to PTC Creo contains the signalling auxiliary application `SignalNative.dll`. After all auxiliary applications have been started, this application will send a notification that **SAP ECTR interface to PTC Creo** can be started. The signalling auxiliary application is included in the delivery of **SAP ECTR interface to PTC Creo**.

Auxiliary applications will start in the order specified in the file `creotk.dat`. Each auxiliary application will start only after the start of the previous application is complete. Consequently, the signalling auxiliary application `SignalNative.dll` needs to be specified last in the file `creotk.dat`.

The startup behavior is further controlled by two environment variables.

- `PLM_RIESS_START_AND_WAIT_ENABLED [true | false]`
  `[true]` enables the startup behavior. **SAP ECTR interface to PTC Creo** will only start after the `CAD ready to start SAP ECTR` notification of the signalling auxiliary application has been received.
- `PLM_RIESS_START_AND_WAIT_MAX_TIME [SEC]`
  The `[SEC]` value specifies the waiting time (in seconds) after which an attempt to start up **SAP ECTR interface to PTC Creo** will be made if no `CAD ready to start SAP ECTR` notification of the signalling auxiliary application has been received. This setting is required if one of the previous auxiliary applications fails to start successfully, preventing the signalling auxiliary application `SignalNative.dll` from sending a notification.

Embedding the signalling auxiliary application

Add the signalling auxiliary application `SignalNative.dll` as the last auxiliary application in the file `creotk.dat`.

1) Use a text editor, e.g., `Notepad++`, to open the file `creotk.dat`.
2) Add the following lines at the end of the file (example):

```
name            CAD ready to start SAP ECTR

startup         dll

exec_file       $PLM_INSTDIR\applications\pro\customize\apps\SignalNative\pma6.0\SignalNative.dll

text_dir        $PLM_INSTDIR\applications\pro\customize\apps\SignalNative\pma6.0\

allow_stop      true

delay_start     false

end
```

3) Save the file `creotk.dat`.

Result: The signalling auxiliary application is started as the last auxiliary application. It will send a notification that all auxiliary applications have been started up and that **SAP ECTR interface to PTC Creo** can be started. The signalling auxiliary application will terminate after it has sent the notification.

Setting the environment variables for the signalling auxiliary application

Do the following to set the environment variables for the signalling auxiliary application `SignalNative.dll`:

1) Use a text editor, e.g., `Notepad++`, to open the start file for PTC Creo Parametric.
2) Add the following lines at the beginning of the file:

```
rem Wait until all auxiliary applications are started
rem ------------------------------------------------
set PLM_RIESS_START_AND_WAIT_ENABLED=true
set PLM_RIESS_START_AND_WAIT_MAX_TIME=180
```

3) Save the start file for PTC Creo Parametric.

Result: **SAP ECTR interface to PTC Creo** will only start after all other auxiliary applications have successfully started. If one or more auxiliary applications could not start successfully, an attempt will be made to start **SAP ECTR interface to PTC Creo** after the specified waiting time.

4.9.2.6 Setting the trace level

SAP ECTR interface to PTC Creo creates a session history and logs any errors that occur during the session. You can define the size and content of the log files by specifying a trace level.

There are three available trace levels:

- `ERROR`
  If the ERROR trace level is set, only errors will be logged.
- `INFO`
  If the INFO trace level is set, errors and additional information will be logged.
- `DEBUG`
  If the DEBUG trace level is set, errors, additional information and debug information will be logged.

Using the trace level

The `ERROR` trace level should only be used in tried and tested systems with proven stability. The `INFO` trace level should be used in the system implementation phase.

If the DEBUG trace level is used, the writing of a large amount of date to the log files will significantly lower the performance of your system. Therefore, you should use the `DEBUG` trace level for the purpose of troubleshooting an error.

4.9.3 Configuring menus and and dialogs

You may configure the menus and dialogs of **SAP ECTR interface to PTC Creo** to achieve the following goals:

- Displaying commands in the SAP PLM menu and right-click menu
- Presetting dialog options
- Displaying detailed SAP PLM information in the PTC Creo Parametric message log

4.9.3.1 Displaying commands in the SAP PLM menu and right click menu

The **SAP PLM** menu of **SAP ECTR interface to PTC Creo** is displayed in the **TOOLKIT** group on the **Tools** tab of the PTC Creo Parametric ribbon.

By default, all available commands are displayed in the **SAP PLM** menu and in the right-click menu.

In the menu tree on the **Menu** page in the configuration dialog of **SAP ECTR interface to PTC Creo**, you can specify the commands to be displayed in the **SAP PLM** menu and right-click menu of **SAP PLM**.

Changes to the **SAP PLM** menu configuration will only become effective after a restart of PTC Creo Parametric.

Synchronizing the SAP PLM menu and the SAP PLM tab

Any options that you remove from the **Menu** page of the **SAP PLM** menu should also be removed from the **SAP PLM** tab on the PTC Creo Parametric ribbon.

Presetting dialog options

Do the following to configure the option presets to be used in the **SAP ECTR interface to PTC Creo** dialogs:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Dialogues** page.
3) Enable the corresponding check boxes [☑] of the option that you would like to see enabled in the dialog box of the command in question.
4) Save your settings.

Result: After a restart of PTC Creo Parametric, the check boxes [☑] of the options in the command dialogs are enabled.

Configuring the SAP PLM menu and the SAP PLM right-click menu

Do the following to configure the **SAP PLM** menu and the **SAP PLM** right-click menu:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Menu** page.
3) Expand the menu tree until you see the desired command.
4) Enable the [☑] check boxes of all commands you want to display in the **SAP PLM** menu and in the **SAP PLM** right-click menu.
5) Disable the [☑] check boxes of all commands you do **not** want to display in the **SAP PLM** menu and the **SAP PLM** right-click menu.
6) Save your settings.

Result: After a restart of PTC Creo Parametric, the **SAP PLM** menu and the **SAP PLM** right-click menu of **SAP ECTR interface to PTC Creo** will be displayed as configured.

Synchronizing the SAP PLM menu and the SAP PLM tab

Any commands that you removed from the **SAP PLM** menu on the **Menu** page should also be removed from the **SAP PLM** tab on the ribbon of PTC Creo Parametric.

Check boxes that cannot be changed

Some command-related check boxes [☑] cannot be changed, i.e., they cannot be enabled or disabled. These check boxes are controlled on a different page of the configuration dialog. The corresponding command will be available in the **SAP PLM** menu depending on the setting on that page.

4.9.3.2 Presetting dialog options

Some commands from the **SAP PLM** menu of **SAP ECTR interface to PTC Creo** allow you to select additional options during their execution by enabling the appropriate check box [☑].

Use the **SAP ECTR interface to PTC Creo** configuration dialog on the **Dialogues** page to specify which option check boxes [☑] will be enabled in each command-related dialog.

You can also specify if the dialog will be displayed at all. If the dialog is not displayed, the behavior of the command is determined by the configuration you have defined by enabling or disabling the corresponding check box [☑]. If you enable the display of the dialog, you may specify the preset check box [☑] to be deactivated in the dialog.

4.9.3.3 Displaying detailed SAP PLM information in the PTC Creo Parametric message log

When the PTC Creo Parametric graphics window is changed, the Message Log of PTC Creo Parametric will display information about the current CAD file in the active graphics window. Optionally, more detailed information can be retrieved from SAP PLM.

Displaying detailed information from SAP PLM

Do the following to configure **SAP ECTR interface to PTC Creo** to display detailed information from SAP PLM in the Message Log of PTC Creo Parametric:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Advanced** page.
3) In the **PTC Creo Parametric Message Log** section, enable the [☑] **Determine detailed PLM information when changing the working window** check box.
4) Save your settings.

Result: When the graphics window is changed, the Message Log of PTC Creo Parametric will display detailed information from SAP PLM.

4.9.4 Configuring the application structure

Use **SAP PLM ▶ ECTR ▶ Show CAD Structure** command to display CAD structures, e.g., assemblies, in the application structure of SAP ECTR. The following configuration options are available for the application structure:

- Setting the initial level of detail
- Displaying files and originals in the application structure using the sorting of the Model Tree
- Displaying drawings in session and in family tables

4.9.4.1 Setting the initial level of detail

The initial level of detail is set in the file `default.txt`.

Setting the initial level of detail of the application structure

Do the following to set the initial level of detail of the application structure:

1) Use a text editor, e.g., `Notepad++`, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.
2) Search for `plm.structure.application.init.strategy.levels.pro`.
3) Set the initial level of detail using an integer value, e.g., `plm.structure.application.init.strategy.levels.pro = 1`.
4) Save the file.

Result: CAD structures are displayed with the initial level of detail of the application structure.

Displaying the entire CAD structure

Setting `plm.structure.application.init.strategy.levels.pro = all` will display the entire CAD structure.

Further expanding CAD structures as required

CAD structures with an initial single or multi-level of detail may be expanded further if required.

4.9.4.2 Displaying files and originals using the sorting of the Model Tree

When passing CAD structures to the application structure, **SAP ECTR interface to PTC Creo** uses the sorting of the Model Tree. When building the application structure, SAP ECTR ignores the structure of the Model Tree and instead uses its own sorting rules. You may define the sorting for your view of the application structure to mirror the sorting of the Model Tree.

Sorting the view of the application structure

Do the following to define the sorting of your view of the application structure to mirror the sorting of the Model Tree.

1) Open a CAD file in PTC Creo Parametric.
2) Select **SAP PLM** ▶ **ECTR** ▶ **Show CAD Structure** to open the application structure.
3) In the **Layout Manager** [⚙], select **Change Tree Sort Order**.
4) Add the `ECTR_CAD_SORT_ORDER` sorting parameter to the **Columns Set**.
5) Enable the [☑] **Ascending** checkbox.
6) Click **OK** to save your settings.

Result: The view of the application structure uses the sorting of the Model Tree.

Displaying the sorting parameter

In the **Layout Manager** [⚙], select **Change Tree Layout** to display the sorting parameter in the application structure.

4.9.4.3 Using additional containers

The application structure is used to display CAD structures, e.g., assemblies and their components. You may configure the application structure to use additional containers (nodes) . The drawings of a component may optionally be displayed inside an additional container named **Drawings in session**. Instances of family tables may optionally be displayed inside an additional container named **Instances**. References may optionally be displayed inside the **References** container.

The following additional containers can be displayed:

- Drawings **Drawings in session**
- Instances of family tables **Instances**
- References **External References**

Displaying drawings, instances of family tables and references in the application structure

Do the following to configure the display of drawings in the application structure:

1) Use a text editor, e.g., `Notepad++`, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.
2) Add the following lines:
```
structure.view.PRO.generic.object.type          = creoType
structure.view.PRO.generic.object.PRO_CONT.name = creoNodeName
structure.view.PRO.generic.object.PRO_CONT.icon = creoNodeIcon
```
3) Save the file.

Result: Drawings stored in memory are displayed inside the container named **Drawings in session**. Instances of family tables are displayed inside the container named **Instances**. References are displayed inside the container named **External References**.

Grouping drawings, instances of family tables and references in the application structure

Do the following to group drawings in session and instances of family tables inside additional containers.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Advanced** page.
3) In the **Application Structure** section, enable the [☑] check boxes of the containers you would like to be displayed in the application structure:
   - **Group instances of the displayed family table into container “Instance”**
   - **Group drwaings in session of the displayed 3D models into container “Drawings in session”**
   - **Group external references of the displayed CAD files into container “External References”**
4) Disable the [☑] check boxes of the containers you would like **not** to be displayed in the application structure.
5) Save your settings.

Result: Drawings in session, instances and references of family tables are grouped inside additional containers of the application structure.

4.9.4.4 Highlighting selected originals in the Model Tree

Any originals selected in the application structure can be highlighted using the **Select** command in the PTC Creo Parametric Model Tree. This option is only available if the file `%PLM_INSTDIR%\applications\pro\customize\config\mapkeys.txt` contains a mapkey entry named `riess_expand_tree`.

Example

An example can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `mapkeys.txt`.

Limitations

The `riess_expand_tree` mapkey and, consequently, the **Select** command can only be executed if the `auto_locate_in_tree` option in the `config.pro` configuration file of PTC Creo Parametric is set to `no`.

Highlighting selected files in the Model Tree

Configure the application structure to ensure that the **Select** command is also available for non-document files. To do so, add the entry `fnc.structure.view.application.generic(sap_select)` below the entry `+ om.popup.menu.INSOB` in the file `%PLM_INSTDIR%\addons\structureview\customize\config\menu.guidef`.

4.9.5 Configuring the loading, opening and editing of originals

The following options are available to configure the behavior of **SAP ECTR interface to PTC Creo** when opening, loading, and editing originals:

- Using a mapkey sequence when opening an original from within SAP PLM
- Loading and opening originals without references from within SAP PLM
- Updating originals prior to editing
- Adding multiple components from SAP PLM to an assembly

4.9.5.1 Using a mapkey sequence when opening an original from within SAP PLM

Mapkeys are macros that consist of a series of PTC Creo Parametric commands. Mapkeys can be customized to match the user interface and provide productivity enhancements. It is possible to group work sequences or common commands into mapkeys. You can initiate a mapkey with a keyboard command or by performing a specific action.

The `mapkeys.txt` file provides mapkeys that can also be used by **SAP ECTR interface to PTC Creo**. When opening originals from within SAP PLM, a mapkey can be executed instead of calling the TOOLKIT API. In this case, when an original is opened, **SAP ECTR interface to PTC Creo** will behave as if a CAD file would be opened directly in PTC Creo Parametric.

Using a mapkey sequence when opening an original from within SAP PLM

Do the following to configure SAP ECTR interface to PTC Creo to use a mapkey instead of calling the TOOLKIT API when opening CAD originals.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Advanced** page.
3) In the **Originals** section, enable the [☑] **Use mapkey sequence when opening originals from SAP PLM** check box.
4) Save your settings.

Result: When opening originals from SAP PLM; the mapkey sequence from the file `mapkeys.txt` will be used.

Opening simplified representations

If you combine the **Use mapkey sequence when opening originals from SAP PLM** function and the `open_simplified_rep_by_default yes` and `open_draw_simp_rep_by_default yes` options and add them as a mapkey to the `config.pro` file. PTC Creo Parametric will display the **Open Representation** selection dialog when an assembly or drawing is opened. This dialog allows you to open simplified representations.

4.9.5.2 Loading and opening originals without references from within SAP

Structures in PTC Creo Parametric may contain references that are not required for opening a file in the CAD system. For example, “External shrinkwraps” may represent complex assemblies. The shrinkwrap references the assembly. When loading from SAP, all referenced objects are loaded from SAP and into the CAD session in PTC Creo Parametric. For large assemblies in particular this may have a considerable impact on the loading time.

SAP ECTR allows the definition of explosion scenarios and explosion rules that can be used to load assemblies without external references, for example.

You may also configure commands in the right-click menu of SAP ECTR to use these explosion scenarios for loading and opening files from SAP, e.g, **Open Original without references**.

The config.pro file

The following options in the config.pro configuration file define how PTC Creo Parametric will handle external references that are not written to the working directory when loaded from SAP:

- `retrieve_data_sharing_ref_parts [no* | yes | ignore_missing]`
- `retrieve_merge_ref_parts [no* | yes | ignore_missing]`

Missing references

If you intend to use explosion scenarios to enable fast loading of CAD structures without external references, make sure to set the `retrieve_data_sharing_ref_parts` and `retrieve_merge_ref_parts` to `ignore_missing` or `no`.

Prerequisite

The following prerequisites must be met to add a custom command to the right-click menu of SAP ECTR.

Table maintenance

Make sure that the following tables in the SAP system are maintained:

- `CDESK_C_EXPLCF`
- `CDESK_C_EXPLFA`

Configuring explosion scenarios

- Explosion scenarios must be configured in the CAD services.

Configuration of SAP ECTR

- `load_scenarios.xml`
  - Each explosion scenario must be combined with an application-specific loading option.
- `menu.guidef` and `menu_macro.txt`
  - An application-specific loading option must be configured for documents of the `PRO` application type.
- `customer.txt`
  - The language-specific description of the loading option needs to be specified.
- `standard_icons.txt`
  - You may assign an icon to the loading option.

Procedures

The following procedures apply to the `FAST_LOAD` explosion scenario. You may define and use your own scenarios as required.

Maintaining the CDESK_C_EXPLCF tables

Do the following to maintain the CDESK_C_EXPLCF tables:

1) Use the SM30 transaction to open the **Maintain Table Views**.
2) Select **Maintain** to maintain the V_CDESK_C_EXPLCF table view.
3) Create ner records for the following CAD fields:
   - RES4
   - SORTF

Result: The CAD fields RES4 (for the **Object Type**) and SORTF (for the **Sort String**) have been maintained and are ready to be used the definition of an **Explosion Scenario** with **Explosion Rules**.

Maintaining the CDESK_C_EXPLFA tables

Do the following to maintain the CDESK_C_EXPLFA tables:

1) Use the SM30 transaction to open the **Maintain Table Views**.
2) Select **Maintain** to maintain the V_CDESK_C_EXPLFA table view.
3) Create new records for the following sort terms, for example:

CAD field	CAD field abbreviation	Short description
SORTF	PRO:C	PTC Creo – Component
SORTF	PRO:Y	PTC Creo – Reference
SORTF	PRO:CA	PTC Creo – Interchange Assembly
SORTF	PRO:CV	PTC Creo - Envelope
SORTF	PRO:CS	PTC Creo – Component – Suppressed
SORTF	PRO:CY	PTC Creo – Reference
SORTF	PRO:CE	PTC Creo – Reference
SORTF	PRO:CN	PTC Creo – Reference
SORTF	PRO:L	PTC Creo – Layout –
SORTF	PRO:G	PTC Creo – Family Table
SORTF	PRO:D	PTC Creo – Drawing - Model
SORTF	PRO:DM	PTC Creo – Drawing – Model Main

Table : Sort terms in the CDESK_C_EXPLFA table – CAD desktop compatibility

Result: The CDESK_C_EXPLFA has been maintained and is ready to be used in the definition of an **Explosion Scenario** with **Explosion Rules**.

Configuring an explosion scenario

Do the following to configure an explosion scenario:

1) Use the `CDESK_CUS` (**ECC**) or `CAD_SRV_CUS` (**S4/Hana**) transaction, respectively, to open the Customizing dialog structure.
2) In the **Group Maintenance**, create the `DEFAULT` group for the `PROE` CAD system if it does not exist already.
3) Enter the users or roles of the group.
4) Create the `FAST_LOAD` explosion scenario for the `DEFAULT` group in the **Explosion Scenario** section and add a short description.
5) Select **Explosion Rules** and define the explosion rules for the FAST_LOAD explosion scenario.

CAD field	CAD field abbreviation	Expl. Bhr.
SORTF	PRO:C	NA: …
SORTF	PRO:Y	ST: …
SORTF	PRO:CA	ST: …
SORTF	PRO:CV	NA: …
SORTF	PRO:CS	ST: …
SORTF	PRO:CZ	NA: …
SORTF	PRO:CY	ST: …
SORTF	PRO:CE	ST: …
SORTF	PRO:CN	NA: …
SORTF	PRO:L	NA: …
SORTF	PRO:G	NA: …
SORTF	PRO:D	NA: …
SORTF	PRO:DM	NA: …

Table: Explosion rules of the FAST_LOAD explosion scenario

Result: The FAST_LOAD explosion scenario has been defined and is ready for use.

Combining explosion scenarios with an application-specific load option

Do the following to combine an explosion scenario with an application-specific load option.

