CAD design supported by PRO.FILE

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PROCAD

Registered Trademarks:

Responsible for Content:

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About this manual

This PRO.FILE manual uses various signs and icons in order to guarantee a good readability and comfortable handling.

Step-by-step instructions:

For quicker finding within the manual, step-by-step instructions are marked with a margin heading.

Menu sequences and function calls
Menu sequences and function calls explained in this manual are marked in bold and in quotation marks.

Example:
"**File**" => "**New**" => "**Document description**"

Buttons and keys
Keys and buttons are highlighted by angle brackets.

Example:
"Confirm with <**OK**>."

Notes and warnings
To highlight special information the following icons are used:

Function call:	
"PRO.FILE" => "Extras" => "Options" => "Performance"	

Example:	
Boxes marked with this icon give subject-relevant examples for the usage of command lines, configuration strings and other software-relevant entries.	

Note:	
Boxes marked with this icon contain useful hints on the operation, configuration or installation of the PRO.FILE software.	

Attention:	
All information given in these boxes is very important and should be read carefully! Non-observance of these hints may lead to wrong functioning, display problems or other negative consequences.	

Important notes:	
The "stop sign" warns you of possible entry or operation errors, which may lead to loss of data!	

Attention – Undo not possible:	
All entries and configurations described in these boxes have to be made carefully, because they cannot be undone!	

1   Welcome to "CAD design supported by PRO.FILE"

This manual is to give you an overview on the functionality and philosophy of the PRO.FILE CAD integrations.

As enhancement to the separate operation manuals of the PRO.FILE CAD integrations, this manual describes fundamental subjects, guidelines and proceedings.

	Note: Function scope of the CAD Integrations The function scope of the PRO.FILE CAD integrations can vary from CAD system to CAD system. Not every functionality described in this manual is available for each CAD integration – especially for 2D systems.

The chapters of these manuals deal with the following topics:

- Better organization of data with PRO.FILE

  This chapter answers basic questions: What is PRO.FILE, how is it structured, what is the central and local storage system behind it?

  - The PRO.FILE CAD integrations

  - The PRO.FILE architecture

  - The local data storage in the PRO.FILE Workcenter

- PRO.FILE parts, documents, projects and structures

  PRO.FILE is the hub for different types of data and information. The arising requirements are covered by the following PRO.FILE object types:

  - The PRO.FILE part

  - The PRO.FILE document

  - The PRO.FILE project

  - Display and manage PRO.FILE structures

- Usage: Starting point CAD design

  This chapter gives basic information on the support of CAD design by which functions.

  - Usage: finding your objects

  - New designs

  - Engineering change design

  - Modification logs

- Usage: Designing in the team (collaboration)

  When designing in a team, the urgent issues are the actuality of the data and the prevention of concurrent changes. This chapter provides the corresponding information.

  - The permissions concept

  - Check in – Check out

  - User permissions

  - Status permissions

- Usage: Create versions/revisions of CAD-relevant data

  The large topic of versions / revisions of CAD documents and bills of materials in PRO.FILE is described in this last chapter.

  - Definitions

  - The display of the version cycle in PRO.FILE

  - The versioning of CAD models with PRO.FILE

  - For the history: Versioning of bills of materials

2   Better organization of data with PRO.FILE

PRO.FILE is an intelligent solution for Product Lifecycle Management (PLM), which administrates all product and development data and the corresponding documents.

- It combines the administration of structured product data with the control of documents and the long-term archival of your technical documents.

- Furthermore, you can control the release and validity of data and documents with PRO.FILE.

- Your benefits: no multiple storage, accelerated processes and less wrong decisions caused by invalid documents.
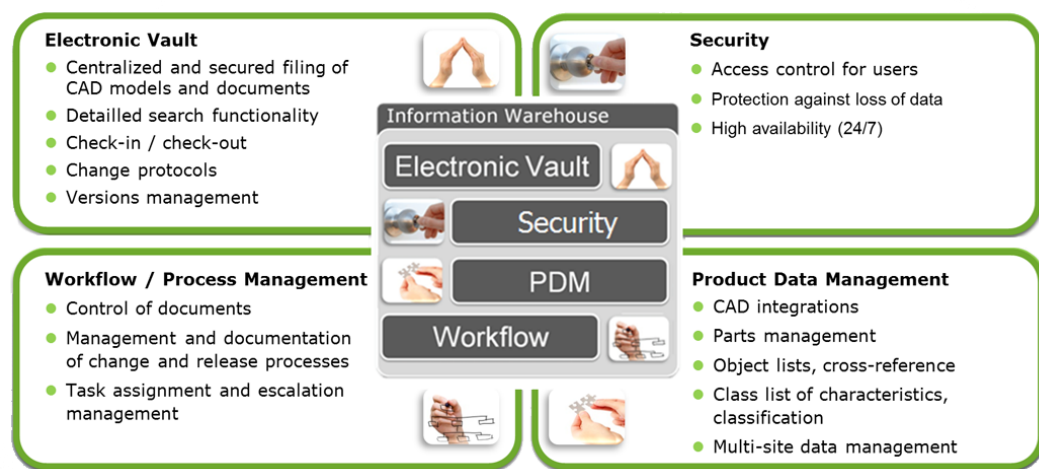
This introductory chapter deals with the special aspects of a close CAD integration with PRO.FILE. It presents the basic storage structure of PRO.FILE, the different options of file storage (encryption) and the possibilities of the Workcenter (structure and functions).

Another topic is the organization of design data with PRO.FILE as well as the benefits for the designer.

The following chapters summarize the basic information for you:

- The PRO.FILE CAD integrations

- The PRO.FILE architecture

  - The central storage system

  - Reliable and automated archival

- The local data storage in the PRO.FILE Workcenter

  - Reason and purpose of multiple work folders in the Workcenter

2.1   The PRO.FILE CAD integrations

PRO.FILE PLM supports the design activities in CAD systems and offers functions for the administration of product data and documents required in the design departments.

- PRO.FILE is an established PDM system for the administration of data and documents for the technical office.

- It works fully integrated in the CAD environment: Drawings and CAD models can be loaded from or saved to PRO.FILE directly from the CAD system.

- When assemblies are saved, PRO.FILE automatically generates bills of materials and proofs of usage. These can be included in the drawing title block along with the product data.

- Interfaces allow the transfer of product data (part master data, bills of material and CAD documents) specified during the design process to ERP systems.

PRO.FILE integrations in the form described in this manual are available for the following CAD systems:

- AutoCAD
- Catia V5
- Inventor
- Creo Elements Direct (ME 10)
- MicroStation

- Creo Parametric
- Solid Edge
- SolidWorks
- Siemens NX

Note: Function scope of the CAD Integrations

The function scope of the PRO.FILE CAD integrations can vary from CAD system to CAD system.

Not every functionality described in this manual is available for each CAD integration – especially for 2D systems.

2.2   The PRO.FILE architecture

PRO.FILE as database-supported PDM/PLM and DMS solution works with the system architecture depicted here.

PRO.FILE is the hub for the secure data exchange between CAD, CRM and ERP systems.
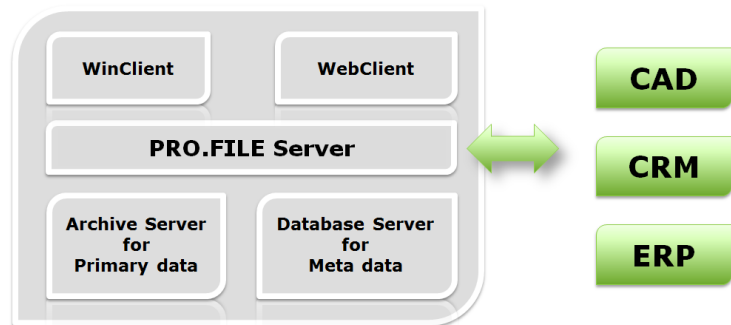
- Access to PRO.FILE is made via the PRO.FILE Windows Client or via the PRO.FILE Web Client. (Note: The connection to CAD integrations is only possible via the PRO.FILE Windows Client).

- The PRO.FILE Server administrates the accesses, checks the permissions, controls data access and data storage.

- By the synchronization of the master data from CAD, ERP, CRM with PRO.FILE, the saved information is handed over the entire ways of the value chain. Data once recorded does not need to be entered again.

2.2.1   The central storage system

The securing of data in PRO.FILE is made via a central storage system.

- All primary data is stored by the PRO.FILE Server on the Archive Server (Electronic Vault).

⇒ Primary data are all files of any given format. PRO.FILE is not limited to specific file formats.

- Metadata are stored on the database server.

⇒ Metadata is the information and describing data from CAD systems, from master records, bills of materials and from the ERP data exchange.

This means: During the saving process in PRO.FILE, data is stored as follows:

CAD data, such as	Information such as
• models	• part master data
• drawings	• bills of materials
• additional files	• proofs of usage

are saved by PRO.FILE:

as primary data on the archive server	as metadata on the database server

The connections between the primary data and the metadata is handled by the PRO.FILE Server.

This central storage system ensures the controlled maintenance of CAD data for secure data storage and efficient working in your company.

These are the benefits for the designers:

- Always up-to-date documents

- Access from all locations

- Available for everyone

- Secured from unauthorized access

See also the chapters "Re-use articles instead of re-designing them" and "Usage: Designing in the team (collaboration)".

2.2.2   Reliable and automated archival

By storing all data and documents in the central storage system of PRO.FILE, you make sure that CAD data, drawings – but also contracts, letters, e-mails and many other document types – are stored and revision-proof.

You thus make sure that important documents in your company do not get lost or become unusable.

Neutral data formats

In this context is also the automated creation of neutral data formats:

- With PRO.FILE you can automatically (via workflow) save CAD documents as copies in TIF, PDF or PDF/A format. These file can then be used, even if the original CAD application is not/no longer available.

- For the creation of these neutral data formats, the PRO.FILE Job Server is used. The PRO.FILE Job Server creates a copy of the original CAD document in TIF or PDF and administrates it along with the original.

- The creation is usually done automatically, e.g. upon release of a CAD document. It can also be triggered by other events or manually. Original and TIF/PDF copy are always managed and versioned in parallel.

Further information can be found in the PRO.FILE manual for the Format Generators.

2.3   The local data storage in the PRO.FILE Workcenter

The CAD system features require the local availability of all related data. The PRO.FILE Workcenter helps you in the administration of components loaded from PRO.FILE and stored locally. It makes the parallel work in several assemblies, projects or databases easier and helps you keep the overview.

The possibilities of the Workcenter:

- CAD data loaded from PRO.FILE is saved intermediately on the user computer in a "work folder".

- The local saving of the CAD drawings makes sure that all required parts and documents required for working with the CAD assembly are available on the user computer.

- This work folder and other local work folder can be created and managed by the user via the Workcenter.

- In the Workcenter you can specify a separate sub-folder for each project / assembly / database, display it in the Windows Explorer, lock and unlock components, view details on parts, documents and bills of materials or delete individual objects from the work folder.

The functions of the Workcenter:

1. When CAD data is opened for the first time from PRO.FILE, the CAD object and corresponding components are saved in the specified Workcenter folder.

2. When a CAD session is closed or when the CAD document is closed within the CAD session, the data remains in the Workcenter, i.e. it is not automatically deleted from this folder.

3. When this CAD data is opened again from PRO.FILE, your local status of the objects is compared with the status in PRO.FILE.

   - If the status has changed in the meantime – i.e. the CAD documents in PRO.FILE are of a more recent date – the locally stored CAD documents are overwritten (depending on the configuration without further inquiry).

   - If you have changed your CAD documents in your CAD sessions, without saving them back to PRO.FILE – i.e. the CAD documents in your Workcenter folder are of a more recent date – a message is displayed, saying that there is a more recent version in your local work folder. The opening process from PRO.FILE is then aborted.

   ⇒ If you still want to open the version from PRO.FILE and discard your local changes, you can delete the desired CAD objects from your CAD session and use the function "Delete local work folder". You can then load the desired object from PRO.FILE.

2.3.1   Reason and purpose of multiple work folders in the Workcenter

When opening CAD objects from PRO.FILE, it may happen that there are already files of the same name in your current work folder. This may happen when working with several projects, or in the context of specific version concepts.

- If you only use one work folder, you cannot load the CAD object from PRO.FILE without overwriting the existing object.

    - But if you want to compare two versions within the CAD session, you can load the second version in a different work folder.

    - Furthermore, you can use multiple work folders when working with different projects. Each project gets a separate work folder. Models from project A may thus have the same name as models from project B without causing conflicts.

> **Attention:**
>
> When working with several work folder, you have to be careful, as it may happen, that the CAD object loaded by you contains components from work folder A and work folder B.
>
> When deleting objects from a work folder, make sure that there are no objects being referenced by assemblies/drawings from other work folder. To avoid the loss of data, you should only delete objects locally that are already saved in PRO.FILE.

3   PRO.FILE parts, documents, projects and structures

In most companies, part master data, bills of materials, CAD data and technical drawings are the backbone of the product data management. Via this data and these documents, the production and later maintenance and service are controlled.

This data is always accompanied by information and files from the Office environment, by technical specifications, e-mails, photos, etc.

The administration of all this data is therefore in the focus of the PRO.FILE functions: Product data management, document management and process-oriented product lifecycle management are united in one system.

In order to effectively categorize the resulting differentiated requirements, different object types are used in PRO.FILE.

The object types:

- Parts: Parts are the basic object for product data management. Parts in PRO.FILE are digital passports describing an article, an assembly or a part, material or item.

- Documents: Whenever a file is saved in PRO.FILE, it is attached to the object type "document". PRO.FILE allows for the administration of any kind of document – the file format is of no relevance.

- Projects: Parts and documents can be assigned to projects. This is to summarize information and data in a project-specific context.

  Example: A project "steam engine" lists all parts and documents or sub-projects belonging to this project.

Note: Other object types in PRO.FILE

The other object types of PRO.FILE – tasks, processes, transmittals and folders – are not part of this manual.
Information on these objects can be found in the manual "Operation PRO.FILE for Advanced" and "Operation PRO.FILE processes and tasks".

The characteristics:

Each object stored in PROFILE, i.e. parts, assemblies, documents or projects, is described by a set of characteristics.

The characteristics:

- enable the systematic description of objects
- are the basis for easy finding
- are recorded automatedly from the CAD system where possible
- have a unique ID
- can be handed over by PRO.FILE to the drawing legend as well as to ERP systems.

The characteristics are saved in the part master record, document master record or project master record in PRO.FILE.

Note: Characteristics are a matter of configuration

The structure and availability of characteristics for the respective object types can be configured by the administrator in the PRO.FILE Management Console.

This chapter is to give you an overview of these PRO.FILE objects and their purpose:

- The PRO.FILE part   ⇒ Benefits of the part management
                            ⇒ The classification of parts via item classes

- The PRO.FILE document   ⇒ The PRO.FILE document – not only for CAD data
                            ⇒ What is a document type in PRO.FILE?
                            ⇒ Visualization in PRO.FILE

- The PRO.FILE project   ⇒ Automatic allocation of data records to a project
                            ⇒ Project-specific permissions via the project role concept

3.1 The PRO.FILE part

The PRO.FILE object "part" consists of a digital data sheet (metadata) for the description of a product / an article / an item / a material / an assembly component / a design part / an assembly.

A "part" in PRO.FILE is described and classified by entering characteristics into the defined input form.

The article is thus identified, as if by a "passport".

A "part" is identifiable by a unique ID and is described by data fields, attributes or by item classes.

The "part" is a linking element in the entire company process: Via the part, the unique "part master data" is handed over in the chain, e.g. from design, production, warehousing, spare parts, sales and customer service.

The PRO.FILE object "part" only consists of this data record of the part description.

- A PRO.FILE "part" can either be a single part, but can also describe a complex assembly or sub-assembly.

- The structure of PRO.FILE parts forms the bill of materials.

- It is possible to attach several PRO.FILE objects "document" to a PRO.FILE part, in order to enhance the description, e.g. by CAD data, drawings, specifications, photos, etc.

