STEP Assembly Manager

Release 5.8

User's Manual

T-Systems

PDTec. Enabling Product Data Technology

Contact

T-Systems Enterprises GmbH
Solution Center PLM
Fasanenweg 5
D-70771 Leinfelden-Echterdingen
Germany

T-Systems Hotline:

Tel: ☎ +49 (0711) 972-43001
Fax: 🖂 +49 (0711) 972-41715
E-Mail : *cadcam.hotline@t-systems.com*
Internet: https://servicenet.t-systems.com/sam

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This edition obsoletes all previous editions.

Trademarks

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Preface

This manual describes the usage of the STEP Assembly Manager (SAM). It contains the description how to store, modify and manage CATIA models and assembly structures and to exchange them via STEP.

About this Manual

This manual is intended for end users of the STEP Assembly Manager (SAM).

Related Documents

The following manuals contain information about installation and administration of the STEP Assembly Manager (SAM):

Manual Title	Version
STEP Assembly Manager *Installation and Administration Guide*	5.8

Organization

This manual contains the following chapters:

Chapter 1 gives an overview about SAM.

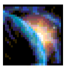
Chapter 2 explains the menu items of the SAM PDM GUI and describes how to handle assembly structures and to exchange them via STEP.

Chapter 3 explains the menu items of the SAM CATIA V4 function and describes how to modify the position of sub-assemblies.

Chapter 4 explains the SAM CATIA V5 module.

Glossary explains the terminology used in the context of SAM.

Conventions Used in this Manual

This font	Is used for document titles and emphasis.
Item 1 > **Item 2**	Is used to describe the menu path to reach a specific function
	This icon is used to identify tips and attention advises.
	This icon is used to identify CATIA related sections and actions.
	This icon is used to identify CATIA V5 related sections and actions.
	This icon is used to identify actions that need confirmation.

CHAPTER 1

Overview

SAM is an application that handles assembly data contained in STEP AP214 Files together with CATIA files (CATIA V4 models, CATIA V5 CATParts, CATProducts and CATDrawings). It consists of a CATIA V4 module (CATIA function), a CATIA V5 module and a separate PDM GUI that communicates with the CATIA modules.

A STEP file used in the context of SAM contains information about the assembly structure and organizational data. Typically it refers to a number of CATIA files or other files, i.e. it does not contain any files, it just contains file names as shown in the figure below. In addition it contains transformation matrices describing the geometric relations between the CATIA files within the assembly structure.

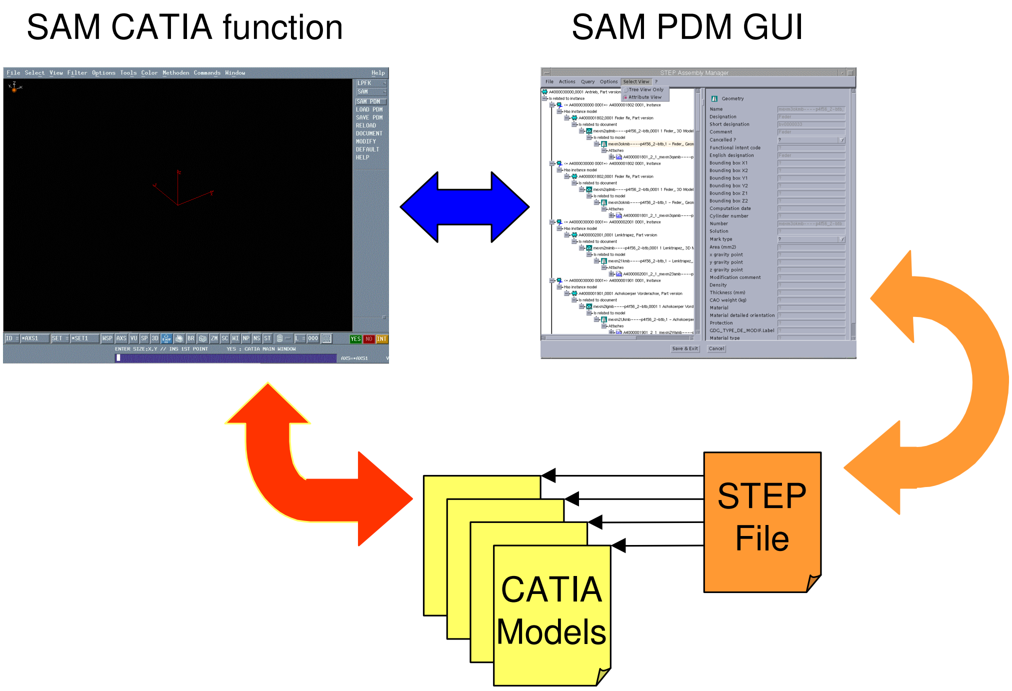


Figure 1: SAM CATIA function and SAM PDM GUI

After a STEP File is loaded into the SAM PDM GUI, it displays the assembly structure and the documents and it tells the SAM CATIA module which CATIA files to load and how to place them. After the information is loaded into CATIA it is possible to change existing CATIA files and their placement and to create new CATIA files. This information can be synchronized with the SAM PDM GUI, so that the STEP file contains the updated information after saving it (e.g. new transformation matrices or new CATIA file references).

Getting started

A typical scenario to use SAM is:

- import a STEP file with an assembly structure referencing CATIA files
- load the assembly structure with the CATIA files into CATIA
- make some modifications in the assembly structure and in the CATIA files
- export the modified assembly structure into a STEP file referencing updated CATIA files

For this scenario, proceed as follows:

- When working with CATIA V4:

In CATIA V4	In the SAM PDM GUI
Call CATIA function "SAM"	
SAM / Menu SAM PDM / YES: Start SAM → SAM PDM GUI pops up	
	Menu File / STEP Import: select STEP file and CATIA Map → CATIMP processes are submitted
	After CATIMP processes are done: click right-hand mouse button on top-level node in the structure view / Expand Models
SAM / Menu LOAD PDM / YES: Read → models are loaded into CATIA	
SAM / Menu MODIFY → modify model positions	
Use standard CATIA functions to modify the model contents	
SAM / Menu SAVE PDM → save modifications in models or structure or both	
	In the structure tree view: click right-hand mouse button on the models you want to export / Activate
	Menu File / STEP Export: Type STEP file name → CATEXP processes are submitted and STEP file is created

- When working with CATIA V5:

In CATIA V5	In the SAM PDM GUI
Start Product Structure Workbench → SAM toolbar is displayed	
Click on "Start SAM PDM" icon in SAM toolbar → SAM PDM GUI pops up	
	Menu File / STEP Import: select STEP file and CATIA target directory → CATIA files are copied to target directory
	After CATIA files are copied: click right-hand mouse button on top-level node in the structure view / Expand Models
Click on "Read from SAM PDM" icon in SAM toolbar → CATIA files are loaded into CATIA	
Use standard CATIA functions to modify the CATIA file contents and positions	
Click on "Update SAM PDM" icon in SAM toolbar → save modifications in CATIA file contents and positions	
	In the structure tree view: click right-hand mouse button on the CATIA files you want to export / Activate
	Menu File / STEP Export: Type STEP file name → STEP file is created and CATIA files are copied to the directory of the STEP file

For a detailed description of the functionality and the user interactions, please see the following chapters.

Note regarding CATEXP format:

- When working with CATIA V4, the CATIA models are typically exchanged in CATEXP (CATIA export) format. At STEP import/export, SAM converts the models from/to CATEXP format using the CATIMP/CATEXP CATIA utilities.
- When working with CATIA V5, the STEP data package may contain CATIA V5 CATParts, CATProducts, CATDrawings and CATIA V4 models. If CATIA V4 models are included, they must be in CATIA model format (.model), as CATIA V5 does not support the CATEXP format.

CHAPTER 2

The SAM PDM GUI

This chapter describes the usage of the SAM PDM GUI.

The SAM GUI has 4 modes:

- Smaragd mode:
  Displays STEP files in a Smaragd like view, imported files have the ending .sma. SAM will use the Smaragd mode for STEP files that you receive from Daimler.
- GDG mode:
  Displays STEP files in a GDG like view, imported files have the ending .gdg. SAM will use the GDG mode for STEP files that you receive from Renault's GDG system.
- VPM mode:
  Displays STEP files in a VPM like view according to Renault's VPM customization, imported files have the ending .vpm. SAM will use the VPM mode for STEP files that you receive from Renault's VPM customization.
- MSF mode:
  Displays STEP files in a view like MSF’s EDB system, imported files have the ending .msf. SAM will use the MSF mode for STEP files that you receive from MSF (Magna Steyr Fahrzeugtechnik).

SAM will automatically select the proper view depending on the STEP file origin.

To start the SAM PDM GUI, first call the SAM CATIA function and press the SAM PDM button.

Loading Files

Initially the SAM window is empty. To load a File, use the File menu in the SAM window (see Figure 2).

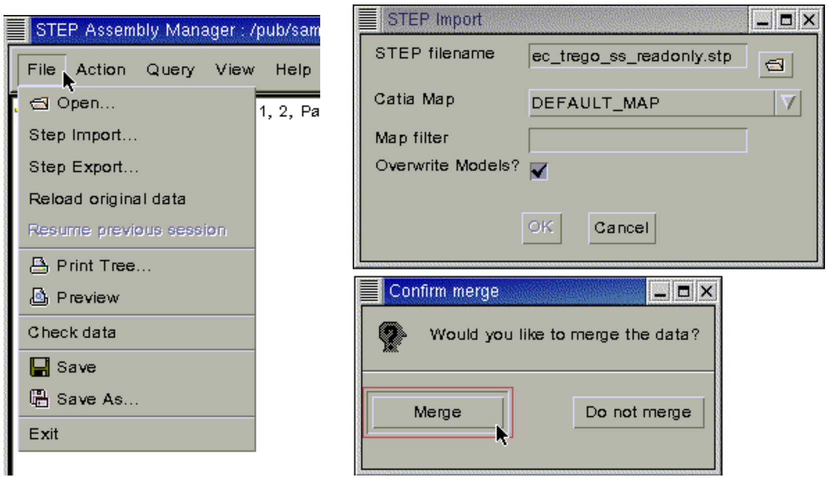


Figure 2: Opening Files with SAM

There are two possibilities to open a file:

- Step Import (if SAM runs with CATIA-V4):
  SAM reads a STEP File (*.stp) or a package containing STEP Files and CATIA models (*.tar.gz or *.tgz). The filename has to be specified in the STEP import dialog window as shown on the right of the figure above.

