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GENIUS TOOLS®

Startup TOOLS

<%MVN_VERSION%>

Overview and guidelines

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1 Introduction

For almost all IT systems, an efficient and consistent setup of the operating environment is essential for profiting from the software in use in a company. The more complex and interlinked a system gets, the more important this factor becomes. The goal of the setup is to minimize the time each user has to spend before they can start their productive work. A well-planned operating environment has a high degree of standardization and thus improves work productivity as well as the quality of the results.

This document contains work guidelines and basic concepts for working with the Startup TOOLS developed by INNEO Solutions GmbH and aims to help you introduce best practices.

1.1 Operating environment and work guidelines

What impact do work guidelines and prior planning have on the product development system?

It is a challenge for companies to create an efficient operating environment for the central CAD software, which is used for developing all mechanical parts. When a new software is introduced and implemented, there is usually a lack of know-how on how to make the best use of the new applications, so that creating the final company-specific environment can be a long and arduous iterative process.

INNEO Solutions GmbH has long-standing expertise in planning and implementing standardized operating environments for PTC software. A coherent set of serviceable configuration options is the first step towards this goal. Startup TOOLS stands for comfortable central management of all relevant configurations and objects. The functionality for synchronizing user environments now also offers a clearly structured installation concept.

Startup TOOLS also includes functionality for project-oriented work in different environments, for license management, for Windchill integration into Creo data management, and for working with additional software modules. Easy availability of a coordinated set of Creo objects such as start parts, assemblies, drawings, parts lists and bend tables reduces training and introduction efforts and lets you work productively from day one, also ensuring consistent data quality.

Startup TOOLS comes with integrated software applications for use within Creo. INNEO customers have come to rely on the modules developed for graphical library management, variant management, parameter management, number and name generation as well as other useful functions such as Mapkey management, tolerance table creation and export of parts lists to Microsoft Excel.

Use the standardized word environment offered by Startup TOOLS to work productively and efficiently right from the start. You will of course have to make some company-specific adaptations, as the example data in Startup TOOLS is based on DIN and ISO standards. However, this is a relatively minor investment. Even the popular Creo manual *3D-Konstruktionen mit Creo Parametric* by Pau Wyndorps uses INNEO's Startup TOOLS, and a free student version is available. It is therefore by no means exaggerated to call the Startup TOOLS and industry standard.

The Startup TOOLS consist of three separate components.

1) GENIUS TOOLS Starter: Applications for central management of operating environments and synchronizing the operating environment from a central Caddepot to a local Cadpool on each application computer

- GENIUS TOOLS Starter App: interface for the users to select projects
- GENIUS TOOLS Project Configurator: administrative tool for configuring projects and managing user access
- GENIUS TOOLS Environment Administrator: administrative tool for creating and managing operating environments

2) Toolkit applications – GENIUS TOOLS for Creo

- GENIUS TOOLS Parameter
- GENIUS TOOLS Library
- GENIUS TOOLS Quick Access, Forms, Dimensions, Inspect etc.

3) Coordinated set of Creo objects

- Start objects
- Drawing frames and header blocks
- Table templates for parts lists, and more

1.2 Company specific environments

Due to different products, company-specific processes, manufacturing technologies and product development strategies, each company has to adapt the operating environment to their specific requirements. These adaptations should always be documented in the form of written guidelines. Step by step, the operating environment can evolve to better support a companies processes and keep all employees informed about how to use the software in a timely and transparent manner.

At least one person has to take on the role of CAD administrator. They have to have in-depth knowledge of the operating environment and keep it up-to-date.

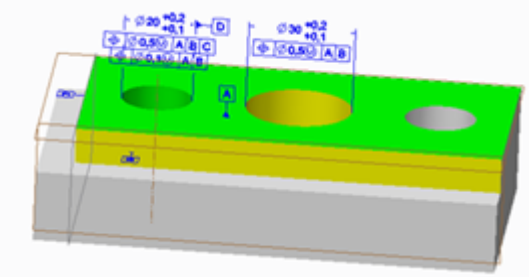
1.3 Thoughts on 2D to 3D transition

The goal of a manufacturing company, as well as its greatest challenge, is to make their products more functional and provide a better quality while at the same time minimizing their costs. New factors such as environmental impact also have to be considered.

Technical drawings are an important part of this process because they contain the information required for manufacturing and assembly. They describe real, three-dimensional products. Engineers are trained to read, interpret and understand standardized drawing formats.

Modern 3D design and development software makes it possible to integrate important information for manufacturing and documentation into the 3D model, for example, shape and position tolerances, material data or parts lists. Information such as tolerancing can be added to the 3D model directly, without even having to create a drawing.

In the future, it is likely that 2D drawings will be used less and less, but 3D information has to be read and interpreted by qualified professionals nonetheless. The real product can be represented most thoroughly and intuitively by a 3D model, as 2D drawings only contain derived information.



3D model with additional information

1.3.1 Advantages of 2D CAD

2D CAD systems have replaced hand-drafting and brought important improvements to drafting work. A 2D CAD system works with lines as the basic elements, with arcs, element variants and easily manipulable helper objects such as line types and layers, as well as functions such as trimming and object properties such as line color or line width. Computer-aided design has made it easier to create drawings, which were formerly created by hand with considerably more effort. In particular, it became much easier to make changes. This development caused a significant rise in work productivity.

These are some of the advantages of 2D CAD:

- easy editing or deleting of existing drawing elements
- automatic dimensioning
- automatic creation of elements with conditions such as parallel or tangential lines

– higher precision
– easy creation of and access to indirect dimensions
– copying and inserting of existing views or drawings
– creation of symbols and groups for repeated elements and patterns

1.3.2 Limits of 2D construction

Two-dimensional design is limited in that the designers have to put their focus on the drawings and on how to correctly represent a model in different view instead of being able to focus on construction features. The three-dimensional model of the finished product that is the base of the drawings is only present in the designers' imagination. Relationships to other models or other projections of the model do not exist in the data, so that it is hard to find any errors in drawing projection. It is also a major challenge to create exploded views, isometric views, detail views and sections because creating the scaling and details implies a lot of manual effort. However, such additional drawings are extremely important to help others understand and visualize the model.

A huge drawback of two-dimensional construction becomes apparent when a lot of drawings have been created and the geometry of the model has to be changed. Each drawing and each view have to be corrected and verified separately.

Two-dimensional drawings are not very useful in providing data for other product development steps. For example, they do not contain information on a model's mass and cannot be used for analyzing stresses or force distribution. NC data or parts list cannot be derived from the drawings.

1.3.3 The product development process

The product development process starts with a concept layout. Right from the start, it is important to identify and resolve any errors. The earlier and error is identified, the easier and cheaper it is to correct, errors found in the production phase being much costlier.

The more designers are involved in creating drawings and the more parameters are necessary for defining references, dimensions and other criteria, the more complex the design process becomes. In the next step, the entire set of drawings with details, different views etc. are verified by testers.

In a 2D layout, it is much more difficult to spot any problems than in a 3D model, as the spatial fit and proper functionality are harder to verify. When there are many moving parts, the potential for collisions is high. Even an experienced tester will need several days to check whether all parts are within tolerance and the assemblies do not have any fit and collision issues. If the tester finds fit or collision issues, they will hand the drawings back to the designers. As there is no automatically managed relation between different documents

or different drawing views, each error has to be corrected in each view, and every single document has to be re-tested and cleared anew. This can cost a lot of time, especially with large assemblies with several hundred parts.

When the drawings have been cleared, each new assembly has to be checked in prototype. Prototypes have to be produced, adding production time to the overall schedule, and assembled. If collision or tolerance errors are found in this phase, the design process starts again, and product development will take significantly longer.

When creating 2D drawings, the designers always have to consider how each part can be represented correctly in each view.

Over the whole of the product development process, the creation of 2D drawings creates a high number of constraints and additional work. The designer has to keep a three-dimensional model of the finished product in their head, creating different views in their imagination and then as a series of drawings. The drawings do not depend on a basic model that can make the work of the designer easier or help them spot mental projection errors. This is where 3D design shows its advantages. The three-dimensional mental model the designer has made can be directly translated into a three-dimensional virtual model.

When 2D CAD drawings are created, the resulting documents are not linked to each other in any way and can thus be easily versioned. However, any technical dependencies between the documents cannot be monitored, so the creator of the drawing has to manage their correctness without external aid.

1.3.4 2D to 3D transition

Digital modeling that uses 3D solids makes it easier for a designer to communicate their design intent to all persons involved, from the design department to the workshop and also to suppliers and customers. Multiple designers can work on the same virtual product and changes will be visible for each one of them.

Creo Parameteric offers a fully parametric 3D design environment. This means that if changes are made to one element, the entire geometry is updated. Also, changes to a part are propagated associatively to the related drawing, assembly and NC model. Looking at drawings specifically, this means that all drawing views will always reflect the current state of the design, no matter how many drawing documents have been created.

This automatic association between different pieces of data is managed using data references. Each assembly contains the names of the assembled parts and their placement constraints, not the part data itself. Also, a drawing contains the names of the represented models and the view definitions instead of the model data itself. When a user opens an assembly or a drawing, the objects are re-calculated in real time and the representation is up-to-date with any changes to the parts. This way of handling the data will only work if each part and each assembly is unique within the company. For example, a standard bolt is only represented by one virtual part, even if it is used thousands of times.

When a 3D solid model is created, even small assemblies imply a complex web of referenced and dependent models and files which ensures that all representations of the model are current. One of the challenges of 3D modeling is data management. On the file system level, only the latest version is stored. Clean versioning or parts usage control are only made possible by using a product data management system (PDM).

Another important advantage of 3D modeling is that there is often no need to produce a physical prototype. Visually intuitive assembly of the 3D part models helps identify issues with part shape, fit and function. Properties such as weight and center of gravity can be analyzed. Errors can be identified and changed in the virtual model.

A current and clearly defined virtual product efficiently supports other processes within the company. Such processes can be started in parallel to the design process based on the virtual model, e.g., production planning, marketing.

2D construction	**3D referenced model construction**
A three-dimensional product is represented by different drawing views. Technical fit between the drawings and the product, and between the different drawings, is only ensured by the three-dimensional model in each co-workers imagination. All changes have to be noted by all co-workers and incorporated into all drawings. This process is complex and error-prone. Even more complexity is added when not only product geometry is considered, but also parts lists, material data, tolerances, moving parts etc. + easily comprehensible drawing tasks - complex communication and change processes - slow and error-prone	A digital, virtual model forms the basis from which all documents (drawing views, parts lists) are derived. All derived documents are automatically kept current with the model. All product properties can be modeled 1. Geometry 2. Parts list 3. Tolerances 4. Material and mass 5. Surface and finish 6. Simulation (collisions, stress etc.) + always up-to-date + automatic changes to all linked documents - high training requirements - complex data mangement
Lines, symbols colors, layers etc. The information for one drawing is stored in one file.	A variety of different elements for different tasks (features, layers, parameters, material data, relations, simplified representations, ...) References = Dependencies The information for one drawing is stored in number of files from three to thousands.