- PRO.FILE parts can also exist without the assignment of PRO.FILE documents, e.g. if a standard part "ISO 4014 – M10 x 60 - 8.8" does not require a drawing.

⇒ The PRO.FILE object "part" is therefore only a record for the description of an article.

⇒ The PRO.FILE object "part" does not describe a file saved in PRO.FILE.

Note:

A PRO.FILE part is not a physical file, but only the description of an article by metadata. A physical file, on the other hand, e.g. a *.prt part is always treated in PRO.FILE as object type "document".

3.1.1   The classification of parts via item classes

For the classification of parts, the PRO.FILE object part offers the powerful feature of item classes.

An item class offers further context-sensitive fields for the detailed description of parts (the description of a screw, for example, requires other fields than the description of a spring). These fields may comprise dimensions, forms, materials, electrical characteristics, physical or chemical characteristics, etc.

What is an item class?

- An item class is the summary of characteristics of a group similar objects.

- The item class facilitates the differentiation of similar but not identical product groups.

- Item classes can describe simple part and assembly families as well as complex, hierarchical product groups.

- PRO.FILE item classes are also hierarchically structured. Information can be queried in a targeted and structured way.

- Item classes can be linked to the parameters of part families in CAD systems.

At a glance:

The following image shows the usage of item classes:

- ①: By the selection "Machinery element" in the field "Item class", a further query is displayed: "Which product group does the machinery element belong to?"

- ②: By replying "spring", the next query is displayed: "What spring category does this spring belong to?"

- ③: By replying "Torsion spring" you can now enter the characteristics of a torsion spring.

Transferred to the example "The PRO.FILE part is the passport of an article " the usage of item classes means:

An article "ball bearing" is described by other characteristics than an article "lever".

- For the article "ball bearing" the corresponding item class therefore offers information such as "inner diameter", "outer diameter" and "width".

- For the article "lever", on the other hand, the corresponding item class offers information such as "color", "D1", "D2" and "B3".

Note:

PRO.FILE item classes can be configured by your administrator in the PRO.FILE Management Console and adjusted to your company-specific requirements.

3.1.2   Benefits of the part management

When an article/assembly is recorded in PRO.FILE as "part", a unique part ID and description is assigned.
This and other information are pure metadata, which is stored in the PRO.FILE database tables.

The benefits:

- Via the part master record, an article can be identified all over the entire life cycle – from design to recycling.

- A part can be described by a variety of documents – CAD data, different drawing views, specifications, simulation data, check protocols and e-mails.
  Via the PRO.FILE part management "n" describing documents can be linked to "1" part master record – the part master record always remains unique.

- If all elements of an assembly are assigned to a PRO.FILE "part", the bill of materials can be directly derived from the assembly structure. These bills of materials are administrated in PRO.FILE via the part master records.

- The part management in PRO.FILE enables the editing of bills of materials without CAD. Bills of materials can already be administrated in PRO.FILE and be enhanced, e.g. by raw or auxiliary materials (paint, glue, lubricants, ...).

- The part management via the part master data allows for direct linking to ERP systems. Data does not need to be entered several times within the process chain, but is handed over from system to system. In case of a bi-directional ERP integration, it is also possible to enter the part master data in the ERP system and to transfer it to PRO.FILE – in PRO.FILE only the assignment to the already existing data is then made.

3.2   The PRO.FILE document
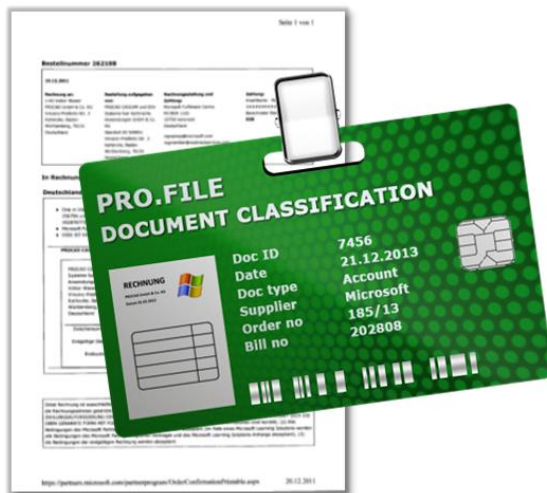
PRO.FILE is capable of storing files of any format on the archive server.

When a file is stored, a unique identification is made via a document description.

Via this PRO.FILE document description:

- documents are quickly found

- classification is made

- documents are identified

Each file saved in PRO.FILE is thus, per definition, of the object type "document".

What does a PRO.FILE document consist of?

The PRO.FILE document consists of two parts:

- Of the PRO.FILE document description as "digital passport" of the document. This metadata is saved on the database server.

- Of the document file. This file is stored on the archive server.

Note: No document file without document description

A document description can be saved in PRO.FILE without a document file.

But it is not possible to save a document file without a document description:

Without "digital passport" no one gets into PRO.FILE!

In the storage path of the archive server, you therefore have the saved files, such as:

- CAD models
- Word files
- e-mails

3.2.1 The PRO.FILE document – not only for CAD data

PRO.FILE takes over the administration of CAD documents via the CAD integrations. Made-to-measure functions make the work of designers with CAD documents easier.

However:

- 75% of documents created in the design process are "normal" documents.

This is why PRO.FILE works with "document types": These document types allow the structuring of a targeted storage of document in PRO.FILE.

> **Note: Configuration of document types**
>
> The structure and availability of document types can be configured by the administrator in the PRO.FILE Management Console.

PRO.FILE handles any type of document

- 3D documents
- 2D documents
- Office documents
- Neutral formats, such as PDF, TIF, etc.
- Specifications
- Minutes of meetings
- Reports
- Mail and e-mail correspondence
- Company-internal and public request + approvals
- Offers, orders, order confirmations, invoices

3.2.2 What is a document type in PRO.FILE?

Different types of documents (drawings, reports, invoices, etc.) require different fields for entering the relevant information.

- The field "invoice date" is of relevance to invoices, but rather not to CAD drawings.

But in order to not all input/output forms having to comprise all fields, but only the fields relevant to the type of document at hand, such information can be classified and sorted via specific document types in PRO.FILE.

For this reason, specific document types can be defined for the storing of different documents.

- Via these document types, the classification of the document attributes is made. The document type also controls the handling of the corresponding document description in PRO.FILE.

- Each of these document types can be assigned separate document characteristics.

- In addition to the fields of the fixed document description (which is identical for all documents) the user can access the fields of the specific document characteristics by selecting the document type.
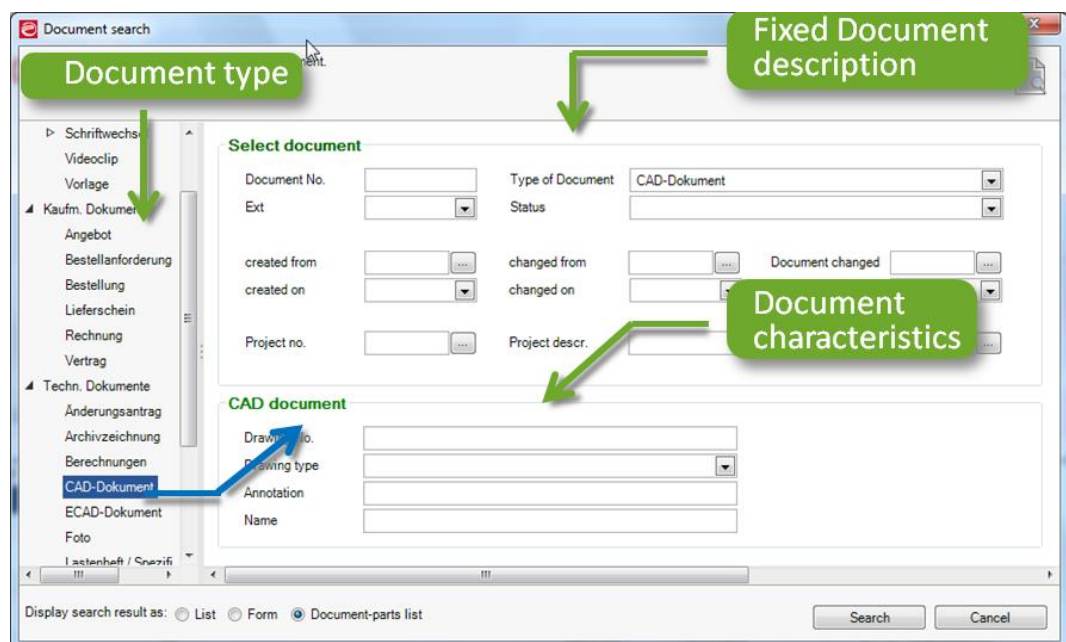
The image above shows: By selecting the document type "CAD document" in the fixed document description, the corresponding document characteristics can be accessed:

Note: Consequences for searches

In order to search for specific characteristics, e.g. of "CAD documents", the document type has to be selected. The selection of the document type makes the fields of the corresponding document characteristics available.

3.2.3   Visualization in PRO.FILE

Documents saved in PRO.FILE can be displayed via a unified view.
The embedding of a viewer, e.g. "AutoVue" in the tabs of PRO.FILE allows the
visualization of up to 450 document formats.

- Word,
  Outlook,
  Excel,
  PDF, graphic
  formats, ...

- PDF/A, 2D
  and
  3D CAD
  models, JT, ...

This display offers
the designer
additional
functions in the
preview without
having to open the
document from
PRO.FILE.
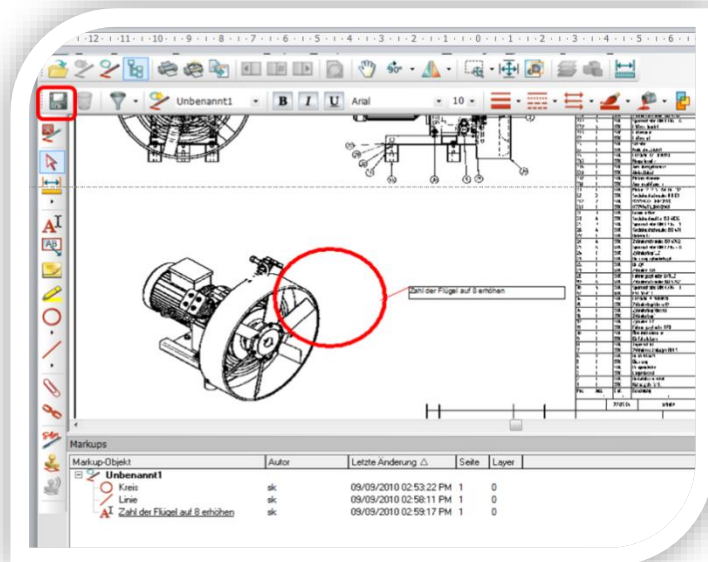
- Zoom, rotate, redline, markup, document comparison

3.3 The PRO.FILE project

To meet company-specific requirements for project-oriented data management, PRO.FILE offers the object type "project".

The object type "project" allows creating project containers for a project-related structured data management and allocating parts, documents and tasks directly to these projects.

Projects are basically used to summarize information and data, parts and document descriptions in a project-specific way.
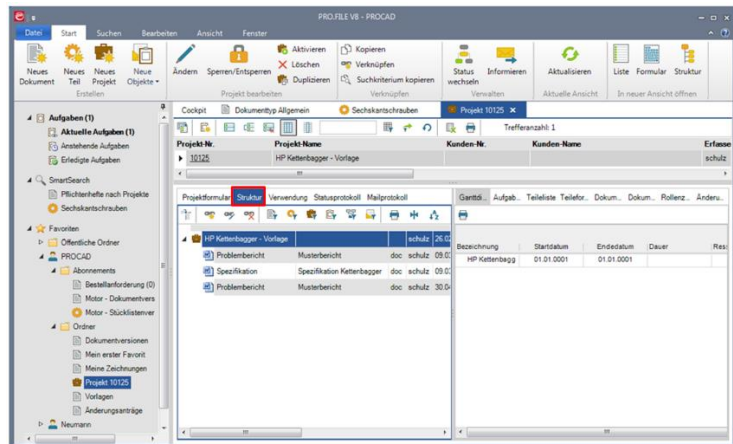
Such data may consist of design data as well as of offers, e-mails, photos, specifications, minutes of meetings, etc.

This way, all data objects related to each other can be:

- attached to a specific project

- accessed, managed and edited in a targeted way via this project.

Projects can be structured hierarchically. It is possible to summarize several sub-projects in one main project.

Working with projects can take place at two levels:

- On the first level, projects can be used as pure "folders" for the structuring of data.

- On the second level, it is possible to use project-specific access permissions on documents, parts, sub-projects, etc. of a project. The pre-defined roles are then assigned to the users involved in the projects and these users get status-dependent permissions via these roles.
  See chapter "Project-specific permissions via the project role concept".

3.3.1 Automatic allocation of data records to a project

If a project is activated in PRO.FILE, all newly created documents and parts are automatically allocated to this project.

These documents and parts are then on the first sub-level of the project structure; manual allocation is not required.

- The activation of a project is made via the "Edit" menu in PRO.FILE.

- An activated project is displayed in the title bar of the PRO.FILE window.

- The deactivation of a project is made via the "File" menu in PRO.FILE.

3.3.2 Project-specific permissions via the project role concept

The usage of project administration in PRO.FILE can be enhanced on the second level by the project role concept.

Project roles are used for the handling of projects with project-specific assignment of permissions.

- Within a project, the selected users can be assigned a project role.

- This project role describes the actions, the user is allowed to perform in the context of the corresponding project (regardless of the user's permissions outside the project or in other projects).

The project roles allow assigning the quick and easy assignment of permissions to the involved employees.

- Via the project role concept, permissions are defined (independent from actual users) for specific project roles.

- In the second step, the defined project roles are assigned to users involved in the project, in order to quickly and easily assign project-specific permissions.

When competencies or involved users change, the resulting permission changes are managed by simply re-assigning the corresponding project role. This way, users in different projects can also assume different project roles with different permissions.

Detailed information on the working with PRO.FILE projects can be found in the manual "Operation PRO.FILE projects and roles".

3.4   Display and manage PRO.FILE structures

One of the important features of PRO.FILE is the administration and display of product structures.

The product structure:

- Shows the graphic display of a product structure in all its assemblies, sub-assemblies and single parts.

- Contains all CAD models, drawings, change requests, etc. that may belong to an assembly or part.

Structures can be displayed in PRO.FILE in dependent tabs or in the structure browser:

Roboter SW			PROCAD	20.11.2013	in Bearbeitung	204117		
Basis			PROCAD	20.11.2013	in Bearbeitung	204049		
Basis			PROCAD	20.11.2013	in Bearbeitung	204050		
CAD-Dokument	Basis	SLDPRT	PROCAD	20.11.2013	in Bearbeitung	106752	3D Model	Basis_00106752.SLDPRT
CAD-Dokument	Basis_00106751	SLDASM	PROCAD	20.11.2013	in Bearbeitung	106751	3D Shape Element	Basis_00106751.SLDASM
Flansch groß			PROCAD	20.11.2013	in Bearbeitung	204051		
CAD-Dokument	Flansch groß	SLDPRT	PROCAD	20.11.2013	in Bearbeitung	106753	3D Model	Flansch groß_00106753.SLDPF
CAD-Dokument	Basis_00106751	SLDASM	PROCAD	20.11.2013	in Bearbeitung	106751	3D Shape Element	Basis_00106751.SLDASM
Motor groß			PROCAD	20.11.2013	in Bearbeitung	204052		
Gehäuse groß			PROCAD	20.11.2013	in Bearbeitung	204053		
CAD-Dokument	Gehäuse groß	SLDPRT	PROCAD	20.11.2013	in Bearbeitung	106755	3D Model	Gehäuse groß_00106755.SLDF
Rotor groß			PROCAD	20.11.2013	in Bearbeitung	204054		
Rotor groß			PROCAD	20.11.2013	in Bearbeitung	204055		
Außenverzahnt			PROCAD	20.11.2013	in Bearbeitung	204056		

How are structures built in PRO.FILE?