1) Use a text editor, e.g., Notepad++, to create the file `%PLM_INSTDIR%\applications\pro\customize\config\load_scenarios.xml`.
2) Add the following lines:

```
<document_load_scenarios
  xmlns:xsi=http://www.w3.org/2001/XMLSchema-instance
  xsi:noNamespaceSchemaLocation="../../../../basis/aux-files/load-
scenarios.xsd">
  <document_load_scenario>
            <name>PRO_FAST_LOAD</name>
            <description>FAST_LOAD_Without_References</description>
            <application_type>PRO</application_type>
            <bod_explosion
                  <explosion_scenario>FAST_LOAD</explosion_scenario>
                  <explosion_rule>
                        <active/>
                  </explosion_rule>
            </bod_explosion>
      </document_load_scenario>
  </document_load_scenarios>
```

Result: The `FAST_LOAD` explosion scenario has been combined with the `PRO_FAST_LOAD` load option. You may add the load option to the right-click menu of SAP ECTR.

Example

An example can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `load_scenarios.xml`.

Configuring an application-specific load option for documents of the PRO application type

Add the `PRO_FAST_LOAD` load option to the (flyout) right-click menu for documents of the `PRO` application type, for example.

1) Use a text editor, e.g., Notepad++, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\menu.guidef`.
2) Add the entry `= ? DOC_DOC_PROADD` to `+ om.popup.menu.DOC.PROFG`.
3) Use a text editor, e.g., Notepad++, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\menu_macros.txt`.
4) Add the following entry:
   `? DOC_DOC_PROADD  = mnu.flyout.doc_doc_riess`.
5) Add the entry `= fnc.doc.open.by.load.scenario(PRO_FAST_LOAD)` to `? DOC_DOC_PROADD`.

Result: The `fnc.doc.open.by.load.scenario(PRO_FAST_LOAD)` load option is displayed in the right-click menu.

Example

Examples can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the files `menu.guidef` and `menu_macros.txt`.

Adding a language-specific description for the load option

Do the following to add language-specific descriptions for the load option.

1) Use a text editor, e.g., Notepad++, to open the file `%PLM_INSTDIR%\customize\dictionary\en\customer.txt`.
2) Add the following line:
   `mnu.flyout.doc_doc_riess=PTC Creo Parametric Originals`
3) Add the following line:
   `fnc.doc.open.by.load.scenario(PRO_FAST_LOAD)= Open original without references`
4) Repeat these steps for other CAD environments.

Result: The right-click menu will display the language-specific description instead of the `fnc.doc.open.by.load.scenario(PRO_FAST_LOAD)` load option.

Assigning an icon to the load option

Do the following to assign an icon to the load option:

1) Use a text editor, e.g., Notepad++, to open the file `%PLM_INSTDIR%\applications\pro\basis\aux-files\standard_icons.txt`.
2) Add the following line:
   `fnc.doc.open.by.load.scenario(PRO_FAST_LOAD) = {0}/sap/parts_partial.png`.

Result: The language-specific description of the load option will be displayed along with the icon.

Example

An example can be found in the directory `%PLM_INSTDIR%\applications\pro\basis\aux-files` in the file `standard_icons.txt`.

4.9.5.3 Checking originals for the latest version prior to editing

Prior to editing an original you can check if its latest version has been loaded by using the command **SAP PLM ▶ Original ▶ Edit**.

Checking for the latest version of originals prior to editing

Configure **SAP ECTR interface to PTC Creo** to ensure that originals are checked for the latest version prior to editing.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Dialogues** page.
3) In the **Edit** section, enable the [☑] **Activate verification "Does a higher version of the document exist in SAP?"** check box.
4) Optionally: In the **Edit** section, enable the [☑] S**how button "Continue" if a higher document version exists** check box.
5) Save your settings.

Result: The software will check if a more recent version of an original is available prior to editing. If a more recent version exists, the user will be able to continue editing of the currently opened version by clicking the **Continue** button.

4.9.5.4 Checking originals for currency prior to editing

You can use the command **Open Original** to load CAD originals from SAP ECTR and open them in PTC Creo Parametric. To start editing the loaded originals, use the command **SAP PLM ▶ Original ▶ Edit**. This command checks that the original is up to date and that there is no newer state in SAP PLM. It also checks whether components included in the structure are current or outdated. **SAP ECTR interface to PTC Creo** can be configured to: a) not check the structure for currency, b) check only one level of the structure for currency or c) check all levels of the structure for currency.

If further checking for currency during editing of an original results in one of the following criteria being fulfulled, the **Edit <original>** will be displayed:

- The selected original is out of date.
- One or multiple originals included in the structure of the original are out of date.
- One or multiple originals included in the structure of the original are missing.

Checking originals for currency prior to editing

Do the following to configure **SAP ECTR interface to PTC Creo** to check whether an original is up to date prior to editing.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Dialogues** page.
3) In the **Edit** section, enable the [☑] **Activate verification "Does out-of-date original exist in SAP?"** check box.
4) Optionally: In the **Edit** section, enable the [☑] **Regard out-of-date originals in structure** check box.

Result: SAP ECTR will check whether originals and their components are up to date prior to editing.

Setting the level of detail for currency checking of the structure of originals

Do the following to set the level of detail for currency checking of the structure of originals.

1) Use a text editor, e.g., `Notepad++`, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.
2) Search for `plm.cad.checkForUpdate.before.CheckoutEdit.multiLevel.PRO`.
3) Enter the desired level of detail:
   single level `false`
   all levels `true`
4) Save the file.

Result: The structure of the original will be checked for currency as per your configuration. This allows for the creation of a list of outdated or missing components. Outdated originals can be updated prior to continuing the editing process, or the editing process can be aborted.

4.9.5.5 Updating originals prior to editing

If the checking for currency of an original prior to editing reveals an outdated original and/or outdated components in the structure, the **Edit <original>** dialog will be displayed. The **Edit <original>** dialog contains the **Continue** and **Cancel** buttons. The **Continue** button allows the editing of the version of the original currently opened in PTC Creo Parametric. The currency checking will be continued with the checking of further criteria.

The **Edit <original>** dialog contains the **Continue** and **Cancel** buttons. The **Continue** button will update all outdated and missing originals and replace them with their newest version, if possible. The selected original will be updated to the newest state from SAP stored within the same version. Any originals contained in the structure will be updated depending on the loading rule defined in SAP ECTR.

SAP ECTR interface to PTC Creo can be configured to display the **Edit <original>** by default. If no outdated or missing originals are found, a message will be displayed to the user that, depending on the defined loading rule, no update is required.

SAP ECTR also provides the following options: to update only selected outdated originals or to force editing without a prior update. In certain use cases it may therefore make sense not to make use of the **Continue** button.

SAP ECTR interface to PTC Creo can be configured to display or hide the **Continue**.

The Continue button in the Save dialog

⚠ If the **Edit** command is called from the **Save** or **Save and Display** dialog, the **Continue** button will not be available, regardless of the configuration.

Displaying the dialog only for outdated or missing originals

Do the following to configure **SAP ECTR interface to PTC Creo** to display the **Edit <original>** dialog only if outdated or missing originals have been found:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Dialogues** page.
3) In the **Edit** section, enable the [☑] **Only show dialog when some originals in the structure are out-of-date acording to the load options in SAP ECTR** check box.
4) Save your settings.

Result: The **Edit <original>** is only displayed if outdated originals have been found in the structure.

Displaying the Continue opton prior to editing

Do the following to configure **SAP ECTR interface to PTC Creo** to display the **Continue** button in the **Edit <original>** dialog:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Dialogues** page.
3) In the **Edit** section, enable the [☑] **Show button "Continue" if originals in the structure oft the originals are out-of-date** check box.
4) Select one of the options from the pulldown menu.
   - **Display for updateable or non updateable originals**
   - **Display only for non updateable originals**
5) Save your settings.

Result: When editing an original, the **Continue** button will be displayed in the **Edit <original>** dialog.

4.9.5.6 Adding multiple components from SAP PLM to an assembly

You can use the **SAP PLM ▶ Original ▶ Add Component** command to add a component from SAP PLM to an assembly via SAP ECTR. **SAP ECTR interface to PTC Creo** can be configured to select multiple components in SAP ECTR using the **SAP PLM ▶ Original ▶ Add Component** command.

Adding multiple components to an assembly

Do the following to configure **SAP ECTR interface to PTC Creo** to be able to add multiple components from SAP PLM to an assembly using the **SAP PLM ▶ Original ▶ Add Component** command.

1) Use a text editor, e.g., `Notepad++`, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.
2) Search for `plm.document.addComponent.singleSelectionOnly.PRO`.
3) Set `plm.document.addComponent.singleSelectionOnly.PRO = false`.
4) Save the file.

Result: The **SAP PLM ▶ Original ▶ Add Component** command can be used to select multiple components in SAP ECTR and add them to an assembly.

The riess_assemble_comp mapkey sequence

Ensure that the `riess_assemble_comp` mapkey exists in the file `%PLM_INSTDIR%\applications\pro\customize\config\mapkeys.txt`.

4.9.6 Configuring how originals are saved to SAP PLM

The following options are available to configure the behavior of **SAP ECTR interface to PTC Creo** when saving originals to SAP PLM:

- Saving drawings in session together with the represented model
- Creating and saving additional originals
- Refreshing the attributes of originals before saving
- Including drawing formats in the document structure
- Deleting old versions from the SAP working directory after saving

4.9.6.1 Saving drawings in session together with the represented model

You can use the **Original ▶ Save** or **Original ▶ Save and Display** commands to save changed originals to SAP PLM. **SAP ECTR interface to PTC Creo** can be configured to save the associated 2-D drawings when 2-D models are saved. In order to use this function, the drawings must be already loaded in the CAD system.

Saving drawings together with the represented model

Do the following to configure **SAP ECTR interface to PTC Creo** to save drawings in a CAD session together with the represented model:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Save** page.
3) In the **Drawing** section, enable the [☑] **Save associated drawings in session together with the displayed model** check box.
4) Save your settings.

Result: When saving CAD models, the associated drawings will also be saved to SAP PLM.

4.9.6.2 Creating and saving additional originals

SAP ECTR displays thumbnails of CAD originals in JPEG format. **SAP ECTR interface to PTC Creo** creates these thumbnails when a CAD original is saved to SAP PLM, and saves them as additional originals associated with the document. In addition to the thumbnails, other additional originals can be created in various formats.

Configuring additional original formats

Do the following to configure the additional originals to be created when saving CAD originals:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Save** page.
3) Enable the [☑] **Create and save additional originals** check box.
4) Enable the [☑] check boxes of the formats to be created when saving CAD originals.
5) Disable the [☑] check boxes of the formats **not** to be created when saving CAD originals.
6) Save your settings.

Result: When you save CAD originals, the system will create and save additional originals in the selected formats.

Enabling thumbnails

To create thumbnails for SAP ECTR, enable the [☑] check boxes of the JPEG formats for 3D models and 2D drawings.

Changing the output quality by using output profiles

You can change the output quality of the `3D JPEG` and `2D JPEG` formats by using output profiles. To use an output profile, enable the [☑] **Use JPEG profile** check box and use the **Customize** option to change the profile according to your requirements.
You can also change the output quality of the `2D PDF` format by using an output profile. To do so, select a PDF output profile in the **Use PDF profile** field.

Sample PDF profile

A sample PDF file can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `example_pro_pdf_export.dop`.

Configuring workstation applications and DTypes

Please note that for the additional originals, the corresponding workstation applications must be available in your SAP system. You also need to define the applicable file extensions and workstation applications in the definition of the used `DType` in the file `dtype.xml` (`primary_application` ▶ `checkin_rules` ▶ `save_direct`).

Using a conversion service

The thumbnails and the additional originals created when saving CAD originals are only a snapshot of the CAD originals shown in the graphics window. Better results can be achieved with corresponding background conversions, e.g., after the release of a document.

4.9.6.3 Refreshing the attributes of originals before saving

When you select the Originals **Original ▶Refresh Attributes** command or open CAD originals from within SAP, values from document data and additional data are transferred to parameters (attributes) of the CAD originals.

It is also possible to refresh the attributes automatically prior to saving the originals in SAP by selecting **Original ▶ Save** or **Original ▶ Save and Display**.

Refreshing the attributes of originals before saving

Do the following to configure that the attributes of CAD originals will be refreshed before saving in SAP:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Save** page.
3) Enable the [☑] **Refresh attributes of original before Save in SAP** check box.
4) Save your settings.

Result: The attributes of CAD originals are refreshed before saving in SAP.

Refreshing the attributes of originals before saving

Enable the **Refresh attributes of original before Save in SAP** option. This ensures that the attributes of the saved CAD originals will be up to date. This is particularly important if the attributes are used in downstream processes, e.g., for conversions.

Configuring the DType and the XML attributes section

Note that the link to the corresponding XML attribute section (`primary_application ▶ attribute_section`) in the file `attributes-from-sap.xml` needs to be included in the definition of the used `DType` in the file `dtype.xml`. The attributes are refreshed depending on the configuration of the corresponding XML attribute section in the file `attributes-from-sap.xml`.

4.9.6.4 Including drawing formats in the document structure

You can use the **Original ▶ Add Format** command to load drawing formats from within SAP and add them to the drawing. Drawing formats added from within SAP can be included in the drawing document structure.

Including drawing formats in the document structure

Do the following to include drawing formats added from within SAP in the drawing document structure.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Save** page.
3) Enable the [☑] **Include Formats in Document Structure** check box.
4) Save your settings.

Result: Drawing formats added from within SAP will be included in the document structure when the drawing is saved.

4.9.6.5 Delete old versions from the working directory when saving in SAP

PTC Creo Parametric does not overwrite CAD files when saving them but creates additional versions of these files in the working directory.

An incremental number is added to the file extension of each version, e.g.:

```
1000004711ppr000.prt
1000004711ppr000.prt.1
1000004711ppr000.prt.2
1000004711ppr000.prt.3
```

When originals are saved in SAP with **Original ▶ Save** or **Original ▶ Save and Display**, the file with the highest incremental number is copied (without the number) and checked in to SAP.

It is possible to have the numbered files deleted automatically after saving the most recent file (without the number) to the SAP working directory.

Deleting old versions from the SAP working directory after saving

Do the following to configure **SAP ECTR interface to PTC Creo** to delete old versions from the working directory after saving them to SAP PLM:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Save** page.
3) Enable the [☑] **Delete old versions (e.g., *.prt.1 - *.prt.n) after Save in SAP from working directory** check box.
4) From the pull-down menu, select one of the following options:
   - **Delete all versions**
   - **Keep highest version**
5) Save your settings.

Result: The numbered files are deleted automatically after saving the most recent file (without its number) to the SAP working directory.

4.9.6.6 Changing the DType

If the object type of a CAD original changes, e.g., because a part (`PRO:P`) becomes a generic part of a family table (`PRO:PG`), the `DType` of the document can be changed. To change the `DType` of a document, use the **Utilities ▶ Change DType [Administrator]** command from the **Administrator** menu.

The `DType` can be changed automatically when a CAD original is saved to SAP PLM.

Possible DType changes

Set the following preference in the file `default.txt`:

- `plm.document.changeDtype.possibleDtypes.for.<SourceDType> = <TargetDType>`

Examples

A sample file is stored in the following directory:

`%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `default.txt`.

- `plm.document.changeDtype.possibleDtypes.for.PROPR = PRNPR`
- `plm.document.changeDtype.possibleDtypes.for.PROPG = PRNPG`
- `plm.document.changeDtype.possibleDtypes.for.PROPI = PRNPI`
- `plm.document.changeDtype.possibleDtypes.for.PROAS = PRNAS`
- `plm.document.changeDtype.possibleDtypes.for.PROAG = PRNAG`
- `plm.document.changeDtype.possibleDtypes.for.PROAI = PRNAI`
- `plm.document.changeDtype.possibleDtypes.for.PRNPR = PROPR`
- `plm.document.changeDtype.possibleDtypes.for.PRNPG = PROPG`
- `plm.document.changeDtype.possibleDtypes.for.PRNPI = PROPI`
- `plm.document.changeDtype.possibleDtypes.for.PRNAS = PROAS`
- `plm.document.changeDtype.possibleDtypes.for.PRNAG = PROAG`
- `plm.document.changeDtype.possibleDtypes.for.PRNAI = PROAI`

Changing the DType

In order to avoid inconsistent data, you should only allow changes between `DType`s that use the same document type.

Make sure to set the correct `application_role` in the `DType` definition. Make sure that only changes between `DType`s of the same object type (part families → parts, assembly families → assemblies) are allowed.

4.9.7 Configuring the import of originals

The following options are available to configure the behavior of **SAP ECTR interface to PTC Creo** when importing originals:

- Hiding the Import On Demand dialog
- Displaying originals together with the document in the Import dialog
- Editing originals after importing
- Automatically adding files to the import list
- Determining CAD parameters regardless of the `DType` definition

4.9.7.1 Hiding the Import On Demand dialog

You can use the **SAP PLM ▶ Original ▶ Import** command to import CAD files into SAP PLM. You will need to select the appropriate `DType` or confirm the suggested `DType` in the **Import On Demand** dialog of SAP ECTR.

If you previously defined exactly one `DType` for each CAD object type, you can skip the **Import On Demand** dialog.

Skipping the Import On Demand dialog

Do the following to skip the display of the **Import On Demand** dialog:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Import** page.
3) Deactivate the [☑] **Always show SAP ECTR Import On Demand Dialog** check box.
4) Save your settings.

Result: The **Import On Demand** dialog of SAP ECTR will no longer be displayed.

DType / CAD object type association

⚠ If the **Import On Demand** dialog of SAP ECTR continues to be displayed, the association between CAD object types and `DType` is ambiguous.

4.9.7.2 Displaying originals together with the document in the Import dialog

The **SAP PLM ▶ Original ▶ Import** command is used to import CAD files from the CAD system into SAP PLM. The file list of the **Identify further files for Import** dialog of **SAP ECTR interface to PTC Creo** displays all CAD files in the working directory. **SAP ECTR interface to PTC Creo** can be configured to display CAD files with a document (originals) in the **Identify further files for Import** dialog.

Displaying originals together with the document

Do the following to configure the **Identify further files for Import** dialog to display originals with a document in its file list.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Import** page.
3) Enable the [☑] **Enable option "Display originals with document"** check box.
4) Save your settings.

Result: In the **Identify further files for Import** dialog of **SAP ECTR Interface to PTC Creo**, the [☑] **Display originals with document** check box is enabled. The **Identify further files for Import** dialog of **SAP ECTR interface to PTC Creo** will display originals with a document in its file list.

Displaying originals together with the document

You can temporarily disable the **Identify further files for Import** dialog of **SAP ECTR interface to PTC Creo** by disabling the [☑] **Display originals with document** check box, if necessary.

4.9.7.3 Editing originals after importing

The **SAP PLM ▶ Original ▶ Import** command is used to import CAD files from the CAD system into SAP PLM. Once the import has successfully completed, the CAD originals have been saved ( ). **SAP ECTR interface to PTC Creo** can be configured to allow the editing of CAD originals immediately after completion of the import ( ).

Editing originals after importing

Do the following to configure the **Identify further files for Import** dialog to allow the editing of CAD originals immediately after completion of the import.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Import** page.
3) Enable the [☑] **Enable option "Edit originals after import"** check box.
4) Save your settings.

Result: In the **Identify further files for Import** dialog of **SAP ECTR Interface to PTC Creo**, the [☑] **Edit originals after import** check box is enabled. The CAD originals are being edited ( ).

Displaying originals together with the document

You can temporarily disable the **Identify further files for Import** dialog of **SAP ECTR interface to PTC Creo** by disabling the [☑] **Edit originals after import** check box, if necessary.

4.9.7.4 Automatically adding files to the import list

The **SAP PLM ▶ Original ▶ Import** command is used to import CAD files from the CAD system into SAP PLM. The file list of the **Identify further files for Import** dialog of **SAP ECTR interface to PTC Creo** displays all CAD files in the working directory. You can use the **Identify Relations** button to determine if a relationship with a file in the file list exists for a particular file. As soon as a CAD file with an existing relationship with another CAD file in the import list is in session, it is automatically added to the import list.