  If you run SAM with CATIA-V4, you need to specify a CATIA Map name where the models (and other files) should be copied after the conversion. If you type in a string into the Map filter field, only the maps that contain this string as substring will be displayed in the Catia Map drop down menu.
  The conversion resolves the references in the STEP file to the files and converts all referenced CATIA model files from CATIA Export format to native CATIA format (*.model) and copies them into the specified CATIA Map.

  When files are copied into maps that already exist in the map, an additional dialog box comes up, that lists all files that will be overwritten, if you select the “Overwrite All” button (see Figure 4, right-hand side).

  If you have already loaded data of the same type into SAM, you will be asked whether you want to merge the data in the opened file with the data which has already been loaded into SAM. If you select Merge, the data will be merged, otherwise the existing data will be deleted before loading the new data.

  If you press the OK (or Merge/Do not Merge) button , the CATIA import will start and a window will pop up that shows the conversion progress (see figure below). Optionally, you can save the messages displayed in the window into a file by selecting “Save log file”. Press “Continue” to finish the STEP import.

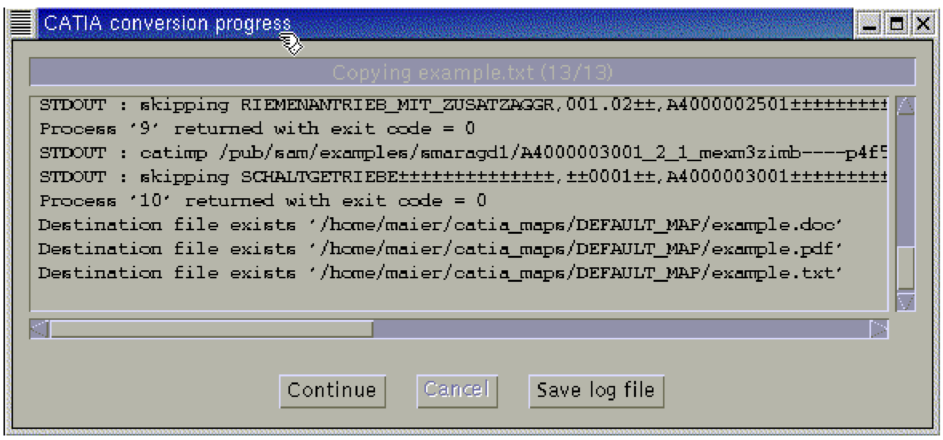


Figure 3: CATIA file conversion window

- STEP Import (if SAM runs with CATIA-V5)
  If SAM runs with CATIA-V5, you can choose a directory where the referenced external files should be copied. If you do not choose a directory, the files will not be copied. Note that no conversion (as in CATIA-V4 mode) will be performed.

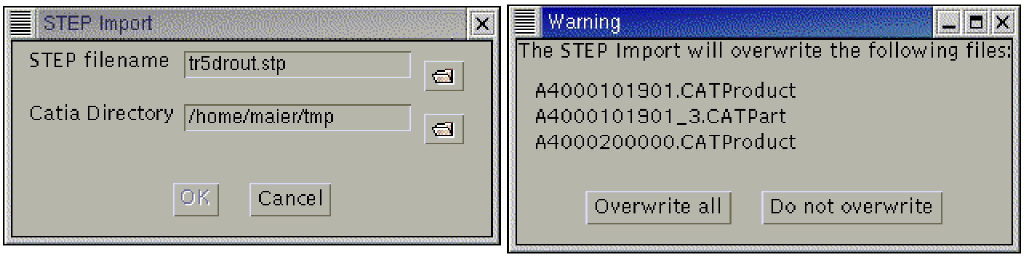


Figure 4: STEP Import Dialog in CATIA-V5 mode

- Open:
  SAM reads a SAM file (*.sma, *.gdg or *.vpm). A SAM file is produced when saving a STEP file. It contains references to already converted CATIA models, i.e. no CATIA model conversion needs to be done when it is opened.
  If you open a file and there is already data of the same type loaded into SAM, you will be asked whether you want to merge the data or whether the existing data should be deleted.

Note regarding installation: Opening a package (*.tar.gz or *.tgz) requires that the commands "tar" and "gzip" are available in the path.
Under Windows, this requirement is not fulfilled by default. In this case, the package first has to be manually opened, e. g. by WinZip. Then the STEP file that is included in the package can be loaded using SAM STEP Import.
See also note in the installation manual with an internet link for some free UNIX tools on Windows.

Merging Data

If you do a STEP Import or an Open File and data is already loaded in SAM, you can choose to merge the existing data with the new data. Merging of data depends on the View and is therefore separately described for each view.

Merging of Smaragd data

If the new file contains a part with the same part number and same revision and sequence, the original part is replaced by the new part (including its documents and files). If the part number is the same, but revision or sequence are different, then an error is reported, because multiple versions of the same part are not allowed in sma-Files. All assembly relations are preserved, even if a part is replaced by a new one. Two assembly relations are merged only if the parent and the child part are the same and if the findnumbers of both relations exists and if they are equal. If the folders of a part have the same types and document description, the folder of the original part including its data is replaced by the new folder.

If the environment variable `SAM_PROPERTY=MANUAL_MERGE=1` is set, you will be asked in the "Confirm merge" window for 3 alternatives:

- Merge automatically
- Start manual merge
- Do not merge

With "Start manual merge", you can compare the new and the potentially resulting merged structure before merging. Both structures are displayed in parallel in 2 sub-windows. Differences in the structures are graphically highlighted. In addition, single modifications can be accepted or reverted.

Merging of GDG data

In GDG mode the following objects are merged:

- Files if the file name is the same
- GEO, P2D, F2D, GDC and M3D if the Number is the same
- INS if Instance number, Revision, Sequence, Father Partnumber, Father revision, Father class, Son Partnumber, Son revision and Son class are the same
- Compositions if Number, Revision and Sequence are the same
- PIE, IND, OCI if Partnumber, Revision and Sequence are the same

Unlike in the Smaragd case, in GDG-mode the original data is preserved and not replaced by the data in the second file.

Merging of VPM data

In VPM mode the following objects are merged:

- Files if the Id, File name, Version and Directory is the same
- Parts if the Partnumber and the Version are the same
- Part relations if the parent and child parts are merged and if the Instance id is the same.

Merging works like in GDG-Mode, i.e. if an object is merged, the attribute values of the original data are preserved.

Merging of MSF data

In MSF mode the following objects are merged:

- Parts if the Document-Id, CAD-Art and CAD-Unterm. and Sheet No. are the same
- Files if the data type is the same.

If an object is merged, the attribute values of the new data is used.

You will be asked in the “Confirm merge” window for 3 alternatives:

- Merge automatically
- Start manual merge
- Do not merge

Manual merge works like in Smaragd-Mode.

Meaning of Symbols in Manual Merge Mode

Object does not exist in the original data set. Object will be added.

Object exists in the original data set, but it is not used in this place.
A usage relation will be added.

Object does no longer exist in the new data set and will therefore be deleted

Object exists in the new data set, but is not used in this place. The usage relation will be deleted.

Attribute values of the relation have been changed (e.g. position).

Blue text: Attribute values are different. The concerned attribute names are marked in red color.

Grey text: Object does not exist.

Version Dependent File Management

In SAM/Smargd and SAM/Public mode, version dependent file names are supported. When importing a STEP file, the reference external files are renamed and the version is appended to the file name to avoid that existing files (of older versions) are overwritten. When loading a file to CATIA, the files are copied to a temporary working directory and renamed to their original name. After the files have been changed in CATIA, they are copied back to the original directory, using the version dependent file name.

SAM/Smaragd creates version dependent file names according to the following rules:

Type of File	File Name	Version Dependent File Name
CATIA-V4	name.model	name#sequence.model
CATPart	name.CATPart	name#sequence.CATPart
CATDrawing	name.CATDrawing	name#sequence.CATDrawing
CATProduct	name.CATProduct	name#partrevision#partsequence#sequence.CATProduct
Others	name.suffix	name.suffix

In SAM/Public each files of all types will be renamed by appending the version of file (if it exists) before the file ending, separated by a hash (#) sign. (i.e. name#version.ending).

The STEP-Export renames the files, so that the created STEP package contains the original file names again.

Setting the location for new files

NOTE: this option is only available in CATIA-V5 mode.

If you want to add new CATIA files to the structure, or if new CATProduct files are automatically generated and added to the structure, SAM needs to know where to store these files. By default these files are stored in the same directory where the SAM structure file (*.sma/*.gdg) is located, but this directory can be manually set, by choosing File->Set new file location as shown in Figure 5.

Once you set the location, it is stored in the SAM structure file (*.sma/*.gdg), so you do not need to set it again next time you open the file.

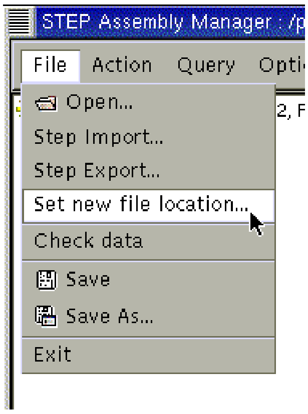


Figure 5: Setting the location for new CATIA files

Browsing and Editing Assembly Data

After you loaded a file into SAM, the data is displayed in a tree view. Initially only the top-level items are displayed. By double clicking one of the top-level items the tree expands one level and can be further expanded by double clicking the child items with a single mouse click on the expansion (+/-) symbols.

In Smaragd mode the expansion state of the tree can be saved automatically. To enable this feature, set “Options->Preferences->Save Tree Expansion State” to “Yes”. If the feature is enabled, the expansion state is saved in an extra file with ending “.bvs”. When you load a Smaragd file next time, the assembly structure tree is expanded in the same way as it was before.

If you can click on a tree item with the right mouse button, a popup-menu will appear and you can select several levels of automatic expansion as shown in the figures below.