Product structures are taken directly from the CAD system.

- Structures and usage lists are automatically generated when CAD data is saved in PRO.FILE.

- The bills of materials is also derived from the structure of the saved PRO.FILE parts - the structure of the assembly thus matches the bill of materials.

- PRO.FILE generates bills of materials in different for-mats: Structure bills of materials, quantity bills of materials, single-level bills of materials, etc.

Structures can be created or adjusted manually:

- Links between projects, documents and parts can be established via the functions "Copy" => "Create link" as well as via drag&drop.

- Links between PRO.FILE parts forma bill of materials. For the handling of bills of materials, the bill of materials editor is available.

3.4.1 Displaying the structures

To display the product structures in PRO.FILE, different display types can be used:

The browser display shows precise references between:

- PRO.FILE parts and documents
- PRO.FILE parts and projects
- PRO.FILE documents, e.g. assemblies, parts and drawings
- PRO.FILE projects (project and sub-projects)

Note: Structures between parts are handled as bills of materials in PRO.FILE.

The tab "Where used" shows:

- where and in which other structures or assemblies the selected document is used.
- where and in which other document and part structures the selected part is used.
- When documents or parts, that are used in assemblies, are changed, PRO.FILE notifies you of all existing usages and thus protects you from unintended changes in design results of others.

The display of the usage list of a PRO.FILE object can be general (shows all superior parts, documents and projects) or object-specific:

- The part usage filters the display for superior part records.
- The document usage filters the display for superior document descriptions.

The bill of materials shows:

- Precise references between part records in PRO.FILE
- Specific bill of materials information, according to the type:
  - Structure bill of materials
  - Single-level bill of materials
  - Quantity bill of materials
  - Design bill of materials

Level 1	Level 2	Level 3	Article no	Designation	Status
2			00924-202040_SW	Oil pump	❌
	2.1		00924-202046	Cover	✅
	2.2		00924-202041	Drive	✅
		2.2.1	00924-071044	Wave	❌
		2.2.2	00924-071052	Stift	✅
	2.3		00924-202047	Case	✅

3.5   Bills of materials

The bill of materials is a central and basic element of product data management. Bills of materials in PRO.FILE results from the linking of part records: A structure of PRO.FILE parts is a bill of materials.

When CAD data is saved to PRO.FILE, the bill of materials can be derived directly from the CAD system. This is where the bill of materials comes to existence, so this is where it is derived from.

The management of bills of materials in PRO.FILE allows for the comprehensive information flow to follow-up systems. By the administration of pure metadata, this information can be directly handed over from PRO.FILE to an ERP system.

The integration of PRO.FILE in ERP systems allows for synchronization of article master data and bills of materials as well as of project structures.

In the opposite direction, it is possible to access ERP data from PRO.FILE.

3.5.1   The automatic creation of bills of materials

When CAD models / drawings are saved to PRO.FILE, product structures are taken directly from the CAD system.

By the allocation of the various elements of the product structure to a PRO.FILE part bills of materials and usage lists are automatically generated. The further update is made from within the CAD system.

PRO.FILE generates bills of materials in different formats: Structure bills of materials, quantity bills of materials, single-level bills of materials, etc.

## 3.5.2   Editing bills of materials in PRO.FILE

Bills of materials are derived automatically from the CAD system to PRO.FILE. The structure of the bill of materials corresponds to the structure of the assembly in the CAD system.

- The structure of the bill of materials can only be changed via the CAD system for these positions.

In PRO.FILE, it is possible to post-process and enhance this bill of materials via the Bill of materials editor.

It is there possible to enhance e.g. the following positions:

- Auxiliary materials
- Testing agents
- Spare parts

- Mounting accessories
- Packaging material
- etc.
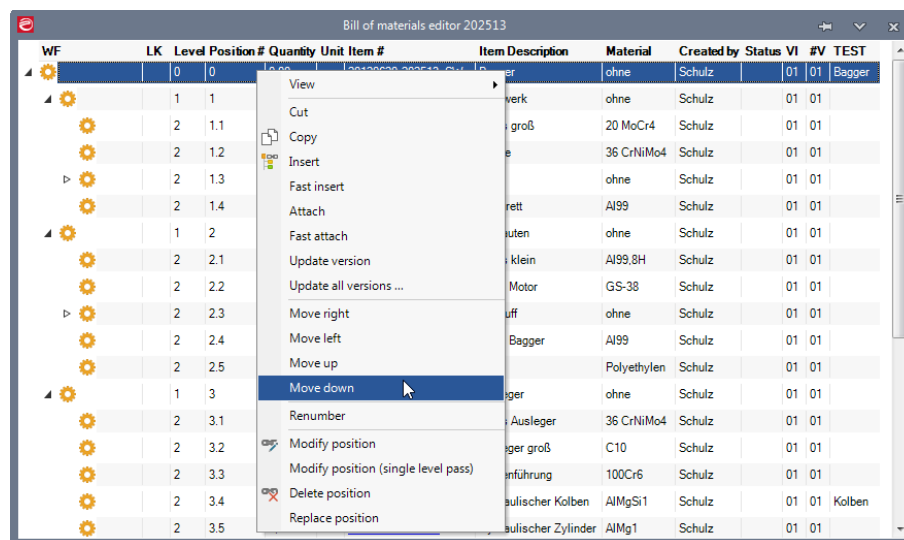
Bill of materials   Via the bill of materials editor in PRO.FILE it is possible:

- To add or move further BOM positions in the structure - even via drag&drop.
- To renumber BOM positions.
- To manage and update BOM descriptions.
- To change, delete or replace BOM positions.
- To update versions.

Detailed information on the bill of materials editor can be found in the manual "Operation PRO.FILE structures and bills of materials".

3.6   Designing with phantom assemblies

The situation: You need the model of a purchase part (e.g. ball bearing), which you use for the design in the CAD system:

- Because you want to check the placement conditions.

- Because the mounting holes in your design depend on the purchase part.

For this purpose, your supplier may give you a simplified assembly containing all required information (size, drillings, mountings, ...).

The components this purchase part consist of are not relevant for the bill of materials – the model is treated like a single part.

For this purpose, PRO.FILE offers the functionality "phantom assembly".

Definition

- Phantom assembly: A phantom assembly acts in PRO.FILE like a single part. The phantom assembly consists of phantom parts but is treated like a single PRO.FILE part.
  If you use a phantom assembly in a superior assembly, the phantom assembly itself will appear in the bill of materials, but not the single parts (phantom parts) of this phantom assembly.

- Phantom part: Parts within a phantom assembly. Phantom parts are virtually invisible, they do not appear in the bill of materials. Phantom parts cannot be used independently.

The function   The administration of phantom parts is available via the PRO.FILE integrations for assemblies and drawings.

Assemblies can summarized under one single part master record in PRO.FILE with all assembly parts contained therein. Via the function for the saving of phantom assemblies in the menu of the PRO.FILE integration, all objects contained in an assembly are saved under the same part master record.

All components yet unknown in PRO.FILE are saved as phantom parts.

- The elements of the assembly can then no longer be loaded individually from PRO.FILE.

- The phantom assembly is thus treated like a single part in PRO.FILE, even though it consists of several parts. The objects contained in this assembly are listed as phantom parts and cannot be explicitly loaded from PRO.FILE.

The following applies for phantom parts:

- Only CAD documents that have not yet been saved to PRO.FILE can be saved as phantoms.

- PRO.FILE treats this assembly like a single part.

- A phantom assembly can have several levels.

- The changing of phantom assembly and part is possible.

- Phantom parts may not be used in other phantom objects (so-called external phantoms).

- Phantom assemblies have only one position number in the bill of materials.

- Changing of these models can only be made within the context.

- A version/revision creation of the phantom (assembly/part) is possible (and also generally made within the context).

Usage of phantom parts from a phantom assembly

If you want to use an element from a phantom assembly in other designs, you have to "cut" this element out of the phantom assembly.

Since you do not open the phantom element directly from PRO.FILE, the proceeding is as follows:

Open the phantom assembly from PRO.FILE. The elements of the phantom assembly are copied to your work folder. From here you can open the required phantom element in your CAD system. If the phantom assembly is still opened in the CAD system, you can open the phantom part directly from within the phantom assembly.

If you now want to save the phantom element anew via "Save", you get the message that the database relation has to be dissolved first.

	Attention: Phantom parts
!	Phantom parts CANNOT be referenced. In order to use a phantom part independently, you have to cut it out of the phantom assembly.

4 Usage: Starting point CAD design

This chapter deals with the following topics:

- Usage: finding your objects
- New designs
- Engineering change design
- Modification logs

4.1 Usage: finding your objects

The various search functions of PRO.FILE support the designer in finding repeated parts and similar objects for re-use.

A prerequisite for this is that the documents and parts are properly classified when saved to PRO.FILE. This classification is the backbone for later searches.
The aim is to make finding an existing part faster than re-designing it.

PRO.FILE offers a wide range of functions for targeted searches:

Searching via…
- Characteristics
- Full-text search
- Relations to tasks, processes and projects
- Relations between parts, assemblies and documents
- Favorites
- Preview in results list

PRO.FILE allows the quick and secure finding of parts, assemblies and documents. This is to prevent unnecessary re-designing and optimize the re-use of parts.

The systematic storage of CAD models, documents and project structures is the basis of finding them again.

- PRO.FILE offers different characteristics searches for parts, assemblies, documents and projects.
- For a 'quick glance' at CAD models and drawings the preview functions and thumbnails can be used.
- The relation between parts and documents as well as contents and structures in digital folders can be displayed via browser structures and lists.

Always the currently valid version	The joint administration of product data in PRO.FILE makes sure that the currently valid version of materials is always available for direct access by the user.

- Production errors and defective goods caused by invalid or outdated documents are avoided.
- PRO.FILE supports the requirements of DIN EN ISO 9000 ff.

The benefits of central storage in PRO.FILE

- Comprehensive display
- Information on editing status
- Quick finding and re-use of design data

Re-use articles instead of re-designing them

The big advantage of PRO.FILE in the CAD design is the simple possibility to find and re-use existing articles instead of re-designing them.

The following images depict the benefits of the usage of a PLM system like PRO.FILE.

Creation, storage, drawing administration	250 €
Creation article no. in ERP	+ 250 €
Supplier data, ordering, goods reception, storage	+ 500 €
Administration costs for one part	**1,000 €**

Without PLM

50 % of the parts are re-designed completely

30 % of the parts are re-used

20 % of the parts are changed slightly

With PLM

30 % of the parts are re-designed completely

40 % of the parts are re-used

30 % of the parts are changed slightly

Sources: VDI, CIMData USA

4.2   New designs

When designing new assemblies and parts and drawings, PRO.FILE supports you with the Checkin wizard.

The Checkin wizard recognizes, which data is not yet saved in PRO.FILE.

The proceeding is as follows:

- Step 1: Create a part master record in PRO.FILE

  By default in PRO.FILE, each CAD document is allocated to a part master record. The part master record consists of attributes and is used e.g. for the creation of bills of materials, for the output of data on the drawing title block or for the transfer to ERP systems.

  Via the Checkin wizard, all required characteristics can now be entered to categorize the article.

  Via the characteristics entered here, the part master record can later be found in PRO.FILE.

- Step 2: Create a document description in PRO.FILE (also automated)

  All files saved in PRO.FILE are saved with the object type "document".

  To save the file in PRO.FILE, the corresponding document description has to be entered. The document is thus described and classified and made available for later usage.

  Note: The document description can also be saved and filled with information from the CAD system automatically. The Checkin wizard of this step 2 is then not displayed.

- Step 3: Allocate the created data to a PRO.FILE project

  In this step, the CAD data just created can be allocated to a specific PRO.FILE project. (This setting is optional).

These steps are repeated for all new assemblies, parts and drawings. PRO.FILE then automatically builds the structure of the CAD data.

When assemblies are saved, PRO.FILE automatically generates bills of materials and proofs of usage. These can be included in the drawing title block along with the product data.

The newly created documents and parts are saved in the configured start status of the PRO.FILE status administration.

4.3   Engineering change design

Apart from the global organization of projects, PRO.FILE offers concepts for engineering change management in the CAD-related design. PRO.FILE supports the 80/20 rule, which is often used in plant engineering and according to which 80% of a customer-specific product of a machinery design are always the same and only 20% have to be newly designed.

Depending on the status and the permission of the user, assemblies or elements of an assembly can be loaded from PRO.FILE and be edited. PRO.FILE monitors these changes and offers them for saving back to PRO.FILE at the end of the editing process.

Depending on the permissions concept, the user can either overwrite or version/revision the previous data.

For the variant and engineering change design, PRO.FILE offers the following functions:

- Managed Copy
- Managed Rename – Renaming in the structure

4.3.1   Managed Copy

Managed Copy helps the designer to only edit the components that need editing.

- Entire machines can be cloned, including all referenced data and drawings.
- Assemblies and parts that are to remain in the new structure are taken over. New components are newly designed.
- New numbers of the cloned machine and the new status in the dependent drawings are updated automatically.
- For all components not selected in Managed Copy, only the references are copied. Existing references thus remain intact.

Managed Copy allows the targeted saving of single modified parts and assemblies as new copy. It is up to the user which elements within an assembly structure are saved as new copy.

Simultaneously the title of the copied components can be adjusted. Finally a bill of materials is derived for the "cloned" assembly.

Managed Copy is widely used in variant design and is there used in the context of revisions.

4.3.2   Managed Rename – Renaming in the structure

With the function "Managed Rename" it is possible to change the file names of CAD models already saved in PRO.FILE, while considering and updating the references.

This function is used e.g. for the following requirements:

- The file name is to be descriptive but the required information for this file name are not yet available (e.g. item no.).

- For performance reasons, the CAD browser is to display the file name. It should therefore also contain PRO.FILE metadata.

Note: Only available without active CAD document

The function "Managed Rename" is only available, if the CAD document is not opened in the CAD system. Only then can the system make sure that the data to be renamed is processed without conflicts.

When the function "Managed Rename" is used, CAD data for renaming is selected by the user in PRO.FILE and saved by the integration in the local work folder. You then have the possibility to adjust the file names in the CAD structure in an overview window.

After confirmation of the window "Managed Rename", the PRO.FILE integration writes the changed file names back to PRO.FILE and updates the document references.

Please note the following for "Managed Rename":

- The renaming can only be made within the selected assembly structure.

- The components to be renamed must not be used in other structures.

- There must not exist any versions of the part to be renamed, since all versions must have the same file name by definition.

- Instances may not be renamed.

- The models to be renamed, as well as the assemblies/drawings containing these models, must be savable in PRO.FILE (permissions).

- CAD documents referenced in an assembly locked by a different user cannot be renamed.

Attention: "Managed Rename" is a modification

The function "Managed Rename" has the same effect on the affected data than a modification: The components renamed in PRO.FILE correspond to the newest editing status. All local statuses of these components are thus outdated and can no longer be saved back to PRO.FILE

4.4   Modification logs

PRO.FILE controls and documents the changes to parts, assemblies and documents. Change and release processes are supported by workflows. For the documentation of the change process, the modification log and a two-level version/revision management are available.

- Time of the change

- User who has made the change

- Reason of the change

Modification log

Doc type	Preview	Article no	Changed by	Date	Remark
CAD document		10098	Smith	05/21/13	Hole diameter from 17 mm to 20 mm
CAD document		10098	Morgan	03/10/12	Bevel from 45º to 35º
CAD document		10098	Barnes	07/08/11	Additional drilling

5　　Usage: Designing in the team (collaboration)

PRO.FILE coordinates the change and release of CAD models, drawings and documents. CAD data, such as models, drawings, bills of materials and usage lists are saved in the electronic vault of PRO.FILE. This is to ensure access to valid and up-to-date materials from all locations, the availability for all authorized PRO.FILE users and protection from unauthorized access.

For the design process in the team, the following additional points apply:

- All relevant data is to be available for the designer (collaboration).