Automatically adding originals to the import list

Do the following to configure the **Identify further files for Import** dialog to automatically add CAD originals to the import list for which relationships with CAD originals in the import list exist.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Import** page.
3) Enable the [☑] **Enable option "Automatically add files to the import list"** check box.
4) Save your settings.

Result: In the **Identify further files for Import** dialog of **SAP ECTR Interface to PTC Creo**, the [☑] **Automatically add files to the import list** check box is enabled. The CAD originals are being edited.

Displaying originals together with the document

You can temporarily disable the **Identify further files for Import** dialog of **SAP ECTR interface to PTC Creo** by disabling the [☑] **Automatically add files to the import list** check box, if necessary.

4.9.7.5 Determining CAD parameters regardless of the DType definition

The `DTyp` definition in the file `dtype.xml` determines the `section` of the file `attributes-to-sap.xml` that will be used for the transfer of parameters from the CAD system to SAP. **SAP ECTR interface to PTC Creo** will then identify the required parameters based on the `section` specified in the `DType`. These parameters must be known to the system during an import before a `DType` is associated. You can configure **SAP ECTR interface to PTC Creo** to identify all existing parameters before an import, regardless of the `DType`. The required parameters will then be selected from the entire list of parameters, and transferred, based on the `DType`.

4.6.7.5 Determining CAD parameters prior to importing regardless of the DType definition

Do the following to configure the **Identify further files for Import** dialog to display originals with a document in its file list.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Import** page.
3) Enable the [☑] **Enable option "Determine CAD parameters independent of the DType definition before import"** check box.
4) Save your settings.

Result: Prior to importing a CAD file into SAP PLM, all parameters of that file are determined.

Parameters and properties

In addition to all parameters in the CAD files, the system will also identify system parameters of PTC Creo Parametric (`Properties`). An overview of the possible `Properties` can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `attributes-to-sap.xml`.

4.9.8 Configuring document structures and material BOMs

4.9.8.1 Aggregating positions in document structures depending on their sort term

Documents of CAD originals that are reused within the CAD structure are aggregated into a single position in the document structure. A quantity indicator that is appended to the position displays the number of uses. You may also configure **SAP ECTR interface to PTC Creo** to aggregate positions in document structures in groups based on their sort term.

Aggregating positions in document structures into groups based on their sort term

Do the following to configure **SAP ECTR interface to PTC Creo** to aggregate positions in document structures into groups based on their sort term.

1) Use a text editor, e.g., `Notepad++`, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.
2) Specify for each required document type that positions in the document structure should be aggregated into groups based on their sort term:
   - `plm.docstructure.itemkey.<DocType>  = <DIRKEY>;<SORTSTRING>`
   - `plm.docstructure.itemkey. <DocType>  = <DIRKEY>;<SORTSTRING>`

   e.g.:
   - `plm.docstructure.itemkey.PPR  = <DIRKEY>;<SORTSTRING>`
   - `plm.docstructure.itemkey.PAS  = <DIRKEY>;<SORTSTRING>`
3) Save the file.

Result: The positions in document structures will be aggregated into groups based on their sort term.

Example

A part is used twice in an assembly (sort term: `PRO:C`) and suppressed once (sort term: `PRO:CS`). In the document structure, the part is listed on two positions: one position with the sort term `PRO:C` and a quantity of 2 and one position with the sort term `PRO:CS` and a quantity of 1.

4.9.8.2 Creating material BOMs

In SAP ECTR, material BOMs are created with the **Bill of Material ▶ Derive Bill of Material <method>** command from the right-click menu. Depending on the selected creation method, parameters for BOM explosion from the CAD system will be evaluated during the creation of the material BOM. In the CAD system, BOM explosion is defined with the following options from the **SAP PLM** right-click menu:

- **Set BOM Explosion Parameter for Component**
- **Set BOM Explosion Parameter inside Original**

The following parameters for material BOM explosion can be set:

- Explosion parameters
  - ○ **Include position**
  - ◉ **Exclude position**
    - ☑ **Move sub position to top**
- Location parameters
  - **Base unit of Measure**
  - **Quantity**

You may configure **SAP ECTR interface to PTC Creo** to display these commands in the SAP PLM right-click menu of the Model Tree. In addition, you can configure which baseline units will be allowed.

Displaying commands for material BOM explosion

Do the following to configure **SAP ECTR interface to PTC Creo** to display the commands for setting the parameters for material BOM explosion in the SAP PLM right-click menu in the Model Tree of PTC Creo Parametric:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Menu** page.
3) In the **Context** section, enable the **Set BOM Explosion Parameter inside Model** and **Set BOM Explosion Parameter inside Original** check boxes.
4) Save your settings.

Result: The commands are displayed in the right-click menu.

Specifying allowed baseline units

Do the following to configure **SAP ECTR interface to PTC Creo** to display only allowed baseline units during the setting of parameters for material BOM explosion.

1) Use a text editor, e.g., `Notepad++`, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\pro_bom_base_unit_of_measure.txt`.
2) Add the allowed baseline units as list of separate lines.
3) Save the file.

Result: The **Base Unit of Measure** pull-down menu of the dialog will only display the baseline units specified in the file `pro_bom_base_unit_of_measure.txt`.

Preventing automatic creation of material BOMs when checking in originals

⚠ SAP ECTR can be globally configured to create or update material BOMs when checking in a CAD structure. Make sure to prevent the use of this global setting for PTC Creo Parametric. To do so, set the following document type-based preferences in the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`:

- `plm.bom.updateAtCheckin.<DocType>  = false`
- `plm.bom.createAtCheckin.<DocType>  = false`

4.9.9 Refreshing attributes and initializing parameters

There are two ways to synchronize (refresh) values in user-defined CAD parameters (**CAD attributes**) and CAD system parameters (**CAD system attributes**) with values in document data, additional document data, and material master data (**SAP fields**):

- Refreshing CAD attributes using values from SAP fields
- Refreshing SAP fields using CAD attribute values

You can create missing parameters in CAD files and assign predefined values.

- Initializing parameters

The following files define how and when attributes will be refreshed using SAP fields:

Configuration of SAP ECTR

- `DTypes.xml`
  - The XML attribute sections in the files `attributes-from-sap.xml` and `attributes-to-sap.xml` need to be specified. The XML attributes sections are used during the refreshing of the CAD attributes and SAP fields, depending on the `DType`.
  - The events used for automatic refreshing of the CAD attributes need to be specified.
- `attributes-from-sap.xml`
  - The XML attribute sections must be defined.
  - Make sure to specify the CAD attributes to be refreshed using SAP fields, grouped by XML attribute sections.
- `attributes-to-sap.xml`
  - The XML attribute sections must be defined.
  - Make sure to specify the SAP fields to be refreshed using CAD attributes, grouped by XML attribute sections.

For further reference

🛈 For further details on how to configure SAP ECTR, please refer to the SAP ECTR documentation.

4.9.9.1 Refreshing CAD attributes using values from SAP fields

Do the following to refresh attributes by using values from SAP fields:

- Manually
  - Refreshing using **Refresh Attributes**
- Automatically
  - Adding a drawing format from SAP using **Add Format**
- Automatically/optionally
  - Checking in CAD originals using **Save** or **Save and Display**
  - Saving CAD originals as a new document
  - Saving CAD originals as a new version
  - Checking out CAD originals from SAP

Prerequisites

⚠ To refresh CAD attributes by using values from SAP fields, the following prerequisites must be met:

Configuration of SAP ECTR interface to PTC Creo

- The **Refresh Attributes** command must be available.
- The **Add Formats** command must be available.
- Optionally: The **Refresh attributes of original before Save in SAP** function must have been enabled.

Configuration of SAP ECTR

- `dtype.xml`
  - The corresponding XML attribute section must be specified in the `DType` definition in the file `attributes-from-sap.xml`.
  - Optionally:
    When defining the `DType`, you must specify the events upon which the attributes will be refreshed automatically.
- `attributes-from-sap.xml`
  - The XML attribute section must be defined.
  - Make sure to specify the CAD attributes to be refreshed using SAP fields in the XML attribute section.

Refreshing the CAD attributes of originals before saving

ℹ CAD attributes can be refreshed before saving the originals to SAP.

Refreshing the attributes of originals before saving

Do the following to configure that the CAD attributes of CAD originals will be refreshed before saving them in SAP: On the Save tab of the **SAP ECTR interface to PTC Creo** configuration dialog, enable the [☑] **Refresh attributes of original before Save in SAP** check box.

Syntax of the configuration file `attributes-from-sap.xml`

⚠ CAD attributes (user-defined CAD parameters) must have the `PROPARAM:` prefix.

Examples

Examples can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `attributes-from-sap.xml`.

4.9.9.2 Refreshing values in SAP fields using CAD attributes

The values in SAP fields will be refreshed automatically using CAD attribute values when you select one of the following commands to save in SAP:

- **Save**
- **Save and Display**
- **Save as New Document**
- **Save as New Version**

Prerequisites

To refresh values in SAP fields using CAD attributes, the following prerequisites must be met:

Configuration of SAP ECTR

- `dtype.xml`
  - The corresponding XML attribute section must be specified in the `DType` definition in the file `attributes-to-sap.xml`.
- `attributes-to-sap.xml`
  - The XML attribute section must be defined.
  - The SAP fields to be refreshed using CAD attributes need to be specified in the XML attribute section.

Syntax of the configuration file `attributes-to-sap.xml`

CAD attributes (user-defined CAD parameters) must have the `PROPARAM:` prefix.

CAD system attributes (system parameters) must have the `PROPROP:` prefix. CAD system attributes can only be transferred from PTC Creo Parametric to the SAP system, not vice-versa.

Examples

Lists with transferable CAD system attributes and examples can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `attributes-to-sap.xml`.

4.9.9.3 Initializing parameters

CAD originals that were migrated to SAP PLM from legacy systems might not contain all required parameters (CAD attributes).

When you select **Original ▶ Add Formats** or **SAP PLM ▶ Additional Functions ▶ Extras ▶ Initialize Parameters** to add a drawing format from within SAP, the missing parameters can be created and initialized with values.

On the **Parameter** page of the configuration dialog of **SAP ECTR interface to PTC Creo** you can define the behavior of **SAP ECTR interface to PTC Creo** when initializing CAD parameters.

In the **Initial Values** section you can specify the parameters to be created and the values to be used for initializing. In addition, you can specify, for each parameter, the target drawing or model (part or assembly) to be used for initializing.

Initializing parameters

Do the following to specify the name, target, and value of parameters to be initialized:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Parameter** page.
3) In the **Initial Values** section, enable the [☑] **Define CAD Component Parameters which are created and initialized using the command "Refresh Component Parameters"** check box.
4) Click the **ADD** button to enable the input field.
5) Enter the parameter name into the **Parameter** field.
6) To initialize the parameter in a drawing, enable the [☑] **Drawing** check box.
7) Enter the desired initial value.
8) To initialize the parameter in a model, enable the [☑] **Model** check box.
9) Enter the desired initial value.
10) Repeat these steps for other parameters you need and then save your settings.

Result: When you select **Original ▶ Add Formats** or **SAP PLM ▶ Additional Functions ▶ Extras ▶ Initialize Parameters** to add a drawing format from within SAP, the missing parameters will be created and initialized with values.

Deleting parameters from the list

To a parameter from the list, right-click to open the right-click menu and select **Delete**.

Parameters without a value

Parameters for which you did not specify an initial value will be created in the enabled target without a value, i.e., they will be blank.

4.9.10 Managing drawing formats in SAP PLM

You can use SAP PLM to manage drawing formats and add them to a drawing directly from within SAP. The title blocks of drawing formats can be used to display drawing parameters and model parameters (CAD attributes) that can be refreshed (udated) using values from SAP fields. Drawing formats can be included in the document structure of the drawing.

4.9.10.1 Configuring how drawing formats are managed in SAP PLM

Managing drawing formats in SAP requires certain prerequisites to be met, as well as various configurations.

Prerequisites

⚠ In order to manage drawing formats in SAP, the following prerequisites must be met:

Configuration of SAP ECTR

- `dtype.xml`
  - The `DType` for drawing formats must be defined.
  - The corresponding XML attribute section must be specified in the `DType` definition for drawings and, if applicable, for drawing models in the file `attributes-to-sap.xml`.
- `default.txt`
  - The selection filters for drawing formats must be defined.
- `attributes-from-sap.xml`
  - Make sure that the XML attribute section for drawings has been defined.
  - Make sure to specify the CAD attributes to be refreshed using SAP fields in the XML attribute section.

Konfiguration SAP ECTR interface to PTC Creo

- The **Add Formats** command must be enabled.
- The CAD attributes that will be created and initialized with the **Add Formats** command must be defined.
- The **Include Formats in Document Structure** function must have been enabled.

Documents for drawing formats

- Documents with drawing formats need to be created.
- The document status of these documents must match the selection filter.

DType for drawing formats

ℹ Like all object types created with PTC Creo Parametric and managed in SAP using SAP ECTR, drawing formats require the creation of a matching `DType`.

Example

A sample definition of the `PROFR DType` can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `dtype.xml` file.

Refreshing attributes

ℹ Use **Add Formats** to add drawing formats. The parameters represented in the title block are refreshed by values from the document and the additional document data.

SAP PLM menu and right-click menu

Do the following to configure the **SAP PLM** menu and the right-click menu: Enable the [☑] **Add Formats** check box.
Including drawing formats in the document structure

Drawing formats added with **Add Formats** need to be included in the drawing document structure.

Do the following to include drawing formats added from within SAP in the drawing document structure.

Documents for drawing formats

To add drawing frames to a drawing using **Add Formats**, the drawing frames need to be checked in into SAP. The document type, document part, and status must match the selection filters defined in the file `default.txt`.

Defining selection filters for drawing formats

The selection of drawing formats available with the **Original ▶ Add Formats** option can be limited in the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt` by using the following criteria as filters:

- **Document type**
  `plm.document.search.drawingFormatDocType.PRO = <DocType>`
- **Document status**
  `plm.document.search.drawingFormatDocStatus.PRO = <DocStatus>`
- **Document part**
  `plm.document.search.drawingFormatDocPart.PRO = <DocPart>`

Excluding filters

If you want to exclude one or multiple filters, you can comment out the corresponding line(s) in the file `default.txt` using the `#` character.

4.9.10.2 Including drawing formats in the document structure

Configure **SAP ECTR interface to PTC Creo** to include drawing formats as positions in the document structure of the drawing document.

Including drawing formats in the document structure

Do the following to configure **SAP ECTR interface to PTC Creo** to include drawing formats as positions in the document structure of the drawing document:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Save** page.
3) In the **Formats** section, enable the [☑] **Include formats in document structure** check box.
4) Save your settings.

Result: Whenever a drawing is saved, the drawing formats used in the drawing will be included in the document structure.

4.9.11 Configuring directories

Each work session with SAP ECTR has a working directory associated to it. In a running CAD session, this directory may be changed only temporarily. The following options are available to configure the behavior of **SAP ECTR interface to PTC Creo** when temporarily changing the working directory and when using CAD files from directories other than the working directory:

- Changing the working directory
- Ignoring files from particular directories

4.9.11.1 Changing the working directory

SAP ECTR interface to PTC Creo links the CAD session to the SAP ECTR working directory. **SAP ECTR interface to PTC Creo** blocks the user from changing the working directory. Some modules, such as PTC Creo Mechanism Design, however, require a temporary change of working directory.

In the **Tempory Directories** section, you can specify which directories PTC Creo Parametric will be allowed to use temporarily as working directories.

Specifying which directories PTC Creo Parametric will be allowed to use temporarily

You can specify which directories PTC Creo Parametric will be allowed to use temporarily.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Directories** page.
3) In the **Temporary Directories** section, enable the [☑] **Specify the working directories PTC Creo Parametric is allowed to change to temporarily** check box.
4) In the **Temporary Directories** section, add a line entry in the **Folder** input panel for each allowed directory.
5) Press the [↵] key to complete your entry.
6) Add more directory paths as required.
7) Save your settings.

Result: PTC Creo Parametric will be able to use the specified working directories.

4.9.11.2 Ignoring files from particular directories

PTC Creo Parametric, as well as some auxiliary applications, e.g., for the simulation of manufacturing processes, use CAD files that are only required for the active CAD session. These CAD files are usually loaded from different directories and, by default, are not managed in SAP PLM.

You may configure SAP interface to PTC CREO to ignore any data from specified directories. These data do not contain a document and will not be saved to SAP PLM. CAD originals or CAD structures using these data can still be managed in SAP PLM as usual.

Specifying directories whose data will be ignored

Do the following to specify the directories whose data should be managed in SAP PLM.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Directories** page.
3) In the **Ignored Directories** section, enable the [☑] **Ignore CAD files from specified directories** check box.
4) In the **Ignored Directories** section, add a line entry in the **Folder** input panel for each ignored directory.
5) Press the [↵] key to complete your entry.
6) Add more directory paths as required.
7) Save your settings.

Result: Any files stored in the specified directories will be ignored.

4.9.12 Using PTC Creo ModelCHECK for saving

On the **ModelCHECK** page of the configuration dialog of **SAP ECTR interface to PTC Creo** you can define the behavior of **SAP ECTR interface to PTC Creo** when it interacts with PTC Creo ModelCHECK.

PTC Creo ModelCHECK can be executed with every **Save** and **Save and Display** operation on CAD originals in SAP.

Optionally, you may block **Save** and **Save and Display** operations for originals with errors in SAP.

Executing PTC Creo ModelCHECK prior to saving in SAP

Do the following to execute PTC Creo ModelCHECK prior to saving in SAP:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **ModelCHECK** page.
3) Enable the [☑] **PTC Creo ModelCheck before saving originals in SAP** check box.
4) In the **Settings** section, enable the [☑] check boxes of the required options.
5) In the **Configurations** section, select the directory containing the PTC Creo ModelCHECK configuration to be used (⊙ radio button option).
6) In the **Results** section, select the target directory where the PTC Creo ModelCHECK result files will be written (⊙ radio button option); if necessary, specify the path and directory name.
7) Save your settings.

Result: PTC Creo ModelCHECK will be executed with every **Save** and **Save and Display** operation on CAD originals.

Settings

In the **Settings** section, enable both check boxes. The result as output by PTC Creo ModelCHECK will be displayed in the internal browser, allowing any errors to be fixed immediately.

Configurations

If you want to run alternative checks (other than those stored in the default path) prior to saving in SAP, enable the **Optional Directory** option (⊙ radio button option) in the **Configurations** section.

Preventing originals with errors from being saved

Do the following to prevent originals with errors from being saved:

1) Open the **SAP ECTR interface to PTC Creo** Configuration dialog.
2) Open the **ModelCHECK** page.
3) Enable the [☑] **Prevent saving originals with errors** check box.
4) From the pull-down menu, select the type of notification to be displayed if originals with errors are encountered.
5) In the **Errors and warnings** section, configure if errors and warnings should be considered.
6) Optionally, specify the number of errors and warnings to be tolerated.
7) Save your settings.

Result: Originals with errors will not be checked in to SAP. A message will be displayed to the user.

Messages

If you select the **Display message** option from the pulldown menu, a message will be displayed when an error occurs. The user must aport the storing of originals with errors.

If you select the **No message** option from the pulldown menu, no message will be displayed when an error occurs. The storing of originals with errors will always be aborted. The user will be notified of aborted storages in the message area of PTC Creo Parametric.

If you select the **Display dialog** option from the pulldown menu, a dialog will be displayed when an error occurs. The user will be able to choose if the storing of originals with errors will be executed or aborted.

4.9.13 Disabling default limitations

The **Advanced** page of the **SAP ECTR interface to PTC Creo** Configuration dialog provides the following options to disable default settings and limitations:

- Disabling the limitation “No renaming of originals in the CAD system”
- Disabling the limitation “Do not create UDF if the reference part has no document”
- Disabling the limitation “Do not save originals as a new document that have been stored in SAP and were locally modified”
- Disabling the limitation “Only refresh attributes in originals that are currently open for editing”

Undesired system behavior and data inconsistencies

⚠ To avoid undesired system behavior and data inconsistencies, **SAP ECTR interface to PTC Creo** by default applies limitations to certain CAD system functions.