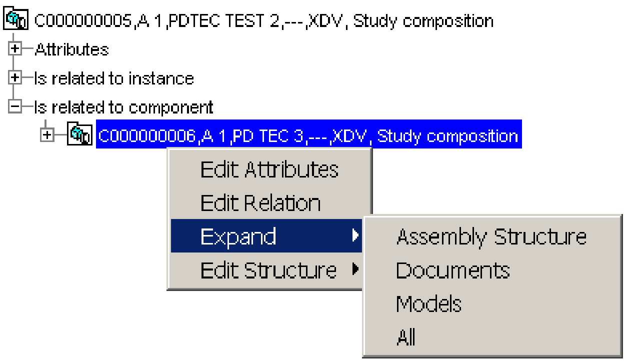


Figure 6: SAM Tree Context Menu in GDG mode

In GDG mode the following expand actions are supported:

- Expand Assembly Structure:
  Expands all assembly components of the selected item.
- Expand Documents
  Expands all assembly components and the documents related to the assemblies or parts.
- Expand Models
  Like Expand Documents, but also expands the items (models) contained in the documents.
- Expand All
  Expands all items (including the files contained in the models)

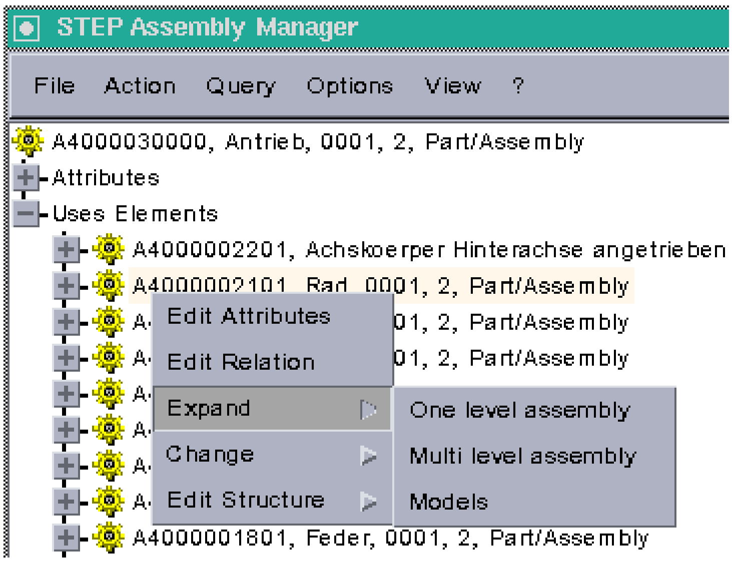


Figure 7: SAM Tree Context Menu in Smaragd mode

In Smaragd mode the following expand actions are supported:

- One level assembly:
  Expands all direct assembly components of the selected item.
- Multi level assembly
  Expands all assembly components

- Data
  Expands the models that are directly attached to the part
- Models
  Expand all models that are attached to the part and all models attached to the parts in the assembly structure.

Using the context menu, you can inspect and edit the attributes of the selected item and the attributes of the relation object between the item and its parent item.

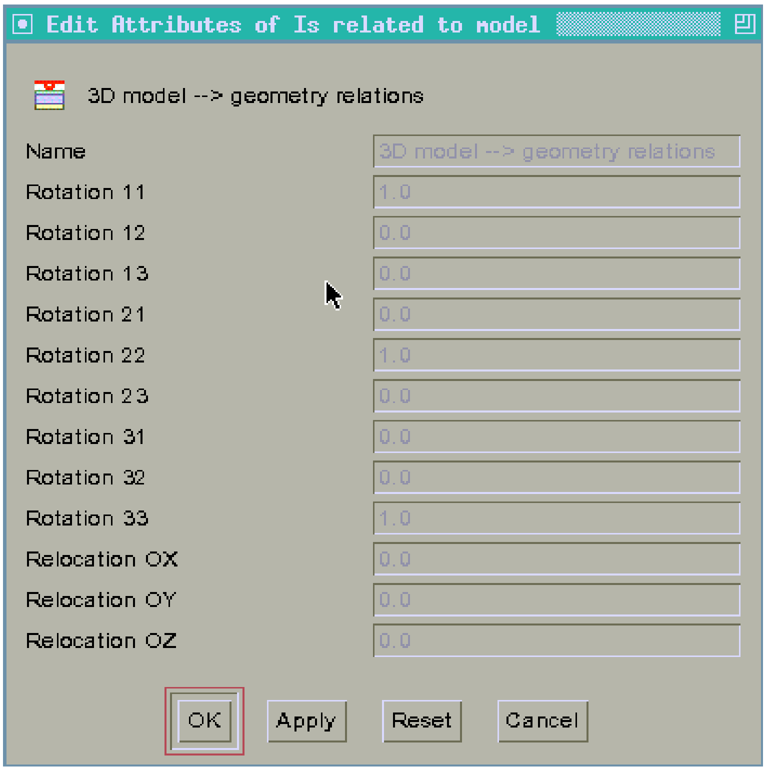


Figure 8: Edit Attributes Dialog Window

Each tree item (parts, assembly, document, etc.) has attributes that are displayed if you open the "Has Attributes" tree node of the item. Alternatively you can switch to a different view, using the menu Select View => Attribute View, that displays all attributes on the right side of the window as shown in the figure below.

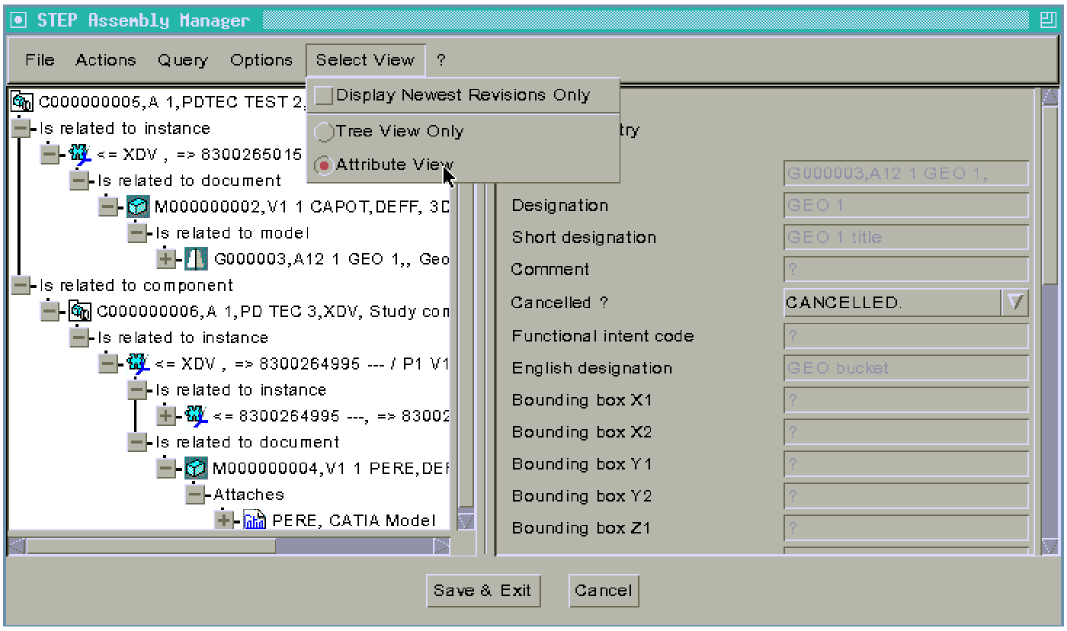


Figure 9: SAM Attribute View

In GDG mode you can use the Select View menu to switch between a view that displays all revisions of parts and documents (which is the default view) and a view that displays the newest revisions only (Select View => Display Newest Revisions Only to switch to this view).

In Smaragd mode you can use the Select View menu to switch between the normal view and an extended relationship view. In the extended relationship view the relations (containing the transformation matrices) are displayed as explicit nodes in the tree, see figure below. If “Options->Preferences->Save Tree Expansion State” is set “Yes”, the expansion state of the tree is kept when switching between different views.

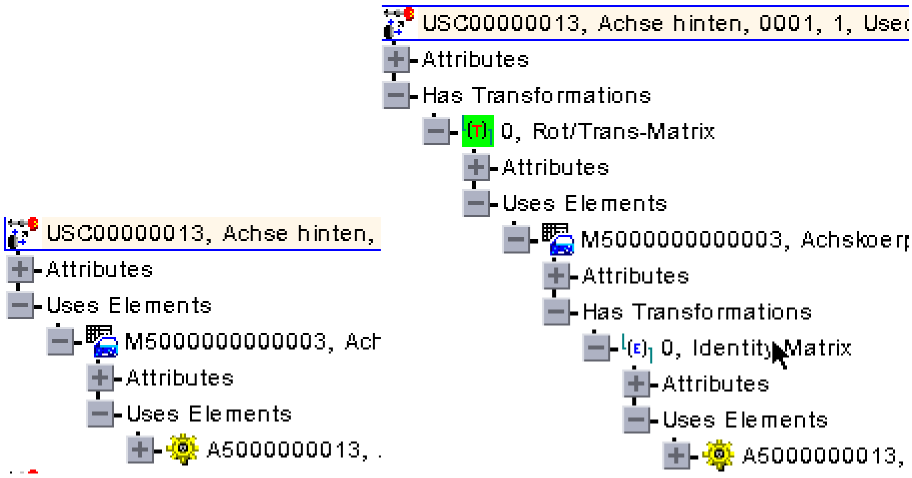


Figure 10: Normal view and extended relationship view (Smaragd mode only)

NOTE: When loading an assembly structure into CATIA, SAM only transfers the structure and the models that are expanded (i.e. visible) in the tree, with the exception that CATIA files are loaded into CATIA if the document or model that contains them is visible.

Selecting Data

By default, all models that are visible in the tree structure are sent to CATIA. In addition you can select specific models that you do not want to display in CATIA by clicking on the model with the right mouse button and choosing “Do not show in CATIA”. The model and its files will be marked by a red , i.e. the model is hidden for CATIA. By clicking again on the model and selecting “Show in CATIA”, the model will be transferred next time you reload the data in CATIA.

Another way to select certain types of model, is the “Actions => CATIA Model Selection” menu: “Show all models” makes all hidden models unhidden. For Smaragd there are two menu entries to hide either 3D or 2D models, using “Show 3D-models only” or “Show 2D-models only”.

Models can also be hidden by the type of the document in which they are contained:

For GDG this is done by using “Do not show ...” and selecting the document type that should not be displayed in CATIA.

For Smaragd there are two menu entries “Show Part Description Primary models only” and “Show Part Desc. Sec. Models only”, to hide models that are contained in Primary Catia Part Description folders or Secondary Catia Part Description folders.