	Challenge: Versioning and managing all object files with all dependencies
Focus: Derived information on a product	Focus: A three-dimensional virtual product

2 Startup TOOLS and operating environments

When a operating environment is created in the Caddepot by GENIUS TOOLS Environment Administrator, either from scratch or by migration, the following directory structure is built. This directory structure is syncrhonized to the Cadpool directory of all local application computers by GENIUS TOOLS Starter.

The table below describes the default directory structure.

Level 1	Level 2	Remarks
._Images		Contains an image (e.g. *<operating_env_name>.png;* 200x70 px) and an icon (*.ico file) for the operating environment
._Information		Contains an information document, the file name of which starts with "alert_" (e.g., *alert_<work_env_name>.pdf*)
.\apps		Contains additional applications
	\gtfc	GENIUS TOOLS for Creo: GENIUS TOOLS Parameter, Library, Quick Access, Dimensions, Forms etc.
	\ui	Freeware GENIUS TOOLS UI File Loader
.\configuration		Company-specific configuration files
	\application	All *protk_xxx.dat* files used in a company for additional toolkit applications
	\database	Contains the GENIUS TOOLS Starter database *sut.db*
	\gt_resource_folder	All configuration data for GENIUS TOOLS for Creo
	\plot	Company plot settings
	\projects	All GENIUS TOOLS Starter project directories
	\standard	Global settings in GENIUS TOOLS Starter
	\units	All GENIUS TOOLS Starter unit directories
	\users	All GENIUS TOOLS Starter user directories

Level 1	Level 2	Remarks
.\data		Data specific to a Creo version or to a client Start models, drawing frames, libraries, symbols, UDF, DTL files etc.
	\companyname	Company-specific data
.\help		Contains the manuals and installation instructions for GENIUS TOOLS for Creo, GENIUS TOOLS Starter and Startup TOOLS.
.\serveronly		Contains additional tools such as GENIUS TOOLS Comma-to-dot or GENIUS TOOLS Purge, as well as log files for errors in the subdirectory *_ErrorLog*. This directory is only in the Caddepot directory, i.e. not on users' computers.
.\software		Software GENIUS TOOLS Starter including *gts.exe* file
.\userdata		Contains directories for user-defined settings, e.g., mapkeys and user images. The subdirectory names correspond to the Login user names. (E.g: image file *userdata\%USERNAME%\%USERNAME%.png*, 100x130px)

3 Basic Creo guidelines

After installing Creo, you can start work immediately and begin to design. You will create parts, assemblies, drawings, mounting instructions, flattened sheet metal parts, NC data, shaded images and many other different types of data records. However, you will soon notice that you have to repeat some steps again and again, that data is being used in multiple contexts, and that the amount of files you create becomes difficult to manage.

Some examples for frequently repeated functions

- **Naming:** each part or assembly name has to be unique across all Creo workstations.
- Repeat **use of the same commands** in the Creo menus: You can create macros.
- Each new part or assembly needs **standard datums, views, parameters, layers** etc.: You can create start objects.
- Instead of continually looking up the **dimensions of bores and grooves, or connecting dimensions**, you can use user-defined features (UDFs).

Some examples for frequently re-used data

- drawing frames with parametric text fields
- DIN and other standard parts
- buy and catalog parts
- templates for parts lists
- material files
- tools for NC machining
- symbols for drawings
- frequently-used texts for drawings

New Creo users will hardly be able to use all the software's powerful features. You will only discover the utility of many functions over time. If you are working alone, you will get to know the extensive configuration options in Creo little by little as you find time to consult the manual.

3.1 Startup TOOLS operating environment

Imagine this situation: A colleague wants to use data you have created. Now, you are trying to figure out how to connect your workstations so that each data record remains unique within the network and you can use you colleague's data as well. At the same time, you know that a third workstation could be added soon, or a new server with enough storage capacity installed.

The Startup TOOLS software created by INNEO Solutions GmbH in 1997 is meant to free Creo users from such administrative considerations. Startup TOOLS come with a coordinated set of objects based on DIN standards, as well as a Creo environment that can be centrally managed and is easy to expand. Company-specific adaptations and additions can be made based with the help of this document.

Startup TOOLS are based on the long-standing experience of INNEO Solutions GmbH with hundreds of Creo users. The general Creo guidelines can be seen as a basis for users to start working with. However, each company will develop specific procedures and guidelines over time. In order to ensure that multiple users can work on a clean set of unique, redundancy-free data records, Startup TOOLS should be used according to a number of basic principles that can be adapted to company-specific requirements and include guidelines for the following aspects

- installation
- data management
- naming
- use of start objects
- use of parameters
- use of layers
- model simplification
- plotter configuration

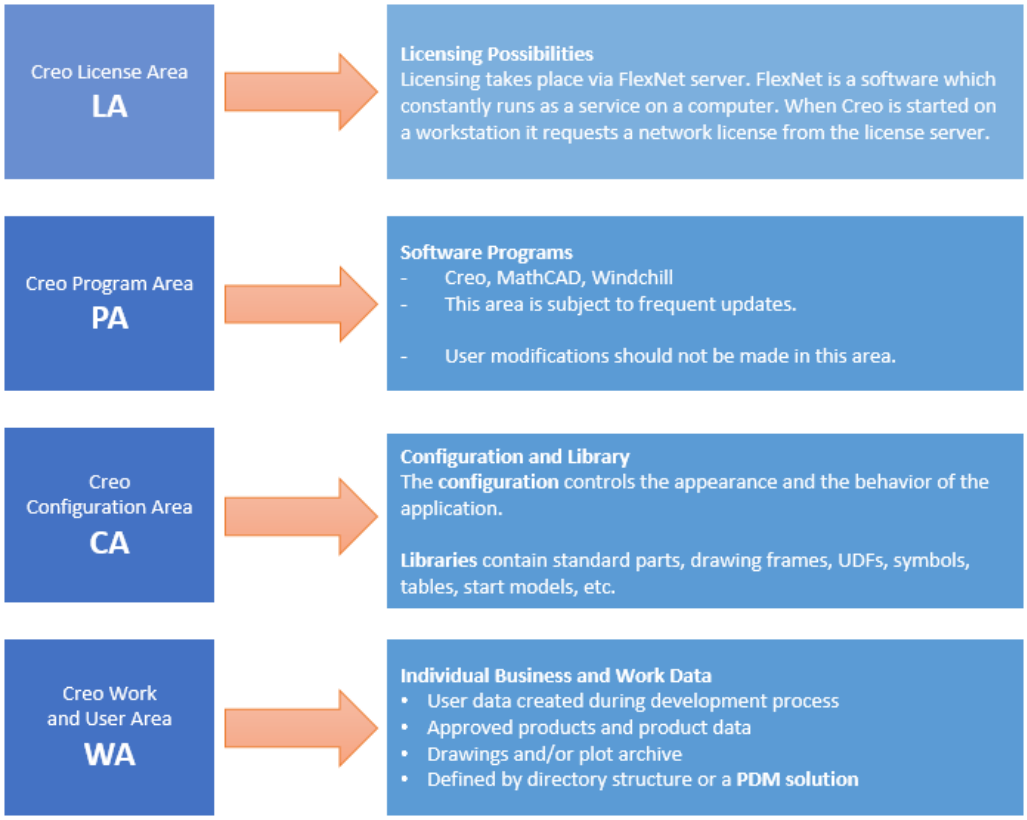
In summary, Startup TOOLS come with a set of objects and are based on a set of concepts. To make good use of the objects, it is best to first understand the concepts behind them.

Please note: Remember that the most basic rule for reference-based 3D development is that file names have to be unique. Define your file naming convention as the first step. It is practically impossible to rename objects later (unless you are using Windchill).

This manual describes the basic concepts behind Startup TOOLS. All document templates provided give the default settings for a standard installation of the Startup TOOLS.

3.2 Installation concept

Successful use of Creo is based on a flexible installation which can be extended quickly and easily to cover additional workstations, but remains centrally managed. Each Creo environment consists of four basic areas as described in the following figure.



Components of a Creo installation

3.2.1 License area (LA)

Different software companies use different strategies to protect their software from unauthorized access, such as dongles, hardware-based license numbers, or license management tools. One such license management tool is FlexNet by Flexera. FlexNet can run on different platforms and manage licenses for different software products. PTC uses FlexNet to manage Creo network licenses. Each basic license is considered as a floating license, that is, the license is freely available within the network. Other Creo modules such as extended assembly management, NC machining or freeform surfaces, can only be used together with a basic license and are called extensions.

Example license setup: A company owns four basic licenses and one NC machining extension. One of the network computers is the FlexNet license server. Any four workstations within the network can use Creo concurrently. One of these workstations can also use the NC machining extension. The computers using the licenses can change at any time. The FlexNet license server monitors that only four basic licenses and only one NC machining extension are in use at the same time.

Starting with version 6.0, Startup TOOLS also uses FlexNet to manage licenses for GENIUS TOOLS Starter and GENIUS TOOLS for Creo.

3.2.2 Program area (PA)

The installation directory contains further subdirectories with program-specific files. The users do not need to make any changes to this directory structure. If the directory structure remains unchanged, a new software version can simply be copied into the old installation directory if the software is updated. Also, the software can be uninstalled without any fear of data loss.

The Creo installation should be made on each workstation.

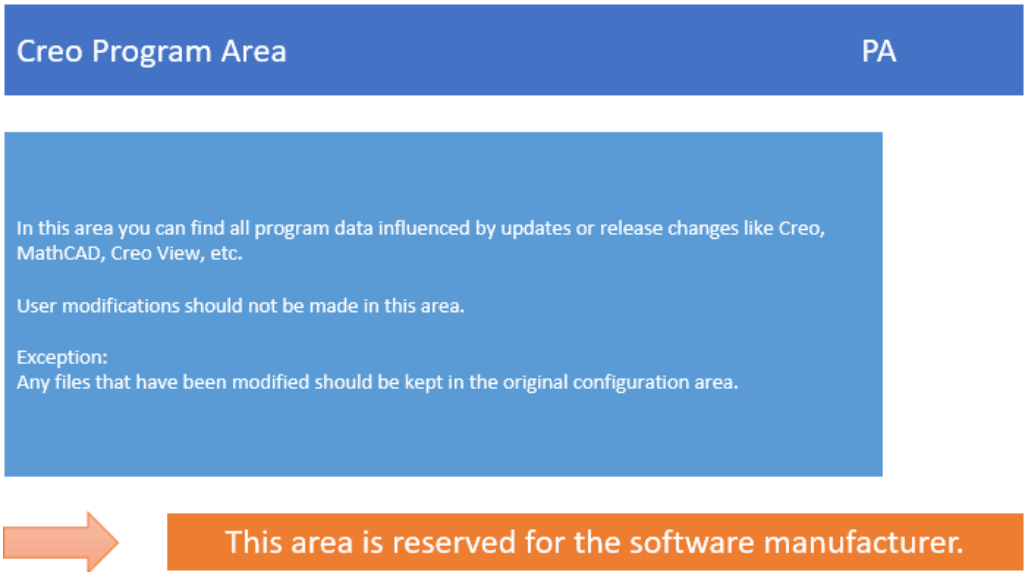
For example: *C:\ptc\creo6*

As Creo is a large application that causes a lot of hard disk activity, it is highly recommended to install Creo locally on each computer.