These limitations can be disabled at your own risk.

4.9.13.1 Disabling the limitation “No renaming of originals in the CAD system”

By default, originals that are already known to SAP and linked to a document info record must not be renamed.

Disabling the limitation “No renaming of originals in the CAD system”

Do the following to disable the limitation “No renaming of originals in the CAD system”:

1) Enable the [☑] **Disable limitation “No renaming of originals in the CAD system”** check box.

Result: CAD files with a document can be renamed in PTC Creo Parametric.

4.9.13.2 Disabling the limitation “Do not create UDF if the reference part has no document”

CAD files are usually renamed during an import. However, UDF reference models of a user-defined feature (UDF) must not be renamed once the UDF has been created. Otherwise, the UDF would lose its link to the reference part and may become unusable. Consequently, **SAP ECTR interface to PTC Creo** by default allows the creation of new UDFs only if the UDF reference model already contains a document.

Disable limitation “Do not create UDF if the reference part has no Document”

Do the following to disable the limitation “Do not create UDF if the reference part has no document”:

1) Enable the [☑] **Disable limitation “Do not create UDF if the reference part has no Document”** check box.

Result: The creation of UDFs will be possible even if the UDF reference model does not contain a document.

4.9.13.3 Disabling the limitation “Do not save originals as a new document that have been stored in SAP and were locally modified”

SAP ECTR interface to PTC Creo allows you to save originals as new documents. The original will still be available in the CAD session after copying. Only originals that have been stored in SAP and were not locally modified may be saved as a new document. This limitation ensures that no locally modified originals exist which might require the user to undo these local modifications. You may disable this limitation if required.

Disabling the limitation “Do not save originals as a new document that have been stored in SAP and were locally modified”

Do the following to disable the limitation “Do not save originals as a new document that have been stored in SAP and were locally modified”:

1) Enable the [☑] **Disable limitation “Do not save originals that are checked in and have been locally modified as a new document”** check box.

Result: Originals that have been stored in SAP and were locally modified can be saved as new documents.

Locally modified originals

Locally modified originals and originals stored in SAP will still be available after copying.
Local modifications will not be undone automatically.

4.9.13.4 Disabling the limitation “Only refresh attributes in originals that are currently open for editing”

SAP ECTR interface to PTC Creo allows the refreshing of attributes (user-defined CAD parameters) regardless of the system configuration. Attributes can be refreshed whenever an original is opened from SAP or prior to storing originals in SAP. Attributes can also be refreshed manually using the **Refresh Attributes** command. Only attributes in originals that are currently open for editing will be refreshed.

Refreshing (updating) attributes when opening originals from SAP

You can specify whether attributes will be refreshed when their originals are opened by using the following XML attributes in the `primary_application` XML element of your `DType` definition:

- `update_attributes_for_ro`
- `update_attributes_for_rw`
- `update_attributes_for_dirty`

If the corresponding XML attribute has a value of “`no`”, no CAD parameters will be refreshed from SAP as a rule when originals are opened from SAP.

If the corresponding XML attribute has a value of “`yes`”, the default limitation of **SAP ECTR interface to PTC Creo** will apply. In this case, parameters will only be refreshed after the original has been checked out and opened for editing (`checked_out_by_me`).

Disabling the limitation “Only refresh attributes in originals that are currently open for editing”

Do the following to disable the limitation “Only refresh attributes in originals that are currently open for editing”:

1) Enable the [☑] **Disable limitation “Refresh attributes only in originals that are open for editing”** check box.

Result: The refreshing of attributes will also be possible for originals that are not open for editing.

4.9.14 Using SmartContainers

SAP ECTR Allows you to display user-defined SmartContainers and add content to them.

- Using SmartContainers for drawings
- Displaying dependent drawings in the “Where Referenced” SmartContainer

4.9.14.1 Using SmartContainers for drawings

Displaying SmartContainer drawings

You can display the **Drawings** SmartContainer for documents associated with parts or assemblies. You can also automatically populate the **Drawings** SmartContainer with part or assembly drawings.

Prerequisite

The following prerequisites must be met to use the **Drawings** SmartContainer:

Configuration of SAP ECTR

- default.txt
  - You need to define the name of the SmartContainer, the macro to be executed, and the icon to be displayed for the SmartContainer.
  - In addition, specify the document types for which you want to display the SmartContainer.
- <macro>.txt
  - The macro specified in the default.txt file must be contained in a file named <macro>.txt.
- dictionary.txt
  - The language-specific dictionary.txt file must contain the display name of the SmartContainer.

Defining SmartContainers

SmartContainer are defined in the file %PLM_INSTDIR%\applications\pro\customize\config\default.txt.

- **SmartContainer name**
  plm.smart.container.<DEF. SMARTCONTAINER>.name = <SMARTCONTAINER>
- **SmartContainer macro**
  plm.smart.container.<DEF. SMARTCONTAINER>.macro = PRO:<macro>.txt
- **SmartContainer icon**
  plm.smart.container.<DEF. SMARTCONTAINER>.icon = <.../Icon>

Example

The file %PLM_INSTDIR%\applications\pro\template-customize\config\default.txt contains the name MACRO_DRW_PRO as an example of the SmartContainer named **Drawings**:

- plm.smart.container.MACRO_DRW_PRO.name = MACRO_DRW_PRO
- plm.smart.container.MACRO_DRW_PRO.macro = PRO:drawings.txt
- plm.smart.container.MACRO_DRW_PRO.icon = {0}/sap/drawing

Displaying a SmartContainer depending on the document type

In the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`, you can specify the document type for which the `SmartContainer` will be displayed:

- **Setting the document type**
  `plm.om.DOC.containers.<DocType> = SMART_CONT(<NAME>);@plm.om.DOC.containers@`

Example

The file `%PLM_INSTDIR%\applications\pro\template-customize\config\default.txt` contains the `PPR` (parts) and `PAS` (assemblies) document types as examples.

- **The PPR document type**
  `plm.om.DOC.containers.PPR = SMART_CONT(MACRO_DRW_PRO);@plm.om.DOC.containers@`
- **The PAS document type**
  `plm.om.DOC.containers.PAS = SMART_CONT(MACRO_DRW_PRO);@plm.om.DOC.containers@`

Preparing the macro

The macro specified in the `default.txt` using the preference `plm.smart.container.MACRO_DRW_PRO.macro = PRO:<macro>.txt` is stored in the `%PLM_INSTDIR%\applications\pro\customize\scripts\macros` folder.

Example

The `%PLM_INSTDIR%\applications\pro\template-customize\config\scripts\macros` file contains the `drawings.txt` sample macro.

Specifying the name of the SmartContainer

In the file `%PLM_INSTDIR%\applications\pro\basis\dictionary\<language abbreviation>\dictionary.txt`, you can specify the language-specific display name of the `SmartContainer`:

- **Naming the SmartContainer**
  `plm.smart.container.dic.<DEF. SMARTCONTAINER> = <Display Name>`

Example

The `%PLM_INSTDIR%\applications\pro\basis\dictionary\en\dictionary.txt` file contains the following example:

- `plm.smart.container.dic.MACRO_DRW_PRO = drawings`

4.9.14.2 Displaying dependent drawings in the Where Referenced container

In SAP ECTR, dependent drawings are displayed in the **Dependent Documents** container of the master document. In addition, dependent drawings can be displayed in the **Where Referenced** container.

Displaying dependent drawings in the Where Referenced container

You can display dependent drawings in the Where Referenced container.

Prerequisite

In order to display dependent drawings in the **Where Referenced** container, you need to define a corresponding preference in the file `default.txt`.

Defining the preference

The `%PLM_INSTDIR%\applications\pro\customize\config\default.txt` file is used to specify that dependent documents will be displayed in the **Where Referenced** container.

- **Setting the preference**
  `plm.om.docContainer.showAdditionalGeometry = true`

4.9.15 Configuring the settings for family tables

Family tables – for parts or assemblies – consist of a generic file and the associated instances. PTC Creo Parametric creates the family table instances at runtime from the generic file. For all family table members, including the generic file, a document info record is created. The documents can be linked to material master records, etc.

The following options are available to configure the behavior of **SAP ECTR interface to PTC Creo** when working with family tables:

- Specifying `DType` for family tables
- Defining the relation between the `DType` for generic files and the `DType` for instances
- Defining allowed changes of the `DType`
- Loading family table instances along with the generic file
- Versioning family table instances along with the generic file

For the configuration, the following files are used:

- `%PLM_INSTDIR%\customize\config\dtype.xml`
- `%PLM_INSTDIR%\customize\config\default.txt`

Prerequisite

⚠ In order to create documents for family tables, the following prerequisites must be met:

- `dtype.xml`
  - The `DType` for generic files must be defined.
  - The `DType` for family table references must be defined.
- `default.txt`
  - The `DType` for generic parts and the `DType` for generic assemblies must be specified.
  - The relations between the `DType` for generic files and the `DType` for the associated instances must be specified.

4.9.15.1 Specifying DTypes for family tables

A `DType` for generic files and for instances must be defined in the file `%PLM_INSTDIR%\customize\config\dtype.xml`.

DType for generic files

ℹ The `DType` for generic files of part family tables must contain the following XML attribute:

- `application_role="PG"`

The `DType` for instances of part family tables must contain the following XML attribute:

- `application_role="PI"`

The `DType` for generic assemblies of assembly family tables must contain the following XML attribute:

- `application_role="AG"`

The `DType` for instances of assembly family tables must contain the following XML attribute:

- `application_role="AI"`

Example

Examples can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `dtype.xml`.

4.9.15.2 Defining the relation between the DType for generic files and the DType for instances

In the file `%PLM_INSTDIR%\customize\config\default.txt`, the `DType` for generic parts and the `DType` for generic assemblies must be specified. In addition, the relation between the `DType` for the generic file and the `DType` for the instances must be defined.

DType for generic files

Set the following preference in the file `default.txt`:

- `plm.document.genericDTypes.PRO = <DType Gen. Parts>;<DType Gen. Assembl.>`

Examples

A sample file is stored in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `default.txt`.

- `plm.document.genericDTypes.PRO = PROPG;PROAG`

DType relations

In the file `default.txt`, specify which `DType` for instances is associated with which `DType` for generic files. Set the following preference:

- `plm.document.variantDType.<DType Generic File>  = <DType Instances>`

Examples

A sample file is stored in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `default.txt`.

- `plm.document.variantDType.PROPG  = PROPI`
- `plm.document.variantDType.PROAG  = PROAI`

4.9.15.3 Defining allowed changes of the DType

If a family table is broken up, the generic file and the family table instances will transform into independent originals. If this becomes necessary, the `DType` needs to be changed. Also, the original of the instance (`*.var.prt` or `*.var.asm`) attached to the document needs to be replaced with the separated and independent original (`*.prt`, `*.asm`). **SAP ECTR interface to PTC Creo** supports you in the break-up of family tables. To this end, you need to configure allowed changes of the `DType` in the file `%PLM_INSTDIR%\customize\config\default.txt`.

Allowed changes of the DType

Set the following preference in the file `default.txt`:

- `plm.document.changeDtype.switch.PRO.<DType>  = <DTpe>`

Examples

A sample file is stored in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `default.txt`.

- `plm.document.changeDtype.switch.PRO.PROPG = PROPR`
- `plm.document.changeDtype.switch.PRO.PROPI = PROPR`
- `plm.document.changeDtype.switch.PRO.PROAG = PROAS`
- `plm.document.changeDtype.switch.PRO.PROAI = PROAS`
- `plm.document.changeDtype.switch.PRO.PRNPG = PRNPR`
- `plm.document.changeDtype.switch.PRO.PRNPI = PRNPR`
- `plm.document.changeDtype.switch.PRO.PRNAG = PRNAS`
- `plm.document.changeDtype.switch.PRO.PRNAI = PRNAS`

Changing the DType

In order to avoid inconsistent data, you should only allow changes between `DType`s that use the same document type.

Make sure to set the correct `application_role` in the `DType` definition. Make sure that only changes between `DType`s of the same object type (part families → parts, assembly families → assemblies) are allowed.

Breaking up family tables

Only break up family tables if it becomes absolutely necessary. In order to avoid inconsistent data in existing CAD structures using family table instances you should first create a new family table version before breaking up the family table.

Do the following to break up family tables:

- Create a new version of the familiy table.
- Successively open all instances, leaving all open instances in memory space.
- Remove all variants from the generic file (delete all lines).
- Save the generic file to SAP.
- Save all open files to SAP.

4.9.15.4 Loading family table instances along with the generic file

When a CAD original is opened from SAP PLM into PTC Creo Parametric, SAP ECTR also opens the CAD originals of the documents included in the document structure of the selected document. However, generic files of family tables do not have a document structure. In order to be able to completely load family tables from SAP PLM, SAP note **2408290** needs to be implemented.

SAP note

2408290 SAPECTR: **Generic objects are loaded from SAP without their instances**

The preference sap.plm.note2408290.checkout.exclude.N

In SAP ECTR, library parts and library assemblies can be tagged with an additional `N` flag in the XML attribute `application_role` of the `DType` definition (`PRO:*IN*`). If you do not want all instances of library parts or library assemblies to be loaded along with the generic file from SAP, you can set the preference `sap.plm.note2408290.checkout.exclude.N=true` in the file `default.txt`.

4.9.15.5 Versioning family table instances along with the generic file

Family tables always need to be managed as a whole. All information on the entire family table is contained in the generic file. It is therefore important to ensure the versioning of the entire family table as a rule. Otherwise, it may result in inconsistent data.

In order to ensure versioning of the entire family table, you need to set the XML attribute `take_instances_for_generic` in the `DType` definition and the preferences `plm.document.newVersion.notAllowedForFamilyMembers.PRO` and `plm.document.variantKeepVersion.PRO` in the file `default.txt`.

The XML attribute take_instances_for_generic

The XML attribute `take_instances_for_generic="yes"` must be set in the XML element `new_version` of the `DType` definition for family tables.

The preference plm.document.newVersion.notAllowedForFamilyMembers

The preference `plm.document.newVersion.notAllowedForFamilyMembers.PRO = true` must be set in the file `default.txt`.

plm.document.variantKeepVersion.PRO

The preference `plm.document.variantKeepVersion.PRO = true` must be set in the file `default.txt`. When a new version of a family table is created from within the CAD system, the version numbers of all family table instances and of the generic file will be incremented together.

4.9.16 Using ballooning functions

SAP ECTR interface to PTC Creo supports you in the creation of BOM balloons in drawings.

A SAP bill of materials for an assembly can only be represented in a drawing if a table with repeat region has been added to the drawing. This table can be used to display component parameters with data from the bill of materials, such as the item number and material number.

By selecting **SAP PLM ▶ Additional Functions ▶ Ballooning ▶ Refresh Component Parameters** you can transfer data from bills of materials to component parameters.

The typical procedure includes the following steps:

- Creating an assembly drawing
- Refreshing component parameters using information from the bill of materials
- Adding a table with repeat region to the drawing
- Creating BOM balloons
- Changing the parameters of BOM balloons

The following commands are available in **SAP ECTR interface to PTC Creo**:

- **Refresh Component Parameters**
- **Add Table**

Prerequisite

⚠ In order to benefit from the ballooning functions, the following prerequisites must be met:

Configuration of SAP ECTR

- `default.txt`
  - The `Ballooning Preferences` must be set in the file `default.txt`.

Configuration of SAP ECTR interface to PTC Creo

- The **Refresh Component Parameters** and **Add Table** commands must be available.
- Component parameters
  - The names of the component parameters must have been defined.
  - The component parameters must have been assigned to the corresponding data type.

Tables

- Table with repeat region
  - A table with repeat region must be have been created.
  - The repeat region of the table must contain the component parameters.

Ballooning preferences in the file default.txt

ℹ Set the following preferences in the file `%PLM_INSTDIR%\customize\config\default.txt`:

- `plm.bom.ballooning.application`
- `plm.bom.ballooning.alternative`
- `plm.bom.ballooning.plant`
- `plm.bom.ballooning.useChangeNoFromDoc`
- `plm.bom.ballooning.usage`

Using values from other preferences

It is possible to use the values of other preferences in the `Ballooning Preferences`. See the following example.

Ballooning preferences

- `plm.bom.ballooning.application = @plm.bom.application@`
- `plm.bom.ballooning.alternative = @plm.bom.alternative@`
- `plm.bom.ballooning.plant = @plm.bom.plant@`
- `plm.bom.ballooning.useChangeNoFromDoc = @plm.bom.useChangeNoFromDoc@`
- `plm.bom.ballooning.usage = @plm.bom.usage@`

On the **Ballooning** page you can define the behavior of **SAP ECTR interface to PTC Creo** during the refreshing of component parameters.

Preparing the tables

Any tables you may want to add to a drawing using the **Add Table** command should be saved in the directory `%PLM_INSTDIR%\applications\pro\templates`.

Sample table

A sample table can be found in the directory `%PLM_INSTDIR%\applications\pro\template_templates` in the file `riess_sap_bill_of_material.tbl`.

On the **Ballooning** page in the **Component Parameter** section of the **SAP ECTR interface to PTC Creo** Configuration dialog you can specify the component parameters to be created and the data from the bill of materials that should be used for refreshing the component parameters.

The following data types can be refreshed:

Type	Description
`MATNR`	Material number (internal representation)
`MATNR_EXTERN`	Material number (external representation – without leading zeros)
`POSNR`	Item number (internal representation – 6 digits)
`POSNR_EXTERN`	Item number (external representation – 4 digits)
`MENGE`	Quantity
`MEINS`	Quantity unit
`POSTEXT_1`	Item text 1
`POSTEXT_2`	Item text 2

Table: Data types to be refreshed from bills of materials

4.9.17 Managing user-defined features in SAP

The CAD systems of the Pro/ENGINEER family are based on features. Users can build complex geometry based on multiple features.

If you often need the same combinations of features or features based on complex sketches, you can create user-defined features (UDF) from them. These UDFs are then available as additional features.

SAP ECTR interface to PTC Creo supports you in your work with user-defined features.

4.9.17.1 UDFs in PTC Creo Parametric

UDF file name

For UDF files, the following naming pattern is used: `<UDFName>.gph`.

Opening a UDF

It is **not** possible to open a UDF directly from your CAD system. Consequently, a direct import of UDFs using the default import method is also not possible.

Creating UDFs

There are two ways to create a UDF:

- **Subordinate** option:
  Changes made to a UDF source model will affect the UDF and thus all parts using it.
- **Stand Alone** option:
  The UDF is no longer dependent on the UDF source part. However, you may include a copy of the UDF source model in the definition of the UDF as a reference part and then, for example, use it as a reference when placing the UDF model.

Reference model file name

The name of the UDF reference model is assigned by the CAD system. For UDF reference models, the following naming pattern is used: `<UDFName_gp>.prt`.

Using UDFs

There are two ways to use a UDF:

- Dependently using the option
  Make feature dependent on dimensions of UDF
- Independently using the option
  Advanced reference configuration

4.9.17.2 Using UDFs in SAP PLM

UDF document type

It is recommended to assign a separate document type to UDFs. This way, you can define UDF-specific permissions, status networks, and classifications.

UDF document structures

The following document structures apply to UDFs:

- UDFs defined as **Stand Alone** without a reference part do not have a document structure.
- UDFs defined as **Stand Alone** with a reference part, the UDF reference model is included in the document structure.
- For UDFs defined as **Subordinate**, the UDF source model is included in the document structure.

Using UDFs in document structures

UDFs used independently with the **Make feature dependent on dimensions of UDF** command will not be added to the document structure.

UDFs used dependently with the **Advanced reference configuration** command will be added to the document structure.

4.9.17.3 DType for UDFs and UDF reference models

UDF files and the included UDF reference models require the creation of `DType` definitions.