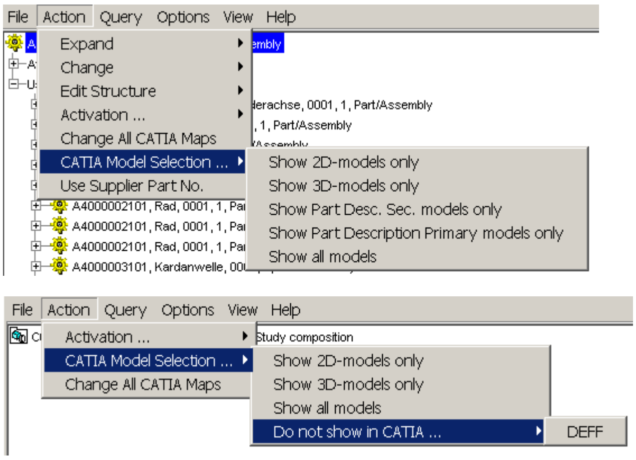


Figure 11: Model Selection in SAM/Smaragd and SAM/GDG

<u>SAM/Smaragd</u>: it is possible to control loading of deactivated models to CATIA. It can be achieved by selecting menu “Options->Preferences” and setting value of the „Load active CATIA Models only“ entry. Allowed values are „Yes/No“.

Querying Data

You can also expand the tree by searching for items with the Query menu. When opening the menu, all types that can be queried are displayed. After you have selected a type, the browser opens a dialog window where you can specify values you want to search for. If a field is empty or if it contains a * character, all values will match the query. Wildcards are supported using the * character.

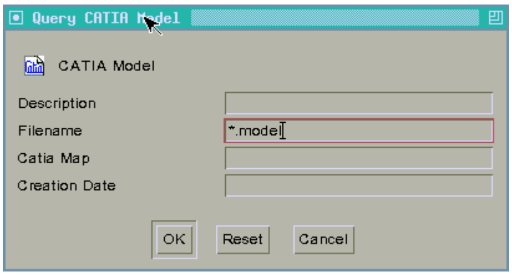


Figure 12: Query Dialog

With the dialog window shown in the figure above a search for all CATIA models that have a file name ending with .model will be started. After you pressed the OK button, all found tree nodes will be expanded and the Query window comes up as shown in the figure below.

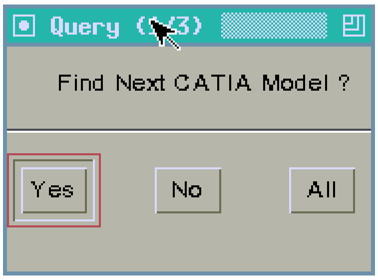


Figure 13: Query Window

By clicking on the Yes button you can jump to the next result of the query. The corresponding item will be selected in the tree. By clicking No, the query aborts, by clicking All, all query results in the tree are selected.

STEP File Header Section Information

A STEP file always has a header section containing organizational data. If you want to see what is contained in this header section, open the menu Options => Show Header Info.

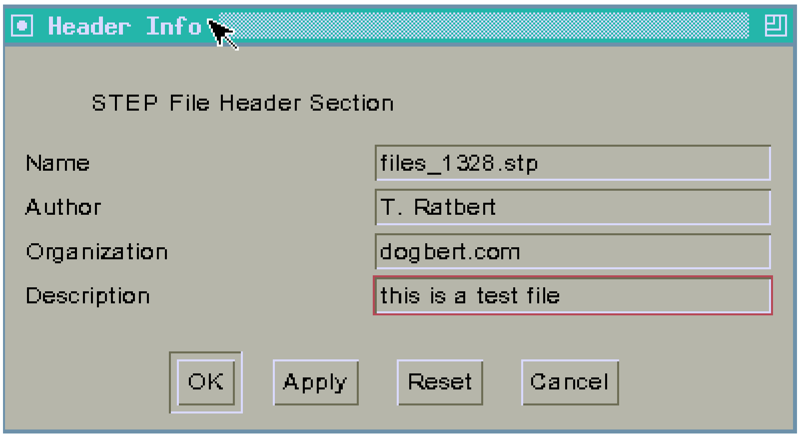


Figure 14: Header Section Dialog Window

A window will appear that displays the header section information of the STEP file which was imported last as shown in the figure above.

Functionality “Check Files”

NOTE: This functionality is only available for SAM/Smaragd.

The functionality *Check Files* checks if the CATIA-Models referenced by SAM are available in the local folder structure. For all file references the available meta data is checked concerning the availability of the physical files on the local disk. As result, a status notification is generated as documentation of the check. The notification can be stored as a log file.

The following model types are checked:

- *Activated models* (is everything available? Have all models which should be contained within a specific export package been imported?).
- *Deactivated models* (are the deactivated models already available (e.g. as an Import Map) and have they been sent within another package in the scope of a specific order?).

In a first step, it is checked whether the respective models are defined as activated or deactivated. The function *Check Files* checks the following possibilities:

- A model is activated but is not found within the folder (File ACTIVE and NOT READABLE),
- A model is deactivated and is not found within the folder (File NOT ACTIVE and NOT READABLE).
- A model is deactivated but is found within the folder (File NOT ACTIVE),

In order to visualize the respective status notification, the status window has to be activated (Menu View → Show Status). The function is then available in the menu *File* (File → Check Files). If one of the cases mentioned above occurs, a respective error message is generated („Errors Found“, **Figure 15**):

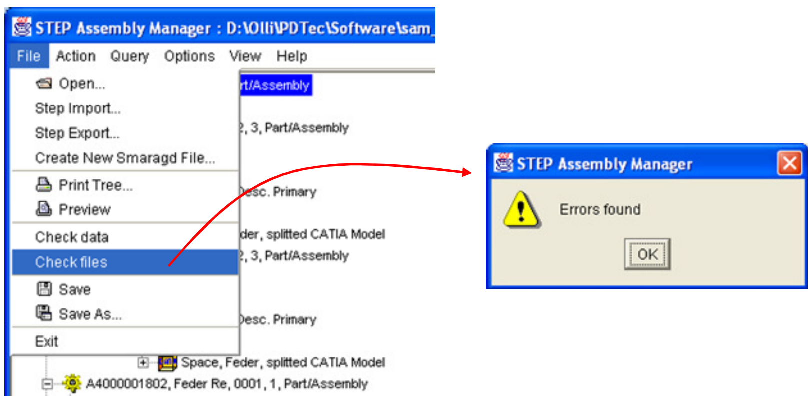


Figure 15: Functionality *Check Files*

Visualization in Main Window and Status Window

The three cases described above are summarized in the following table:

Event	Message in status window
case 1: model file is activated but is not found within the folder	File ACTIVE and NOT READABLE: <file name>
case 2: model file is deactivated and is not found within the folder	File NOT ACTIVE and NOT READABLE: <file name>
case 3: model file is deactivated but is	File NOT ACTIVE: <file name>

found within the folder	

Case 1: File ACTIVE and NOT READABLE

In the main window, all models that were not found are marked („File not found“); in the status window, the inconsistencies are displayed (**Figure 16**):

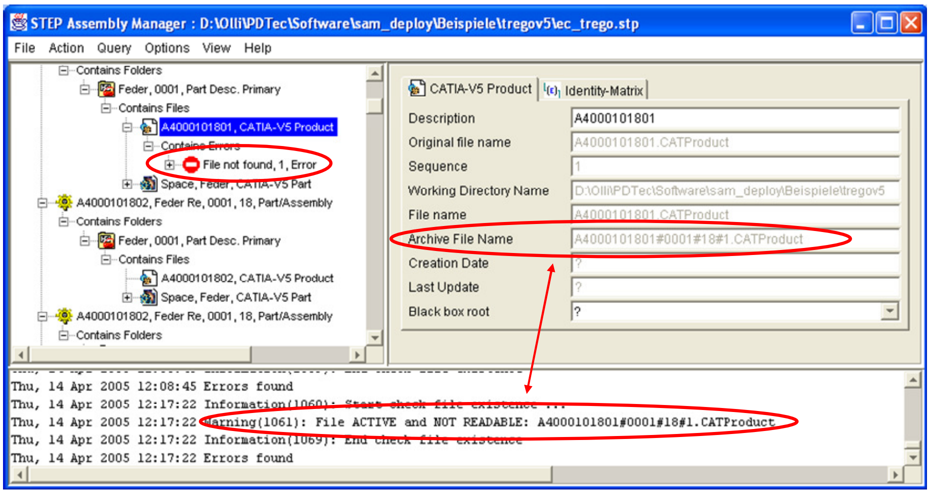


Figure 16: File ACTIVE and NOT READABLE

Case 2: File NOT ACTIVE and NOT READABLE

In the main window, all models that were not found are marked („File not found“); in the status window, the inconsistencies are displayed (**Figure 17**):

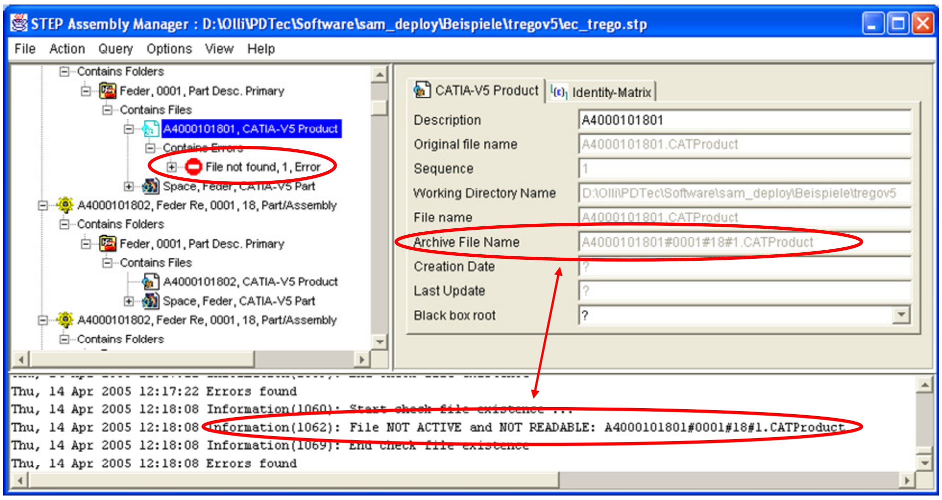


Figure 17: File NOT ACTIVE and NOT READABLE

Fall 3: File NOT ACTIVE

In the main window, all deactivated models are displayed in a different colour (pale); in the status window, the inconsistencies are displayed (**Figure 18**):