Unfortunately, PTC does not fully adhere to the philosophy of separating the software from the configuration. Important configuration files are located in the program area directory structure, so that many configuration solutions, including GENIUS TOOLS Starter, require write access for the current Windows user to various subdirectories for setting configuration options.

Write access is required in

- *<creoinstalldir>\Common Files\text* (*config.pro*, *config.sup*, *creo_parametric_admin_customization.ui*)
- *<creoinstalldir>\Parametric\bin* (PSF files)



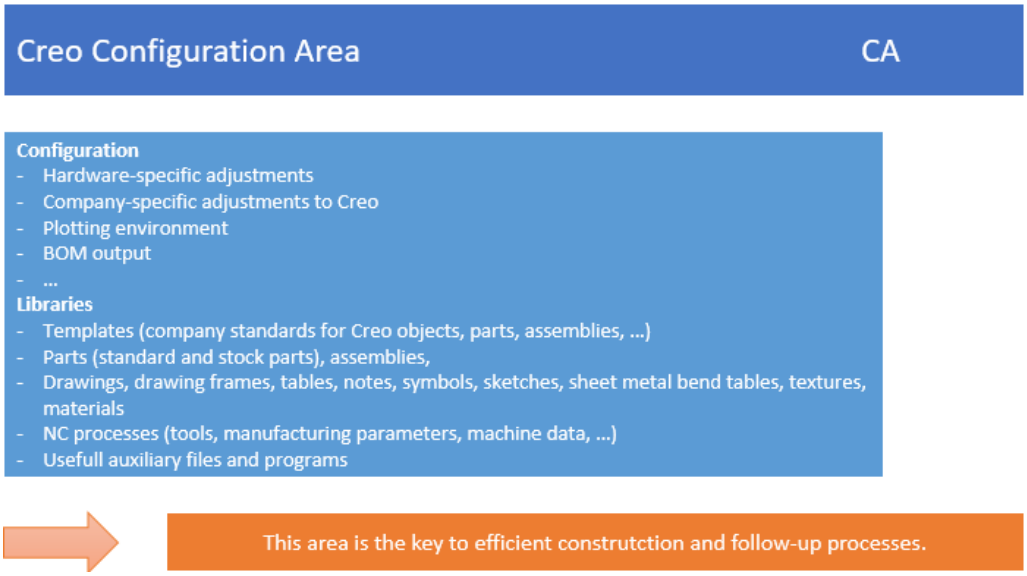
In a Startup TOOLS operating environment, the program area is located in *\software*.

3.2.3 Configuration area (CA)

Use the configuration area to tune your Creo environment. From a Startup TOOLS perspective, the configuration area corresponds to an operating environment.

The master configuration area should only exist once within the company (across all locations): in the operating environment in the Caddepot directory.

In order for user to have access to the same configurations, the operating environment is synchronized from the Caddepot to each client computer by GENIUS TOOLS Starter.



3.2.4 Work and user area (WA)

Design results such as parts, assemblies and drawings make up the added value of a company and form one of its central assets. It is highly recommended to store such data centrally and configure a daily backup. To prevent the connection between a central data server and the workstations from becoming the system's bottleneck, the network speed should be ≥ 1 Gbit/s and the data server should have a fast RAID storage system. The hardware requirements are determined by the number of workstations and the amount of Creo data (number of parts and complexity of products).

The work and user area consists of three parts:

The Home directory: Each user should be routed to this directory when they open Creo. The home directory can be seen as a sort of sandbox. Files stored in the home directory can be used for practice or tests.

The project work space: This is the storage location for current projects. For complex products, you should divide the data into subdirectories according to the assembly hierarchy. If you want to work on project data, first change your Creo working directory

from the home directory to one of the directories of the project via *File* > *Manage session* > *Set working directory*.

Finished data: This storage location is organized in the same way as the project work space, but only contains objects cleared for production. This means that parts, assemblies and drawings will be opened in read-only mode. Typically, moving objects from the project work space to the finished data area is not done via Creo, but on the file system level by an authorized user.

Warning: Never copy a Creo object into another directory under the same name. Never rename a Creo object on the file system level.

Each part, each assembly and each drawing has a file name. In order to open an assembly file, you also need the part files for the components. To open a drawing file, you also need the files for the parts and assemblies represented in the drawing.

You must never change the name of a part or assembly that is already in use.

You must not create part files with the same name in different directories.

Only if you follow these rules, it becomes possible to introduce a PDM system later on, or at least the introduction process will be much smoother. If you can, work with a product data management system (PDM) right from the start.

If you are looking for an easy way to make data accessible to all company staff, consider storing the data in a digital archive in a universal format, such as PDF or TIFF for drawings, 3D PDF for parts and assemblies. The data can then be read or printed on any PC. Creo data can also be viewed on any PC using the free software tool *PTC Creo View Express*, including parts, assemblies and drawings.

Tip: Creo files have a complex net of references binding them to each other. Managing Creo files on the file system level is only reasonably feasible up to a certain point. Consider using a product data management system, for example a Windchill solution.

Creo Work and User Area WA

Individual corporate and user data

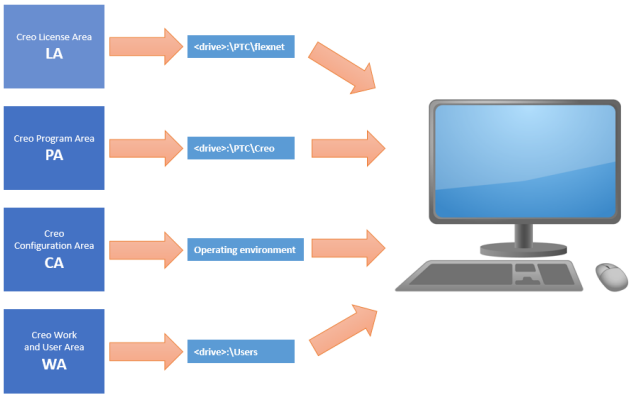
User data and information
- Personal workspace
- Files currently under development

Approved product data files
- In the original CAD format
- In universal formats (PDF, TIFF, STEP, HPGL, VRML, etc.)
- In deliverable formats (IGES, STEP, VDA, JT, etc.)

Data management. Large amounts of data should be managed in a PDM system.

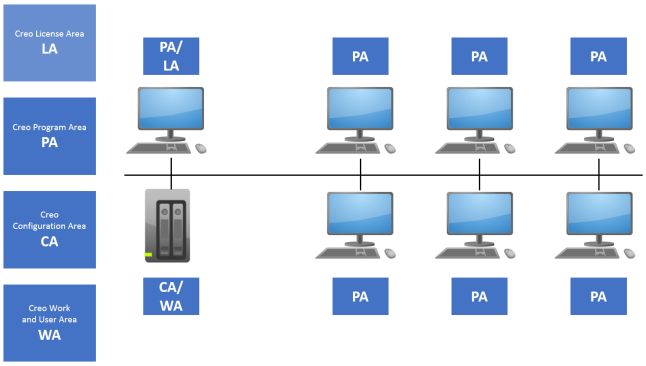
3.2.5 Network areas

If you are working on an individual computer, all required storage areas need to be located on this computer.



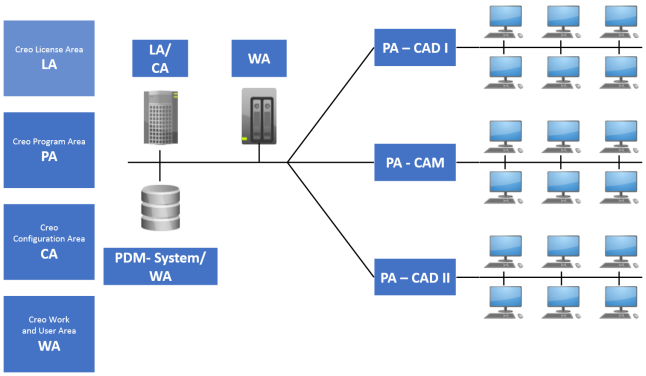
Areas on an individual workstation

The setup in a larger network could look like in the figure below.



Areas in the network

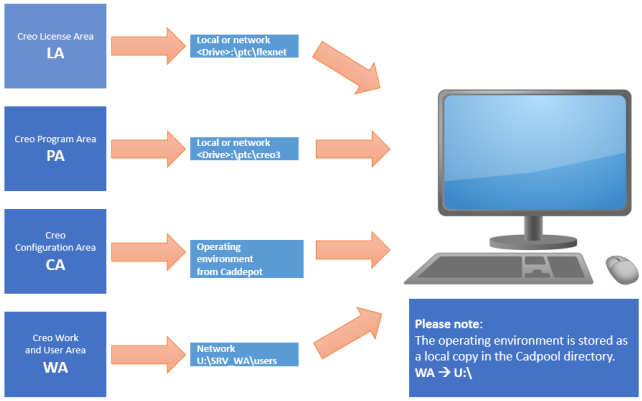
More complex structures are also supported.



Areas in the network

Independently of the number of workstation that use Creo, each of the workstations needs to have access to the LA, PA, CA and WA. The license area does not have to be installed or copied to each computer. The license server just needs to be known when installing Creo. The configuration and work areas contain data that each user needs to access. These areas should only exist once as a master copy in the company network.

As a result, each workstation has the same setup:



Directory structure on a workstation

3.3 Configuration concept

For information on the configuration concept, please refer to the documentation for the GENIUS TOOLS Starter configuration tool.

3.4 File storage concept

The information in this section is based on the information given under Work and User Area (WA) 16. The directory structure for each project depends to a large extent on the type of product under development and on the company's organizational structure. As a general rule, do not save more than 300 to 400 files in one directory.

Standard parts, buy parts, catalog parts and repeat parts have to be considered as special cases in the file storage concept. Creo comes with an easy-to-use search function for such

parts using lists of properties. However, the parts have to be managed in a library structure. The Startup TOOLS library structure is stored in the directory defined in the configuration option `pro_library_dir`. To integrate new parts into the library structure, you have to move them into the corresponding directory structure and initialize them using the MNU files there. If you create new library directories, you have to include them in the *search.pro* file. The library structure is extremely important for working efficiently in Creo.

Please note: Never copy data between directories. You can move objects. When moving objects, take care to move only the latest version and delete any old versions.

Only a product data management system (such as Windchill) can ensure consistent data management in the long term and manage successive versions or where-used lists for objects. On the file system level, only the latest, current version of the objects is present.