Constraint

A UDF file and the corresponding UDF reference models or UDF source models form a unit. Once a UDF file has been created, the UDF file and the corresponding UDF reference model or UDF source model must not be renamed. Renaming the UDF file will make a UDF unusable.

Defining the DType for UDFs

You should define a `DType` for UDF files to ensure that original files associated with it cannot be renamed.

To do so, use the XML attribute `rename_template_filename="no"` in the XML element `document_create`.

Defining the DType for UDF reference models

You should define a `DType` for UDF reference models to ensure that original files associated with it cannot be renamed.

To do so, use the XML attribute `rename_template_filename="no"` in the XML element `document_create`.

Defining the DType for UDFs – `function_group`

If you want to deploy UDFs via user-defined options and macros in the working directory, you should consider using a UDF-specific functional group, e.g., `function_group="PROUF"`, in the XML attribute `function_group` of the XML element `program_options` in the `DType` definition for UDF files.

Note

The following procedures include instructions on how to add a custom command that depends on the `function_group` contained in the `DType` definition to the right-click menu. The custom command is only displayed for documents whose `DType` definition contains the `function_group`.

Defining the DType for UDF source models

UDF source models of UDFs defined as subordinate UDFs do not require a separate `DType` definition. However, UDF source models need to be known as documents in SAP before they can be used as a source for deriving a UDF.

Examples

The `%PLM_INSTDIR%\applications\pro\template-customize\config\dtype.xml` file contains sample `DType` definitions.

- Sample definition of the `DType` for UDFs → `DType PROUF`
- Sample definition of the `DType` for a UDF reference model → `DType PROPH`

4.9.17.4 Including UDFs in the document structure

Use the **Include User-Defined Features in Document Structure** function o specify that UDF documents should be included in the document structure.

Including user-defined features in the document structure

Do the following to configure user-defined features to be included in the document structure:

1) Enable the [☑] **Include User-Defined Features in Document Structure** check box.

Result: When saving in SAP, user-defined features (UDFs) will be included in the document structure.

Import UDF

By enabling the [☑] **Include User-Defined Features in Document Structure** check box, the **SAP PLM ▶ Additional Functions ▶ Extras ▶ Import UDF** command is also enabled.

4.9.17.5 Deploying UDFs to the working directory

In order to use UDFs as features in parts, the UDF files and their respective reference or source parts need to be deployed to the working directory. You may also deploy UDFs from the SAP ECTR right-click menu with a custom command.

Prerequisites

The following prerequisites must be met to add a custom command to the right-click menu of SAP ECTR:

Configuration of SAP ECTR

- `dtype.xml`
  - The `DType` definition for UDF files, e.g., for `PROUF`, needs to include its own `function_group`.
- `menu.guidef`
  - The `+ om.popup.menu.DOC.PROUF` right-click menu of the `function_group` must be defined in the file `menu.guidef`.
- `dictionary.txt`
  - The descriptions of the appropriate custom command must exist in the language-specific `dictionary.txt` files.
- `standard_icons.txt`
  - An icon may be associated with the custom command in the file `standard_icons.txt`.
- `pro_udf_fnc_doc_checkout_view.txt`
  - A macro must be defined that is run from the right-click menu.

Changing the function_group in the UDF definition of the DType

Do the following to change the `function_group` in the UDF definition of the `DType`, e.g., in `PROUF`:

1) Use a text editor to open the file `%PLM_INSTDIR%\applications\pro\customize\dtype.xml`.
2) In the `DType` definition for UDF files, e.g., `PROUF`, add the XML attribute `program_options` XML element with the `function_group`.

Result: The `DType` definition for UDF files has its own `function_group`.

Example

An example can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `dtype.xml` in the `PROUF DType` definition.

Defining the right-click menu for UDF files

Do the following to define the `+ om.popup.menu.DOC.PROUF` right-click menu for UDF files in the file `menu.guidef`:

1) Use a text editor to open the file `%PLM_INSTDIR%\applications\pro\customize\menu.guidef`.
2) Add the following entry:
   `fnc.execute.macro(pro_udf_fnc_doc_checkout_view.txt)`

Result: The right-click menu for UDF files has been defined to depend on the `function_group`. The `pro_udf_fnc_doc_checkout_view.txt` function will be displayed in the right-click menu.

Example

An example can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `menu.guidef`.

Adding descriptions of the custom command to the dictionary

Do the following to add suitable descriptions of the custom command to the language-specific `dictionary.txt` files:

1) Use a text editor to open the file `%PLM_INSTDIR%\applications\pro\basis\dictionary\en\dictionary.txt`.
2) Add the following entry:
   `fnc.execute.macro(pro_udf_fnc_doc_checkout_view.txt)=Transfer UDF`
3) Repeat the above steps for all other languages as required. For example, add the entry `fnc.execute.macro(pro_udf_fnc_doc_checkout_view.txt)=UDF` to the file `%PLM_INSTDIR%\applications\pro\basis\dictionary\de\dictionary.txt` for German.

Result: Language-specific descriptions have been associated with the `fnc.execute.macro(pro_udf_fnc_doc_checkout_view.txt)` function. The language-specific description of the `pro_udf_fnc_doc_checkout_view.txt` function will be displayed in the right-click menu.

Example

An example can be found in the directory `%PLM_INSTDIR%\applications\pro\basis\dictionary\en` in the file `dictionary.txt`.

Associating an icon with the custom command

Do the following to associate an icon with the custom command:

1) Use a text editor to open the file `%PLM_INSTDIR%\applications\pro\basis\aux-files\standard_icons.txt`.
2) Add the following line:
   `fnc.execute.macro(pro_udf_fnc_doc_checkout_view.txt) = {0}/sap/download.png`

Result: The  icon will be displayed in front of the language-specific description of the `pro_udf_fnc_doc_checkout_view.txt` function in the right-click menu.

Example

An example can be found in the directory `%PLM_INSTDIR%\applications\pro\basis\aux-files` in the file `standard_icons.txt`.

Defining macros

Do the following to define a macro to deploy the UDF file along with its reference or source part to the working directory:

1) Create a macro, e.g., `pro_udf_fnc_doc_checkout_view.txt`, with the following content:

```
k= KEYLIST_FROM_CONTEXT("active","selected")
set1 = CREATE_SET(k)
p = PARAMETER_MAP("doc_filter_fields")
p.DTYPE = "PROPH"
set2 = TRANSFORM_SET_WITH_PARAMETERS( set1, "expand_structure" ,p )
kx = KEYLIST_FROM_SET( set2 )
CALL_OMF("fnc.doc.checkout(view)",kx)
```

2) Save the file `pro_udf_fnc_doc_checkout_view.txt` to the directory `%PLM_INSTDIR%\customize\scripts\macros`.

Result: The `pro_udf_fnc_doc_checkout_view.txt` macro can be run by selecting the **Deploy UDF** command from the right-click menu.

Example

An example can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\scripts\macros` in the file `standard_icons.txt`.

Refreshing component parameters

Do the following to define the names of the component parameters to be refreshed:

1) Enable the [☑] **Define CAD Component Parameters which are created and initialized using the command "Refresh Component Parameters"** check box.
2) Click the **ADD** button to enable the input field.
3) Enter the parameter name into the **Parameter** field.
4) Select the parameter type from the **Type** field.
5) Enable the [☑] check box.
6) Repeat these steps for other parameters you need and then save your settings.

Result: By selecting **SAP PLM ▶ Additional Functions ▶ Ballooning ▶ Refresh Component Parameters** you can transfer data from bills of materials to component parameters.

Disabling individual parameters

Disable the [☑] checkbox in the corresponding parameter line.

4.9.18 Using external libraries

External libraries containing purchased parts, standardized parts, catalog parts etc. are available from various manufacturers. Many customers also create and use their own libraries for corporate use. Originals from these libraries are stored in SAP PLM as documents. However, it may sometimes be necessary to additionally deploy these libraries to external server drives. If a user opens CAD structures, e.g., assemblies, from SAP PLM that contain originals from these libraries, these originals are not explicitly loaded from SAP PLM. PTC Creo Parametric uses search paths to locate originals in external libraries and directly loads them from their respective library directories.

Using external libraries

⚠ Compared to a purely internal management of parts in SAP PLM, the use of external libraries requires more administrative effort as well as a suitable configuration.

You should therefore consider the following questions before using external libraries:

- What benefits do you expect from using external libraries?
- What are the issues or challenges that you plan to resolve by using external libraries?
- Are you prepared to accept the risks that arise from possible inconsistencies, redundancies etc. or can you at least minimize these risks?
- Did you evaluate all possible alternatives?

Required configurations for the use of external libraries

ℹ The following configurations are required if you want to use external libraries:

- Optionally: Defining special document types for external libraries
- Creating separate `DType` definitions for external libraries
- Publishing library paths to SAP ECTR
- Specifying the relations between `DType` definitions of family tables
- Assigning write and read permissions to library directories
- Specifying search paths to the libraries in PTC Creo Parametric

4.9.18.1 Using special document types for external libraries

You will need to configure additional document types in SAP if you would like to use your own status networks and classifications for external libraries. Using your own document types for libraries will also allow you to assign permissions (e.g., create permissions, modify permission, etc.) to library documents that differ from those for other CAD documents.

4.9.18.2 Creating separate DType definitions for external libraries

Documents from external libraries usually use a different object type and different relative paths compared to other documents. You should therefore create separate `DType` definitions for external libraries.

Creating separate DType definitions for external libraries

Do the following to create `DType` definitions for external libraries in the file `%PLM_INSTDIR%\applications\pro\customize\config\dtype.xml`:

1) Create separate `DType` definitions for:
   - External library parts
   - External library parts – Family tables – Generic files
   - External library parts – Family tables – Instances
   - External library assemblies
   - External library assemblies – Family tables – Generic files
   - External library assemblies – Family tables – Instances
2) In the `DType` definitions of the external libraries, specify that SAP ECTR should use relative paths. In the XML element `primary_application`, set the XML attribute `use_relative_path="yes"`.

Result: For all relevant `DType` definitions, SAP ECTR will use the paths to the external libraries as specified in the file `default.txt`.

4.9.18.3 Publishing library paths to SAP ECTR

To make the paths to the external libraries known to SAP ECTR, you need to specify them in the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.

Library paths

Set the following preference in the file `default.txt`:

- `plm.library.PRO.<Library> = <Path>`

Examples

A sample file is stored in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `default.txt`.

- `plm.library.PRO.Library1= X:\\...\\Library 1 Screws`
- `plm.library.PRO.Library2= X:\\...\\Library 2 Bearings`

4.9.18.4 Specifying the relations between DType definitions of family tables

In the file `%PLM_INSTDIR%\customize\config\default.txt`, the `DType` for generic parts and the `DType` for generic assemblies must be specified. In addition, the relation between the `DType` for the generic file and the `DType` for the instances must be defined.

DType for generic files

Set the following preference in the file `default.txt`:

- `plm.document.genericDTypes.PRO = <DType Gen. Parts>;<DType Gen. Assembl.>`

Examples

A sample file is stored in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `default.txt`.

- `plm.document.genericDTypes.PRO = PRNPG;PRNAG`

DType relations

In the file `default.txt`, specify which `DType` for instances is associated with which `DType` for generic files. Set the following preference:

- `plm.document.variantDType.<DType Generic File>  = <DType Instances>`

Examples

A sample file is stored in the following directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the file `default.txt`.

- `plm.document.variantDType.PRNPG  = PRNPI`
- `plm.document.variantDType.PRNAG  = PRNAI`

4.9.18.5 Assigning write and read permissions to library directories

By default, CAD originals in external libraries should only be managed by users with the required permissions. You therefore need to make sure that only authorized persons have write access to library directories.

To do so, set the following options in the `config.pro` configuration file of PTC Creo Parametric:

- `file_open_default_folder  working_directory`
- `override_store_back  no`

4.9.18.6 Specifying search paths to the libraries in PTC Creo Parametric

To make the paths to the external libraries known to PTC Creo Parametric, add the paths to the `config.pro` file using the `search_path` option. Alternatively, you can set the search paths in the file `search.pro`, To do so, add the path to the file `search.pro` to the `config.pro` file using the `search_path_file` option.

4.9.19 Using macros

SAP ECTR provides the option to use macros. The purpose of a macro is to process generic queries on a `SmartList` and to populate a `SmartContainer` or `SmartFolder`. In addition, macros may be used to map and add simple specific functions to pop-up menus.

Macros can be used in the following contexts:

- Right-click / flyout / pop-up menu
- `SmartContainer`
- `SmartList`
- `SmartFolder`

Examples of macros in the document pop-up menu

Use cases for macros added to the document pop-up menu.

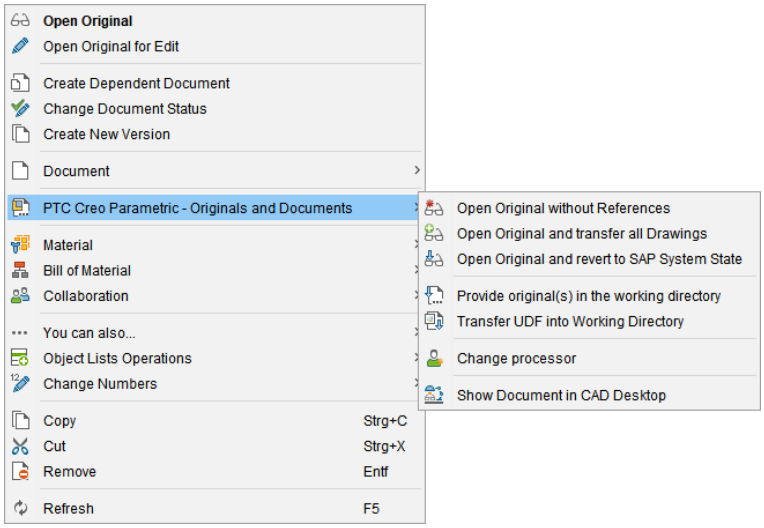


Figure: Macros in the document pop-up menu

For further reference

For further information on macros, please refer to:

https://wiki.scn.sap.com/wiki/display/PLM/Tutorials+and+Enablement+Content

4.9.20 Configuring cloning

4.9.20.1 Configuring the display of cloning options

The Clone window of SAP ECTR provides four alternative options for the cloning of originals from PTC Creo Parametric. You can configure SAP ECTR to display only a specified subset of these options.

Configuring the display of cloning options

Do the following to configure the Clone function of SAP ECTR to display only a subset of specified cloning options:

1) Use a text editor, e.g., Notepad++, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\menu.guidef`.
2) At `+ om.popup.menu.CLONEPOS.PROFG` , comment out any options that you do not want to be displayed by inserting a hash character (`#`) at the beginning of the corresponding line.

Result: The Clone dialog will only display the specified options.

Additional options

`+ om.popup.menu.CLONEPOS.PROFG`
Do not add any additional cloning options. **SAP ECTR interface to PTC Creo** supports only the options listed in the file `PLM_INSTDIR%\applications\pro\customize\config\menu.guidef`.

4.9.20.2 Running the cloning process in a separate CAD session

CAD originals need to be loaded to a PTC Creo working session in order to be cloned. To avoid undesired side effects on CAD originals already in memory, the work memory needs to be flushed before cloning. If you have additional PTC Creo Parametric licenses available, you can run the Clone function in a separate CAD session.

Running the cloning process in a separate CAD session

Do the following to configure **SAP ECTR interface to PTC Creo** to run the Clone function in a separate CAD session.

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Advanced** page.
3) In the **Cloning** section, enable the [☑] **Perform cloning function in additional CAD session** check box.
4) Save your settings.

Result: The Clone function will be run in a separate CAD session.

4.9.20.3 Path to the Rename executable for cloning

The file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt` must contain the path to the executable file that will be run to rename originals cloned from PTC Creo Parametric:
`plm.adv.clone.renamerProgram.PRO = @PLM_BINDIR_PRO@\\CadRenamer.exe`

4.9.21 PTC Creo Unit Technology support

PTC Creo Parametric features the PTC Creo Unite Technology to support working with CAD files created in other CAD systems. These functions are included in the PTC Creo Collaboration Extensions.

CAD files created in other CAD systems, such as CATIA, Siemens NX or SolidWorks can be opened in PTC Creo Parametric. This also applies to CAD files created in other PTC applications, e.g., Creo Elements/Direct Modeling. When you try to open a file created in a different CAD system it is first converted “on the fly” into a format (`*.creo`) that can be read by PTC Creo Parametric. Converted files are stored in the working directory with their original file name and extension, followed by the additional extension `*.creo`. The original source CAD application is still recognizable from the original file extension.

Any files in the working directory that were already converted into a `*.creo` file existing in that directory earlier will not be recalculated. Existing `*.creo` files will also not be recalculated even if the original CAD file is modified in the native application. In order to force a recalculation or reconversion you will need to delete the `*.creo` file.

If CAD originals that were not created in PTC Creo are loaded into the working directory from SAP PLM, **SAP ECTR interface to PTC Creo** will delete any corresponding `*.creo` files. This is done to ensure that only the most recent geometry is converted.

SAP ECTR interface to PTC Creo supports you in working with PTC Creo Unite Technology and the PTC Creo Collaboration Extensions by:

- Adding `*.creo` files to a document using drag & drop
- Opening `*.creo` files from SAP PLM in PTC Creo Parametric
- Preventing the deletion of `*.creo` files

4.9.21.1 Adding *.creo files to a document using drag & drop

If you are using the PTC Creo Collaboration Extensions, it may be useful to attach the `*.creo` files created at runtime by PTC Creo Parametric as additional originals to the documents of the CAD originals. In SAP ECTR, `*.creo` files can be dragged and dropped to the document of a CAD original created in a different CAD system. In order to enable PTC Creo Parametric to open `*.creo` files directly from within SAP ECTR they must have the `PRO` workstation application assigned to them.

Assigning the PRO workstation application to *.creo files

Do the following to configure SAP ECTR to automatically assign the PRO workstation application to a `*.creo` file if it is dragged and dropped to a document as an additional original.

1) Use a text editor, e.g., Notepad++, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.
2) Search for `plm.document.originals.extension.WsAppl.creo`.
3) Set `plm.document.originals.extension.WsAppl.creo = PRO`.

Result: `*.creo` files that are dragged and dropped to the document of a CAD original created in different CAD system are assigned the `PRO` workstation application.

4.9.21.2 Opening *.creo files from SAP PLM in PTC Creo Parametric

When CAD originals are opened from SAP ECTR, it uses the CAD system in which they were created. If there is no running instance of the CAD system, SAP ECTR will attempt to start the CAD system in question.

If you want to open CAD originals created in a different CAD system or the additional `*.creo` originals in PTC Creo Parametric, you will need to use a special Open command from the pop-up menu of the respective document.

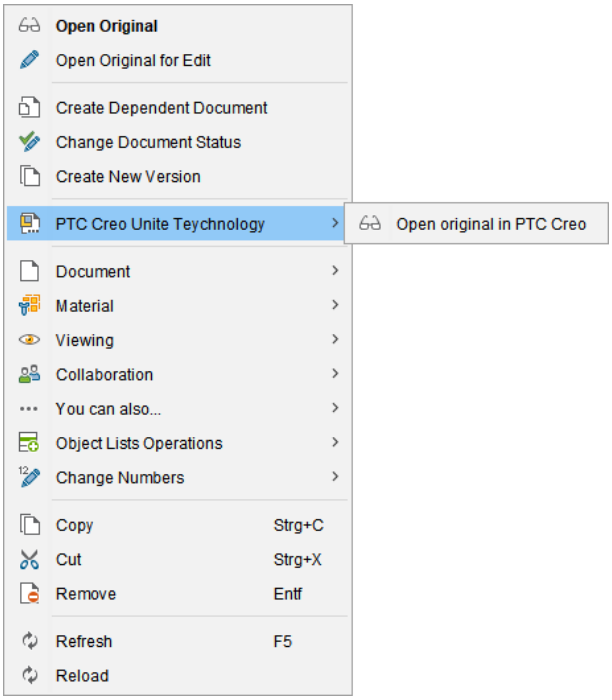


Figure: Additional pop-up menu command

Macro DOC_DOC_CREOUNITE for PTC Creo Unite Technology

The file `%PLM_INSTDIR%\applications\pro\template-customize\config\menu_macro.txt` contains the following macro that you can easily copy to the file `menu.macros.txt` for the required application.

```
# ? DOC_DOC_CREOUNITE   = mnu.flyout.doc_doc_riess_creounite
#                       = fnc.doc.open.fast.view(PRO)
```

Adding the macro DOC_DOC_CREOUNITE to the pop-up menu

Do the following to add the macro `DOC_DOC_CREOUNITE` to the pop-up menu for the application whose CAD originals you would open directly in PTC Creo Parametric.