- At the same time, concurrent or unauthorized changes have to be avoided, especially if several designers are working at the same assembly at the same time.

- It has to be made sure that each employee – also at different locations – knows which CAD model / which drawing is current and valid.

With the Checkin-Checkout procedure, PRO.FILE makes sure that CAD models and documents are only edited by one person at a time. This is to make sure that changes are not overwritten by other colleagues.

Each change to a document or CAD model is recorded in a modification log.

The intelligent PRO.FILE change and release management supports you via workflows that document each change precisely. At the same time, it is made sure that the data in your company remain up to date.

5.1　　The permissions concept

The PRO.FILE permissions concept allows the controlled and user-defined access to data and documents. Via this permissions concept, unauthorized or conflicting access resulting in the disturbing of company processes is prevented.

It is made sure that only authorized users access or even edit the data. The permission concept thus offers protection from unauthorized access, data loss and accidental deleting.

The permissions concept is based on three pillars:

- Check in – Check out: Locking of documents that are in the editing process to protect from concurrent changes.

- User permissions: The permissions of users are controlled by function access rights specifying which users or groups may perform which action.

- Status permissions: Via the PRO.FILE status network you can specify at what stages of the product lifecycle PRO.FILE objects may be edited or deleted by specific users or groups.

5.2 Check in – Check out

The Check in – Check out functionality protects the CAD data from concurrent changes. A CAD document may at a specific point in time…

- only be edited by one person,
- but viewed by any desired number of persons.
- PRO.FILE then shows who is editing the document.

If a user wants to edit a CAD object,

- the object has to be locked at the beginning of the editing process to avoid concurrent changes.
- This is the only way to make sure that several users are not making simultaneously changes to the same drawing.
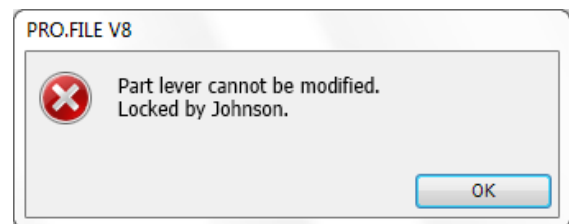
- To make an object available again once the editing process is finished, the end of the editing process has to be signaled to the database by unlocking the document after saving the file.

The PRO.FILE commands "Lock"/"Unlock", "Save" and "Display" offer the user all functions required for the controlled editing of the CAD objects.

In analogy to the function "Lock" you can remove the lock flag for documents you have locked by using the function "Unlock".

Unlock all your objects after having saved your changes to PRO.FILE, so that other colleagues (provided they are authorized) can also make their changes to the design.

Note:

You can only unlock documents you have locked yourself. The right to unlock other documents is reserved for administrators.

5.3 User permissions

The second level of the permissions concept of PRO.FILE are the permissions assigned to PRO.FILE users or user groups.

Via the PRO.FILE Management Console, you can specify, which users are to be permitted to use which functions.

For this, as displayed in the sample image - function access rights are activated or de-activated for each user/user group. The displayed user, for example, ma not creation versions of bills of materials.
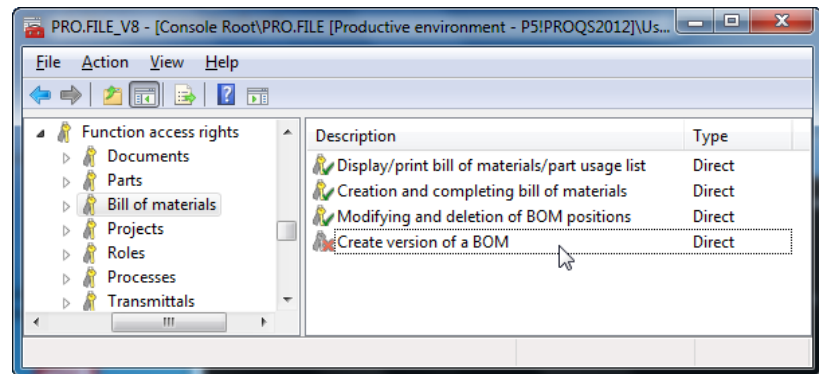
These permissions are assigned as a basis. The fact that a user has a function access right does not necessarily mean that he/she can use the function at every time for every object. An example:

- A user groups has the basic right to delete bill of materials positions.

- Even if the user group has this right, this should not be the case for "released" documents.

This is where the third pillar of the permissions concept becomes relevant.

5.4   Status permissions

The concept of status administration

PRO.FILE the organization of company processes can be described by different statuses. A status describes the area of competence. A part, document or project can thus be moved within the defined status chain in a defined order or sequence to a different status.

The PRO.FILE status administration thus makes sure that a part, document or project due for approval or editing reaches the correct employees and that these users then have the required permissions.

An example for different statuses describing the typical lifecycle of a part from design to production (or archival) is shown in the following image.
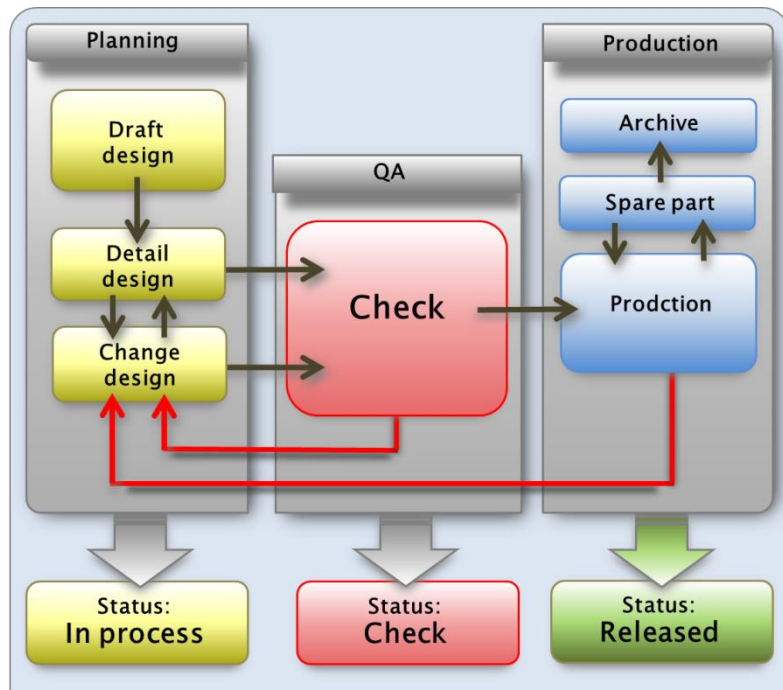
The PRO.FILE status management contains the following points:

• Modeling of editing processes for parts, document and projects

• Release processes

• Integration of e-mail

• User and access rights

Via the usage of the user and access rights it is made sure that only specific users and groups have the permission to perform a status change, and also the permissions for viewing, editing, etc. for documents and parts are controlled for each respective status.

The functionality   With the PRO.FILE status administration you can describe the company-wide organization and process structures. Via the status administration, the following access aspects are controlled:

- who   (user/group/role) has access

- when   (status)

- how   (allowed PRO.FILE functions)

- to what   (view and write permissions for PRO.FILE objects).

PRO.FILE allows the administrator to configure this status administration directly via the PRO.FILE Management Console.

The status administration uses two basic elements:

- Status and   • Status change

The following picture shows a simple example of a status system:

- Status:
  Each document, part or project is always in a specific status in PRO.FILE. For each status it is specified which users/groups can use which functions in this status. These rights assigned to the users/groups within the status thus control the access permissions to the document part or project in this status.

- Status change:
  In order for an object to make its way from status to status, the different statuses are linked by status changes. Using the status changes, objects can be transferred in another status.
  For a status change, a user also requires the corresponding permission. It is thus clearly specified which users/groups can make the status change.
  Status changes are always precisely assigned to a specific start status and a specific target status. From the image below you can see that a document can be moved from status "In processing" to "Check" but not directly from "In processing" to "Release".

At the moment the status change from start status to target status is made, the permissions change accordingly:

- Before the status change, the permissions of the start status apply.

- After the status change, the permissions of the target status apply.

Status changes are always "one-way roads", not bi-directions: If you want to be able to go from status A to status B and from status B to status A, you need two status changes.

- Start status:
  If we imagine a system of status and status changes, the "way" of parts, documents projects through this system needs a starting point.

  This is why a document, part or project always has a defined start status upon creation:

  - By default, this is the start status of the user – each user has a start status assigned in the user administration. It therefore depends on the user creating the document, in which status the new object is created.
    An example: Designer A creates a document in start status "Design in processing". QA member B creates document in start status "QA creation".

  - For specific document types you can specify separate start statuses, which are then preferred to the start status of the user. (The start status of a document type always outranks the start status of a user).
    An example: Designer A creates a drawing (document type "drawing") in start status "Design in processing". If the same user creates a check report (document type "check report"), it may start in the status "QA creation".

5.4.1   What does a status control?

A status describes a specific stage within the status chain. The various statuses are linked via status changes. This allows to describe specific organization processes in PRO.FILE.

The most simple example for such a process is: "In processing" => "Check" => "Release".
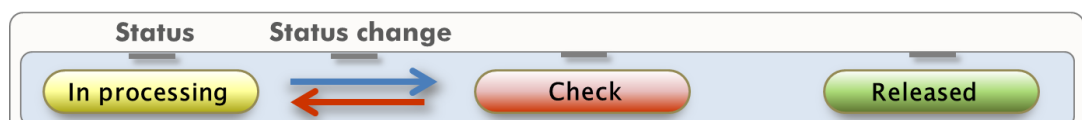
To put it more simple: A document, part or project goes from the designer's desk to the validator's desk and then to production.

For each status the status-dependent permissions are assigned: Which user may do what in which status?

- The designer may view and edit a document description and file in the status "In processing". (Maybe even delete? Or is this only allowed to the Head of design?)

- If the document is "Released" (and maybe even already being produced), it must be made sure that the designer can no longer make changes to this document (though he probably may still be allowed to view it, like the members of the production team).
  The validator too may no longer make changes to the document in this status.
  For reasons of product liability, these documents have to be archived and may no longer be changed.

- For members of the production team it would be best if they could only see documents in the status "Released", but not documents in other statuses.

The search for documents in other statuses would then be unsuccessful because they do not have the permission to see documents in other statuses than "Released". This would prevent drawings still in the design process, and therefore not yet "Released" land in the production.

Via the PRO.FILE Management Console, the administrator can specify, which users are to be permitted to use which functions in the different statuses.

For the requirements in the example above, the following settings would be the result:

Status	User	may create	may view	may edit	may delete
In processing	Designer	✓	✓	✓	✓
	Validator	✗	✓	✗	✗
	Production	✗	✗	✗	✗
Check	Designer	✗	✓	✗	✗
	Validator	✗	✓	✓	✗
	Production	✗	✗	✗	✗
Released	Designer	✗	✓	✗	✗
	Validator	✗	✓	✗	✗
	Production	✗	✓	✗	✗

Note: The PRO.FILE status administration allows for the configuration of far more rights than the rights mentioned in this example.

5.4.2   Status change top-down

If an assembly or other structure of linked parts, documents or projects is to be moved to a different status, it may be tedious to move each object in the structure separately.

For this requirement you can use the function "Status change – Top-down". All parts, documents and project linked underneath the selected object are then moved to the desired status.

When you select this function, a window is displayed, in which you can select which linked objects are to be transferred to the target status as well.

- None: No parts, documents or projects are included in the status change.
- 1. Level: Only the parts, documents or projects from the first sub-level are included in the status change.
- All: None: All parts, documents or projects of the sub-structure are included in the status change.

This setting can be made separately for documents, parts and projects.

Note:

If the status changed is to be made for several objects, some of which have different start statuses, the status change is only made for those objects, for which the change to the desired target status is allowed.

Real-time simulation of the function "Status change – Top-down"

By the function "Check status change – top-down" the user can simulate the structured status change. This function does not perform the actual status change - it is only a simulation.

You can thus check whether the status change you want to make can be performed.

A status change cannot be made:

- if the object is currently being changed or is locked.
- if an object cannot reach the target status from its current status.
- if the user is not authorized to make a specific status change in question.

After the function "Check status change – top-down" PRO.FILE displays a list of errors that would occur in the case of an actual status change. This is to give you the opportunity of resolving these problems before making the status change.

As it is only a simulation, the data structure is not altered in any way.

6   Usage: Create versions/revisions of CAD-relevant data

PRO.FILE offers its users a sophisticated concept for versions and revisions. Benefits are the easy and secure advancement of the design process and a significant reduction of metadata.

The following explanations of the versioning with PRO.FILE CAD integrations us some basic definitions. These definitions are summarized in the sub-chapter. Basically, the following applies:

*   With PRO.FILE you can version documents as well as bills of materials.

*   PRO.FILE makes use of a two-level version and revision concept for the documentation of editing status of documents and product data.

*   The classification starts with a version of a document / a bill of materials. Changes after the release create a revision.

Function call: Control of version and revision

The general behavior of the versioning and revisioning is controlled in the PRO.FILE Management Console in the are "Configuration" => "Parameter" => "PRO.FILE" => "Document" => "Version / Revision counter".

It has to be explicitly noted that the versioning of documents follows a different version concept than the versioning of bills of materials. This is due to the usage purpose:

*   The versioning of documents is used to leave a specific status of a document untouched and to create a new version for further editing. Within linked structures, reference is always made to the newest version (this applies for PRO.FILE base; CAD integrations may offer the user the option to update the references).
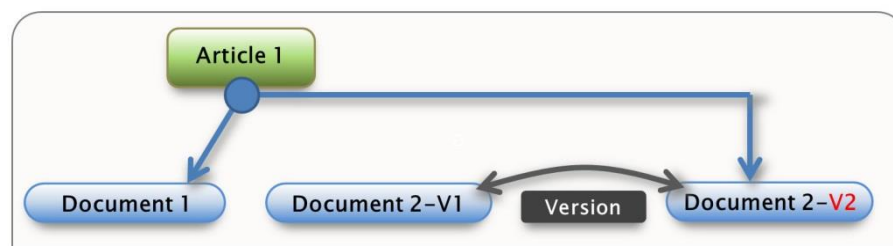
- The versioning of bills of materials is used for the product history, in order to prove which bill of materials was valid at a specific time. Reference is therefore always made to the original version.
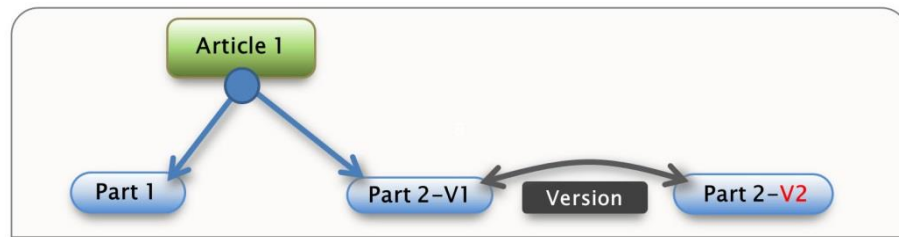
The composition and this the history of the older versions of this bill of materials must, of course, not be changed. Only if the superior object is versioned, it will include the versioned parts of the assembly:
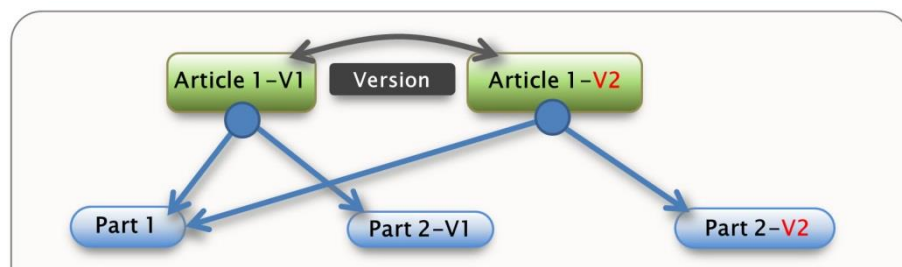
In order to make this difference evident, the name "Bill of materials versioning" has been chosen in PRO.FILE. Although the versioning of parts is used for the versioning of bill of materials, the actual focus of the process is not on the versioning of parts but on the versioning of the corresponding bill of materials.