Preliminary considerations for the file storage concept

– analysis of products (structure and amount of data)

– analysis of construction tasks (special design or serial production etc.)

– analysis of organization and workflow (number of colleagues in the same project etc.)

– analysis of upstream and downstream interfaces

Document templates for guidelines

– Document template: Construction file storage

– Document template: Libraries

Use of the file storage concept

The file storage concept is used during everyday Creo work.

3.5 Naming concept

One of the most frequent steps when working with Creo is naming a part, assembly, or drawing. Creating unique names is a key to using Creo efficiently. The basic Creo philosophy of unique data records demands that each part and each assembly can only be present once within a network (company-wide).

Please note: Each part, each assembly and each drawing has a file name. In order to open an assembly file, you also need the part files for the components. To open a drawing file, you also need the files for the parts and assemblies represented in the drawing.

The system of references in Creo means that you should absolutely never change the name of a part or assembly that is already in use.

Also, you must not create part files with the same name in different directories.

Drawings automatically get the name of the part or assembly that they represent. A part and an assembly can have the same name in theory, but this should not be done as the drawings for both would get the same name as well.

There are two key requirements for file names:

1. overall uniqueness
2. readability for the users

File names that are not unique within the company should only be accepted if names are managed externally, e.g., by different clients.

File names that are not readable often occur when Windchill is used or the ERP part number determines the file name.

There are different ways to ensure company-wide uniqueness of names.

In companies that work with an ERP system, items in the ERP have a unique ID that can also be used as a name for parts and assemblies. However, this may not be feasible in the design phase, when there are no ERP items yet that the designer can refer to. The designer can only use ERP IDs for names if they can retain numbers in the ERP in real time.

Another way to achieve uniqueness is assigning a sequence of construction numbers. The construction number could start with the designers initials so that each person has their own sequence of numbers.

Independently of which numbering system you want to use, you should always add a descriptive name as well, for example *56645_hood* or *KUH57871_shaft*.

Tip: You can use GENIUS TOOLS Name Generator to create unique numbers and add descriptive names in a consistent pattern.

You can also add more information to unique names by using classification codes or drawing numbers. Codes can convey information on the individual part or on its place or function in the product.

Information on the individual part can come in different categories, e.g. function, sheet metal or cast part, material, color etc. Information on the part's relation to the product could be the product name, hierarchy level in the assembly, function, or assembly number. Names including classification codes can look like *SHM_ST_34_873A_29* or *FKH123_001_003_32*.

Please note: File names cannot contain special characters or umlauts. Only use letters, numbers and underscore characters.

In any case, try to use the naming convention already in place in your company, or only adapt it slightly for use in Creo.

When working with Creo, there are a number of special cases that need to be considered in the naming concept such as draft objects, flattened sheet metal parts, joined or welded assemblies.

Use the Creo flexibility functions for parts that come in different shape states (e.g., springs), and for assemblies that should be represented in different states (e.g., cylinder positions).

Please note: The company-specific naming concept has to be written down and available to all colleagues at any time.

Buy parts, foreign library parts or other files coming in from different companies have to be brought in line with the naming concept before use. You can also use GENIUS TOOLS Name Generator to re-name existing models.

Preliminary considerations for the naming concept

– How do you guarantee unique file names?

– How do you ensure file name readability?

– analysis of existing keys (ERP ID, classification number, drawing number etc.)

Document template for guidelines

– Document template: File naming

Use of the naming concept

The naming concept is used whenever Creo objects such as parts, assemblies, drawings, manufacturing files, mounting drawings, flattened sheet metal etc. are named.

3.6 Parameter concept

One of the strongest advantages of managing unique data records in Creo lies in the fact that not only geometry data but also administrative information is unique. This means that information that appears in drawing headers, parts lists or BOM balloons in drawings of assemblies has to be created and updated in only one place.

Administrative parameters in Creo are variables or parameters whose values are stored in the part or in the assembly. Such parameters can contain, for example, the name and description, material, mass, or drawing number.

When a drawing is created for a part or an assembly, the drawing header can be filled in automatically. When parts are assembled, an up-to-date parts list for the assembly is

available with a mouse click. The format for parts lists or bills or materials can be freely configured (number of columns and column content). BOM balloons with any piece of information from the parts list can be created automatically for each component. All objects are connected associatively.

You can use GENIUS TOOLS Parameter to create and fill in parameter forms according to a company-specific definition.

Warning: In Creo 3.0, a new parameter concept has been introduced that makes it easier to use Startup TOOLS in international companies with multiple UI languages. The objects affected by this change (files, folders) are marked with the suffix *_int_de*.
It is not possible to automatically migrate an old operating environment to the new concept. If you want to use the updated functionality, consider using Model Processor for migrating your data. Please consult with INNEO Solutions GmbH.

Preliminary considerations for the parameter concept

– analysis of existing text fields

– analysis of parts list formats

– analysis of construction-relevant fields in the ERP

Document template for guidelines

– Document template: Parameters

Use of the parameter concept

The parameter concept is used in the start objects, for standard and buy parts, in drawing headers, in all types of parts lists and bills of material, BOM balloons and parametric drawing texts, as well as when defining your parameter definitions in the parameter manager.

3.7 Layer concept

Layers are an established concept in 2D systems. 2D systems use layers for dimensioning, crosshatching, line types, parts, helper elements etc. In Creo, the main application for layers is not in drawings, but in working with parts and assemblies. Layers are used for showing or hiding sets of objects to have a clear and easy representation for different types of tasks.

There are two basic ways to create layers and assign objects to them.

Manual layer creation

Use the Creo layer dialog to create new layers and name them.

After a new layer has been created, you can assign objects to the layer, e.g., datum planes, curves or points, remove or change assigned objects. Objects on a layer can be shown or hidden. Each layer can have rules that define which types of objects should automatically be assigned to it.

Automated layer creation and object assignment

Creo can be configured to use layer templates (*config.pro* settings), which means that layers will be created automatically and assigned certain types of objects. For example, if you create the first thread in your model, a layer *Thread* will be created that contains the newly created element. All threads created later will also be assigned to that layer. Showing or hiding layers still is a manual step.

Use the configuration option `default_layer_model` to specify a model name (for parts, assemblies and drawings) for a model that contains rule-based layers. These will be imported into a new model if you apply the template.

In contrast to 2D systems, Creo cannot hide an individual geometry from a part, but only complete parts when you are in assembly mode. For example, if all bore features are placed on the layer *Bores*, then the layer is hidden, only the axes of the bores will be hidden, while the bores themselves will still be shown. Individual geometry elements such as bores or rounds can only be removed from the visual representation with the Creo functions *suppress* and *resume*. This functionality can be combined with layers.

For example, the layer *Details* could contain small elements of a part. If you apply the suppress function to the layer, all elements of the layer will be suppressed in one step. If you apply the resume function to the layer, all elements will be resumed in one step.

This functionality is often used for standard parts. Typically, components are added to an assembly with the minimum required geometry. If a detailed representation is required, for example, for documentation, additional elements can be shown using the resume function. All elements of a layer are selected if you select a layer, then click *Select elements* in the context menu.

The function *isolate* will show only objects that are placed on such selected layers.

By grouping layers, you can show or hide more than one layer in one step.

Preliminary considerations for the layer concept

– automated layer creation via *config.pro*

– construction layers for parts, assemblies and drawings with their names and meaning

– model to use with the configuration option `default_layer_model`

– grouping of layers

– rule-based layers in start objects

Document template for guidelines

– Document template: Layers

Use of the layer concept

The layer concept manifests in the start objects and the layer template models, and is used for standard parts and buy parts.

Warning: Do not create an overly complex layer concept. Also, users should always know where to find information on the meaning and use of layers.

3.8 Simplification concept

With current desktop computer technology, it is not possible to achieve satisfactory performance when rendering products that consist of many thousand parts and sub-assemblies or of may very complex parts in all detail. Technological development is very fast, and an up-to-date computer – Intel Core i7/32 GB RAM/high-end graphics – can work with more than 500 000 parts at a time without problems. With 64bit operating systems, working memory is not an issue anymore.

However, it is not always necessary or expedient to call up all geometry and design information at the same time. A surfeit of information with surfaces and lines for 10 000 parts on a 24-inch screen hinders the designer's work. In traditional 2D drawings, no single drawing would hold all information for all levels of detail at the same time, either.

In principle, Creo lets you represent a complex product in all detail, there are no restrictions built into the software for model size or number of parts. There are several simplification strategies to always provide the proper depth of information and not too much.

Basically, there are three types of simplification as described below.

Part simplification

Parts can be simplified by only displaying a defined set of features, surfaces and zones. The simplified part is defined by a separate name.

When you make changes to the original part, the simplification is adapted automatically. Simplified parts can in turn be used in simplified views of assemblies.

This strategy can be used, for example, in representing complex cast parts such as cylinder blocks or machine beds.

Assembly simplification

A simplified assembly is also defined by a separate simplification name. Creo provides support for selecting parts to include in the simplification by different criteria. Parts or assemblies can be replaced by their simplifications. There is a wide range of functionality that works with simplified assemblies, such as envelopes, associative selection rules, interchange assemblies, dynamic re-loading of components, reference manager, skeleton models etc.

Drawing simplification

A simplified drawing is also defined by a separate simplification name. You can use rules to define the visibility of different drawing views on different sheets of a drawing.

How to use simplification

– Immediately upon opening a model or drawing to load only the required information

– Simplified parts can be used in simplified assemblies

– You can create drawings of simplified parts or assemblies

Simplification is especially relevant in a top-down design workflow. In top-down design, the main draft is prepared first, consisting of the main assembly with skeleton or layout information (AAX license required), then an assembly structure and drafts of the main components of the product. The most important tool in this design phase is the construction manager.

In the next step, the simplified assemblies are created. It is important to use consistent naming conventions. For example, you could define simplified representations in the start assemblies, e.g. the simplification *Skeleton* which will always load skeleton drafts only, or the simplification *Standard* which is empty by default and only contains the components that should be visible by default in the next higher level of representation.

Then, the work can easily be split to several designers who add increasing layers of detail to their assigned assemblies and their parts. The representation of the overall product will be created automatically in this process. Also, there are some default simplifications that are always available in Creo, e.g., graphics representation and geometry representation.

Example: Four designers work on a truck
The first step is to create the assembly that will later contain the entire truck. Then, empty sub-assemblies are created for the main construction blocks (vehicle frame, cabin, drivetrain, body). A skeleton draft for each main construction block is created in the overall assembly. This skeleton draft contains relevant characteristics such as dimension bounds, installation space etc., and interface information. Afterwards, the constraints from the overall assembly are taken up in the assemblies for the main sub-assemblies using

publish and copy geometries (AAX license required), and further constraints created in the skeleton drafts of the main sub-assemblies.