1) Use a text editor, e.g., `Notepad++`, to open the file `%PLM_INSTDIR%\applications\pro\template-customize\config\menu_macro.txt`.
2) Locate the macro `DOC_DOC_CREOUNITE` and copy it to your clipboard.
3) Use a text editor to open the file `%PLM_INSTDIR%\applications\pro\customize\config\menu_macro.txt` and insert the macro at the appropriate position.
4) Remove the hash (#) character at the beginning of each line.

```
#_____________________________________________________________
# Menu Macros PTC Creo Unite Technology
#_____________________________________________________________

? DOC_DOC_CREOUNITE      = mnu.flyout.doc_doc_riess_creounite
                         = fnc.doc.open.fast.view(PRO)
```

5) Save the file `menu_macro.txt`.
6) Use a text editor to open the file `%PLM_INSTDIR%\applications\<Application>\customize\config\menu.guidef`.
7) Locate the section `+ om.popup.menü.<Function Group>` and insert the macro `DOC_DOC_CREOUNITE`.

```
+  om.popup.menu.DOC.<Function Group>  = ----------------------------
                                       = ? DOC_GENERAL
                                       = ----------------------------
                                       = ? DOC_DOC_CREOUNITE
                                       = ----------------------------
```

8) Add the macro to all other `<Function Group>` pop-up menus as required.
9) Save the file `menu.guidef`.

Result: The document pop-up menu now contains a command to open CAD originals from other CAD systems in PTC Creo Parametric.

<Application>

The `<Application>` placeholder stands for the installation directory of the relevant SAP ECTR interface to <CAD system>, e.g., `SLW`; `CAT`; `UGS` etc.

<Function Group>

The `<Function Group>` placeholder stands for the relevant `function_group` of the `DType` for which the pop-up menu is displayed.

Language-specific commands

You can define a language-specific setting for the display of the macro `DOC_DOC_CREOUNITE`. To do so, you need to edit the language-specific files `%PLM_INSTDIR%\customize\dictionary\<Sprache>customer.txt` and add the following text, for example:

```
mnu.flyout.doc_doc_riess_creounite = PTC Creo Unite Technology
fnc.doc.open.fast.view(PRO) = Open original in PTC Creo
```

Icons

You can set an icon to be used for the display of the macro `DOC_DOC_CREOUNITE` in the pop-up menu. To do so, you need to edit the file `%PLM_INSTDIR%\ applications\pro\basis\aux-files standard_icons.txt` and add the following text, for example:

```
fnc.doc.open.fast.view(PRO)   = {0}/sap/display
```

4.9.21.3 Preventing the deletion of *.creo files

If the PTC Creo Collaboration Extensions are to be used, you should usually prevent the `*.creo` files from deletion.

SAP ECTR interface to PTC Creo can be configured to prevent `*.creo` files of CAD originals that are to be used with the PTC Creo Collaboration Extensions from being deleted.

Preventing the deletion of *.creo files

Do the following to configure **SAP ECTR interface to PTC Creo** to prevent `*.creo` files of CAD originals that are to be used with the PTC Creo Collaboration Extensions from being deleted:

1) Use a text editor, e.g., `Notepad++`, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.
2) Search for `PRO.delete.PTCCreoUniteTechnologyFiles.atcheckout`.
3) Set `PRO.delete.PTCCreoUniteTechnologyFiles.atcheckout = false`.

Result: When CAD originals from other CAD systems are loaded, their corresponding `*.creo` files will not be deleted.

PRO.delete.PTCCreoUniteTechnologyFiles.atcheckout

This option is used to specify if `.creo` files will be deleted when CAD originals from other CAD systems are loaded:

Value	Description
`false`	`*.creo` files will not be deleted.
`true`	`*.creo` files of CAD originals created with the `ACD`, `ACE`, `ACM`, `CAT`, `UGS`, INV, `SLE`, or `SLW` CAD applications will be deleted.
`<XXX>;<YYY>;…`	`*.creo` files of CAD originals created with the specified CAD applications will be deleted.

4.9.22 Using the CAD Desktop compatibility mode

CAD Desktop and SAP ECTR use different sort terms for components and references in their document structures. To avoid sorting issues, **SAP ECTR interface to PTC Creo** can be run in CAD Desktop compatibility mode. In CAD Desktop compatibility mode, **SAP ECTR interface to PTC Creo** will use the same sort terms as CAD Desktop when transferring data to SAP ECTR.

4.9.22.1 Activating the CAD Desktop compatibility mode

CAD Desktop compatibility mode

⚠ You should only use the CAD Desktop compatibility mode if it is absolutely necessary, e.g., if SAP ECTR and CAD Desktop need to be temporarily run in parallel.

Activating the CAD Desktop compatibility mode

Do the following to activate the CAD Desktop compatibility mode:

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Advanced** page.
3) Enable the [☑] **Enable CAD Desktop compatibility** check box.
4) From the pull-down menu, select one of the following options:
   - **Interpret merge & inheritance as a reference**
   - **Interpret merge & inheritance as a component**
   - **Interpret merge component / inheritance as reference**
5) Save your settings.

Result: **SAP ECTR interface to PTC Creo** will run in CAD Desktop compatibility mode.

4.9.22.2 Using the master group identifier “M”

With SAP ECTR, documents with CAD Drawings can be created as dependent documents of the master document of the represented model. In the default setting, SAP ECTR marks an existing relation between a dependent document and its master model by adding a `PRO:CD` sort term to the master document in the document structure. The “**D**” part of the sort term is called a master group identifier.

In CAD Desktop compatibility mode, the relation between a represented model and its drawing is marked with the `PRO:D` sort term. If a drawing is related to multiple models, the leading model will be marked with the `PRO:DM` sort term. In CAD Desktop compatibility mode, the `PRO:DM` sort term may also be used to map the relation between the master document and its dependent drawing document.

Using the master group identifier “M”

Do the following to use the master group identifier `M` in CAD Desktop compatibility mode:

1) Use a text editor, e.g., `Notepad++`, to open the file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.
2) Locate `sap.plm.document.mastergroup.identifier.read.PRO` and `sap.plm.document.mastergroup.identifier.write.PRO`.
3) Set the master group identifier `M`.
   - `sap.plm.document.mastergroup.identifier.read.PRO = M`
   - `sap.plm.document.mastergroup.identifier.write.PRO = M`
4) Save the file.

Result: The master group identifier `M` will be used.

4.9.22.3 Transferring BOM explosions from legacy data

Legacy data created with CAD Desktop may contain BOM explosion markers in the **Object Type** (`Reserve Field 4`) or at the corresponding position of the the document structure in the **Sort String** (`SORTF`).

When legacy data created with CAD Desktop are stored in SAP PLM after editing, SAP ECTR overwrites the BOM explosion identifiers from CAD Desktop.

You can configure **SAP ECTR interface to PTC Creo** to transfer the BOM explosion from legacy documents and document structures to model parameters (`SAP_BOM_ORG`) and assembly component parameters (`SAP_BOM_POS`) during **Save** and **Save and Display** operations.

CAD Desktop **BOM identifiers**	**SAP ECTR** **BOM explosion parameter value**
W	EXCLUDE_AND_KEEP_CHILDREN
WU	EXCLUDE_INCLUSIVE_CHILDREN

Table: BOM explosion parameter

Additional SAP transport required

⚠ Note that the functionality for transferring BOM explosion identifiers from legacy documents and legacy document structures to model parameters (`SAP_BOM_ORG`) and assembly component parameters (`SAP_BOM_POS`) requires that the transport for the **SAP note 299107** was implemented in your SAP system.

Transferring BOM explosions from legacy documents

Do the following to configure **SAP ECTR interface to PTC Creo** to transfer the BOM explosion identifiers from legacy documents and document structures to model parameters (`SAP_BOM_ORG`) and assembly component parameters (`SAP_BOM_POS`):

1) Open the **SAP ECTR interface to PTC Creo** configuration dialog.
2) Open the **Advanced** page.
3) In the **Bill of Material** section, enable the [☑] **Transfer BOM explosion from legacy documents and document structures into model and assembly component parameters** check box.
4) Save your settings.

Result: BOM explosion identifiers from legacy documents and document structures will be transferred to to model parameters (`SAP_BOM_ORG`) and assembly component parameters (`SAP_BOM_POS`) during **Save** and **Save and Display** operations.

4.9.23 Displaying icons depending on the status

In SAP ECTR, document-related information is displayed with icons appearing in a defined order.

Figure: Document icons

The icon at the third position can be controlled by setting the preference `plm.icon.group.doc.<Status>` in the file `default.txt`.

plm.icon.group.doc

The corresponding preference `plm.icon.group.doc.<Status>` can be set in the file `%PLM_INSTDIR%\customize\config\default.txt` or `%PLM_INSTDIR%\applications\pro\customize\config\default.txt`.

plm.icon.group.doc	Icons
`default`	/ /
`released`	
`blocked`	
`inchange`	
`archived`	

Table: plm.icon.group.doc and icons

Example

Status-dependent icons

```
plm.icon.group.doc.AC   = default
plm.icon.group.doc.IW   = default
plm.icon.group.doc.RE   = released
plm.icon.group.doc.RJ   = blocked
```

4.9.24 Setting preferences in the file default.txt

The file `%PLM_INSTDIR%\applications\pro\customize\config\default.txt` contains several configurable preferences to control the behavior of **SAP ECTR interface to PTC Creo** and SAP ECTR.

4.9.24.1 Setting the application timeout

plm.control.waitForSessionInMs.PRO

⚙ Maximum waiting time for an application to send a “successfully started” reply to SAP ECTR.

Value	Description
`[<Time>]`	Waiting time in milliseconds (default: 30,000)

4.9.24.2 Prevent originals loaded to CAD from being reloaded from SAP

plm.doc.checkout.reloadInCadForbiddenForApplications

⚙ Do not reload originals loaded to the CAD system using the PRO workstation application from SAP.

Value	Description
`@plm.doc.checkout.reloadInCadForbiddenForApplications@;PRO`	Prevent originals loaded to the CAD system using the workstation application from being reloaded from SAP.

4.9.24.3 Required preferences

The following preferences are required by **SAP ECTR interface to PTC Creo** to ensure smooth operation and must not be changed:

- Clone – Path to `Rename` executable
  `plm.adv.clone.renamerProgram.PRO = @PLM_BINDIR_PRO@\\CadRenamer.exe`
- Family tables – Family table instances are versioned along with the generic file.

  `plm.document.newVersion.notAllowedForFamilyMembers.PRO = true`
- Family tables – When a new version of a family table is created from within the CAD system, the versions of the family table instances and of the generic file will be kept in synch.
  `plm.document.variantKeepVersion.PRO = true`
- File names
  `plm.control.partnameRules.applExtension.PRO = prt`

SAP ECTR displays the working directory in the **Desktop** window. Any files without a document info record are displayed as unknown files. Set the following preferences if the saved versions of CAD files, e.g., *.prt.1, *.prt.2, …, should not be displayed in the **Desktop** window.

- Disable the display of the saved version

```
plm.tree.workdir.unknown.files.exclude.PRO.1=.*\\.prt\\.[0-9]+
plm.tree.workdir.unknown.files.exclude.PRO.2=.*\\.asm\\.[0-9]+
plm.tree.workdir.unknown.files.exclude.PRO.3=.*\\.drw\\.[0-9]+
plm.tree.workdir.unknown.files.exclude.PRO.4=.*\\.sec\\.[0-9]+
plm.tree.workdir.unknown.files.exclude.PRO.5=.*\\.lay\\.[0-9]+
plm.tree.workdir.unknown.files.exclude.PRO.6=.*\\.cem\\.[0-9]+
plm.tree.workdir.unknown.files.exclude.PRO.7=.*\\.rpt\\.[0-9]+
plm.tree.workdir.unknown.files.exclude.PRO.8=.*\\.dgm\\.[0-9]+
plm.tree.workdir.unknown.files.exclude.PRO.9=.*\\.frm\\.[0-9]+
plm.tree.workdir.unknown.files.exclude.PRO.10=.*\\.gph\\.[0-9]+
```

This section describes how to display the SAP PLM menu in PTC Creo Parametric and explains which commands can be executed from the SAP PLM menu.

5 How to use the menu and commands

There are several places in PTC Creo Parametric, where the **SAP PLM** menu for **SAP ECTR interface to PTC Creo** is displayed:

- in the **TOOLKIT** group on the **Tools** tab of the ribbon
- in the right-click menu of the Model Tree
- On the quick access toolbar
- SAP ECTR application structure

Optionally, the **SAP PLM** menu can be displayed at the following user-defined locations:

- on a user-defined tab of the ribbon
- on the General tab of the ribbon

Command Scope

Commands can be selected from the **TOOLKIT** group on the **Tools** tab of the **SAP PLM** menu or from the right-click menu of the Model Tree. Depending on where the command was selected, the scope of the command is different: it either applies to files or to originals.

5.1 Display of the commands

5.1.1 TOOLKIT tool in PTC Creo Parametric

The **SAP PLM** menu is accessible from the **TOOLKIT** group on the **Tools** tab of the ribbon.

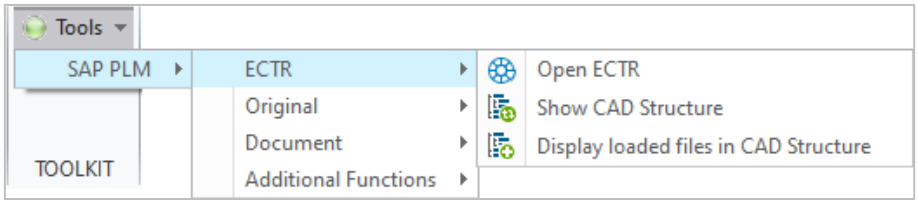


Figure: SAP PLM Menu as a TOOLKIT Tool

Modules without ribbon

The **SAP PLM** menu is available on the **Tools** tab in nearly all modes of PTC Creo Parametric.

Some modules of PTC Creo Parametric do not have a ribbon (e.g., REPORT, DIAGRAM). In these cases, the **SAP PLM** menu is directly integrated with the menu bar on the upper left.

Obsolete modules

The standard installation of PTC Creo Parametric ≥ 4.0 does not include any modules without a ribbon (e.g., REPORT, DIAGRAM). To reenable these obsolete modules, set the `config.pro` option `enable_obsolete_modes` in PTC Creo Parametric 4.0 to `yes`.

5.1.2 Right-click menu of the Model Tree in PTC Creo Parametric

In each mode, the **SAP PLM** menu is embedded automatically into the right-click menu of the Model Tree (right mouse button).

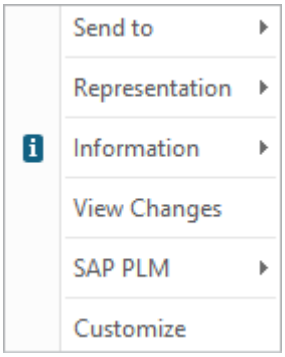


Figure: SAP PLM menu in the right-click menu of the Model Tree

5.1.3 Quick access toolbar in PTC Creo Parametric

You can customize the **SAP PLM** menu to add it to the quick access toolbar.

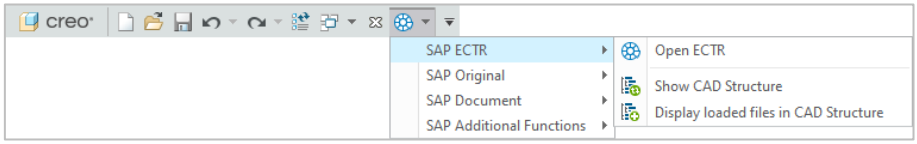


Figure: SAP PLM menu on the quick access toolbar

5.1.4 Custom tab in PTC Creo Parametric

The commands from the **SAP PLM** menu can be displayed on a custom tab of the ribbon.

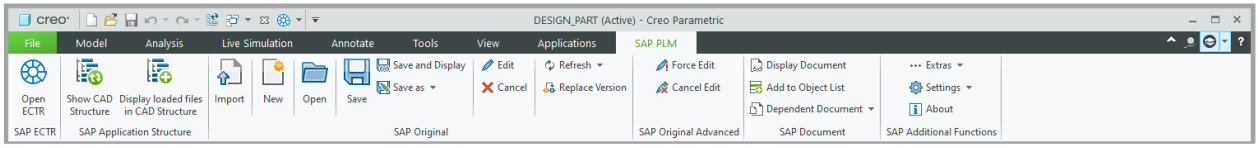


Figure: Commands from the SAP PLM menu in a custom tab

Configuring a custom tab

You can configure a custom **SAP PLM** tab. For each mode in PTC Creo Parametric, you can create individual user-defined tabs. On the custom tab, the commands may be arranged in a different order than in the **SAP PLM** menu.

5.1.5 Application structure in SAP ECTR

A selection of commands from the **SAP PLM** right-click menu of the Model Tree is also accessible from the right-click menu of the SAP ECTR application structure.

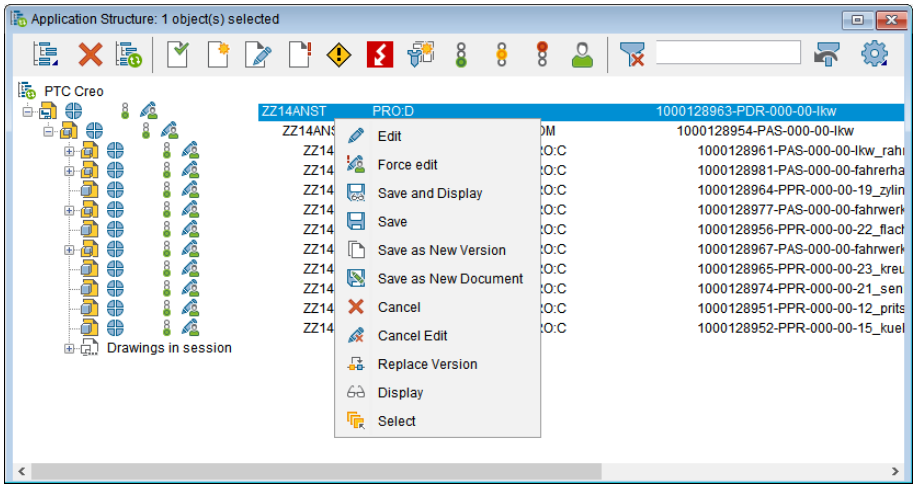


Figure: Commands from the SAP PLM menu of the application structure

5.2 Scope of commands selected from the SAP PLM menu

A command will apply to different files or originals depending on whether it is selected from the **SAP PLM** menu of the ribbon or from the **SAP PLM** right-click menu.

Commands on the **SAP PLM** tab of the ribbon and commands in the quick access toolbar will apply to the files or originals in the active graphics window.

Commands in the **SAP PLM** right-click menu will apply to the files or originals selected in the Model Tree.

5.2.1 Current or active original

You can activate assembly components in the Model Tree. The active component is highlighted in the Model Tree by a special icon. Commands selected from the **SAP PLM** menu on the **Tools** tab or from the quick access toolbar will always apply to the active component. If no component is active, the command applies to the file currently open in the graphics window.