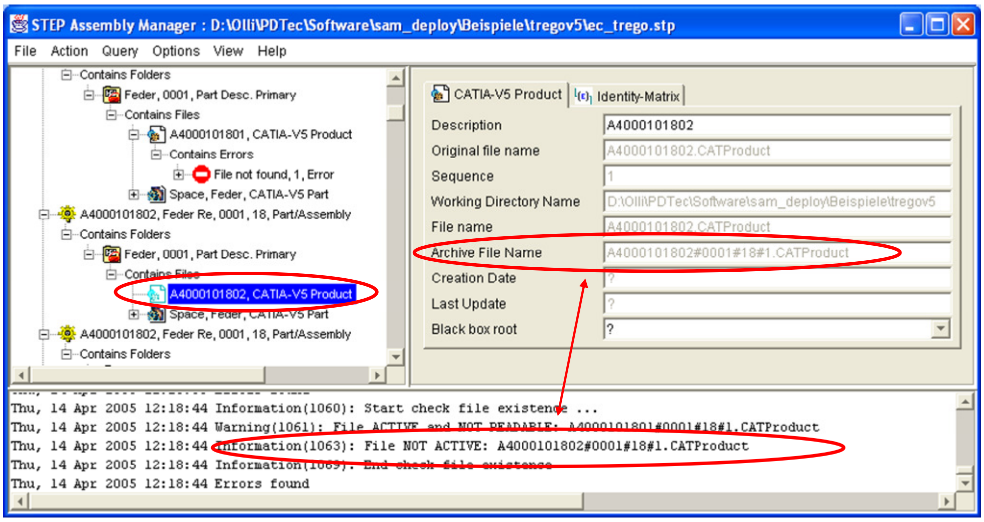


Figure 18: File NOT ACTIVE

Save Notifications as Log File

All information displayed in the status window can be saved as a log file. The respective functionality is available by clicking the right mouse button (context menu → Save Log).

The log file for the cases described above has the following format:

```
Thu, 14 Apr 2005 12:17:22 Errors found
Thu, 14 Apr 2005 12:18:08 Information(1060): Start check file existence
Thu, 14 Apr 2005 12:18:08 Information(1062):
File NOT ACTIVE and NOT READABLE: A4000101801#0001#18#1.CATProduct

Thu, 14 Apr 2005 12:18:08 Information(1069): End check file existence
Thu, 14 Apr 2005 12:18:08 Errors found
Thu, 14 Apr 2005 12:18:44 Information(1060): Start check file existence
Thu, 14 Apr 2005 12:18:44 Warning(1061):
File ACTIVE and NOT READABLE: A4000101801#0001#18#1.CATProduct

Thu, 14 Apr 2005 12:18:44 Information(1063):
File NOT ACTIVE: A4000101802#0001#18#1.CATProduct

Thu, 14 Apr 2005 12:18:44 Information(1069): End check file existence
Thu, 14 Apr 2005 12:18:44 Errors found
```

The log file is saved with the suffix *.log*.

Creating and Replacing CATIA Files in the Assembly Structure

If you have loaded the data into CATIA, you can extend and modify the assembly structure data by changing the placement of existing CATIA files, by adding new CATIA files or replacing existing ones. Changing the placement of a CATIA file updates the transformation matrix attributes of the affected items in SAM. Adding or replacing new CATIA files requires some user interaction which is described in this section.

When SAM is notified by CATIA that a new CATIA file should be added (for CATIA-V4 mode see *chapter 3, DOCUMENT CREATE*, for CATIA-V5 mode see *chapter 4, Create in SAM PDM*), a window comes up for selecting the location of the CATIA file (see next Figure). The new CATIA file will be created as a child component of the selected node, if the node is a document. If the node is a file, it will replace the selected file.

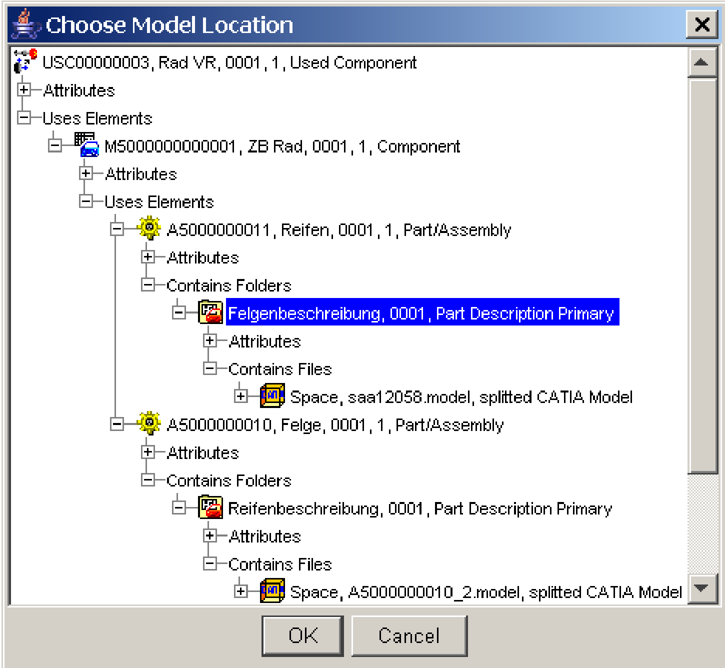


Figure 19: CATIA file destination selection

After you selected the location of the new CATIA file, another window comes up for entering the data of the new CATIA file.

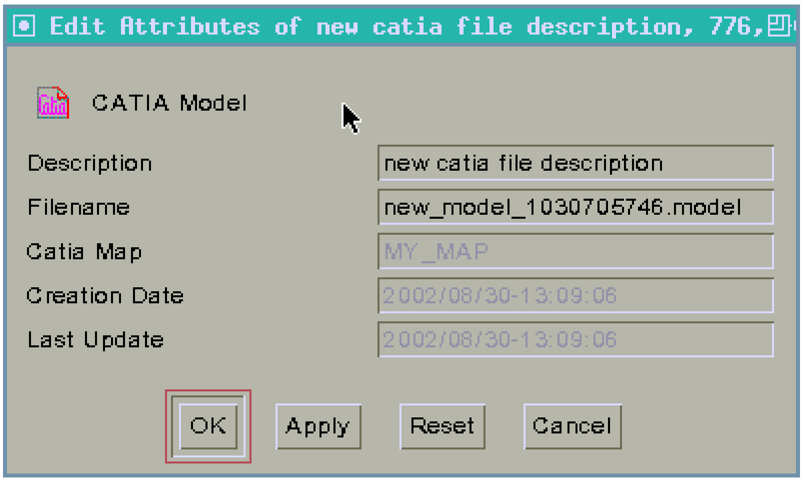


Figure 20: Dialog Window to put in new CATIA model data

In CATIA-V4 mode, the CATIA Map of the new model has to be specified.

In CATIA-V5 mode, the directory of the new CATIA file (CATPart or CATDrawing) has to be specified.

After entering the file name of the new CATIA file and the description, press the OK button and the CATIA file is displayed in the tree view.

Adding existing Files to the Assembly Structure

NOTE: This functionality is only available for SAM/Smaragd and SAM/GDG.

To add an existing File to the assembly structure, open the context menu on a Smaragd Folder and choose “Edit Structure”.

In CATIA-V4 mode, choose “Add CATIA-V4 File” (only for SAM/Smaragd) to add a CATIA V4 model, or choose “Add File" to add files of other formats.

In CATIA-V5 mode, the menu entry “Add CATIA-V4 File” does not exist. Choose “Add File" to add CATIA files or files of other formats.

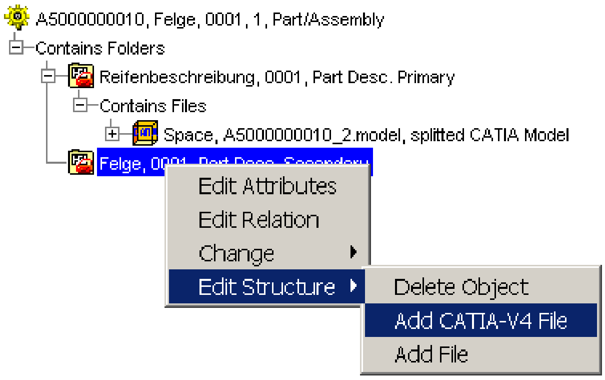


Figure 21: Context menu to add new files to Smaragd Folders

If you open the context menu on a CATIA model or file and choose Edit Structure->Add (CATIA-V4) File, the existing file will be not added but replaced by the newly selected one.

If you choose "Add CATIA-V4 File", a dialog window will appear to choose the CATIA Map and the CATIA-Model as shown in Figure 22.

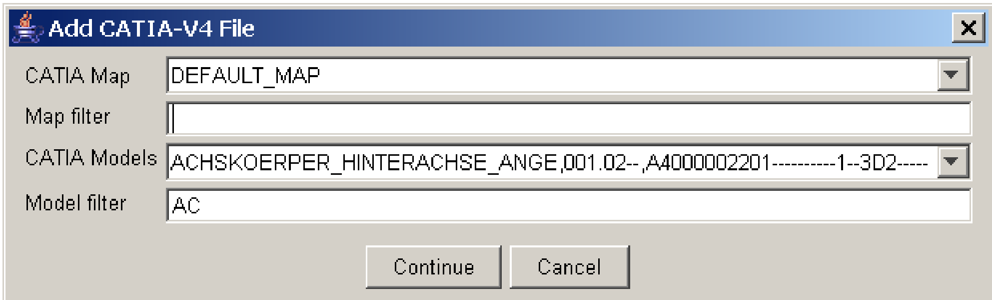


Figure 22: Adding existing CATIA-V4 Files

If you choose "Add File", a dialog window will appear to enter the filename, its description and the type of the file as shown in Figure 23.

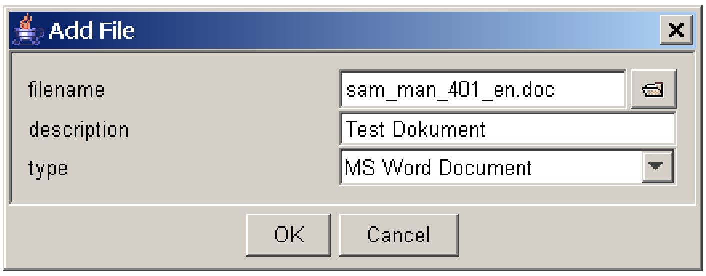


Figure 23: Adding existing other Files

In CATIA-V5 mode, the following CATIA file types can be specified:

CATIA-V5 Part (CATPart)

CATIA-V5 Product (CATProduct)

CATIA-V5 Drawing (CATDrawing)

Splitted CATIA Model (CATIA V4 model)

After selection “OK” in the dialog window, the file will be added to the structure. For CATIA files you need to reload the structure in CATIA to display it.