In the next step, simplified representations are created in the overall assembly and its main sub-assemblies, and the simplified sub-assemblies added to the overall assembly. Then, each designer can work on their assigned sub-assembly, create further sub-assemblies with skeleton drafts as required and design individual parts. Each main sub-assembly is designed in more and more detail, while simplified representations ensure that the overall truck can always be represented as well.

In this way, Creo provides easy access to the state of the overall construction as well as to the overall parts list at any time. It is also possible at any moment to check whether the individual designers' work fits together and all constraints are met.

This functionality is unique in the field of CAD software. Its advantages can only be fully realized if all company guidelines are met consistently, the philosophy of top-down design is clear to everyone involved, and all designers are proficient Creo users.

Preliminary considerations for the simplification concept

– What is the basic structure of the finished products?

– What are the main constraints (overall draft and main references)?

– Which persons will work on which sub-assemblies?

– Which components are subject to frequent changes?

Document template for guidelines

– Document template: Simplifications

Use of the simplification concept

– in large assemblies

– in top-down design workflows

– for complex parts

– when several persons work on the same product

3.9 Start object concept

Start objects are ordinary parts, assemblies, sheet metal parts, drawings etc. that come with defined settings. Startup TOOLS include a selection of start objects in their libraries. The following *config.pro* settings are especially important for start objects and are set to the values given below in Startup TOOLS:

start_model_dir	$GTS_DATA\library_dir\start_model_dir

template_solidpart $GTS_DATA\library_dir\start_model_dir\sut_int_de_start_prt.prt

template_sheetmetalpart
$GTS_DATA\library_dir\start_model_dir\sut_int_de_start_smt.prt

template_designasm
$GTS_DATA\library_dir\start_model_dir\sut_int_de_start_asm.asm

template_drawing
$GTS_DATA\library_dir\start_model_dir\sut_int_de_start_drw.drw

Please note: Each new part, assembly and drawing should be based on a start object.

What are start objects good for?

Company standards are automatically included in each new part and assembly.

– units

– tolerances

– basic features

– views

– parameters

– relations

– layers

– simplifications

– material

Start objects help save time as basic settings for objects do not have to be created again and again. Start objects can and should also be used when creating components in an assembly.

Preliminary considerations for the start object concept

– Use the document templates

– Document template: Parameters

– Document template: Layers

– Document template: Simplifications

Document template for guidelines

– Document template: Start objects

Use of the start object concept

When creating each new part, assembly or drawing.

3.9.1 Creating new start objects

To create a new start model, create the required Creo model, a part, for example, and copy it to the start model directory.

Example
Create a part with the features, parameters, layers, views etc. that you want to incorporate. Name it, for example, *company_start*.

Copy the part *company_start.prt* to the directory *$GTS_DATA\library_dir\start_model_dir*

Change the *config_*_pro* in the project directory to include the start object
`template_solidpart $GTS_DATA\library_dir\start_model_dir\company_start.prt`

Please note: You need write access to the above-mentioned directories.

3.10 Plot concept

There are two basic approaches to plotting from Creo:

– using Creo printer or plotter drivers

– using Windows printer or plotter drivers

Creo drivers		Windows drivers	
Advantages	Disadvantages	Advantages	Disadvantage
tried-and-tested drivers, often available for older models as well	limited selection of hardware for which PTC provides a driver	any hardware for which there is a Windows driver can be used	plot method is new to Creo, some options always have to be set manually

Plotter settings are independent of the driver used. Plotter settings are defined in PCF and PNT files that are typically placed as specified in the configuration option `pro_plot_config_dir`.

If there are no plotter setting files, parameters such as paper size, printed section or scaling have to be set manually when plotting. If PCF files are used, combinations of plotter settings can be selected from a drop-down menu.

Preliminary considerations for the plotting concept

– Use the document templates

– Document template: Parameters

– Document template: Layers

– Document template: Simplifications

Document template for guidelines

– Document template: Plotter settings

Use of the plotting concept

When outputting drawings.

3.10.1 Plotter configuration files (PCF files)

You can specify all plotter settings, or only some of them, in a plotter configuration file. The name of the PCF file, or the name specified in the PCF file under `button_name` is displayed in the drop-dopwn list *Plotter* in the *Print* (or *Settings*) dialog.

The Creo manual contains a full list of settings that can be defined in a PCF file. Some of these settings can also be made in the *config.pro* configuration file. To avoid confusion, it is recommended to only specify `pro_plot_config_dir (<caddepot>\configuration\plot)` or `quick_print_plotter_config_file` in the *config.pro* file.

3.10.2 Pen assignment files (PNT files)

When you plot an element from Creo, a pen is assigned based on the default system color for the selected element type. The system will use the line width and type assigned to a certain color.

Example: The system uses pen 1 for elements that are represented in white in Creo and use the same line width. If your plotter supports 8 pens and you want to use all pens for plotting, set the configuration file option `use_8_plotter_pens` to *Yes*.

Pen assignment is calculated internally even for inkjet plotters or laser printers. However, you can directly influence line widths and colors.

Example: You want to pring an A1 drawing on an A4 laser printer. Line widths need to be reduced significantly. The PNT file could look like this:

```
!============================
!= Line widths for scaling down =
!============================
```

```
pen 1 thickness 0.025 cm; color 0.0 0.0 0.0; drawing_color
pen 2 thickness 0.01 cm; color 0.0 0.0 0.0; edge_highlite_color
pen 3 thickness 0.01 cm; color 0.0 0.0 0.0; half_tone_color
pen 4 thickness 0.01 cm; color 0.0 0.0 0.0; highlite_color
pen 5 thickness 0.01 cm; color 0.0 0.0 0.0; letter_color
pen 6 thickness 0.01 cm; color 0.0 0.0 0.0; attention_color
pen 7 thickness 0.01 cm; color 0.0 0.0 0.0; section_color
pen 8 thickness 0.01 cm; color 0.0 0.0 0.0; dimmed_color
```

The Creo manual contains a full list of settings that can be defined in a PNT file.

3.10.3 Printer/plotter drivers from Creo

Printers for Creo drawings need to be able to process Postscript, HPGL or the Calcomp format. Plotters and printers supported by Creo are listed in the hardware configuration or can be viewed under *Plotters* > *Other plotters*.

You have to configure plotters and printers in Windows or make them available via the network before Creo can use them. If a printer is installed under Windows, it can be configured for Creo using PCF configuration files. PCF files look similar to the following example.

```
!============================
! Configuration single-sheet plotter
!============================
plotter DESIGNJET750C
plotter_command print /d:\\computername\printername
allow_file_naming yes
button_name "Single-Sheet DJet450C bw.
button_help "For DesignJet750C
delete_after_plotting no
interface_quality 3
pen_table_file p:\config\plot\a4_la4.pnt
plot_access create
plot_drawing_format yes
plot_names yes
plot_roll_media no
plot_segmented no
plot_sheets current
plot_translate 1 0.5
```

The *config.pro* configuration file, under `pro_plot_config_dir`, points to the directory in which to look for PCF files.

Create a new file and enter the settings for the new printer or setup. The printer or plotter has to be mentioned in the list of printers cleared by PTC and entered in the PCF file under the same name as in that list.

If you want to print to a Postscript file, configure the plotter *Postscript*. You can also use other file formats to print to such as TIFF, JPG or CGM. The entry *PLOTTER_COMMAND* corresponds to the print command you would use when sending the file via the DOS command prompt.

Example for a network plotter: `PLOTTER_COMMAND print /d:\\"Server_name"\"Share_name"`

The name of the PCF file will appear as an available printer or plotter in the *Print* menu.

3.10.4 Printer/plotter drivers from Windows

If you want to use Windows drivers for plotters or printers, the PCF file has to have the following entry:

```
plotter ms_print_mgr
```

Define your plot in Creo (whole drawing or section, segmented, scaled, pen assignment file etc.), then select the output device from the Windows pring manager.

If you want to plot drawings to A4 laser or inkjet printers, some additional settings are required.

Whole drawing to A4 using Windows print manager

When you output to a laser or inkjet printer, there is no way to print a properly standardized DIN A format. There are two workarounds.

1. Scale the drawings (for example, 0,931 for your laser printer; 0,9 for your inkjet printer).
2. Scale the drawing frames so that they do not exactly meet the DIN format anymore. If you often output A4 and A3 formats to A3 printers, this is the preferred solution as the drawing itself is not scaled and can be used for measuring. By default, the Windows print manager is set to landscape orientation, which can be used for all Creo landscape formats. If you want to print A4 in portrait orientation, remember to manually set the orientation in Windows print manager. If larger formats are sent to a printer, use scaling (laser printer 0,931; inkjet printer 0,9). Scaling can be defined via a PCF file. Also remember to reduce the line widths in scaled-down drawings, or the drawings will not be readable.

Configuring line widths

If you output a drawing 1:1 (e.g., A4 to A4 or A3 to A3), configure the correct line widths (*a4_la4.pnt*). For better readability, set the color to black for all element types, otherwise, some lines will appear light gray.

a4_la4.pnt

```
pen 1 thickness 0.05 cm; color 0.0 0.0 0.0; drawing_color
pen 2 thickness 0.025 cm; color 0.0 0.0 0.0; edge_highlite_color
pen 3 thickness 0.025 cm; color 0.0 0.0 0.0; half_tone_color
pen 4 thickness 0.035 cm; color 0.0 0.0 0.0; highlite_color
pen 5 thickness 0.025 cm; color 0.0 0.0 0.0; letter_color
pen 6 thickness 0.025 cm; color 0.0 0.0 0.0; attention_color
pen 7 thickness 0.025 cm; color 0.0 0.0 0.0; section_color
```

```
pen 8 thickness 0.025 cm; color 0.0 0.0 0.0; dimmed_color
```

If you output drawings scaled down, adapt the line widths as given in *a3_a2_a1_a0_la4.pnt*.

a3_a2_a1_a0_la4.pnt

```
pen 1 thickness 0.025 cm; color 0.0 0.0 0.0; drawing_color
pen 2 thickness 0.01 cm; color 0.0 0.0 0.0; edge_highlite_color
pen 3 thickness 0.01 cm; color 0.0 0.0 0.0; half_tone_color
pen 4 thickness 0.01 cm; color 0.0 0.0 0.0; highlite_color
pen 5 thickness 0.01 cm; color 0.0 0.0 0.0; letter_color
pen 6 thickness 0.01 cm; color 0.0 0.0 0.0; attention_color
pen 7 thickness 0.01 cm; color 0.0 0.0 0.0; section_color
pen 8 thickness 0.01 cm; color 0.0 0.0 0.0; dimmed_color
```

Plotting A4 landscape to A4

In the *Print* menu, select *A4 to LaserA4*.

Click *OK* in the menu, then *OK* in the Windows print manager.