Detailed information on the subject "Versioning with PRO.FILE" can be found in the following chapters:

- Definitions

- The display of the version cycle in PRO.FILE

- The versioning of CAD models with PRO.FILE

- For the history: Versioning of bills of materials

6.1   Definitions

This section is to describe the terms "version", "revision" and "variant", which are of relevance to the following chapters.

Version   Intermediate design status on the design path from one revision to the next.

In PRO.FILE, versions depict design statuses within a revision. Access to a version is always possible with the design status typical for the version.

A version represents a development usable for the designer, as a possible basis for further design developments. Versioning creates "backup copies" which can be used in case that further design developments have gone "into the wrong direction". This is to ensure the required flexibility in product development.

The versioning concept results in two options for the saving of opened versions:

- When the function "Save version" is used in the CAD integration, a copy of the object is created. The differentiation between the changed object and the original is represented by a version index increased by "1" while keeping the same material or item number.

- Smaller changes to a version may be implemented via the function "Save" without creating a new object version.

Revision   Released, 'frozen' result of a change. There is one revision for each release status.

A revision in PRO.FILE comprises an entire development cycle. It usually starts as version in the status in processing and ends in the status 'released' (the status 'released' must be defined as a release status before in the PRO.FILE Management Console). Released PRO.FILE objects (parts, assemblies, documents) can no longer be changed - unless a new revision is created.

This is done as soon as a version in a release status is saved as a new version. In this case, the revision index is increased, the version index is set back to a defined initial value. The 'old' revisions remain as backup copies.

Variant   The difference between versions and revisions on the one hand and variants on the other hand is the following:

- The terms revision or version contain information on the interchangeability of parts or sub-assemblies within products. Interchangeability is given if a changed object can be exchanged with its predecessor in all superior structures at random while still matching in form, fit and function.

- A variant is an independent object that is not interchangeable with its predecessor in form fit and function.

6.2   The display of the version cycle in PRO.FILE

To allow the designer keep an overview on the current version status, PRO.FILE visualizes the version/revision status via counters and initial values.

Counters

PRO.FILE allows numeric and alphabetic counting of versions and revisions.

The counting starts with 1 (numeric) or A (alphabetic) or with an initial value.

Within PRO.FILE, the versioning and revisioning of CAD documents is made from within the corresponding CAD integration. Here the user can start the action <New version>.

Versioning

If a new version of a document is created, the version counter is increased by one step. Each version is thus visible in PRO.FILE and can be accessed, if required. If a document has been versioned several times, the document description always shows the current version status.

The previous version are displayed in PRO.FILE via the tab "Versions". If a version with the status <in processing> is moved to the status <Released>, it can no longer be changed. To obtain a document instance that can be edited, a new version has to be created. In this case, a revisions in created and the version counter is reset to the first value ("1" or "A"):

Revision and version cycle in PRO.FILE
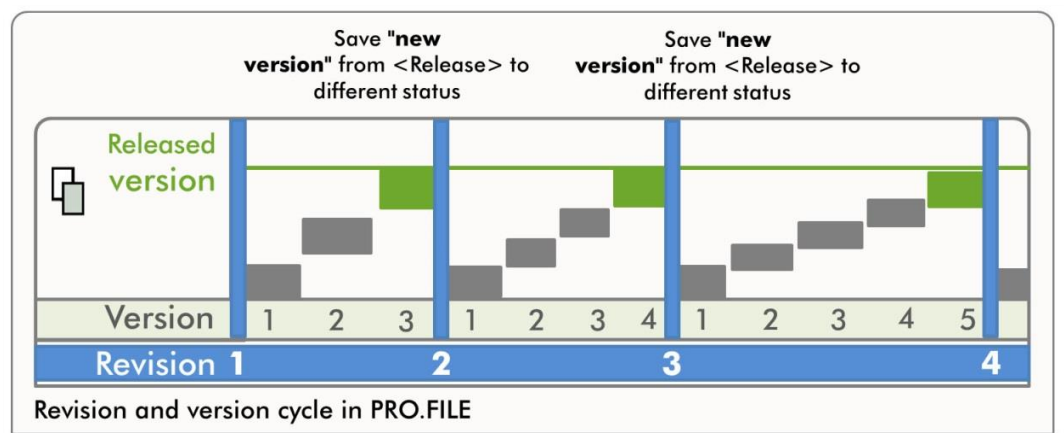
Note: Continuous version counting

Via the parameter "Activate version and revision counting", you can specify that the counting of versions is to be continued after a release status. For this the parameter has to be set to the value "CONTINUOUS".

Initial value

The initial value describes the first document created in the status <in processing>.

The initial value is not to be confused with the first value of the regular counting. It rather shows that the current document has not been edited so far.

The status <in processing> is here the first possible status is the system, for which an initial value can be set.

The initial value can be set for the version counting as well as for the revision counting in the PRO.FILE Management Console (Parameter "Initial value for the version counting" and "Initial value for the revision counting").

Recommended initial values:

▬	Minus	*	Asterisk
▭	blank	~	Tilde

The following is an example of the behavior of the version and revision counter in PRO.FILE if the initial value "-" is used:

Initial value = ▬	Version counter (numeric)	Revision counter (alphabetic)
Newly created	▬	▬
Released	1	▬
New version	▬	A
Released	1	A
New version	▬	B

As you can see, there are no redundancies in the display of the version cycle.

For the initial value, you should not select a character used in the general counting, as this may lead to confusions in the differentiation of versions and revisions.

The following negative example illustrates this problem: "A" is used as initial value, although the revision counter works alphabetically.

Initial value = [A]	Version counter (numeric)	Revision counter (alphabetic)
Newly created	[A]	[A]
Released	[1]	[A]
New version	[A]	[A]
Released	[1]	[A]
New version	[A]	[B]

As you can see from the colored highlighting in the table above, the version status is not always unique, since the first value of the counting is identical to the initial value of the revision counter.

6.2.1 Configuration examples of the versioning and revisioning

The general behavior of the versioning and revisioning is controlled in the PRO.FILE Management Console in the are "Configuration" => "Parameter" => "PRO.FILE" => "Document" => "Version / Revision counter".

Depending on the set parameters the system behaves differently. In the following, the effects of these parameters are explained with examples.

Example for a configuration without initial value

Parameters	Value	Explanation
Activate version/revision counting	YES	This parameter specifies whether versions of document descriptions are to be counted.
Type of the version counting	NUMERIC	Specifies whether the version counting is to be alphabetic, numeric, with or without initial value.
Places of version counting	[1]	This parameter specifies the number of places used for the version counting (e.g. 4 => 0015)
Initial value of the version counting	[—|]	The set value is NOT read, because a version counting without initial value has been selected.
Type of the revision counting	ALPHABETIC	Specifies whether the revision counting is to be alphabetic, numeric, with or without initial value.

Parameters	Value	Explanation
Places of the revision counting	1	This parameter specifies the number of places used for the revision counting (e.g. 4 => 0015)
Initial value of the revision counting	_|	The set value is NOT read, because a revision counting without initial value has been selected.

Result of a versioning/revisioning:

Action:	Result in PRO.FILE	
First-time creation of a document	Version 1	Revision A
versioned five times	Version 6	Revision A
This version released	Version 6	Revision A
The released version is versioned => revision	Version 1	Revision B
A new version created	Version 2	Revision B

Example for a configuration with initial value

Parameters	Value	Explanation
Activate version/revision counting	YES	This parameter specifies whether versions of document descriptions are to be counted.
Type of the version counting	NUMERIC WITH INITIAL VALUE	Specifies whether the version counting is to be alphabetic, numeric, with or without initial value.
Places of version counting	1	This parameter specifies the number of places used for the version counting (e.g. 4 => 0015)
Initial value of the version counting	_|	If the parameter "Type of the version counting" is set to "... WITH INITIAL VALUE", the initial value for the version counting can be specified.
Type of the revision counting	ALPHABETIC WITH INITIAL VALUE	Specifies whether the revision counting is to be alphabetic, numeric, with or without initial value.
Places of the revision counting	1	This parameter specifies the number of places used for the revision counting (e.g. 4 => 0015)
Initial value of the revision counting	_|	If the parameter "Type of the revision counting" is set to "... WITH INITIAL

Parameters	Value	Explanation
		VALUE", the initial value for the revision counting can be specified.

Result of a versioning/revisioning:

Action:	Result in PRO.FILE	
First-time creation of a document	Version -	Revision -
versioned five times	Version 5	Revision -
This version released	Version 5	Revision -
The released version is versioned => revision	Version -	Revision A
A new version created	Version 1	Revision A

6.3 The versioning of CAD models with PRO.FILE

For the versioning of CAD models with PRO.FILE, different usage cases can be distinguished, in which the versioning/revisioning can be used:

- Save changes on part level or create a version
- Versioning of instances of a part family
- Versioning of drawings

- Changing the revision status
- Versioning of parts in the context of assemblies
- Save changes to assemblies or create a version

6.3.1 Save changes on part level or create a version

To save changes on part level to PRO.FILE, the user can choose between the functions "Save" and "Save version".

To explain the difference of the functions and their effects, let's take a look at the following example:

A part in PRO.FILE is to be edited. It is in a status "in processing". It is now loaded from PRO.FILE for editing and then changed. To save the changes, the user can now choose:

Save changes   A part is loaded from PRO.FILE and changed. To save the changes you select "Save".

Part"Version 1" change and "Save"

By this command, the part is saved back to PRO.FILE. No change is made in the version number.

Save changes as version

Due to the modification, the designer decides to save the part as new version.

To do so, the designer selects the command "Save version".

By this command, the part is saved back to PRO.FILE as new version. The version counter is increased by one step. The old version remains intact in PRO.FILE.

Part "Version 1" change and "Save as version"

6.3.2 Versioning of instances of a part family

A significant part within a part family is changed. The changes affect the entire part family.

If the original part family is now used further on, the new part family has to be versioned.

To do so, the designer selects the command "Save version".

A single instance cannot exist on its own. It is derived according to the specifications of the part family.

Consequently, when a part family is versioned, all instances are versioned.

6.3.3 Versioning of drawings

It is to be noted that the usage of the command "Save version" for a part does not automatically create a version of the drawing of the part in PRO.FILE.

Part "Save as version"

To update the existing drawing, PRO.FILE offers the same function as for assemblies.

Via the function "Open with newest versions" when a drawing is selected from PRO.FILE, the system will search for newer versions of the contained components. In the second step the drawing is generated with the newest versions.
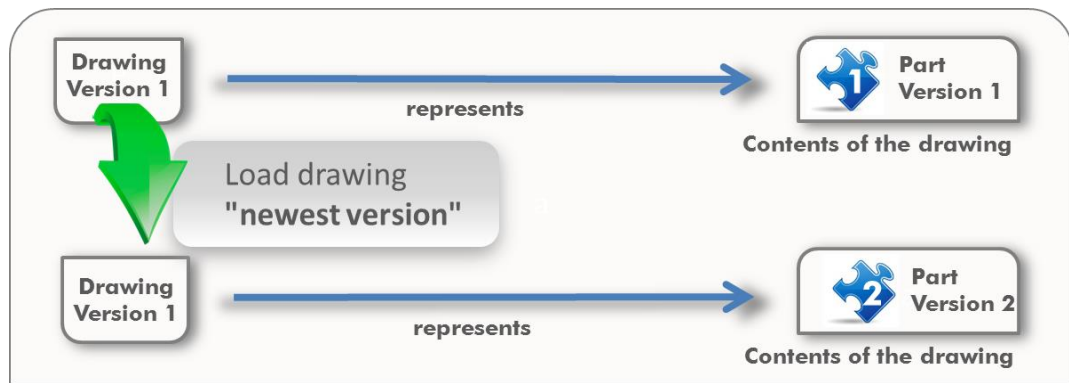
In the next step, the drawing is to be saved. To do so, you can use again the functions "Save" and "Save version".
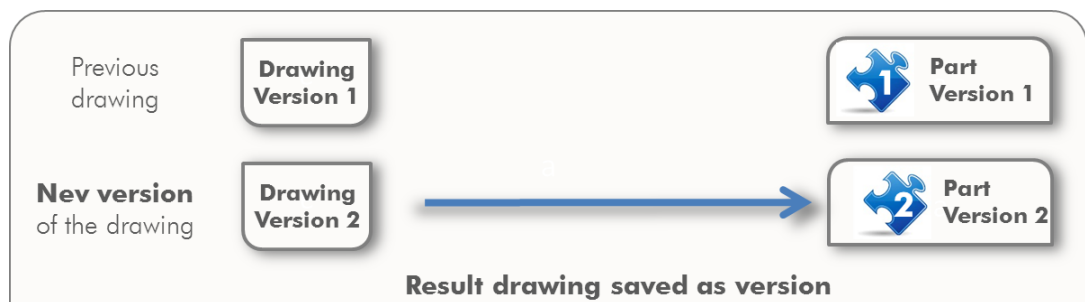
Save

The command "Save" overwrites the existing drawing. The version counter is not changed.
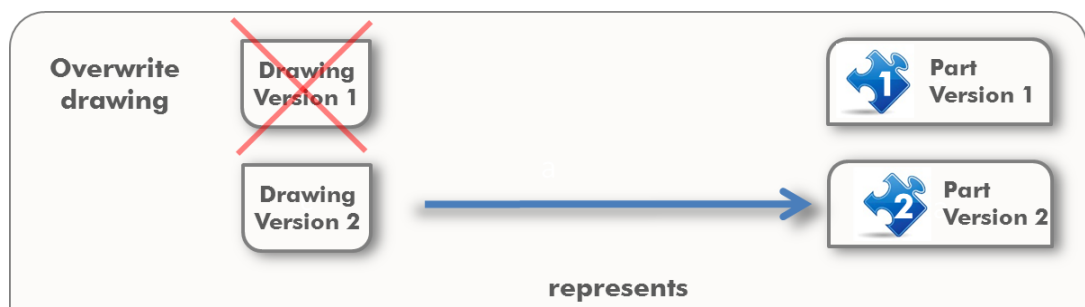
Save version

The function "Save version" creates a new version of the drawing referencing to all new part version displayed in the drawing. The version counter is increased by one step. The previous version of the drawing is still available and documents an earlier design state.

6.3.4   Versions within assemblies

Some CAD systems allow the usage of different component versions within an assembly or drawing. For a clear reference, the CAD component has to be saved with a different file name when the version is created. The versioning with new file names is only possible for real modelle. In the PRO.FILE Management Console, you can specify whether a new file name is to be created for versions.

Note:

This type of versioning is supported by the CAD systems Creo Parameteric, Siemens NX, Inventor, SolidWorks and Solid Edge. It is used during the creation of single versions and during Managed Versions.

Function call:

The configuration of new file names for versions is made in the PRO.FILE Management Console via the parameter "**Create new file name when versioning**" under

"**Configuration**" => "**Parameter**" => "**CAD**" => "**Integration**" => "**<CAD system>**" => "**Other**"

Proceed as follows

1. Set the value of the parameter "Create new file name when versioning" in the PRO.FILE Management Console to the value "1" or "2".

2. Restart PRO.FILE.

3. Create a new version of the CAD component via the integration.

4. Use the integration function "Replace component" to insert the new component version.

The parameter "Create new file name when versioning" has the following effects:

*   Depending on the value setting, the file name is generated as follows:

    *   0 = No new file name is generated.

    *   1 = A new file name is always generated.

    *   2 = A dialog is displayed, asking whether a new file name is to be generated. When Managed Version is used, the dialog is displayed for each component to be versioned.

*   If the file name is identical to the file name of the old version, the document record ID is attachted.
    Example: Old file name myassembly_00107565.iam, new file name myassembly_00107565_00107568.iam

*   The model from the CAD integration is saved locally with the new file name.

*   The file is saved to PRO.FILE as a version of the original document.