Changing CATIA Maps or Directories

NOTE: In CATIA-V4 mode, the CATIA models are stored in CATIA Maps. In CATIA-V5 mode, the CATIA files are stored in directories. This section describes the functionality to change the CATIA Maps (CATIA-V4 mode). In CATIA-V5 mode, the functionality to change the directories for CATIA files and other files is analogous (use "Change directory" instead of "Change CATIA Map").

Description to change the CATIA Maps:

After import, you can manually change the CATIA map for a file. There are three ways to do this in SAM:

Open the context menu on a CATIA file and select Change CATIA Map. This will change the CATIA map for the selected file.

Select one or more CATIA files with the left mouse button (holding the Control Key after having selected the first file) and select Change CATIA Map in the Action menu. This will change the CATIA maps for all selected files.
NOTE: To change all CATIA Maps in a (sub-)assembly, expand the Assembly tree (using Expand All), select the first item of the assembly and click on the last item of the assembly holding the shift key. This will mark all items of the assembly. After that, use Change CATIA Map in the Action menu.

Select Change All CATIA Maps in the Actions menu. This will change the CATIA maps for all files referenced by the STEP data.

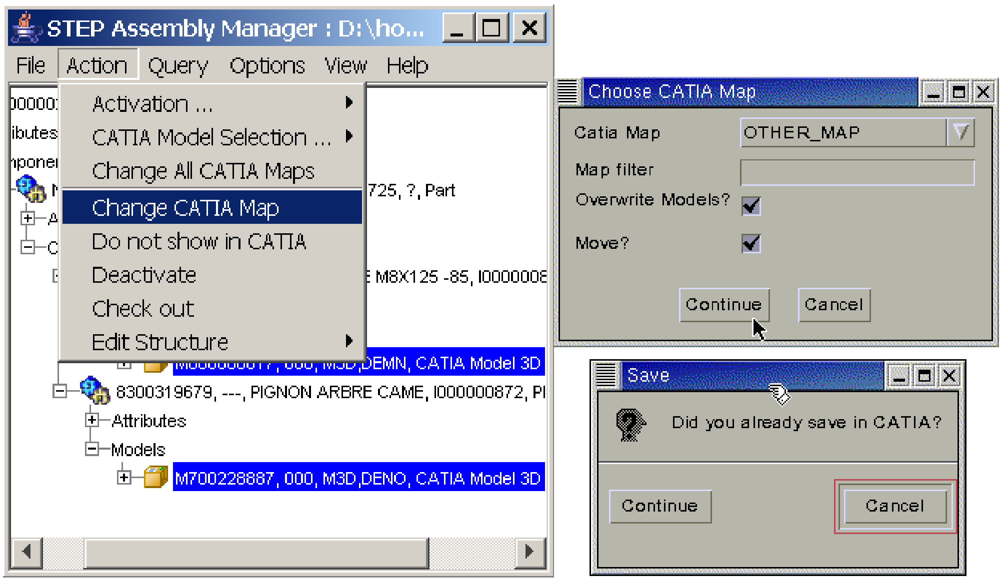


Figure 24: Changing CATIA maps

After you have selected one of the above menu buttons, you have to choose the new CATIA map. In addition there are two options:

Overwrite Models?: If this checkbox is switched on, existing models with the same name will be overwritten.

Move?: If this checkbox is switched on, the model is moved from the original map to the new map, otherwise it is copied.

NOTE: Before changing the map of a model, you should first save in CATIA using the SAM CATIA function. After you have changed the map, you should reload the structure in CATIA.

Creating and Editing the Assembly Structure

If you want to change or prune the assembly structure, you can also use the context menu by clicking with the right mouse button on a part and selecting Edit Structure. Depending on the type of the part and its position in the assembly structure several actions that can be applied on the part are provided (see next figure):

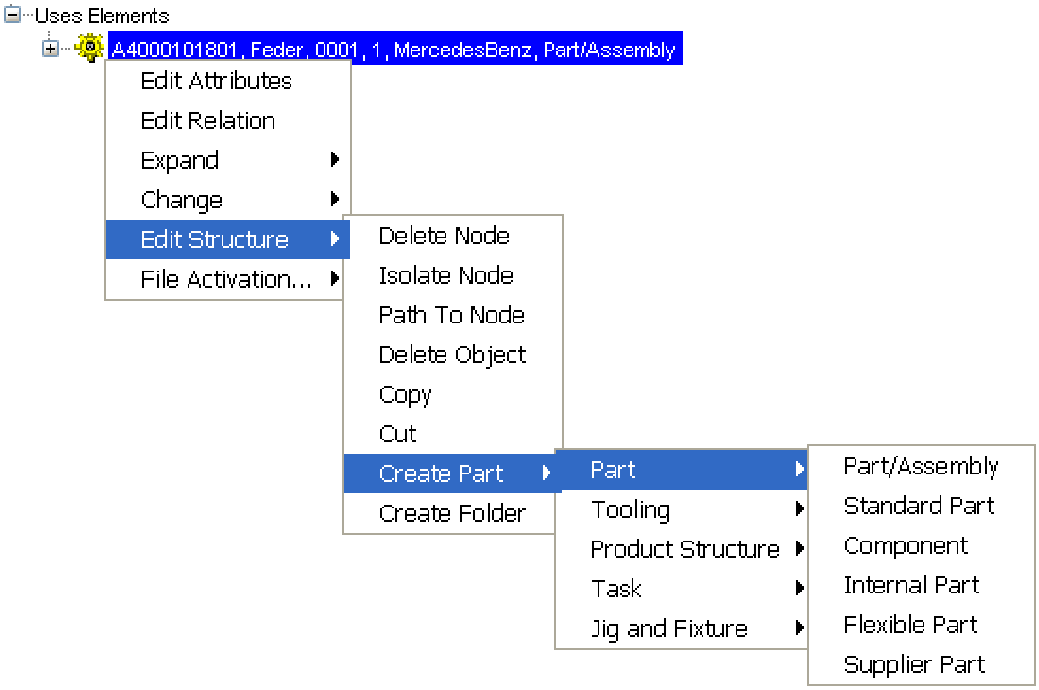


Figure 25: Part Actions

- Delete Node:
  Deletes the usage of the part in the assembly structure.
- Isolate Node:
  Delete all parts that are not sub-parts of the current part. The current part will become the new top-level part.
- Path to Node:
  Delete all parts that are not members of the path from the current part to the toplevel assembly.
- Delete Object:
  Deletes the part including all its usages in the assembly structure
- Copy
  Copy the part to the clipboard, so that it can be pasted later.
  NOTE: Copying does not mean to copy the part instance, but to copy the usage of the part. If the part is pasted later, you will get a new usage of the part, not a new part.
- Cut
  Delete the usage of the part, but put it into the clipboard so that it can be pasted later.
- Paste
  Paste a copied or cut part so that it becomes a child part of the current part.
- Create Part
  Create a new part that is a component of the Part on which the action was applied to. In Smaragd mode, the Create Part Menu contains sub menus for choosing the specific part type.
  NOTE: After the part has been created, you need to assign a valid part number.

If you select more than one item (by holding the control key or shift key pressed when selecting an item with the left mouse button), you can use the Action menu to perform an action that is then applied to all selected items.

Smaragd mode specific actions

You can use SAM to create new Smaragd assembly structure files (*.sma). To create such a file, choose menu button “File->Create New Smaragd File”.(see Figure 26). A file save dialog box will come up, so that you can choose the directory and name for the new file. After that, you will get a predefined structure with a part. You can then modify and extend this structure (see previous section).

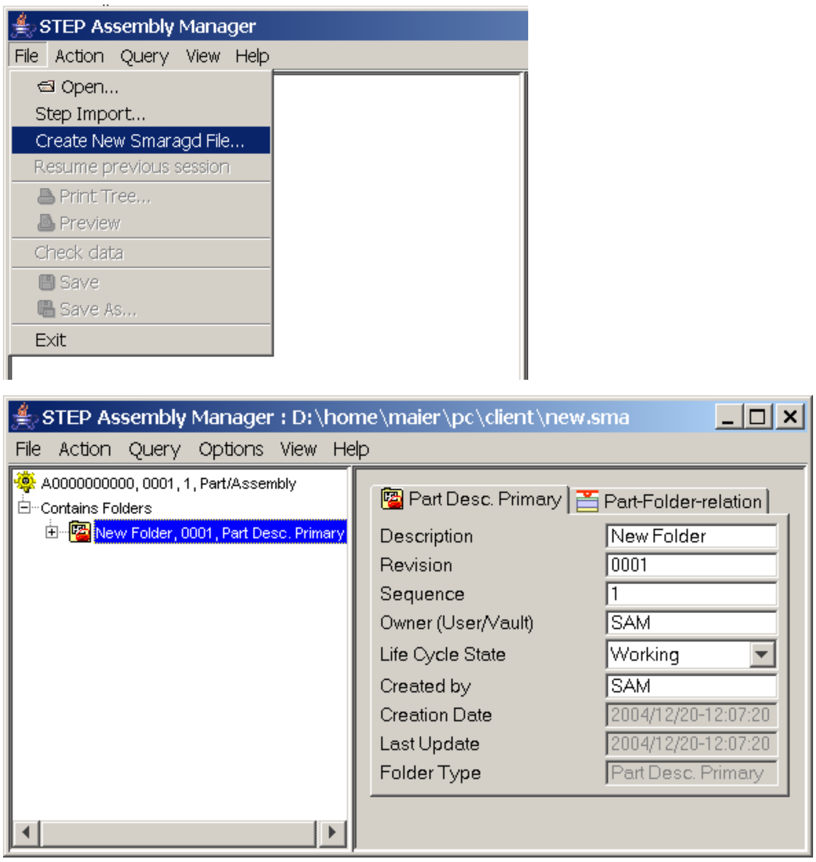


Figure 26: Creating new Smaragd files

In Smaragd mode SAM supports actions for changing the type of parts and documents and editing the product structure.