PCF file: *a4_laser_a4.pcf*

PNT file: *a4_la4.pnt*

If a scaled-down drawing format is used, comment out the entry `plot_scale` in the file *a4_laser_a4.pcf*.

a4_laser_a4.pcf

```
!===================================
!= Plotter configuration for Windows print manager =
!= Format A4 to laser printer A4 =
!===================================
plotter ms_print_mgr
button_name A4 to LaserA4
button_help Whole drawing to A4
interface_quality 3
delete_after_plotting no
plot_file_dir c:\temp
plot_names yes
pen_table_file p:\config\plot\a4_la4.pnt
plot_drawing_format yes
plot_scale plot 0.931
```

Plotting A4 portrait to A4

In the *Print* menu, select *A4 to LaserA4*.

Click *OK* in the menu, then select portrait orientation > *OK* in the Windows print manager.

PCF file: *a4_laser_a4.pcf*

PNT file: *a4_la4.pnt*

If a scaled-down drawing format is used, comment out the entry `plot_scale` in the file *a4_laser_a4.pcf*.

Plotting A3 to A4

In the *Print* menu, select *A3 A2 A1 A0 to Laser A4*.

Click *OK* in the menu, then *OK* in the Windows print manager.

PCF file: *a3_a2_a1_a0_laser_a4.pcf*

PNT file: *a3_a2_a1_a0_la4.pnt*

a3_a2_a1_a0_laser_a4.pcf

```
!==================================
!= Plotter configuration for Windows print manager =
!= Formats A3 A2 A1 A0 to laser printer A4 =
!==================================
plotter ms_print_mgr
button_name A3 A2 A1 A0 to LaserA4
button_help Whole drawing to A4
interface_quality 3
delete_after_plotting no
paper_size a4
plot_file_dir c:\temp
plot_names yes
pen_table_file p:\config\plot\a3_a2_a1_a0_la4.pnt
plot_drawing_format yes
plot_scale plot 0.931
```

Plotting A2 to A4

In the *Print* menu, select *A3 A2 A1 A0 to Laser A4*.

Click *OK* in the menu, then *OK* in the Windows print manager.

PCF file: *a3_a2_a1_a0_laser_a4.pcf*

PNT file: *a3_a2_a1_a0_la4.pnt*

Plotting A1 to A4

In the *Print* menu, select *A3 A2 A1 A0 to Laser A4*.

Click *OK* in the menu, then *OK* in the Windows print manager.

PCF file: *a3_a2_a1_a0_laser_a4.pcf*

PNT file: *a3_a2_a1_a0_la4.pnt*

Plotting A0 to A4

In the *Print* menu, select *A3 A2 A1 A0 to Laser A4*.

Click *OK* in the menu, then *OK* in the Windows print manager.

PCF file: *a3_a2_a1_a0_laser_a4.pcf*

PNT file: *a3_a2_a1_a0_la4.pnt*

Plotting sections of drawings

It is a common requirement to print one view or some part of a large drawing in A0, A1 or A2 to an A4 format without scaling. To do so, proceed as follows.

In the *Print* menu, select *Section to A4*

Click *OK* in the menu, then select the corner points of the required section, *OK* in the Windows print manager.

PCF file: *ausschnitt_auf_a4.pcf*

PNT file: *a4_la4.pnt*

ausschnitt_auf_a4.pcf

```
!==================================
!= Plotter configuration for Windows print manager =
!= Section to A4
!==================================
plotter MS_PRINT_MGR
button_name Section to A4
button_help Define section to print to A4
plot_drawing_format YES default
plot_segmented NO default
plot_roll_media NO default
plot_handshake YES default
plot_label NO default
create_separate_files NO default
plot_with_panzoom YES default
rotate_plotting NO default
allow_file_naming YES
plot_name YES
interface_quality 3 default
```

```
plot_destination file default
pen_table_file p:\config\plot\a4_la4.pnt
plot_sheets current default
paper_size A4 default
paper_outline YES default
```

Plotting drawings in color

If you do not specify any colors in the PNT files, the printed colors will correspond to the screen colors.

Tip: Create a PDF file with colored lines.

Plotting TIFF drawings

The TIFF format is a universal graphics format (*tiff.pcf*). TIFF allows lossless compression for very small file sizes for drawings.

Please note: The TIFF snapshot output function in the *File* menu is not the same as a TIFF print output. To achieve small file sizes in TIFF print output, go to the *config.pro* file and set `tiff_compression g4`.

4 Mapkeys

A mapkey is a macro for Creo that binds a frequently used sequence of commands to a certain key or combination of keys. Mapkeys can be saved in the configuration file *config.pro*. Each mapkey has a unique combination of keys that starts the macro. You can define a mapkey for any task that you frequently need to complete in Creo.

Mapkeys are one of the many ways to automate Creo, along with TRAIL files, Pro/Program or custom applications.

In order to be able to make use of the mapkeys in your environment, the users have to know them. Mapkeys can also be assigned and icon and displayed in the command bar. Or you can provide even more ease of use by adding mapkeys to GENIUS TOOLS Quick Access, making them available directly at the mouse position with localized tooltip help. You can also create so-called intelligent mapkeys which contain variables such as model parameters or operating system variables and provide even more flexibility.

4.1 Startup TOOLS Mapkeys

Important mapkeys for controlling the user interface:

– q … Open or close the model tree

– y … Open or close the browser

Function keys			
F3	Fit model or drawing	**F7**	Redraw image
F4	Model default view	**F8**	Shade model
F5	Open orientation window	**F9**	Load model tree configuration from tree.cfg
F6	Save model tree configuration to tree.cfg	**F12**	Save model view with name “Gut”

General options			
ds	Save object	**fa**	Color theme custom
db	Clean up object (keep only latest version)	**fd**	Color theme dark
x	Regenerate (re-calculate) object	**fs**	Color theme Standard
q	Model tree on/off	**fw**	Color theme black/white
y	Browser on/off	**fu**	Color theme original (fa)
cd	Change work directory	**ad**	Temporary simplification for selected parts
ot	Open part	**da**	Remove temporary simplification
oz	Open drawing	**sd**	Search/open drawing
ob	Open assembly	**sa**	Cancel sketch
bh	Load browser start page		
Model views			
gg	View “Gut”	**uu**	View from below
vv	View from front	**ex**	Assembly exploded view
rr	View from right	**zu**	Leave assembly exploded view
oo	View from above	**tra**	Color transparency off
hh	View from back	**tre**	Color transparency on
ll	View from left	**ff**	Colors on/off
Show or hide reference elements			
ae	Reference planes, axes and coordinate systems on/off	**pp**	Reference points on/off
ee	Reference planes on/off	**nn**	3D notes on/off
aa	Reference axes on/off	**tt**	Reference labels on/off
kk	Reference axes on/off		

Object information			
im	Model mass	**mm**	Measure distance
is	Create summarized Excel parts list for assembly		
Load drawing frame			
f0	Load A0 frame	**f3**	Load A3 frame
f1	Load A1 frame	**f4**	Load A4 frame
f2	Load A2 frame		
Drawing options			
ge	Create group	**tn**	Edit entire note (multi-line)
po	Get position number for part in assembly drawing		
Bores			
z12	Through hole with cylindrical counterbore M12 Medium; available for the following diameters by using the corresponding numbers: 3,4,5,6,8,10,12,16,20,24	**s12**	Tapped blind hole; available for diameters as given for z
d12	Through hole M12 Medium; available for diameters as given for z	**b12**	Bore through all; available for diameters 3 through 29 mm
m12	Tapped through hole; available for diameters as given for z		

Mapkeys for GENIUS TOOLS for Creo (ToolKit application)			
autostart	Is executed on starting GENIUS TOOLS (on Creo start)	**createdrw**	Is executed by the functionality *Open/create drawing* when it has finished

Mapkeys for GENIUS TOOLS Library			
tbxinote	Formatting for reference lines when inserting notes in the library browser		

5 Document templates

5.1 Document template: Installation overview

Computer	Installed software	Tasks / Users	Shared resources
Name: \\CADSERVER System: Windows 2016 Server  CA / WA	Startup TOOLS under C:\inneo	Data server: Central user management Administration for - operating system - Creo	Caddepot c:\inneo\caddepot \\CADSERVER\caddepot D:\users \\CADSERVER\users Plotter \\CADSERVER\Djet450C
Name: \\LIZSERVER System: Windows 7 Professional  LA	FLEXNET licensing C:\ptc\flexnet	Creo license management	Laser printer \\LIZSERVER\Ljet5p
Name: \\CAD1 System: Windows 7 Professional  PA	Creo under D:\ptc\creo6 Startkey parametric Modules: Creo basic Cadpool: C:\cadpool	Product development, drawing creation; - Ms. Hofer - Mr. Franke	Used resources: \\CADSERVER\caddepot (U:)\\CADSERVER\users \\CADSERVER\Djet450C \\LIZSERVER\Ljet5p
Name: \\CAD2 System: Windows 10 Professional  PA	Creo under: D:\ptc\creo6 Startkey parametric1 Modules: Creo basic Startkey paramet-ric aax Modules: Foundation Adv+AAX. Cadpool:	Product development, drawing creation, mounting instructions - Ms. Wagner - Mr. Karst	Used resources: \\CADSERVER\caddepot (U:)\\CADSERVER\users \\CADSERVER\Djet450C \\LIZSERVER\Ljet5p

```
C:\cadpool
```

5.2 Document template: App directory conventions

Names of installation directories and start files:

Product	Installation directory	Remarks
Creo	`C:\ptc\creo6`	On all computers
ProductView	`C:\ptc\creoview`	On all computers
Creo libraries	`%GTS_DATA%\library_dir`	Master libraries only in the Caddepot directory
FLEXnet licensing	`C:\ptc\flexnet`	Only on the license servers

Applications with protk.dat / creotk.dat

Registry files for additional applications for Creo such as *protk.dat* or *creotk.dat* should be placed in the <*OperatingEnv*>*configuration**application* directory with a shorthand code.

Examples: *protk_gtfc.dat, protk_mpuser.dat, protk_profile.dat, protk_partsolutions.dat*

The registry files can then be used in the *config.pro* building blocks for the projects.

Example

```
protkdat      $GTS_ROOT_DIR\configuration\application\protk_stools.dat
```

Each additional application should be located in a separate directory under *%GTS_ROOT_DIR%\apps*.

Environment variables

`%GTS_DATA%` ... Points to the data package for the current project

`%GTS_ROOT_DIR%` ... Directory path at Startup TOOLS / Creo execution

5.3 Document template: User environment

The configuration area (CA) is synchronized from the Caddepot to the Cadpool directory by GENIUS TOOLS Starter.

The work and user area (WA) has to be connected as a network drive on each Creo workstation.

```
(u:) \\CADSERVER\users
```

By default, all users except the administrator have read-only access to the directory structure within the Caddepot directory. The Creo administrator has write access. Additional write access privileges for the users are required for the following functionality:

– using Name Generator

– creating UDF forms

– making user-specific settings in the *config.pro*

If you want to use this functionality, it is recommended to give all users write access to the directory *CADSERVER**caddepot**userdata*.

Please note: The full functionality of GENIUS TOOLS for Creo can only be used if all users can write to the directory .*userdata* and its subdirectories.