Attention:

If you are using different file names within a version chain, you can use the opening functions "with version browser", "with newest versions", "with newest, released versions" and the function "Replace version" only to a limited extent. This is due to the fact that these functions only consider versions within the version chain that have the same file name. The opening function that is used can be influenced via the parameter "Version load options dialog" in the PRO.FILE Management Console.

A corresponding message is displayed in the dialogs:

```
"There is a newer visible version <version number>. This
version cannot be used due to the different file name <file
name>."
```

The dialogs of the CAD intetgration, e.g. the document list, Managed Copy or Managed Version, contain the column "Version". This column shows the number of the currently version as well as the overall number of versions for this document.

To use a version with a different file name in your assembly, please use the integration function "Insert component".

## 6.3.5   Changing the revision status

A development or change cycle of a part is closed when the part is "Released". Upon release, only the workflow status changes. The version counter is not changed.
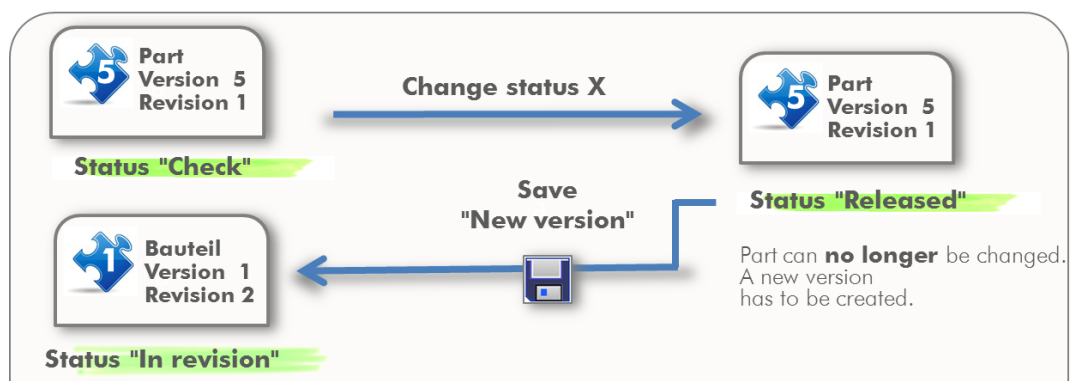
If it is discovered that a released part has to be changed, a new change cycle has to be set up. This is done as follows:

1. The CAD model of the affected part or the document is loaded in the corresponding application.

2. The changes are made in the CAD program.

3. The model is saved back via the function <Save version>. In this case, the new revision of the part or document is created automatically:

⇒   The revision counter is increased by one step.

⇒   The version counter is reset to the start value.

⇒   The status of the part corresponds to the start status defined for the active user.


Revision and version cycle in PRO.FILE
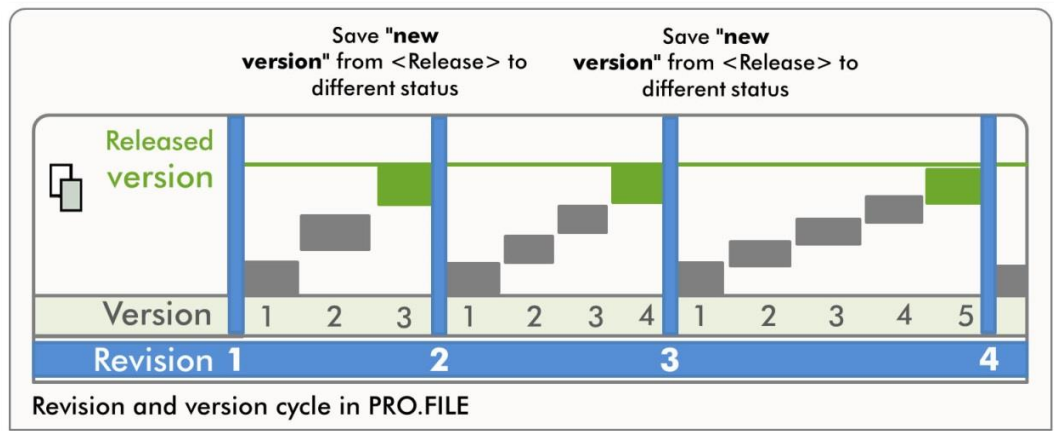
6.3.6   Versioning of parts in the context of assemblies

When a part is versioned in PRO.FILE, the reference of the assemblies to the previous part remains intact.

New part versions are at first edited independently without an assembly referencing it.

If the editing of the part is finished, the designer explicitly has to make sure that the new version of the part is used in the superior assemblies. For this, the superior assembly has to be opened from PRO.FILE with the function "Open with newest versions" and then saved back.
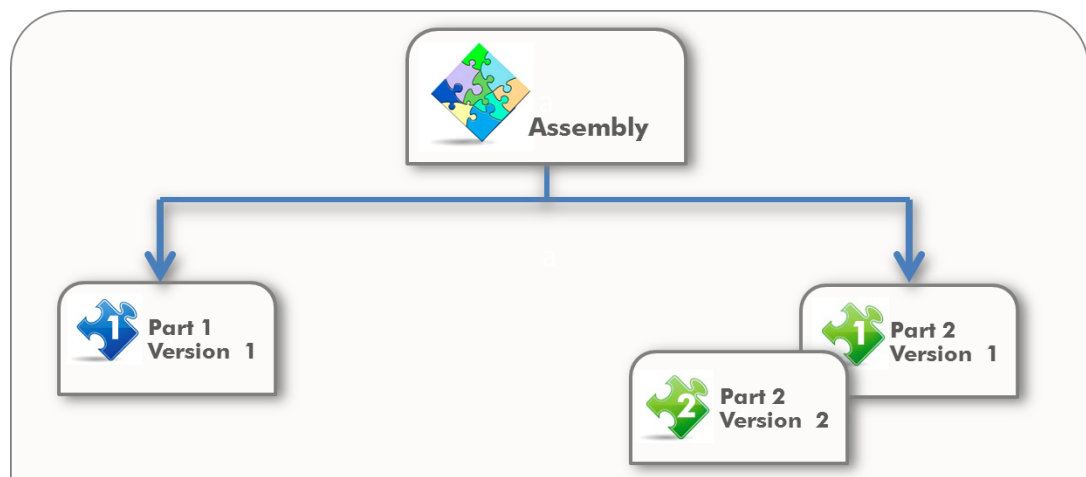

Opening assemblies from PRO.FILE

To generate an assembly with the newest versions, it has to be loaded from PRO.FILE again.

To do so, you can choose from three different options for opening objects from PRO.FILE in the CAD program:

- Open ("as saved")
- Open with newest versions
- Open with released versions

When "Open with released versions" is selected, PRO.FILE checks whether there are newer, released versions of parts, which are contained in the assembly, in PRO.FILE. After this check, the new structure of the current assembly is detected.
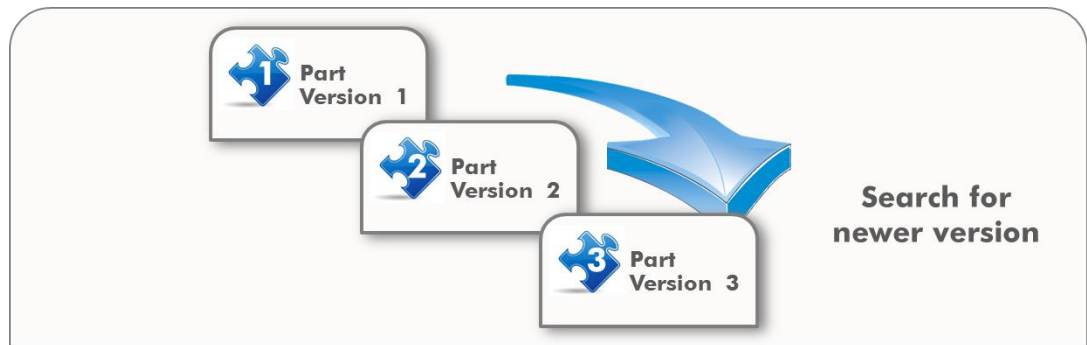
After that, starting from the found newest version of a part, the permissions and statuses are checked.

Opening an assembly with versions of the parts "as saved"

Via the function "Open", the assembly is loaded in the constellation of component version that was last saved. Changes to parts documented by newer versions are ignored.
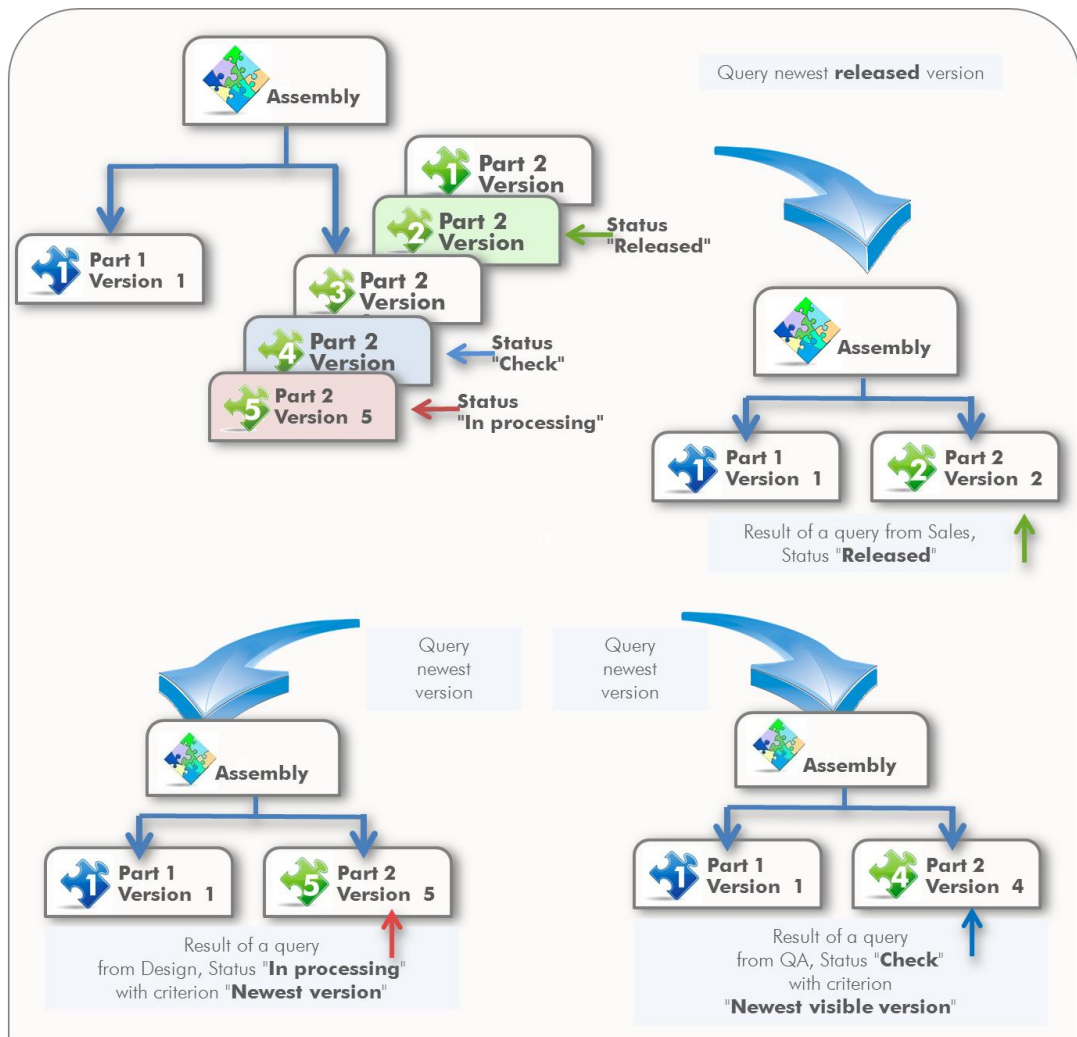
Opening an assembly with newest versions

When this function is used, PRO.FILE uses for the structure of the assembly at all levels the newest saved version of parts that can be found. As a consequence, PRO.FILE checks for every part whether the user has the required reading permissions for each found version. If the user does not have the required permissions for the found part version, PRO.FILE goes back one step in the version list and repeats the check. This is repeated until a part version is found, for which the user has the required permissions. If no such part version is found down to version 1, the user gets an error message from the system. Thus, only the versions visible for the user are loaded.

Opening an assembly with released versions

When this function is used, PRO.FILE uses for the structure of the assembly at all levels the newest saved version of parts that is in a released status. Here too, the check is made, whether the user has the required permissions for seeing the part.

Example: Effects of the different opening methods on the displayed result.

6.3.7 Save changes to assemblies or create a version

After loading an assembly from PRO.FILE, the assembly in the CAD system contains - according to the user permissions - the newest version of all used parts. When saving the assembly, you can decide how the assembly is to be handled. You can choose between two ways of proceeding:

- Overwrite the assembly with the newest changes => "Save"

- Create new version of the assembly => "Save as new version"

Save

When using the function "Save", the existing assembly is updated with the newest version of parts without changing the version counter of the assembly. In this case, the references are updated to the new part versions and the assembly is saved with the existing name and overwritten in PRO.FILE.

> **!** Attention: Risk of data loss
>
> When this function is used, all possibilities of recovering the old editing status of the assembly are lost.

Save version

When the function "Save version" is used, a new version of the assembly is saved to PRO.FILE. The structure tree (references) of the assembly is built with the currently loaded parts. The references are set to the newest versions of the parts and the assembly is saved as new version in PRO.FILE. The old version of the assembly also remains in PRO.FILE, with its previous editing status.
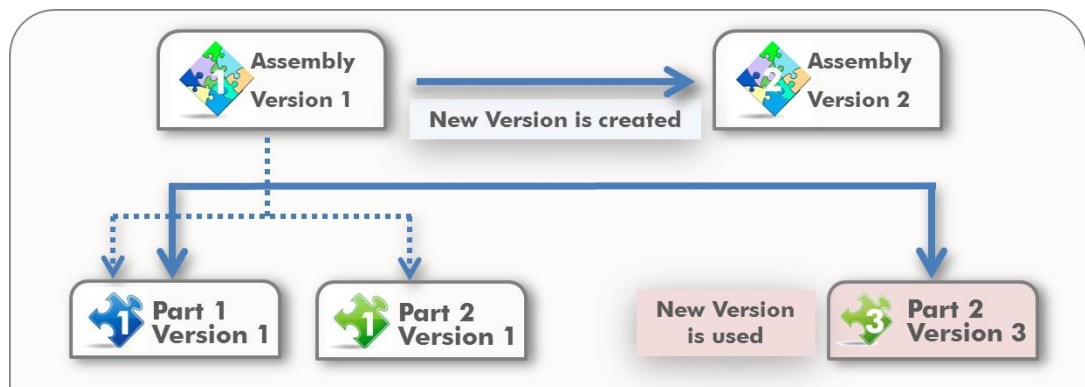
> **!** Attention: No update across assemblies!
>
> If a part is used in several assemblies, each assembly has to be update via this function (or via the function "Replace version").
>
> This has to be noted, especially for updating single-level designs and for the usage of multiple different usage of components.

Cross-assembly versioning with "Replace version"

It may be required to use the new version of the part at a specific time in all superior assemblies.

In case that the part is to be replaced in all superior assemblies, the handling can be made easier by the PRO.FILE function "Replace version".