To change the type of a folder, click with the right mouse button on a folder, so that the context menu comes up as shown in the figure below. Select Change and choose the folder type you want to have.

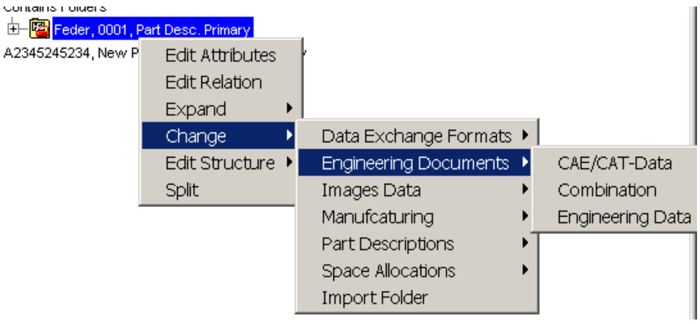


Figure 27: Changing the type of a Smaragd Folder

In the same way you can change the type of Parts.

In addition, the following actions are available:

- Create Folder
  Create a new folder attached to the part
- Merge
  This action is only available if the part has more than one folder. In this case all folders except the first are deleted and the files contained in the deleted folders are attached to the remaining folder.
- Split
  This action is available for folders containing more then one file. If you apply this action to a folder, a new folder will be created for each contained file.

Functionality "Create Toplevel Arrangement"

If a structure contains more than one toplevel element, an additional element can be created as superior element (one level above the existing elements). Therefore, the functionality *Create Toplevel Arrangement* is available in SAM. The functionality can be called from the menu *Action*. If a loaded structure consists of only one toplevel element, the functionality is deactivated.

The following attributes are created in order to show the needed information:

- Part Number
- Nomenclature
- Revision
- Sequence

An example is shown in
Figure 28:

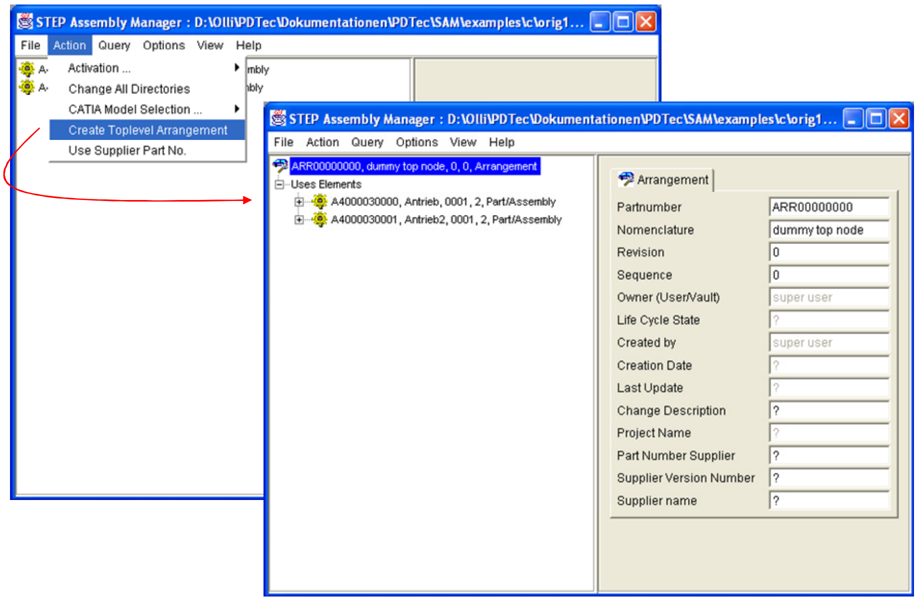


Figure 28: Functionality *Create Toplevel Arrangement*

Saving and Exporting Data

After you have imported and modified STEP data, you can save the file containing the assembly structure by using the File => Save or File => Save As menu entries. SAM saves the files in a native format which corresponds to the format of the originating system. However, this format is fit for saving data only, not for exporting and sending data back to the original sender. Once you have saved a file in native SAM format, you can open it again using the File => Open menu entry.

If you want to export data and send it back, you first have to decide which (CATIA) files should be included and which not. In GDG mode only the CATIA models that have been created or replaced in the assembly structure will be included into the export package by default. In Smaragd mode, all models are activated by default. If you want to send only particular files, you can do this by activating or deactivating them. You can either do this globally by using the menu entries Action => Activation => Activate All Models or Action => Activation => Deactivate All Models or Action => Activation => Deactivate Unchanged Files or you can do it for a particular file, model or document by clicking with the right mouse button on the item that should be (de)activated and selecting Activate or Deactivate in the context menu (see figure below).

In Smaragd mode there is an additional sub menu to deactivate CATIA catalogue files (see “Special option for CATIA catalogues

If the environment variable "SAM_SYNC_CATALOG_DIR" points to a CATIA catalogue directory, the catalogue parts will be handled in a special way at “Synchronize/Update”. Then each part in the CATIA structure will be checked whether it exists under the same name in the catalogue directory. If yes, it will be created in SAM-PDM as read-only (lock symbol) and deactivated (pale).

How to define a default part class”, Page 64).

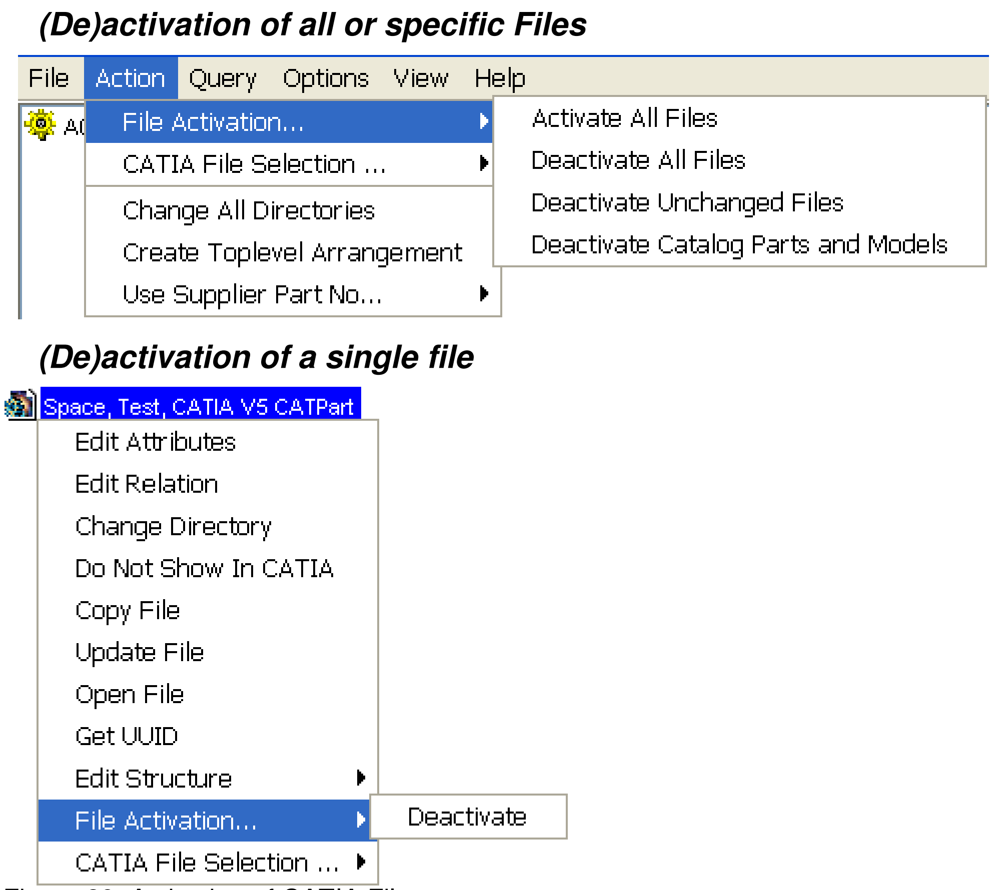


Figure 29: Activation of CATIA Files

If an item is deactivated it changes its color. If you activate it again the color will change to the original color. If you (de)activate a document or a model all containing files will be (de)activated.

If you want to send the data back, the activated CATIA files need to be converted into CATIA Export format (CATIA-V4 mode) and the assembly structure data has to be converted into STEP format. You can do this by selecting File => Step Export. After entering a file name and pressing the OK button, the CATIA export processes are started (CATIA-V4 mode), the results of the conversion are displayed in a similar window as for the import. In CATIA-V5 mode, the activated CATIA files are copied to the directory of the STEP file but not converted into CATIA export format. In GDG mode, after the conversion is finished the files will be packed into an archive (*.tar.gz) that contains the STEP file and all referenced external files that are to be included.

Note regarding installation: Packing into an archive (*.tar.gz) requires that the commands "tar" and "gzip" are available in the path.
Under Windows, this requirement is not fulfilled by default. In this case, the files created at STEP Export are copied into the specified target directory, but no archive is created.
See also note in the installation manual with an internet link for some free UNIX tools on Windows.

NOTE: Deactivated CATIA models are classified in the STEP file as "reference" to indicate that they are referenced but not included in the data package.

Exporting Structures to 4DNavigator

In GDG-Mode you can export a structure to 4DNavigator. Initially, the structure tree has to be expanded, so that all CATIA models that should be loaded into 4D-Navigator are visible in the tree. After that, choose File->Save As 4DNav Scene to create a wrl file (VRML) that can be loaded into 4DNavigator (see Figure 30).