The access privileges under `U:\` depend on each company's workflow.

For the configuration to be copied from the server to the workstations on each program start, the directories <*drive*>*ptc**creo6*\<*Version*>*Parametric**bin* and <*drive*>*ptc**creo6*\<*Version*>*Common Files**text*, with the files

- *config.pro*
- *config.sup*
- *creo_parametric_admin_customization.ui*

have to be writable (depending on the settings in Project Configurator).

Each user has to have access to the printers or plotters they need.

A desktop shortcut for the GENIUS TOOLS Starter App is created on each workstation.

Checklist

	User				
	Franke	Hahn	Lange	Maier	Kroll
Cadpool present?	x	x	x	x	x
Write access for *CADSERVER**caddepot*\ *userdata* for all users?	x	x	x	x	x
Drive U:\ and access rights present? Working directory present?	x	x	x	x	x
No write protection on configuration files and directory?	x	x	x	x	x
Plotter / printer configured?	x	x	x	x	x
Online help available?	x	x	x	x	x
Plotter / printer tested?	x	x	x	x	x

5.4 Document template: Construction file storage

Recommendation:

Working directory for each user on the data server:

– <drive>:\users\franke

– <drive>:\users\lange

– <drive>:\users\kroll

– <drive>:\users\hahn

– ...

Directory structure for active projects on the server.

Example:

– <drive>:\users\projects

– <drive>:\users\projects\vehicle1000

– <drive>:\users\projects\vehicle1000\chassis

– <drive>:\users\projects\vehicle1000\drivetrain

– <drive>:\users\projects\vehicle1000\cab

– <drive>:\users\projects\vehicle1000\body

– ...

– <drive>:\users\projects\vehicle2000

– ...

For standard parts, buy parts, semifinished and finished products, in-house standard parts etc., also refer to the document template for libraries.

5.5 Document template: Library data

By default, all libraries are located under:

%GTS_DATA%\library_dir (configuration option `pro_library_dir`).

Directory structure:

Directory	Description
\library_dir\crosshatch_dir	crosshatch definitions Configuration option `pro_crosshatch_dir`
\library_dir\format_dir	drawing frames Configuration option `pro_format_dir`
\library_dir\note_dir	drawing notes Configuration option `pro_note_dir`
\library_dir\symbol_dir	2D symbols Configuration option `pro_symbol_dir`
\library_dir\table_dir	drawing tables, drawing headers, dimension tables Configuration option `pro_table_dir`
\library_dir\group_dir	UDFs (user-defined features): - holes - grooves - threads Configuration option `pro_group_dir`
\library_dir\sketcher_palette	parametric 2D sketches Configuration option `sketcher_palette_path`

\library_dir\standards	invariable standard parts (DIN; ISO; EN etc.)
\library_dir\start_model_dir	start part, start assembly, start drawing Configuration option `start_model_dir`
Additonal directories can be created:	
\library_dir\buy_parts	catalog models
\library_dir\	
\library_dir\	

5.6 Document template: File name conventions

File names have to be unique company-wide. Number generators can help you keep them so.

You can use GENIUS TOOLS for Creo Name Generator, Windchill or a combination of both.

5.6.1 Internal parts and assemblies

Example 1

```
56645_hood
----- -----------
|           | Descriptive name does not have to be unique.
| Number from GENIUS TOOLS Name Generator!
```

Example 2

```
FKH123_001_003_32
------ --- --- --
|      |      |  | 2-digit variant number
|      |      | 3-digit part number
|        | 3-digit assembly number
| Unit number
```

Drawings of parts and assemblies get the same name as the part or assembly they represent.

Standard parts

```
DIN912M20X15ST8_8
----- ---- ------
|      |     | Unique object parameters
|      | Standard number
| Standard type
```

5.6.2 Special objects

Object type	Naming rule
Flattened sheet metal parts	Sheet metal part name + suffix flat
Mounting assemblies (Pro/Process for assemblies)	Assembly name + suffix mon
Semifinished products to be processed (e.g., profiles))	
Welded assemblies	
NC products	
NC materials	

5.7 Document template: Parameter concept

Parameters in parts, assemblies and drawings

Parameter	Type	PRT/ ASM	DRW	Remark	Old para-meter
CAD_APPROVED_BY	STR	x	x	Approver	PRUEFER
CAD_APPROVED_BY_(X)	STR	x	x	Revision approver (last change)	-
CAD_APPROVED_ON	STR	x	x	Approval or issuing date	PRUEF_DAT
CAD_APPROVED_ON_(X)	STR	x	x	Revision approval date (last change)	-
CAD_CHANGE_NO	STR	x	x	Change number	-
CAD_CHANGE_NO_(X)	STR	x	x	Revision change number (last change)	-
CAD_CHANGE_TEXT	STR	x	x	Change description	-
CAD_CHANGE_TEXT_(X)	STR	x	x	Revision change description (last change)	-

CAD_CREATED_BY	STR	x	x	First creator	BEARBEITER
CAD_CREATED_ON	STR	x	x	First creation date	BEARB_DAT
CAD_ITERATION	STR	x	x	Current iteration	-
CAD_LIFECYCLE_STATE	STR	x	x	Lifecycle status	-
CAD_MODIFIED_BY	STR	x	x	Revision creator	-
CAD_MODIFIED_BY_(X)	STR	x	x	Revision creator (last change)	-
CAD_MODIFIED_ON	STR	x	x	Revision date	-
CAD_MODIFIED_ON_(X)	STR	x	x	Revision date (last change)	-
CAD_REVISION	STR	x	x	Current revision	-
CAD_REVISION_(X)	STR	x	x	Revision (last change)	-
CATEGORY	INT	x		Classification label	KATEGORIE
CLASSIFICATION	STR	x		Model type (standard part, buy part, free issue)	-
COATING	STR	x		Surface coating	-
COLOR	STR	x		Color, e.g., RAL	-
DEPARTMENT	STR	x		Company unit	-
DESC_SEMIFINISHED	STR	x		Semifinished part description	HALBZEUG
DESCRIPTION_1_DE	STR	x	x	Model name, German	BENENNUNG
DESCRIPTION_1_EN	STR	x	x	Model name, English	NAME_EN
DESCRIPTION_2_DE	STR	x	x	Model description, German	BEZEICHNUNG
DESCRIPTION_2_EN	STR	x	x	Model description, English	DESCRIPTION_EN
DIMENSION	STR	x		Overall dimension (box size)	ABMESSUNG

DOC_TYPE_DE	STR		x	Document type, German	-
DOC_TYPE_EN	STR		x	Document type, English	-
DRAWING_NO	STR		x	Drawing number (DRW parameter leads)	ZEICHNUNGS NUMMER
HARDNESS	STR	x		Hardness value	-
LANGUAGE	STR		x	Drawing language, ISO3166-1 code	-
MASS	REAL	x		Mass	MASSE
MATERIAL	STR	x		Material in parts list	MATERIAL
MC_CHECKTYPE	STR	x	x	Model type for ModelCheck	-
PART_NO	STR	x		ERP number	IDNR
PART_NO_SEMIFINISHED	STR	x		ERP number for semifinished parts	IDNR_ROH
STANDARD	STR	x		Standard for standard parts	NORM
STANDARD_DIMENSION	STR	x		Standard for dimensions	-
STANDARD_EDGE	STR	x		Standard for edges	-
STANDARD_INFO_DE	STR	x		Standard information for standard parts, German	NORMINFO
STANDARD_INFO_EN	STR	x		Standard information for standard parts, English	-
STANDARD_SURFACE	STR	x		Standard for surfaces	-
STANDARD_TOL_CLASS	STR	x		Standard for general tolerances (e.g. ISO 2768)	-
SUPPLIER	STR	x		Supplier	-
&format	-		x	Drawing format (Creo system parameter)	&format

&sheet	-	x	Drawing sheet number (Creo system parameter)	&sheet
&total_sheets	-	x	Drawing number of sheets (Creo system parameter)	&total_sheets

Parameters in drawing headers

Drawing header per ISO 7200.

Green fields contain information from the drawing document. All other information has its source in the selected model.

DRW Zeichnungsdokument CRI_ZYLINDERBLOCK_2000	Erstellt 22.12.2002 T.Ant	Geändert 28.06.2014 X.Worker	Genehmigt - -	Lebenszyklus-Status wird bearbeitet	Version B 2
type:1 Modelldokument CRI_ZYLINDERBLOCK_2000	Erstellt 12.12.1998 T.Ant	Geändert 28.04.2014 X.Worker	Genehmigt 09.08.2014 F.Sun	Lebenszyklus-Status freigegeben	Version C 0
Abteilung (Modell) DEV-LPZ	Maße nach DIN EN ISO 14405	Allgemeintoleranzen ISO 2768-mK	Oberfläche ISO 1302 Kanten ISO 13715	Werkstoff CuZn39Pb0,5 Beschichtung	
Artikelnummer PNO-LPZ-0002021253 Halbzeugnr. Halbzeug -		Titel, zusätzlicher Titel (Modell) Zylinderblock 2000 ZB / 2000 :64			
INNEO®		Zeichnungsnummer DNO-00123			Masse 0.461 kg
		Dokumentenart Fertigungszeichnung	Format A3	Maßstab 2:1	Sprache DE Blatt 1/1

5.8 CATEGORY list

Overview of the CATEGORY values delivered with Startup TOOLS.