5.2.2 Right-click menu of the Model Tree

Components can be selected in the Model Tree of PTC Creo Parametric. Commands in **SAP PLM** right-click menu of the Model Tree will apply to all components selected in the Model Tree.

5.2.3 User-defined tab

Custom tabs can contain commands from the **SAP PLM** menu and from the **SAP PLM** right-click menu. Depending on its origin, the command will apply to the current or the active file or original or to the selected component.

Creating a user-defined tab

Only add commands from the **SAP PLM** menu to user-defined tabs. This ensures that the commands in the user-defined tab will apply to the same scope as the commands in the **SAP PLM** menu on the **Tools** tab.

Context menu

Commands in the **SAP PLM** menu will always apply to the current or active file or original. The **SAP PLM** context menu only applies to the components selected in the Model Tree.

5.2.4 Right-click menu in the application structure

The application structure allows the selection of files without a document or originals with a document. Commands in the right-click menu of the application structure will apply to the files or originals selected in the application structure.

5.2.5 Files without document or material

Files without a document can only be displayed in the application structure or imported to SAP.

If a command cannot be executed because there is no document yet, a corresponding message is displayed for the selected command.

File without document!
Action can not be executed.

If a document exists for an original, but there is no material assignment, you can assign materials to it. Commands that affect assigned materials or bills of materials cannot be executed if no material has been assigned.

If a command cannot be executed because no material has been assigned yet, a corresponding message is displayed for the that command.

Document without material assignment!
Action can not be executed.

5.2.6 Including originals in the document structure of the scope

The scope of some commands can be extended to include further originals in the document structure.

If the document structure contains further originals that could be included in the command's scope of application, a corresponding option ([☑] check box) is displayed when you select the command:

Include all <…> in the document structure

5.2.7 Removing originals in memory from the scope

The scope of some commands could affect other originals that are in memory, but this is not intended and could lead to data inconsistencies.

If a command affects other originals in memory and this is not intended, remove all other originals from memory.

A corresponding message is displayed for the selected command.

All originals will be removed from memory

5.2.8 Alternative effects on the scope

Some commands can have alternative effects on the files or originals in the scope.

If a command has alternative effects on the files or originals in its scope, the alternatives are suggested with ⊙ radio buttons when you execute the command.

5.3 Application of ribbon commands

The ribbon and quick access toolbar of **SAP ECTR interface to PTC Creo** contain commands for the following contexts:

- **SAP ECTR**
- **SAP Original**
- **SAP Document**
- **Additional Functions**
- **SAP User-Defined Feature**

SAP PLM menu

The structure of commands described in this chapter reflects the order in which they appear in the **TOOLKIT** on the **Tools** tab of the **SAP PLM** menu.

5.3.1 SAP ECTR context commands

SAP ECTR interface to PTC Creo provides the following commands in the **SAP ECTR** context:

- **Open ECTR**
- **Show CAD Structure**
- **Display loades files in CAD Structure**

5.3.1.1 Open ECTR

When to use

Use this command when you are working in PTC Creo Parametric and want to bring SAP ECTR to the front to display information about documents, materials, etc.

How to use

Do the following to open SAP ECTR from within the CAD system:

1) Click the **Open ECTR** icon.

Result: SAP ECTR is launched and brought to the front. The PTC Creo Parametric CAD system remains active.

5.3.1.2 Show CAD Structure

When to use

Use this command when you are working in PTC Creo Parametric and want to display the CAD file from the active graphics window in the SAP ECTR application structure.

How to use

Do the following to display the CAD structure, e.g., of an assembly, in the SAP ECTR application structure.

1) Click **Show CAD Structure**.

Result: The CAD structure, e.g., of an assembly, is displayed in the SAP ECTR application structure.

5.3.1.3  Display loaded files in the CAD structure

When to use

Use this command when you are working in PTC Creo Parametric and want to display the CAD file from the working session in the SAP ECTR application structure.

How to use

Do the following to display the files and originals loaded in the CAD system in the SAP ECTR application structure.

1) Click the **Display loades files in CAD Structure** icon.

Result: The files loaded in the CAD system will be displayed in the SAP ECTR application structure grouped by their file extensions.

5.3.2 Commands in the SAP Original Copy context

SAP ECTR interface to PTC Creo provides the following commands in the **SAP Original Copy** context:

-  **Import**
-  **New**
-  **Open**
-  **Save**
-  **Save and Display**
-  **Save as New Version**
-  **Save as New Document**
-  **Edit**
-  **Cancel**
-  **Refresh Originals**
-  **Refresh Attributes**
-  **Replace Version**
-  **Cancel Edit**
-  **Force Edit**

5.3.2.1 Import

When to use

Use this command when you are working in PTC Creo Parametric and want to import existing CAD files to SAP PLM.

Scope of the Import command

Use the **Import** command only to import CAD files created in the current CAD session. Import with the command **Import** e. g. components newly created in the context of an assembly.

Out-of-Scope

Do **not** use the **Import** command …

- … for the initial migration of existing data to SAP!
- ... for data exchange with external partners!
- ... in collaboration scenarios SAP!

How to use

Do the following to create documents for the CAD files in session and import the associated originals from the CAD system to SAP:

1) Create a new file or open a file.
2) Click the **Import** icon.
3) Note the import scope in the **Import** dialog.
4) Use the **Identify Relations** command to identify any additional files that also need to be imported.
5) Add the files you determined to the import scope.
6) Import the files listed in the import scope by clicking **Import**.
7) In the SAP ECTR dialog, confirm the import by clicking **OK**.

Result: The files have been imported to SAP and the originals are displayed in the CAD system ( ).

Identifying relations

Usually, files are renamed when they are imported to SAP, provided the system is configured accordingly. Files and originals that use renamed files must have been loaded into the CAD system memory. This is the only way to ensure that these originals know the new names of the referenced files.

For this reason, make sure to identify the relations before performing the import. Add the files you determined to the import scope.

Click **Identify Relations** to load the files into memory and to determine their relations to the files in the import scope.

Using help information

For this dialog, help information is available.

Select **Help** to display information on the meaning of the symbols used in the **Identify further files for Import** dialog.

5.3.2.2 New

When to use

Use this command when you are working in PTC Creo Parametric and want to create a new document (info record and new original).

How to use

Do the following to create a new original and its associated document info record in SAP PLM and open the original in the CAD system:

1) Click **New**.
2) In the SAP ECTR dialog, select the desired `DType` and the desired template.
3) Specify all required settings in the SAP ECTR dialog.
4) Click **OK** to create a new original and document info record.

Result: A new document (original and associated document info record) is created. The document is displayed in SAP ECTR in the active folder. The new original is opened in PTC Creo Parametric where you can edit it ( ).

Document type and start model

By selecting the desired `DType` and template, you can define properties such as the document type of the new original and the CAD start model, e.g., a start part or a start assembly.

Active folder

In SAP ECTR, the new document will be displayed in the active folder and marked with a `Dirty Flag` ( ), for instance because its attributes may have not been refreshed yet.

5.3.2.3 Open

When to use

Use this command when you are working in PTC Creo Parametric and want to search for an existing document (document info record and original) in SAP PLM and then open the original directly from within PTC Creo Parametric. As an alternative, you can run the document search from the SAP GUI.

How to use

Do the following to search for an original in SAP and open it in the CAD system:

1) Click the **Open** icon.
2) Specify your search criteria in the SAP ECTR dialog.
3) In the SAP ECTR dialog, click **OK**.
4) From the results displayed in the SAP ECTR dialog, select the desired documents.
5) In the SAP ECTR dialog, click **OK**.

Result: The originals of the selected documents are available in the session directory and are opened in the CAD system.

5.3.2.4 Save

When to use

Use this command when you are working in PTC Creo Parametric and want to save an intermediate state of an original in SAP PLM and then continue your work on the original in PTC Creo Parametric.

How to use

Do the following to save an original to SAP and continue editing in the CAD system:

1) Click the **Save** icon.
2) Add locally changed originals to the **Save** list using the **Save** dialog or ignore originals that have been changed locally.
3) Click **Save** to save your originals.

Result: The new original is saved in SAP and is ready for editing ( ).

Dirty flag

If the document has been marked with a `Dirty Flag` ( ), the `Dirty Flag` will be removed.

Prerequisite for saving

You can only save originals to SAP that are edited in SAP ( ).

Saving locally changed originals

Locally changed originals are originals that have been edited in the CAD system but not in SAP ( ). If the document structure of the original to be saved includes originals that are edited in SAP ( ), they will also be saved automatically. Originals that have only been changed locally cannot be saved.

The **Save** dialog opens.

You can add locally changed originals to the **Save** list by clicking **Edit** or exclude them from saving by clicking in the ☑ **Ignore <x> modified originals** column. Ignored originals are not checked in to SAP when saving.

The **Save** button is only available after you have edited or ignored all locally changed originals.

Saving originals with files without a document in the structure

Originals cannot be saved if they reference a file without a document. In this case, the **Save** dialog displays a conflict message.

Files without document in structure

To solve the conflict, click **Import** or **Ignore**.

5.3.2.5 Save and display

When to use

Use this command when you are working in PTC Creo Parametric and want to save an original in SAP PLM but do not want to edit that original in PTC Creo Parametric.

How to use

Do the following to save an original to SAP and switch from Edit mode to Display mode. This will enable other users to edit the original in the CAD system.

1) Click the **Save and Display** icon.
2) Add locally changed originals to the **Save** list using the **Save** dialog or ignore originals that have been changed locally.
3) Click **Save and Display** to save your originals.

Result: The original has been saved to SAP and is no longer in Edit mode ( ). The saved originals in the structure are no longer in edit mode either ( ).

Dirty flag

If the document has been marked with a `Dirty Flag` ( ), the `Dirty Flag` will be removed.

Locally changed originals and files without a document

For locally changed originals, originals with files without a document in the structure, and for files without a document, the **Save** und **Save and Display** command behaves like the **Save** command.

5.3.2.6 Save as new version

When to use

Use this command when, while working in PTC Creo Parametric, you want to create a new document version for an original.

If the document for which you want to create the new version has been edited ( ), the changes will be applied to the new version of the original. The original of the initial version will be reset to the version last checked in ( ). This means that the initial original remains in SAP PLM as checked in the last time.

If dependent documents exist for the document, new versions will be created for the dependent documents as well, provided the system has been configured accordingly.

How to use

Do the following to save an original as a new version in SAP:

1) Click the **Save as new Version** icon.
2) In the SAP ECTR dialog, click **Yes**.

Result: A new document version of the original has been created. The original of the new document version has been checked in to SAP ( ) and loaded in PTC Creo Parametric.

5.3.2.7 Save as new document

When to use

Use this command when you are working in PTC Creo Parametric and want to create a new document and a new original. The initial original is copied and then renamed according to the applicable rules.

If the original for which you want to create the new document has been edited (), the changes will be applied to the new original. The original of the initial document will be reset to the one last checked in (). This means that the initial original remains in SAP PLM as checked in the last time.

How to use

Do the following to save a document (original and document info record) as a new document in SAP:

1) Click the **Save as New Document** icon.
2) Edit the settings in the SAP ECTR dialog, if required.
3) In the SAP ECTR dialog, click **OK**.

Result: The new document has been saved. The new original has been checked in to SAP () and loaded in PTC Creo Parametric.

5.3.2.8 Edit

When to use

Use this command if you are working in PTC Creo Parametric and want to edit a displayed original in SAP.

How to use

Do the following to request an exclusive lock in SAP in order to edit the original in the CAD system.

1) Click the **Edit** icon.
2) Note the message in the **Edit <filename>** dialog, if applicable.
3) In the **Edit <filename>** dialog, click **Continue**.

Result: The original is in Edit mode () in SAP.

Dirty Flag

If the document has been marked with a `Dirty Flag` ( ), the `Dirty Flag` will be removed.

5.3.2.9 Cancel

When to use

Use this command when you are working in PTC Creo Parametric and want to discard edits to an original in memory and the working directory and retrieve the state last checked in to SAP PLM instead.

How to use

Do the following to cancel editing the original, discard the latest modifications and reload the original from SAP:

1) Click the **Cancel** icon.
2) Select the desired effect of the command (option with ⊙ radio button).
3) Add all originals in the document structure to the scope of this command ([☑] check box), if applicable.
4) Click the **Continue** icon.
5) Click **Remove** to remove all originals from memory.
6) Open the closed originals that use the original directly in PTC Creo Parametric by selecting **File ▶ Open**.

Result: Edit mode in SAP PLM has been canceled. The original has been replaced by the state last saved for this version in SAP.

Options

The ⊙ **Close original in CAD** option loads the original from SAP PLM without opening it in the CAD system.

The ⊙ **Reload original from SAP system** option loads the original from SAP PLM and opens it in the CAD system.

5.3.2.10 Refresh originals

When to use

Use this command when you are working in PTC Creo Parametric and want to load the current state of an original from SAP PLM. This command is helpful, for instance, if your assembly uses components which are obsolete because another user changed them. With this command, you can only replace originals of the same document version.

How to use

Do the following to load originals modified by other users from SAP PLM and replace the originals in the CAD system:

1) Click the **Refresh Originals** icon.
2) Click **Remove** to remove all originals from memory.

Result: The original changed by another user is now open in the CAD system.

5.3.2.11 Refresh attributes

When to use

Use this command when you are working in PTC Creo Parametric and want to refresh the attributes of an original from SAP PLM.

How to use

Do the following to refresh the attributes of the original loaded in the CAD session with values from SAP PLM:

1) Click the **Refresh Attributes** icon.
2) Add all originals in the document structure to the scope of this command ([☑] check box), if applicable.
3) Click the **Continue** icon.

Result: The parameters in the original and in the SAP object fields (e.g., document and material) have been refreshed.

Detailed information

For details on how to configure SAP ECTR, please refer to the SAP ECTR documentation.

Configuration

Depending on the configuration specified in the files `attributes-from-sap.xml` and `attributes-to-sap.xml`, the attributes from SAP fields are transferred to the CAD parameters or vice versa.

Syntax of the configuration files

In the XML element `APPL_ATTRIBUTE`, make sure to use the `PROPROP:` prefix for system parameters and the `PROPARAM:` prefix for user-defined parameters.

Examples

Examples can be found in the directory `%PLM_INSTDIR%\applications\pro\template-customize\config` in the files `attributes-from-sap.xml` and `attributes-to-sap.xml`.

System parameters

System parameters can only be transferred from PTC Creo Parametric to the SAP system.

5.3.2.12 Replace version

When to use

Use this command when you are working in PTC Creo Parametric and want to load the current version of an original from SAP PLM. For instance, this command can be used if you use components in your assembly for which newer versions (|) exist. This command can only be used to replace originals of other document versions.

How to use

Do the following to replace an original in the CAD system with an original of another document version from SAP.

1) Click **SAP PLM** ▶ **Original** ▶ **Replace Version**.
2) Click **Remove** to remove all originals from memory.
3) Select the desired version in the SAP ECTR dialog.
4) Confirm your selection by clicking **OK**.

Result: The original in the CAD system has been replaced with an original of another document version.

5.3.2.13 Force edit

When to use

Use this command when you are working in PTC Creo Parametric and want to force the editing of an original in SAP.

How to use

Do the following to force an exclusive lock in SAP in order to edit the original in the CAD system:

1) Click the **Force Edit** icon.
2) Note the message in the **Edit <filename>** dialog, if applicable.
3) In the **Edit <filename>** dialog, click **Continue**.

Result: The original is in Edit mode ( ) in SAP.

5.3.2.14 Cancel edit

When to use

Use this command when, while working in PTC Creo Parametric, you want to cancel editing a CAD original without discarding the local changes.

How to use

Do the following to cancel editing the original by releasing the exclusive lock without discarding local changes to the original:

1) Click the **Cancel Edit** icon.
2) Add all locked originals in the document structure to the scope of this command ([☑] check box), if applicable.
3) Click the **Continue** icon.

Result: Editing in SAP has been canceled, but local changes to the original have not been discarded.

5.3.3 Commands in the Document context

SAP ECTR interface to PTC Creo provides the following commands in the **Document** context:

- **Display Document**
- **Add to Object List**
- **Open Original of Dependent Document**
- **Create Dependent Document**

5.3.3.1 Display document

When to use

Use this command when you are working in PTC Creo Parametric and want to display, and possibly edit, the document of the current original in the **Object Browser** of SAP ECTR.

How to use

Do the following to display the document associated with an original:

1) Click the **Display Document** icon.

Result: The document associated with the original is displayed in the **Object Browser** of SAP ECTR.

5.3.3.2 Add to object list

When to use

Use this command when, while working in PTC Creo Parametric, you want to add the document associated with the current original to an existing or new object list. Object lists are helpful if you want to collect and jointly edit several documents, e.g., to change their document status.

How to use

Do the following to add the document associated with the current original to an object list of SAP ECTR.

1) Click the **Add to Object List** icon.
2) In the SAP ECTR dialog, select an object list or create a new one.
3) Select the desired option in the SAP ECTR dialog.

Result: The document has been added to the object list.

Temporary object lists

When you exit SAP ECTR, all object lists created during the session will be deleted. They will **not** be available in a new session.

5.3.3.3 Open original of dependent document

When to use

Use this command when, while working in PTC Creo Parametric, you want to open the original of a dependent document. If a dependent drawing document exists for the document associated with the current original, e.g., a part, you can open the dependent drawing directly from SAP PLM in PTC Creo Parametric.

How to use

Do the following to open the original of a dependent document from SAP PLM (e.g., open a drawing for a part):

1) Open a part or an assembly for which a dependent document exists.
2) Click the **Open Original of Dependent Document** icon.

Result: The original of the dependent document (this is usually the drawing) is loaded from SAP PLM and opened in the CAD system.

5.3.3.4 Create dependent document

When to use

Use this command when, while working in PTC Creo Parametric, you want to create a drawing document that depends on the document associated with the current original. Depending on the configuration of your system, dependent documents follow the master document and can only be versioned together with their master document, for example.

How to use

Do the following to create a new original and a document as a dependent document of the current original in SAP PLM (e.g., create a drawing with document for a part):

1) Open a part or assembly for which a dependent document exists.
2) Click the **Create Dependent Document** icon.
3) Specify all required settings in the SAP ECTR dialog.
4) In the SAP ECTR dialog, select the desired `DType` and the desired template.
5) Click **OK** to create a new original with its document info record.

Result: A new dependent document (original with its document info record) has been created. The document is displayed in SAP ECTR in the **Dependent Documents** container of the active folder. The new original has been opened in PTC Creo Parametric and is ready for editing ( ). In the SAP ECTR, the indicates that dependent documents of this document exist.

5.3.4 Commands in the Additional Functions context

SAP ECTR interface to PTC Creo provides the following commands in the **Additional Functions** context:

- **Assign Material**
- **Display Material**
- **Load Drawings**
- **Load Family Table Instance**
- **Load References**
- **Delete Old Versions**
- **Initialize Parameters**
- **Open Log Directory**
- **Open Working Directory**
- **Settings)**
- **Trace Level**
- **About**
- **Add Component**
- **Add Formats**
- **Add Drawing**
- **Select Leading Model**
- **Add Table**
- **Refresh Component Parameters**

Flagging a material as exclusive

You can flag a material link as exclusive. If the document of the original is linked to multiple materials, the **Display Material** command will apply to the material marked as **exclusive** in SAP ECTR. If no material was flagged, the commands will apply to the first material listed in the material links list of the document.

5.3.4.1 Assign material

When to use

Use this command when you are working in PTC Creo Parametric and want to assign a material existing in SAP PLM to the document associated with the current original. SAP ECTR indicates that the document is linked to one or more materials. If multiple materials are linked, you can flag one of the material links as the exclusive material link.

Existing material links

Icon / Overlay	Meaning
	A material link exists.
	Multiple material links exist.
	No material link has been flagged as exclusive yet.