Via the function "Replace version", a search is made for all assemblies referencing the predecessor version of the current part. The reference can then be update to the new part version.
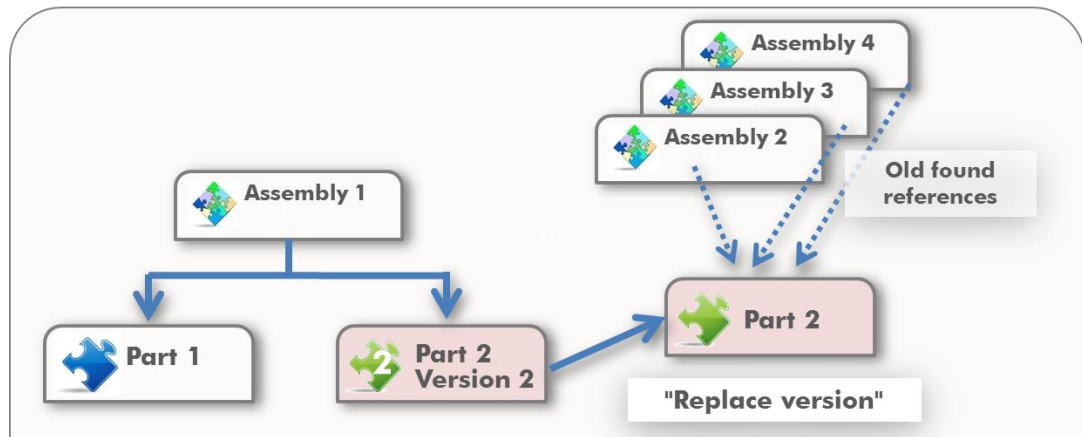
PRO.FILE creates a special documents list, in which all assemblies are listed, which reference the predecessor version of the part.

You can then select the assemblies to be updated. The CAD information X*used shows how often this assembly is in turn used in other assemblies.

In all selected assemblies, the references are then updated by references to the current part (new version). Before a components is exchanged in an assembly, PRO.FILE checks whether the user is authorized to change this assembly.
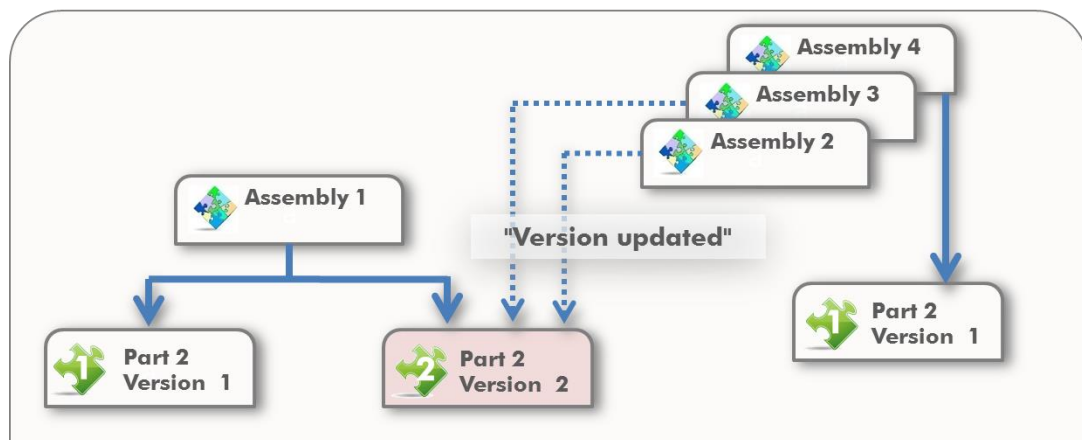
In the example above, the assemblies 2 and 3 were selected for updating and the assembly 4 is kept with the old structure.

This proceeding should be applied if only the parts are to be versioned, but the assemblies are always to reflect the current version. The assemblies are not versioned. By the versioning of the parts, a part history is created. The benefit of this is that the predecessor version of the part can be accessed at any time. The disadvantage is that the corresponding assembly (previous version status) cannot be generated.

This proceeding may be applied in the product development during the design phase. To meet the requirements stipulated by DIN ISO, the intermediate status can be documented by the creation of TIFF drawings of the assembly.

6.4   For the history: Versioning of bills of materials

PRO.FILE allows for the versioning of bills of materials. But since the term "versioning of bills of materials" may lead to different interpretations, this chapter is to give you an overview of the basic concept and the resulting possibilities of usage. This is to make sure that you can use the versioning of bills of materials to your requirements and advantage.

This description consists of four sections:

- The aim of the versioning of bills of materials:
  What can the versioning of bills of materials be used for? How can it be used? What are the recommendations of PROCAD?

- Concept of the versioning of bills of materials:
  Some theory. What happens during the versioning of bills of materials in PRO.FILE?

- The usage of the versioning of bills of materials:
  How can I access the functions for the versioning of bills of materials? How can I update versions? What happens to my assemblies, used parts and attached documents? Which order has to be observed?

- An example of the versioning of bills of materials:
  A direct example for clarification of the complex topic of the versioning of bills of materials.

Important note::

It is strongly recommended that you create a concept for the versioning of parts lists, which contains actual company circumstances, requirements and the specified structure that you require, before it is introduced.

Once this concept has been created, all users should receive the relevant teaching according to this concept.

6.4.1   The aim of the versioning of bills of materials

In PRO.FILE a bill of materials describes the parts and the structures that are combined to make up an assembly.

As assemblies are subject to constant changes and regularly being modified during the lifecycle, the bill of materials is also continually being modified and updated. This means that during its existence, an assembly can have different version of the bill of materials.

Therefore, parts, are replaced by other parts, are simply deleted, and are joined by newly added parts. For a number of reasons, (Quality management, spare part management, etc.), it is therefore a great advantage to know how the bills of materials look within the assemblies, at all times. In other words:

Which version of the bill of materials was valid at what point in time for the assembly?

This is the exact task for which the versioning of bills of materials for PRO.FILE has been developed. Using the versioning of bills of materials, you have the ability to:

* Give information on the nature of a part or an assembly in the past.

* Keep records of modifications to assemblies within quality management and/or the legal requirements.

* Ensure that documents are made available for maintenance, service and the provision of spare parts.

!	**Attention:**
	Two different versions of a bill of materials cannot both be up to date and valid at the same time.
	It is only the newest version of the bill of materials that is valid.
	Bill of materials versions can therefore not be used for the administration of variants.

6.4.2   Concept of the versioning of bills of materials

In PRO.FILE, a bill of materials is always created for a specific assembly, and is permanently linked to the part description of this assembly. Therefore, the versioning of bills of materials in PRO.FILE can only be carried out with the help of a new version of the PRO.FILE part description for the specific assembly.

By this versioning of the bill of materials from the part description, the metadata, fields, and item classes, the single-level bill of materials, and the direct relationship to documents of the selected assembly are copied.

This means: A new part description containing a new database identity number, which has a version relationship with the "old", assembly, is created for the new version of the bill of materials. At the start, this new version of the assembly always has the same bill of materials as the old version.

In addition to the bill of materials, the new version of the assembly takes on the relationship to the documents of the previous version of the assembly. These are then marked with a special link type (soft link). This enables the installed CAD integration to recognize that the document is not the direct document for this version of the assembly. The "strong", link to the attached document still remains to the original version of the assembly.

Thanks to this principle, the new version of the bill of materials can now be called up via the new version of the part description. The bill of materials and the links to the old versions remain unchanged in the release status, and can always be viewed in this original status.

This allows a better creation and handling of a production history than in earlier PRO.FILE Versions. The individual versions of the bills of materials can be easily set apart from each other.

Furthermore, you can later replace older versions of parts with the newest versions when modifying the versioned bill of materials. Or you can replace them with other parts. And therefore change the newer version of the bill of materials in contrast to the old version.

> **Note:**
>
> Every new version of a part description has its own database identity number. For this reason, the database identity number is not suited for the use as article number.

Version and revision counter for parts

To enable you to tell the versions apart from one another, a version and revision counter is used for the parts, in the same way as one is used for the documents. If a version is created from a part description in a release status, the revision counter is increased. If a part description is versioned in a status that is not a release status, the version counter is increased.

Search for bill of materials versions

A bill of materials version is linked to the related version of the part description, and is selected via this part description.

A search for part descriptions without the use of the identity number, will always find the newest visible versions. The older versions are then shown additionally in the dependent tab 'Versions'. If configured accordingly, it is possible to show the newer versions here as well.

Specific versions of an assembly can be identified using the database identity number.

> **Attention:**
>
> The PRO.FILE versioning of bills of materials **is not** article versioning!

As each bill of materials is uniquely linked to a specific assembly and the corresponding PRO.FILE part description, the versioning of a bills of materials in PRO.FILE is carried out via the versioning of its related part description.

Even though the versioning of bills of materials is carried out in this way you should not confuse it with general article versioning.

It is important to remember the basic principles of bill of material versioning:

- The versioning of bills of materials is carried out to fix already existing bills of materials to an assembly, and to create a new version of the bill of materials when modifications are made. The history of the old versions of bills of materials should remain unchanged.

- Therefore the existing bill of materials of a superior assembly must remain unchanged even when a subordinate part is versioned.

- This means that after the versioning of a subordinate part, the bill of materials should not automatically refer to this new version of the part (P2/V2), but to the original version (P2/V1):
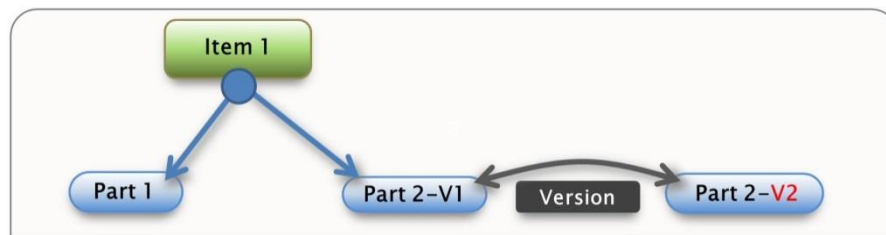
In this way, the PRO.FILE versioning of bills of materials behaves in a completely different way than a general article versioning:

- A general article versioning would refer to the new version.
- The PRO.FILE versioning of bills of materials, on the other hand, refers to the original version (see image above).

The following information should help to support this.

6.4.3   The usage of the versioning of bills of materials

There are several steps that must be carried out in the correct order for the versioning of bills of materials; these are as follows:

Step 1   Create a new version of the bill of materials via the PRO.FILE part description of the superior assembly.

- To carry out the versioning of a bill of materials in PRO.FILE, you should call up the part description of the assembly for which the bill of materials version is to be created.
- You can then selection the function "New version/revision" either from the "Edit" menu in PRO.FILE or via the context menu (right mouse button).
- The metadata and the existing bill of materials are copied into the new version of the part description, and the version and revision counters are increased accordingly.

Step 2   Identify the documents linked to assembly

- Documents that were attached to the previous versions of the assembly are now visible through a "soft" link, under the new version of the assembly.
- All documents, the contents of which are affected by the versioning of the bill of materials of an assembly, and which are to be modified, must also be versioned.

Step 3   Adjustment of the versioned bill of materials

- There are now two possible ways of adjusting the versioned bill of materials:

- You can use the bill of materials editor in PRO.FILE, and the function "Update Version" from the corresponding "Edit" menu.

- You can use the bill of materials alignment from the PRO.FILE – CAD integration, of course after the components have been exchanged in the CAD assembly.

Note:

The versioning of bills of materials in complex structures must always be carried out from the bottom upwards.

Further descriptions of how the part description behaves as a superior assembly or used part with a new bill of materials version, and how to handle linked documents can be found in the following sub-chapters:

6.4.4   The versioning of bills of materials of assemblies

Once an assembly has been versioned (menu "Edit", => "New Version/Revision"), although a new part description with a new database identity number is created, the references of the existing bill of materials are duplicated by this versioning process:

The new version of the assembly does not only have the same metadata as the original part description of the assembly, but at the beginning also the same bill of materials.
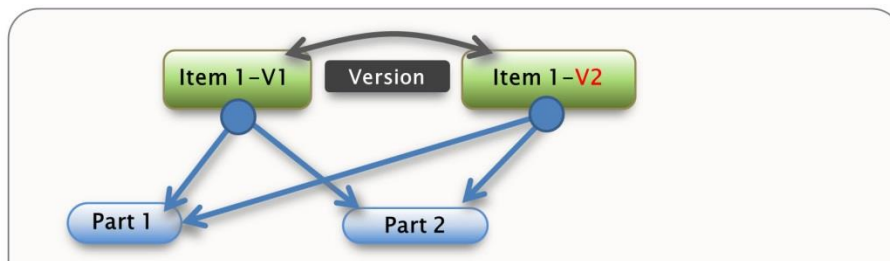
After this, modifications can be carried out in the bill of materials of the new assembly version without the bill of materials of the previous version being changed.

You can ensure a complete bill of materials history by ruling out later modification of the bill of materials using the release statuses and the related status-dependent permission from the status administration.

Versioning from the viewpoint of the built in parts

If a used part is versioned in PRO.FILE, a new part description is created with a new database identity number, and version identifier.

The bill of materials the existing part was contained in remains. The superior assemblies, which contain the original version of the part description, remain unchanged:
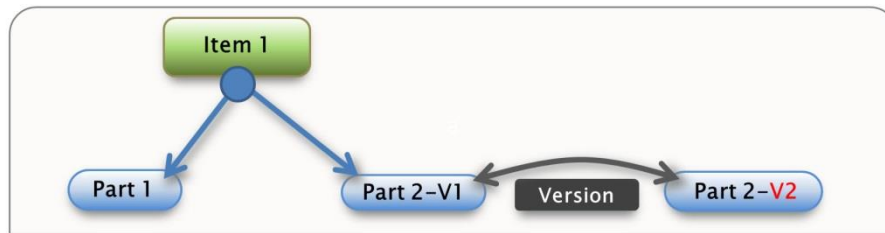
Attention:

When a subordinate part is versioned, the bill of materials does not automatically refer to the new version of the part. Only in this way is the original state of the bill of materials kept as the old version, which guarantees a flawless bill of materials history.

The versioning of a single part and an assembly are both carried out using the same function in PRO.FILE, "Edit" => "New Version/Revision". The single part has no bill of materials.

A new version of a part description can only be created from the newest version.

Note:

No older versions of a part description can be versioned.

Updating the new part version in the bill of materials:

When modifying the bill of materials of a superior assembly in PRO.FILE, it is possible to swap old versions of built in parts for the newest versions of the parts. The function, "Update Version", from the bill of materials editor can be used here, or it can be carried out in the CAD integration via the bill of materials alignment.

The bill of materials then no longer uses the old version of the part description; it uses the new version in PRO.FILE.

This is only useful for the required bill of materials history if the versioning is carried out on two levels: This means, for the used part, as well as for the superior assembly. Only after the versioning of the assembly, the new version of the used part should be updated in the versioned bill of materials.

Attention: Careless operation can endanger the bill of materials history

If the built in part alone is versioned, and not the bill of materials of the assembly to which it belongs, and then the new version of the built in part is updated in the existing bill of materials, this is only a modification of the bill of materials.
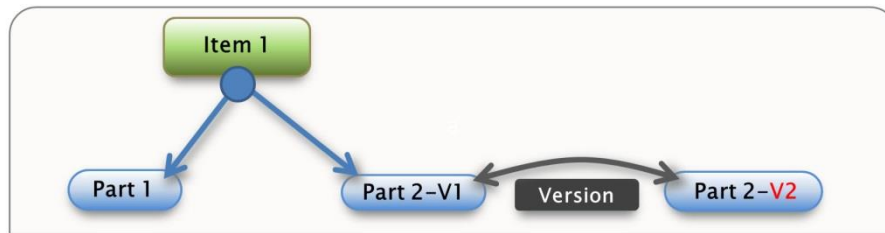
The bill of materials position is versioned, and not the bill of materials. It is then not possible to trace the bill of materials history.

This error can be avoided by using status-dependent permissions. If the current bill of materials is in a release status, no modifications can be carried out by default. Modifications can only be carried out by versioning the bill of materials.

A detailed practical description of the process of updating the bill of materials can be found in the following chapter "For the history: Versioning of bills of materials".

6.4.5 Handling documents in the versioning of bills of materials

If the part description of the assembly is versioned with the bill of materials, a new part description with a new version number is created for the assembly. At the beginning, the new part description always has the same links to the documents as its previous version.

For the document assignment and the access authority within PRO.FILE, it is necessary that the linked documents have a unique "Father", (also e.g. to fill out the title block of a drawing).