CATEGORY	DE	EN
0	allg. Baugruppe	assembly
1	allg. Bauteil	solid part
2	allg. Blechteil	sheetmetal part
5	Schweissnaht	welding seam
20	Schrauben	screw
25	Muttern	nut
30	Scheiben	washer
35	Sicherungen	securing parts

40	Stifte	pin
45	Federn	spring
50	Lager	bearing
55	Dichtelemente, O-Ringe, Wellendichtringe	seal
60	Profile	beam
65	Buchsen	bushes
80	Zubehoer	accessories
100	Blech	plate
110	Rohr & PIPING-Rohr	piping tube
120	geschmied. Rohr	
130	Rund	round
140	Schmiedewelle	
170	Boden	dished head
200	Behaelterflansche	vessel flange
201	Ruehrwerksflansche	
202	Blindflansche	blind flange
203	Blockflansche	block flange
205	Vorschweissflansche	welding neck flange
206	Gewindeflansche	screw flange
207	Flachflansche	
210	T-Stuecke	tee
220	Rohrboegen	elbow
230	Kappen	cap
240	Reduzierungen	reducer
270	Sattelstutzen, weldolet	sattel nozzle
320	Handstellventile	hand valve

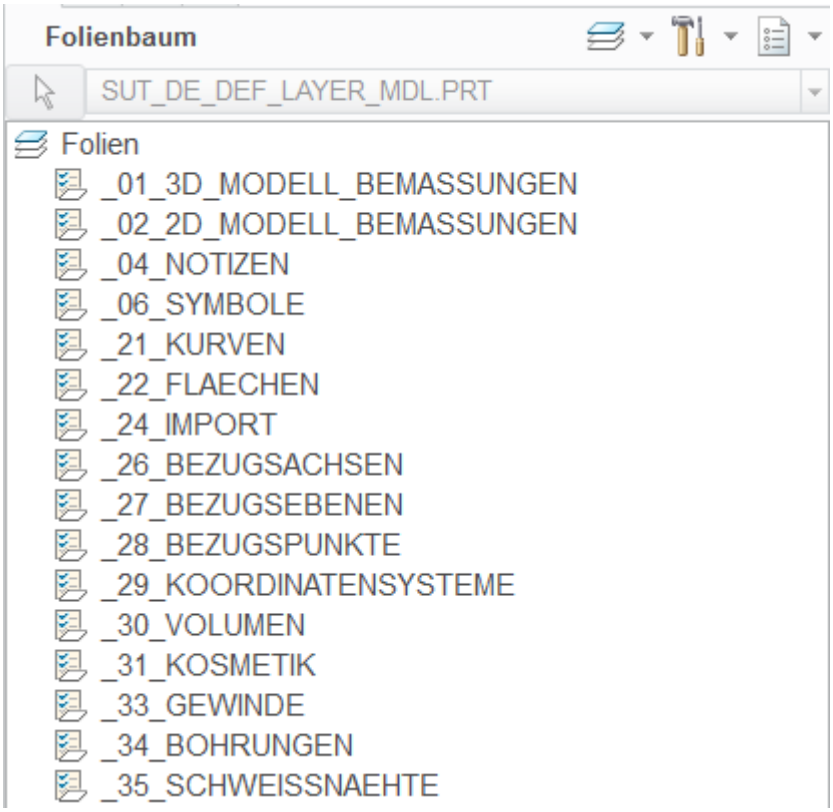
330	Motorstellventile	motor valve
340	pneum. Stellventile	pneumatic valve
350	Sicherheitsventile	seafty valve
360	Rueckschlagklappen	check valve
420	Fuellstandsanzeiger	level indicator

5.9 Document template: Layer concept

All objects and elements in Creo can be placed on one or more layers. Layers can be shown or hidden. You can create rules to automatically assign elements to layers on creation or later on.

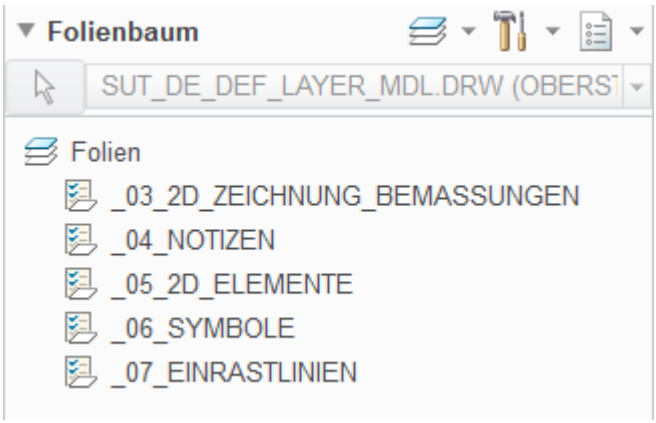
Rules for layer assignment on object creation can either be defined by using the command `DEF_LAYER` in the *config.pro* file, which sets a standard for all sessions, or by using a layer model. A layer model can be crated for each mode — PRT (part), ASM (assembly), DRW (drawing) — all of the same name, with the appropriate file name extension for each mode. Within the layer model, the layer rules can be created in a user-friendly way via the search tool. The layer model is then assigned to the session via the *config.pro*.

```
default_layer_model $GTS_DATA\config\sut_de_def_layer_mdl
```



Layer model sut_de_def_layer_mdl.prt

Layer model sut_de_def_layer_mdl.asm



Layer model sut_de_def_layer_mdl.drw

Layers can also be created manually to group construction information. Use the start models to add a consistent set of layers.

Layer name	Description
_00_HILFEN	For helper geometries, hidden by default
_00_PLATZIERUNG	For elements that are used for placing parts in an assembly
_00_PRT_START_FEATURES	Start elements for a part
_00_ASM_START_FEATURES	Start elements for an assembly

Please note: If you want to work with layers, the naming rules, meaning and intended use for the layers need to be defined in a document that all colleagues have access to. Take care to inform all colleagues of how to use the layer functionality.

5.10 Document template: Simplifications

Simplifications in the start assembly

Name	Description
Detailed_Low	Content defined by user
Detailed_Medium	Content defined by user
Detailed_No_Geom	Content defined by user. However, you should comply with the descriptive name and not represent any geometry.
It is highly recommended to create additional simplifications for large assemblies.	

5.11 Document template: Start objects

Properties	Start part	Start assembly
Units	Millimeter Kilogram Sec (mmKs)	
Accuracy	Absolute 0.01	
Tolerance	ISO/DIN Medium	
Views	3D; Back, Left, Top, Right, Bottom, Front	

Parameters	Refer to the parameters document template	
Layers	Refer to the layers document template	
Features	C_P...Coordinate system YZ_P_RECHTS...Plane XZ_P_OBEN...Plane XY_P_VORNE...Plane	C_A...Coordinate system YZ_A_RECHTS...Plane XZ_A_OBEN...Plane XY_A_VORNE...Plane
Material	STAHL_ALLGEMEIN (Steel, general) (Density 7.85E-6 kg/mm³)	
Relations (after regenerating)	MATERIAL=MATERIAL_PARAM("SELECT_EN_NAME") MASS=FLOOR(PRO_MP_MASS,3)	MASS=FLOOR(PRO_MP_MASS,3)

5.12 Document template: Plot settings

Creo accesses the following output devices:

Device	Locally connected and shared
A1 Color Plotter DesignJet 450C	At computer CADSERVER Network access `\\CADSERVER\Djet450C`
Laser Printer HP LaserJet 5p	At computer LIZSERVER Network access `\\LIZSERVER\Ljet5p`

Files for plotter and printer configuration (PCF, PNT) are located at

`<drive>:\stools\configuration\plot` or `$GTS_ROOT_DIR\configuration\plot`

Pen assignment file a4_la4.pnt for default line widths

```
pen 1 thickness 0.05 cm; color 0.0 0.0 0.0; drawing_color
pen 2 thickness 0.025 cm; color 0.0 0.0 0.0; edge_highlite_color
pen 3 thickness 0.025 cm; color 0.0 0.0 0.0; half_tone_color
pen 4 thickness 0.025 cm; color 0.0 0.0 0.0; highlite_color
pen 5 thickness 0.025 cm; color 0.0 0.0 0.0; letter_color
pen 6 thickness 0.05 cm; color 0.0 0.0 0.0; attention_color
pen 7 thickness 0.025 cm; color 0.0 0.0 0.0; section_color
pen 8 thickness 0.025 cm; color 0.0 0.0 0.0; dimmed_color
```

Pen assignment file a3_a2_a1_a0_la4.pnt for downscaled output

```
pen 1 thickness 0.025 cm; color 0.0 0.0 0.0; drawing_color
pen 2 thickness 0.01 cm; color 0.0 0.0 0.0; edge_highlite_color
pen 3 thickness 0.01 cm; color 0.0 0.0 0.0; half_tone_color
pen 4 thickness 0.01 cm; color 0.0 0.0 0.0; highlite_color
```

```
pen 5 thickness 0.01 cm; color 0.0 0.0 0.0; letter_color
pen 6 thickness 0.01 cm; color 0.0 0.0 0.0; attention_color
pen 7 thickness 0.01 cm; color 0.0 0.0 0.0; section_color
pen 8 thickness 0.01 cm; color 0.0 0.0 0.0; dimmed_color
```

Outputting to A4 paper via Windows print manager

```
a4_laser_a4.pcf
a3_a2_a1_a0_laser_a4.pcf
```

Other outputs

When plotting drawings on the paper roll plotter, the configuration file *rolledjet750sw.pcf* is used.

If you need to plot colored drawings or parts of drawings or have other printing and plotting requirements, individual adjustments need to be made. Consult the manual or ask for user support.

6 Functions with subscription license

Starting with Startup TOOLS 6.0.1 the functions available to users differ depending on the licenses for a product.

The following functions are available with a subscription license for GENIUS TOOLS Starter.

Release	Function / module	Description
6.0.1.0	Dynamic access to Windows user management with LDAP/Active Directory (lightweight directory access protocol)	Creates access to Windows user management and enables live queries so that user assignment is always up-to-date. Users thus do not have to be created and maintained manually. ⇨ Less maintenance work
6.0.1.0	Configuring units	Adds a group element (“unit”) that can easily reflect complex configurations such as for company sites and units. ⇨ Easier configuration for companies with many sites and/ or units ⇨ Allows for a reduction of projects
6.0.1.0	Access to directory “users”	Adds a group element (“users”) that can easily reflect complex configurations for many users. ⇨ Less maintenance work
7.0.0.0	Selecting Creo startkey when starting a project	Provides a project with a choice from several Creo startkeys (start command that opens Creo with a defined license package). Users can start a project with a default startkey or select another one when opening a project in GENIUS TOOLS Starter App. ⇨ Allows the reduction of projects
7.0.0.0	Apps projects	Creates projects that run on any other program.

		Assigning a project directory and batch files is possible. ⇨ GENIUS TOOLS Starter App can be made the central access point for users.
7.0.1.0	Operating satellites in GENIUS TOOLS Starter Service	Enables the connection of satellite servers to a main server and their automatic synchronization. ⇨ Faster connection of user computers to a synchronized satellite server ⇨ Reducing queries from network to main server

The following functions are limited to subscription licenses for either GENIUS TOOLS Library or GENIUS TOOLS Parameter.

Release	Function / module	Description
6.0.1.0	CS Assembler	Automatically adds a number of components to an assembly by deploying a coordinate system (CS). ⇨ Efficient construction of assemblies
7.0.0.0	Load Save Converter	Converts Creo objects from previous Creo-, Wildfire- or Pro/ENGINEER versions to the currently used version. ⇨ Using models from older Creo versions
7.0.0.0	Inspect Revision	Stores all versions of inspection symbols on a drawing. Easily creates an overview of a revision history. (DIN 6770) ⇨ Revision history of inspection symbols
7.0.0.0	Open base model	Opens geometric base models that are reference sources for features with one click in the context menu of a feature. ⇨ Efficient construction of models
7.0.0.0	Select surfaces by color	Surfaces of the same color can be selected with one click and be colored or otherwise modified thereafter. ⇨ Efficient construction of models

7.0.0.0	Extend relations	Adds functions to model relations that define parameters for models and – in Creo 7.0 – also for bodies. GT_UpdateBodyParamMaterial(): Body parameter retrieves material parameter GT_CalculateBodyBoundingBox(): Body parameter retrieves bounding box values GT_CalculateBoundingBox(): Model parameter retrieves bounding box values ⇨ Efficient construction of models

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