Table: Icon and overlays for material links

How to use

Do the following to assign an existing material from SAP PLM to the document of the current original:

1) Click the **Assign Material** icon.
2) Search for the desired material in the SAP ECTR dialog.
3) Confirm your selection by clicking **OK**.

Result: This links the material to the document associated with the original.

Assigning multiple materials

Using the **Assign Material**, it is possible to link multiple materials to the document associated with the original one after the other.

5.3.4.2 Display material

When to use

Use this command when, while working in PTC Creo Parametric, you want to display or edit the material associated with the current original.

How to use

Do the following to display the material that is assigned to the document of the current original:

1) Click the **Display Material** icon.

Result: The material will be displayed in the **Object Browser** of SAP ECTR.

5.3.4.3 Load drawings

When to use

Use this command when you are working in PTC Creo Parametric and want to load all drawings in the working directory from PTC Creo Parametric. Loaded drawings can be displayed in the SAP ECTR application structure, for example.

How to use

Do the following to load all drawings from the working directory to memory.

1) Click the **Load Drawings** icon.

Result: All drawings in the working directory will be loaded in PTC Creo Parametric.

5.3.4.4 Load references

When to use

Use this command when you are working in PTC Creo Parametric and want to load all references of the active or selected CAD model to PTC Creo Parametric.

How to use

Do the following to load the references of a CAD file to memory:

1) Click the **Load References** icon.

Result: The references will be loaded in PTC Creo Parametric.

References

In PTC Creo Parametric, references are created using the functions of the **Get Data** functional group.

5.3.4.5 Load family table instances

When to use

Use this command when you are working in PTC Creo Parametric and want to load all instances of a family table to PTC Creo Parametric.

How to use

Do the following to load the family table instances of the selected generic file to memory:

1) Click the **Load Family Table Instances** icon.

Result: The family table instances are loaded in PTC Creo Parametric.

5.3.4.6 Delete old versions

When to use

Use this command when you are working in PTC Creo Parametric and want to delete the saved versions from the **SAP ECTR interface to PTC Creo** working directory. Each time you save a version in PTC Creo Parametric the version number appended to the extension of the file in the working directory is incremented by 1, e.g., `prt0001.prt.1` → `prt0001.prt.2`.

How to use

Do the following to delete any old versions (e.g., *.prt.1 - *.prt.n) created during saving in PTC Creo Parametric from the working directory:

1) Click the **Delete Old Versions** icon.

Result: The old versions created during saving in PTC Creo Parametric have been deleted.

Automatically deleting versions that were created by saving

Versions that were created during saving in PTC Creo Parametric can be automatically deleted when the **Save** command in SAP is used. Enable this functionality on the **Save** page of the **SAP ECTR interface to PTC Creo** Configuration dialog.

5.3.4.7 Initialize parameters

When to use

Use this command when, while working in PTC Creo Parametric, you want to add and initialize missing parameters of an original. Using this command, you can, for instance, add new parameters to CAD originals that have been migrated from legacy systems. The new parameters can be initialized using the specified initial value.

How to use

Do the following to create missing parameters in CAD files and initialize the parameters using predefined values:

1) Click the **Initialize Parameters** icon.
2) Add all originals in the document structure to the scope of this command ([☑] check box), if applicable.
3) Click the **Continue** icon.

Result: The parameters have been created and initialized in the CAD file.

Configuration

To configure the **Initialize Parameters** command, select **SAP PLM ▶ Additional Functions ▶ Settings ▶ Settings**. You can configure whether missing parameters in the model or drawing will be initialized using the specified initial value.

5.3.4.8 Open log directory

When to use

Use this command when you are working in PTC Creo Parametric and want to open the log directory of **SAP ECTR interface to PTC Creo**.

The log files for **SAP ECTR interface to PTC Creo** are stored in the log directory `C:\Users\<USERNAME>\.riess\SAPInterface`. These files might be required when you request support.

How to use

Do the following to open the **SAP ECTR interface to PTC Creo** log directory in Windows Explorer:

1) Click the **Open Log Directory** icon.

Result: The **SAP ECTR interface to PTC Creo** log directory has been opened in Windows Explorer.

5.3.4.9 Open working directory

When to use

Use this command when you are working in PTC Creo Parametric and want to open the working directory of **SAP ECTR interface to PTC Creo**.

How to use

Do the following to open the **SAP ECTR interface to PTC Creo** working directory in Windows Explorer:

1) Click the **Open Working Directory** icon.

Result: The **SAP ECTR interface to PTC Creo** working directory has been opened in Windows Explorer.

5.3.4.10 Settings

When to use

Use this command when you are working in PTC Creo Parametric and want to change the settings for **SAP ECTR interface to PTC Creo**.

How to use

Do the following to configure the settings for SAP ECTR interface to PTC Creo:

1) Click the **Settings** icon.
2) Make the desired settings.
3) Click **Next** or **Back** to browse to the next or previous page and specify the configuration settings as required.
4) Click **Save** to complete the configuration.

Result: **SAP ECTR interface to PTC Creo** has been configured and the settings have been saved.

Closing the configuration dialog without saving

Click to close the configuration dialog without saving your edits.

5.3.4.11 Trace level

When to use

Use this command when you are working in PTC Creo Parametric and want to change the trace level of the log files for **SAP ECTR interface to PTC Creo** at runtime.

How to use

Do the following to set the trace level of **SAP ECTR interface to PTC Creo**:

1) Click the **Trace Level** icon.
2) Select the desired trace level (⊙ radio button).
3) Confirm your selection by clicking **OK**.

Result: The trace level of **SAP ECTR interface to PTC Creo** has been set.

5.3.4.12 About

When to use

Use this command when you are working in PTC Creo Parametric and want to display information on **SAP ECTR interface to PTC Creo**.

How to use

Do the following to display information on **SAP ECTR interface to PTC Creo**:

1) Click the **About** icon.

Result: The system displays information on **SAP ECTR interface to PTC Creo**.

5.3.4.13 Add component

When to use

Use this command when you are working in PTC Creo Parametric and want to add components from SAP PLM to your assembly.

How to use

Do the following to search for a new component in SAP and insert it into an assembly from SAP PLM:

1) Open an assembly.
2) Click the **Add Component** icon.
3) Use the SAP ECTR dialog to search for the component via its document or material or using the search options displayed there.
4) Place the component in the assembly.

Result: The component is now included in the assembly.

5.3.4.14 Add format

When to use

Use this command when you are working in PTC Creo Parametric and want to add a drawing format from SAP PLM to your drawing.

How to use

Do the following to select a drawing format in SAP PLM aus and add it to the drawing:

1) Open a drawing.
2) Click the **Add Formats** icon.
3) Select the desired drawing frame from the SAP ECTR dialog.

Result: The drawing format has been added to the drawing and the attributes have been refreshed.

Drawing and/or represented model without a document

If no document exists for a drawing and/or represented model, the attributes will not or only partly be refreshed.

5.3.4.15 Add drawing model

When to use

Use this command when you are working in PTC Creo Parametric and want to add a drawing model from SAP PLM to your drawing.

How to use

Do the following to search for a drawing model in SAP and add it to a drawing:

1) Open a drawing.
2) Click the **Add drawing model** icon.
3) Use the SAP ECTR dialog to search for the drawing model via its document or material or using the available search options.

Result: The drawing model has been added to the drawing.

5.3.4.16 Select leading model

When to use

Use this command when you are working in PTC Creo Parametric and want to select the leading model of a drawing containing multiple drawing models to PTC Creo Parametric.

How to use

Do the following to select the leading model of a drawing containing multiple drawing models:

1) Open a drawing with multiple models.
2) Click the **Select Leading Model** icon.
3) Highlight the leading model in the **Select the leading drawing model** dialog.
4) Confirm your selection by clicking the **OK** icon.

Result: The leading model of the drawing has been selected.

Leading model of a drawing

If you want to transfer (refresh) parameters (attributes) from the represented model of a drawing to the document of the drawing, you should note that the leading model must be selected if the drawing contains multiple models. The parameters of the selected model will be transferred.

5.3.4.17 Add table

When to use

Use this command when, while working in PTC Creo Parametric, you want to add a table with repeat region to your drawing. Then, you can create BOM balloons.

Creating BOM balloons based on data from the bill of materials

Do the following to create BOM balloons from the bill of materials:

Step		Application
1	Add Table	**SAP ECTR interface to PTC Creo**
3	Creating BOM balloons	PTC Creo Parametric
3	Displaying component parameters in BOM balloons	PTC Creo Parametric

Table: Steps for BOM balloons

How to use

Do the following to add a BOM balloons table with a repeat region for component parameters to a drawing:

1) Click the **Add Table** icon.
2) Place the table in the drawing.
3) Update the table in PTC Creo Parametric, if required.

Result: The table has been added to the drawing and shows the bills of materials. The ballooning functions of PTC Creo Parametric now allow you to create BOM balloons. The balloons can be used to display component parameters, e.g., the item number from the repeat region.

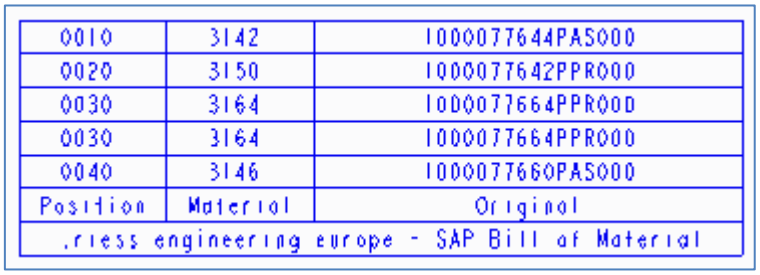


Figure: Table with repeat region

Refreshing component parameters

When you add a table, the required data is automatically transferred from SAP PLM to the component parameters. This has the same effect as the **Refresh Component Parameter** command.

Creating BOM balloons

Do the following to create BOM balloons in the drawing:

1) Create all required balloons by using the corresponding table and balloon commands in PTC Creo Parametric.

Result: The balloons are displayed in the drawing. The balloons show the internal numbers assigned by PTC Creo Parametric.

5.3.4.18 Refresh component parameters

When to use

Use this command when, while working in PTC Creo Parametric, you want to create or refresh component parameters in the active assembly. The component parameters are populated with data from the bill of materials, e.g., the item number and material number.

How to use

Do the following to populate the component parameters for BOM balloons with data from the bills of materials:

1) Click the **Refresh Component Parameter** icon.

Result: The component parameters are populated with data from the document BOM.

Command scope

The **Refresh Component Parameter** command can only be used on assemblies. For material linked to the assembly document must exist a material BOM. If you select the command from within a drawing, it is executed for the assembly represented in the drawing.

Refreshing BOM balloons based on data from the bill of materials

If you have already added a table with repeat region to your drawing, you can use the **Refresh Component Parameter** command to explicitly refresh the component parameters and update the BOM balloons with data from the bill of materials.

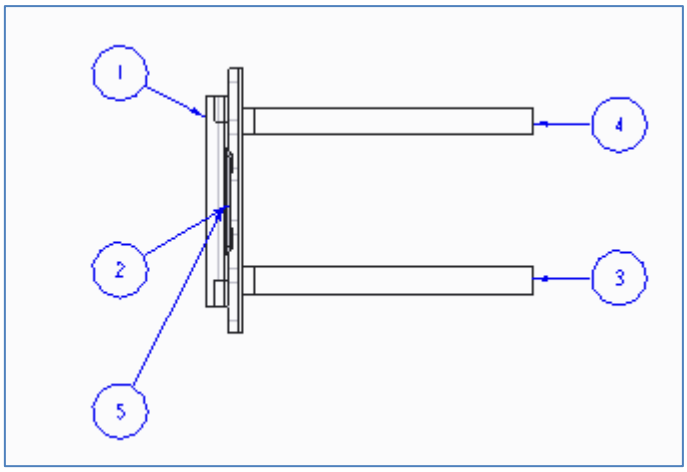


Figure: Balloons with internal numbering

Displaying component parameters in BOM balloons

Do the following to display the item numbers from the BOM in the balloons:

1) Select the table.
2) Right-click to change the table properties.
3) Change the BOM balloon parameters.
4) Click the table column that contains the item numbers from the bill of materials.

Result: The balloons now show the item numbers from the bill of materials.

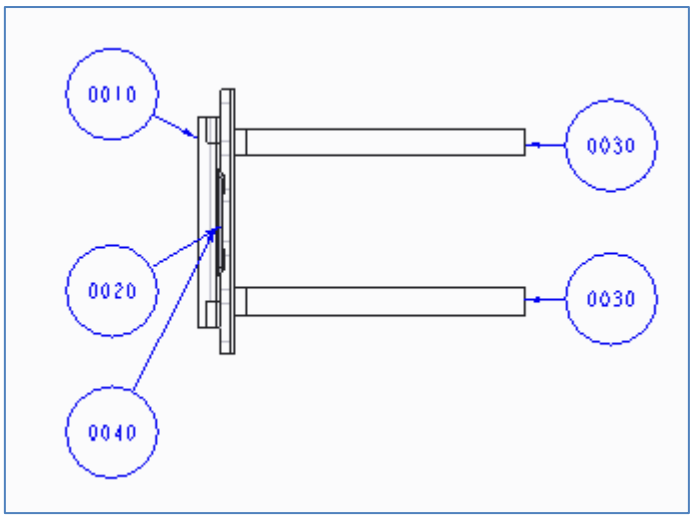


Figure: Balloons with BOM item numbers

5.3.5 Commands in the SAP custom features context

PTC Creo Parametric does not support direct open of UDF files (`*.gph`). Consequently, UDF files cannot be imported to SAP or edited directly from within PTC Creo Parametric. However, UDF files can be imported, edit and saved from the working directory.

SAP ECTR provides the following functions for working with UDFs.

-  **Import UDF**
-  **Edit UDF**
-  **Cancel UDF**
-  **Cancel Edit UDF**
-  **Save UDF**
-  **Save and Display UDF**
-  **Save as New Version**

Using UDFs

UDFs stored in SAP need to exist in the working directory before they can be used. Do the following to ensure this: Define a search query for UDFs in SAP ECTR. Highlight the desired UDFs and their corresponding reference parts or source parts and select the **Document ▶ Transfer Original(s)** command from the right-click menu.

Deploying UDFs to the working directory using a custom command

You can also deploy UDFs using a custom command from the right-click menu of SAP ECTR.

UDF source parts

The UDF geometry is defined in the UDF source part. UDFs that are created with the **Subordinate** option are dependent on their UDF source parts. If the UDF source part is renamed after the definition of the UDF, the UDF can no longer be used. **SAP ECTR interface to PTC Creo** therefore checks if the UDF source part is known to SAP. Originals that are known to SAP are usually not renamed.

UDF reference parts

The UDF geometry is defined in the UDF source part. UDFs that are created with the **Stand Alone** option are not dependent on their UDF source parts. However, you may include a copy of the UDF source model in the definition of the UDF as a reference part and then, for example, use it as a reference when placing the UDF model. Like the UDF source part, the UDF reference part must not be renamed once the UDF has been defined. Therefore, **SAP ECTR interface to PTC Creo** uses a `DType` for UDF reference parts that does not rename originals during document creation.

UDF reference model file name

The name of the UDF reference model is assigned by the CAD system. For UDF reference models, the following naming pattern is used: `<UDFName_gp>.prt`.

5.3.5.1 Import UDF

When to use

Use this command when you want to import existing UDF files to SAP PLM from the working directory.

How to use

Do the following to create documents for the UDF files and import the associated originals from the CAD system to SAP:

1) Click the **Import UDF** icon.
2) In the **Import UDF** dialog box files, select the UDFs that you want to import.
3) Click the **Continue** icon.
4) Import the files listed in the import scope by clicking **Import**.
5) In the SAP ECTR dialog, confirm the import by clicking **OK**.

Result: The UDF files have been imported to SAP and the originals are displayed in the CAD system ( ).

5.3.5.2 ☐ <Command> UDF

When to use

Use the commands for UDFs if you want to edit existing CAD files to SAP PLM from the working directory, save them as new version etc.

How to use

Do the following to use the commands for UDFs:

1) Click the **<Command> UDF** icon.
2) In the **<Command> UDF** dialog box files, select the UDFs on which the UDF command should be used.
3) Click **Continue**.
4) Perform any further actions required to execute the command.

Result: The selected command has been applied to the UDF files.

5.4 Application of commands in the SAP PLM right-click menu in the Model Tree

The **SAP PLM** right-click menu in the Model Tree of PTC Creo Parametric shows a selection of commands that are also available in the SAP PLM menu. Commands in the right-click menu are used in the same way as their counterparts in the **SAP PLM** menu. Note that the commands in the right-click menu have different areas of application compared to the commands in the **SAP PLM** menu.

The following commands are only available in the **SAP PLM** right-click menu of the Model Tree.

- **Set BOM Explosion Parameter for Component**
- **Set BOM Explosion Parameter inside Model**

The following options can be used to set the parameters for BOM explosion:

- Explosion parameters
  - **Include position**
  - **Exclude position**
    - **Move sub position to top**
- Location parameters
  - **Base Unit of Measure**
  - **Quantity**

Parameters for BOM explosion for component

The following assembly component parameters can be set for the component:

Parameters	Value
SAP_BOM_POS	EXCLUDE_AND_KEEP_CHILDREN EXCLUDE_INCLUSIVE_CHILDREN
SAP_BOM_UNIT_POS	<Base Unit> from the file pro_bom_base_unit_of_measure.txt
SAP_BOM_QUANTITY_POS	<Quantity>

Parameters for BOM explosion in the model

The following model parameters can be set in the model:

Parameters	Value
SAP_BOM_ORG	EXCLUDE_AND_KEEP_CHILDREN EXCLUDE_INCLUSIVE_CHILDREN
SAP_BOM_UNIT_ORG	<Base Unit> from the file pro_bom_base_unit_of_measure.txt
SAP_BOM_QUANTITY_ORG	<Quantity>

5.4.1 Set BOM explosion parameters for component

When to use

Use this command if you are working in PTC Creo Parametric and want to set the parameters for BOM explosion of a component at its position within an assembly component parameter.

How to use

Do the following to set the parameters for BOM explosion of a component at the component's position:

1) Highlight a file or original in the Model Tree of PTC Creo Parametric
2) In the right-click menu, click **Set BOM Explosion Parameter for Component**.
3) In the **Set BOM Explosion Parameter for Component** dialog, set the parameters for BOM explosion and the SAP BOM position.
4) Confirm your selection by clicking **Continue**.

Result: The parameters for BOM explosion will be written to the assembly component parameters. The parameters for BOM explosion and positioning will be saved to SAP when the assembly is saved, and taken into account when a BOM is derived.

Deriving BOMs

Use the `Advanced Bill of Material maintenance` commands to derive the BOMs in SAP ECTR using the parameter settings for BOM explosion.

5.4.2 Set BOM explosion parameters inside model

When to use

Use this command if you are working in PTC Creo Parametric and want to set the parameters for BOM explosion for a component within a model parameter inside the model.

How to use

Do the following to set the parameters for BOM explosion of a component inside the model:

1) Highlight a file or original in the Model Tree of PTC Creo Parametric.
2) In the right-click menu, click **Set BOM Explosion Parameter inside Model**.
3) In the **Set BOM Explosion Parameter inside Model** dialog, set the parameters for BOM explosion and positioning.
4) Confirm your selection by clicking **Continue**.

Result: The parameters for BOM explosion will be written to the model parameters. The parameters for BOM explosion and positioning will be saved to SAP when the assembly is saved, and taken into account when a BOM is derived.

Deriving BOMs

Use the `Advanced Bill of Material maintenance` commands to derive the BOMs in SAP ECTR using the parameter settings for BOM explosion.

Parameters for BOM explosions for component vs. inside model

⚠ If parameters for BOM explosion of a component exist as assembly component parameters, the assembly component parameters will be taken into account with priority when a BOM is derived. If no parameters for BOM explosion exist as assembly component parameters, the parameters for BOM explosion inside the model will be taken into account.