Therefore a "strong connection", is created between the document and the original part description, and a "weak connection", is assigned to the versioned part description:
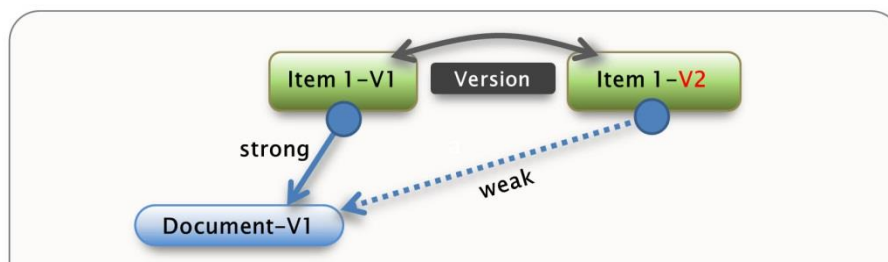
The "strong" connection enables the clear assignment, whereas the "weak", connection only shows the documents that exist for this article.

If the part description of the assembly is then versioned again, an additional weak connection is created between the part description and the attached document.
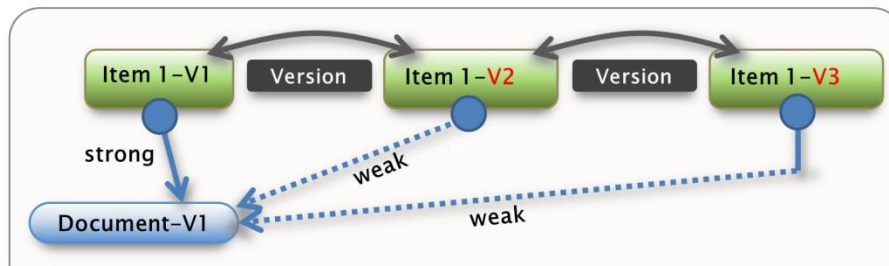
The behavior of the attached documents when versioning:

If the attached document is now versioned, a new document description and version number are created. This new document description is attached to the newest version of the related part description, to which the previous version of the document description was linked.

The existing "weak connection" for the newest version of the part description to the "original" document description is deleted and replaced by a "strong connection" to the newest version of the document description:
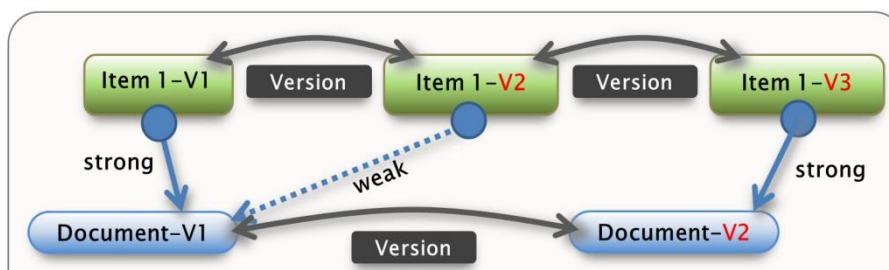
Version a document or not?

The diagram above requires the following additional information:

The article 1 exists, with the link to a document 1. Two versions of this article 1 are created, one after the other. Both have a weak connection to the document 1. Then a version 2 is created from document 1.

The newest version of the article 1/version 3 now has a strong connection to document version 2. The original article 1/version 1 still has a strong connection to document version 1.

Only article 1/version 2 has a weak connection to document version 1, and therefore no own version of the drawing.

Depending on the requirements of the versioning of the part description, it is always necessary to version the related documents as well. Only after this has been completed, you should create another version of this part description:

There are constellations that require their own version of the document, e.g. drawings with version number of the article in the title block. An attached operation manual on the other hand does not need to be versioned, if e.g. only the material of a screw has been changed which will have no effect on the operation.

Generally, attached documents must always be versioned in the following cases:

- If the current metadata of a part is to be displayed in the title block (e.g. the revision counter).

- If the bill of materials on a drawing is to be updated.

- If other parts or other versions of parts are used in CAD assemblies.

!	Attention:
	With documents that are dependent on one another, the order which is defined for the document versioning in PRO.FILE must be observed during the versioning procedure. See also the operation manuals for PRO.FILE or the corresponding CAD integration.

i	Note:
	The versioning of older document versions is still allowed for document descriptions. The resulting new versions are linked to the up-to-date part versions. Part versions that are in between keep their "weak" connections.

6.4.6   An example of the versioning of bills of materials

Below, is an example of the versioning of bills of materials in PRO.FILE:

Example: A version of an existing car is built with a carburetor. This version is no longer up to date; from now on only cars with fuel injection pumps are to leave the production line. Therefore, a new version is created, which includes the fuel injection pump. The old version with the carburetor will no longer be delivered.

A new version of the engine is now created. This new version has the fuel injection pump added to it. After this, a new version of the car containing the new version of the engine with the fuel injection pump is created. The result of this is as follows: The present version of the car now contains the new version of the engine, and the old version of the car still exists with the old version of the engine.

- Starting point
  The present version of the part description of the car is revision 1/version 0. The same applies to the part description of the engine and the carburetor. Both the car and the engine have their own design drawing with the revision 0A/version0A:
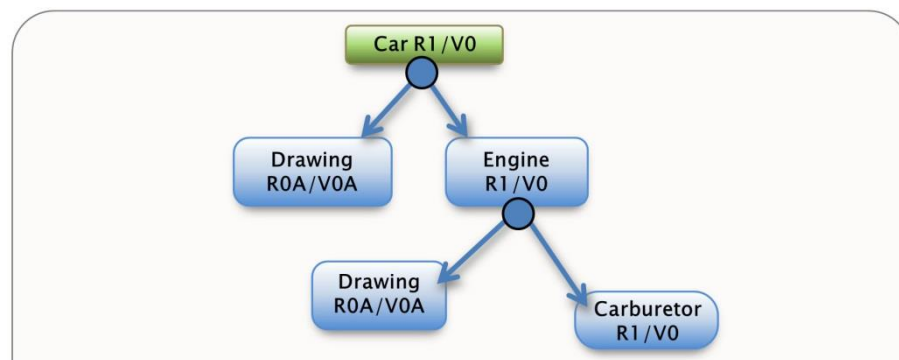
- **Step 1: Versioning the engine (Object type "Part")**
  Using the PRO.FILE-function "New Version", from the "Edit"-menu of PRO.FILE, you can create a new version of the part description for the engine:
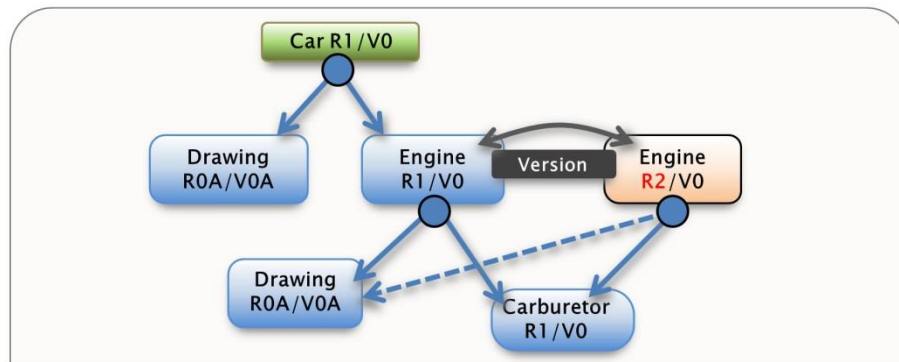

  The new version has the same bill of materials as well as the "weak" connection to the same documents as that of the original version of the part description of the engine.

- **Step 2: Versioning the engine drawing (Object type "Document")**
  A new version of the engine drawing is now created. This is done via the CAD integration:
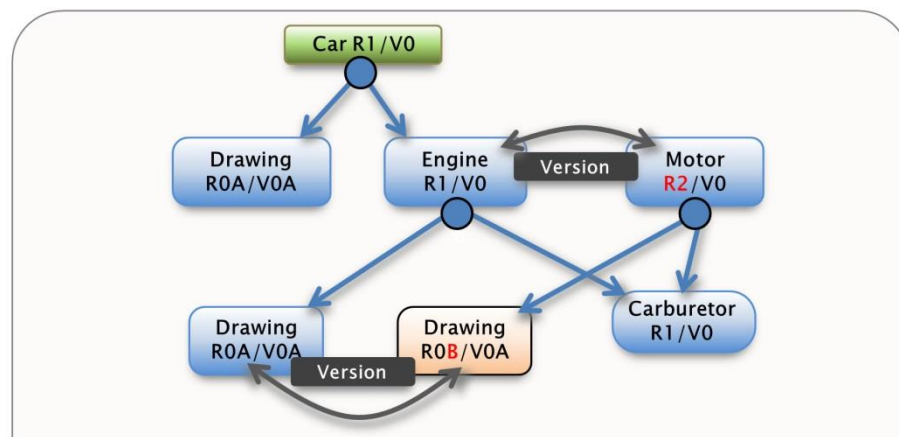
  The new version of the drawing is automatically attached under the new version of the engine. The drawing R0B/V0A of the engine R2/V0, can now be modified.

- **Step 3: Replacing carburetor with fuel injection pump (Bill of material modification)**
  Using the PRO.FILE bill of materials functions, the bill of materials for the engine R2/V0 must now be modified. The carburetor is replaced by the fuel injection pump:
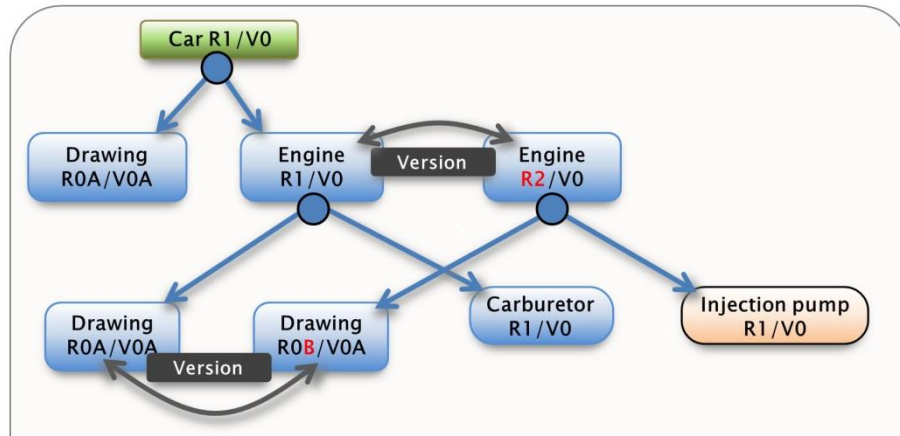
- Step 4: Versioning the car (Object type "Part")
  Before the new car can be designed, a new version of the part description is created for the car with the revision 2/version 0. This is also done via the PRO.FILE function "New Version" from the PRO.FILE "Edit"-menu:
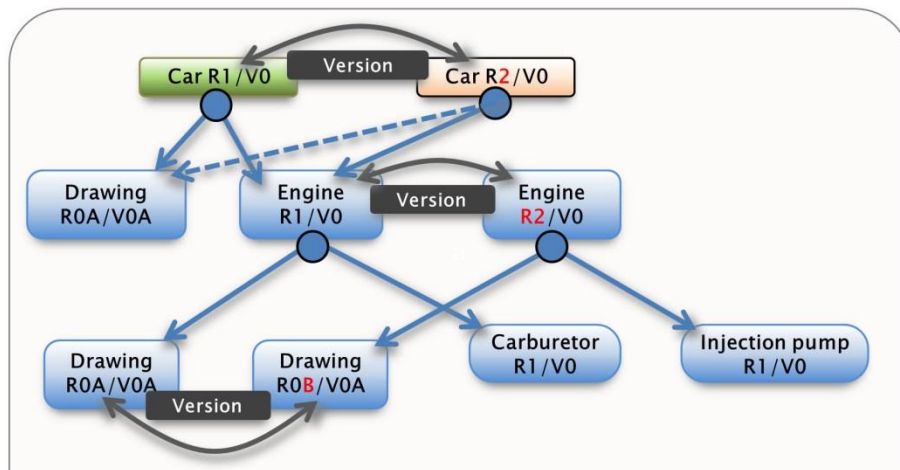
- Step 5: Versioning the car drawing (Object type "Document")
  Finally, a new version of the design drawings of the car is created. This is done via the CAD integration:
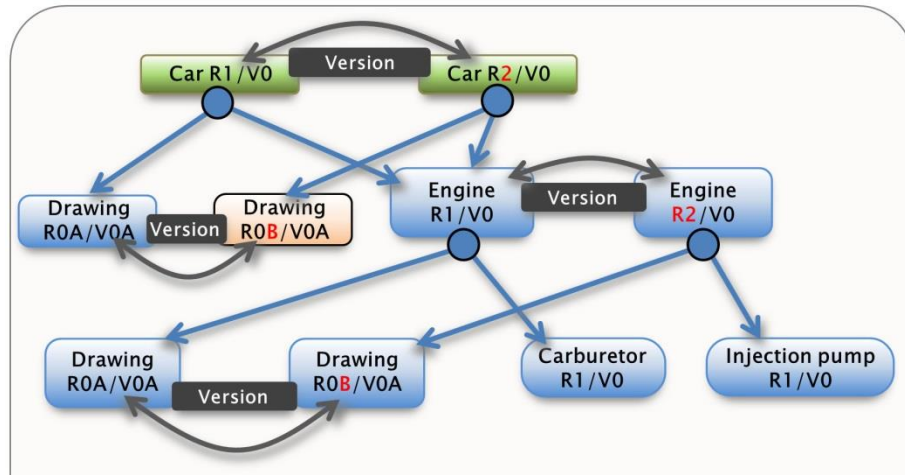
As for the engine drawing, the new version of the design drawing is automatically attached under the new version of the car. The drawing R0B/V0A of the car can now be edited in the CAD system.

- Step 6: Updating the new bill of materials for the car with the new engine
  In the deciding step, the bill of materials must now be updated with the newest version R2/V0 of the car with the new version of the engine R2/V0 assigned to it.

  The modification of the bill of materials is carried out using the command for the update of a bill of materials position in the latest version of the engine. This command can be found in the PRO.FILE bill of materials editor: "Update Version".
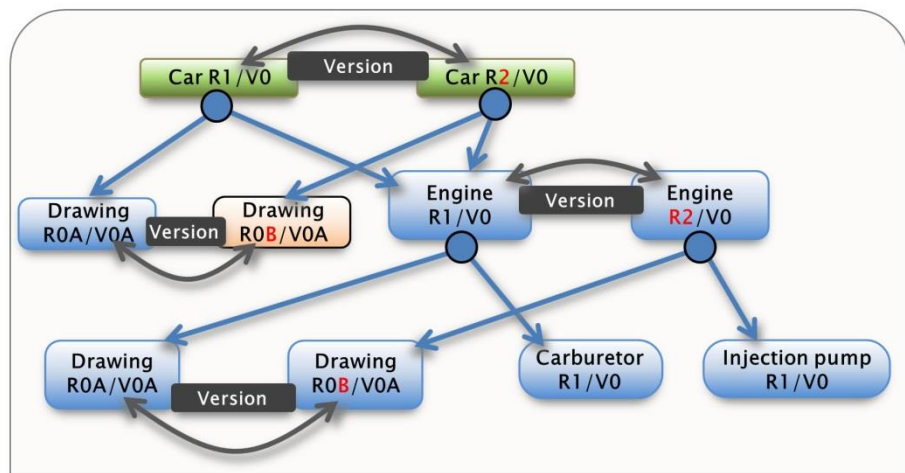
- Result
  The final result in PRO.FILE is the car with the engine containing the fuel injection pump. The latest version of the car R2/V0, shows the versioned drawing R0B/V0A, as well as the newest version of the engine R2/V0 with the fuel injection pump R1/V0.
  The version of the car containing the carburetor is now outdated, and is no longer produced – the PRO.FILE search functions now only find the new version R2/V0 of the car.

⚠️	Achtung:
	Only one version of the car can be the currently valid version at a time. The car "Fuel injection pump", must always be used as the latest version, and not as a variant.

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