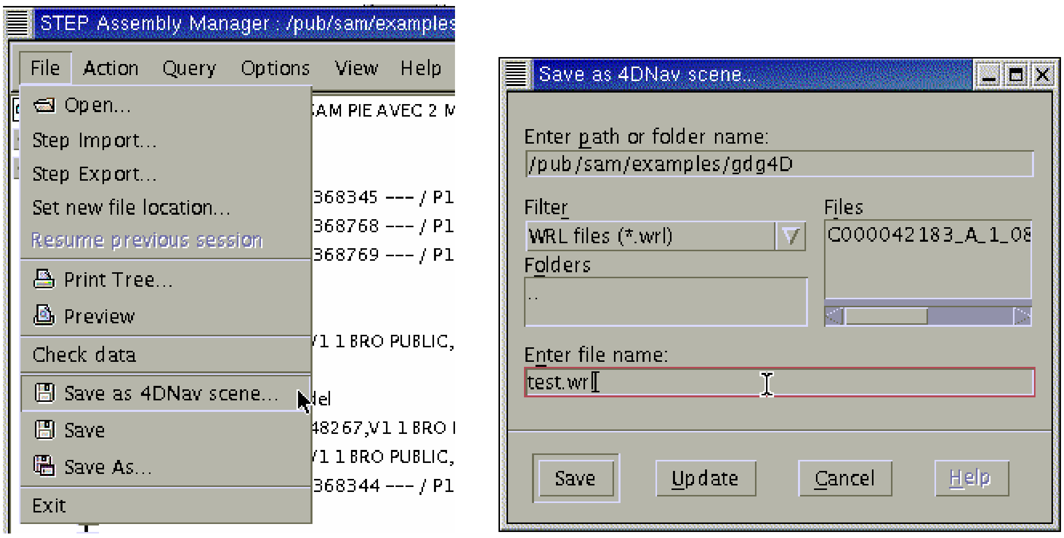


Figure 30: Export to 4DNavigator

Read-Only Items

A node in the tree may be marked as read-only. In this case its attributes cannot be modified and, if it is a CATIA file, it is read-only in CATIA (in CATIA-V4 mode, the CATIA model is loaded in "lock as passive" mode). If a node is read-only, it is marked by a small lock symbol as shown below. The read-only information is contained in the STEP file and cannot be removed by SAM.

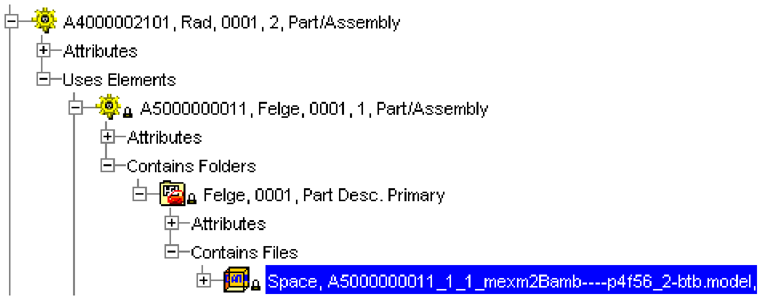


Figure 31: Read Only Parts and Models

CATIA Model Locking

If SAM is started in file locking mode, it provides functionality for checking in and checking out files and models. By default, all models and files are read-only. If you want to modify a CATIA model or a file, you need to check it out first. You can do this by clicking on the model icon with the right mouse button and choose “Check Out” in the context menu.

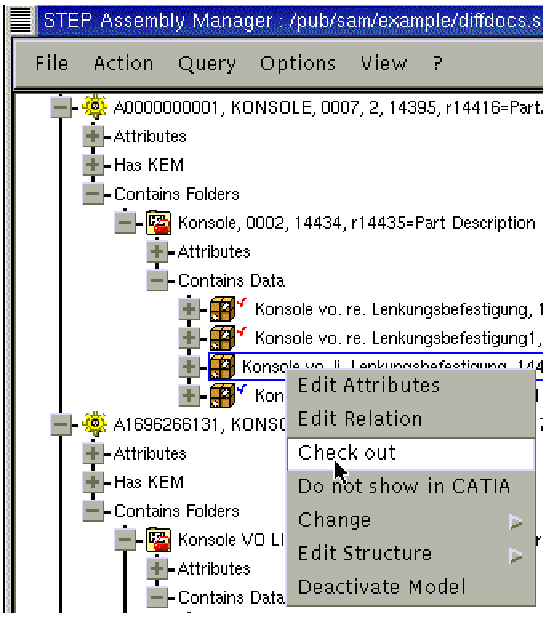


Figure 32: Checkout of CATIA Models

After you checked out the model, it will be marked with a blue and it is read-write. If you see a model marked with a red , somebody else has already checked out the model, and you cannot check it out, unless it is checked in by the one who has checked it out. For models that have been checked out by other users, the context menu contains an additional entry “Show Checkout Info”, if you select this, you will see who has checked out the file on which computer and when.

When you have finished your work on the model, you can check it in so that other users can work on the model.

Non CATIA File Handling

If you open the context menu for a non CATIA File, it will show the entries “Open File”, “Copy File” and “Update File”.

With “Open File”, you can start an application that is associated with this file type and view or edit the file.

With “Copy File”, you can create a copy of the file in another directory, edit or change it and then update it, i.e. copying it back to its original location, using “Update File”.

Copy Trafos (SAM/Smaragd)

This function is only available in the "Extended Structure Display".

In the context menu of a Trafo the menu item "Edit Structure" has a submenu that lets you copy the selected Trafo object into the buffer.

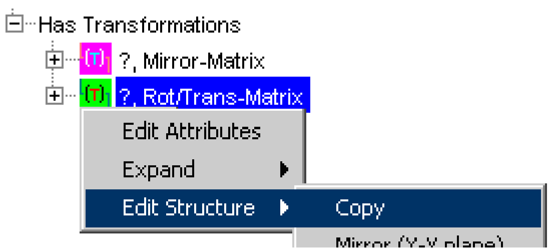


Figure 33: Copy Trafo into the buffer

Through the menu "Paste" the currently selected Trafo will be replaced with the one copied into the buffer before.

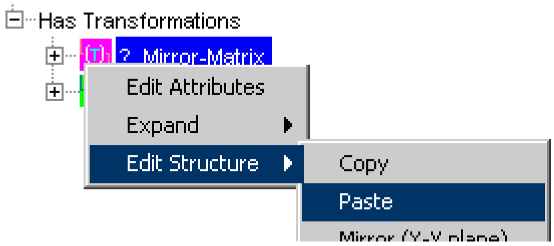


Figure 34: Paste Trafo from the buffer

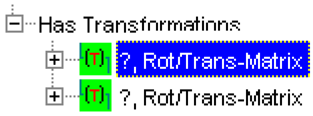


Figure 35: Inserted Trafo (original Trafo was replaced)

Mirror Trafos (SAM/Smaragd)

This function is only available in the "Extended Structure Display".

In the context menu of a Trafo the menu item "Edit Structure" has 3 submenues "Mirror (X-Y plane)", "Mirror (Y-Z plane) and "Mirror (Z-X plane)".

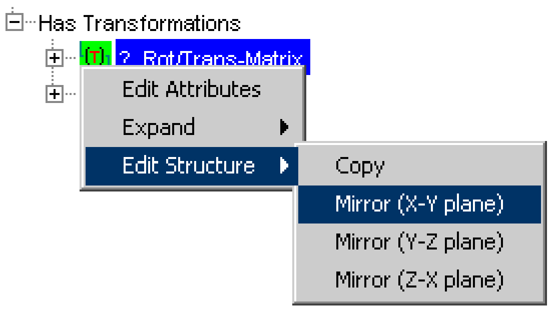


Figure 36: Context menu for the mirroring of a Trafo

Through the choice of one of these submenues the Trafo will be mirrored by the selected plane and the Trafo type will be changed accordingly.

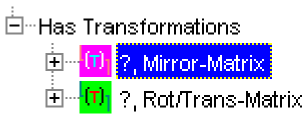


Figure 37: Mirrored Trafo

CHAPTER 3

The SAM CATIA V4 Function

This chapter describes the functionality of the CATIA V4 module SAM together with the SAM PDM GUI.

Menu structure in CATIA

The following figure shows the menu structure of the SAM CATIA module. In this structure it is possible to see which menu items can be reached through this CATIA function.

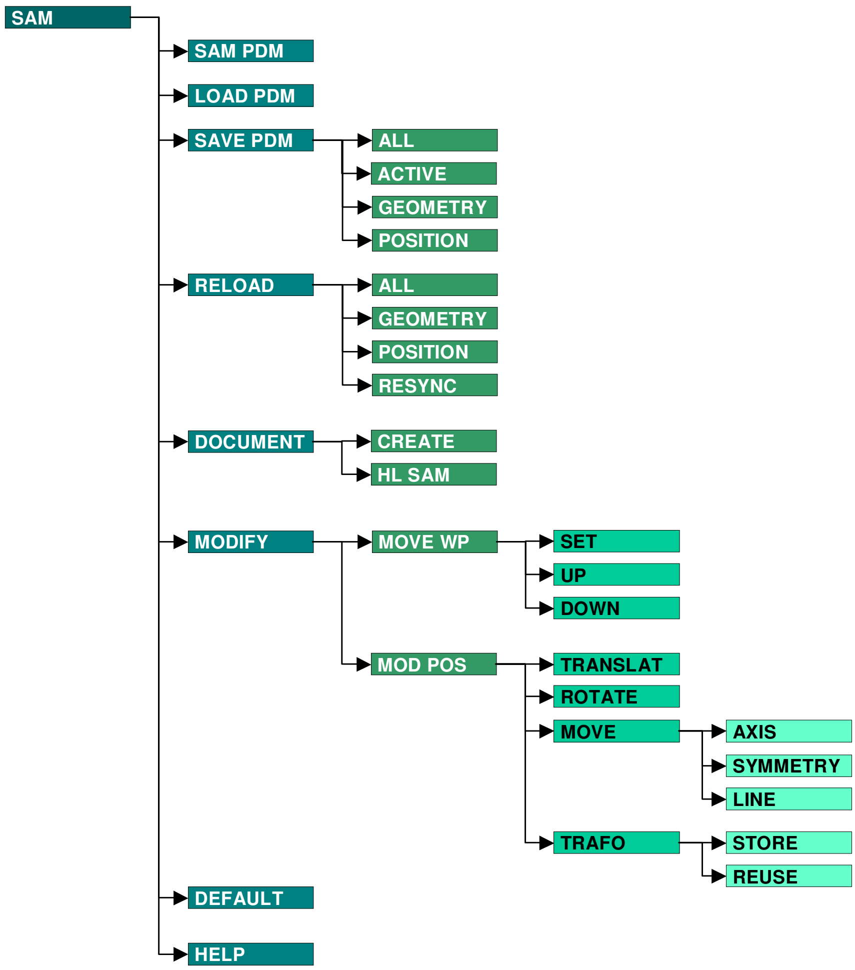


Figure 38: SAM CATIA function menu structure

Usage of the SAM CATIA function

The SAM function is designed conformably to the look and feel of the CATIA Motif user interface. If you are familiar with the CATIA user interface, you will be able to work with the SAM function just as well as usual.

If any message appears on the status line, you can type

/help

on the CATIA command line to get more information about this message.

To get help for the active menu point, you can type

/samhelp.

General

The SAM function manages hierarchical assembly structures of CATIA models.

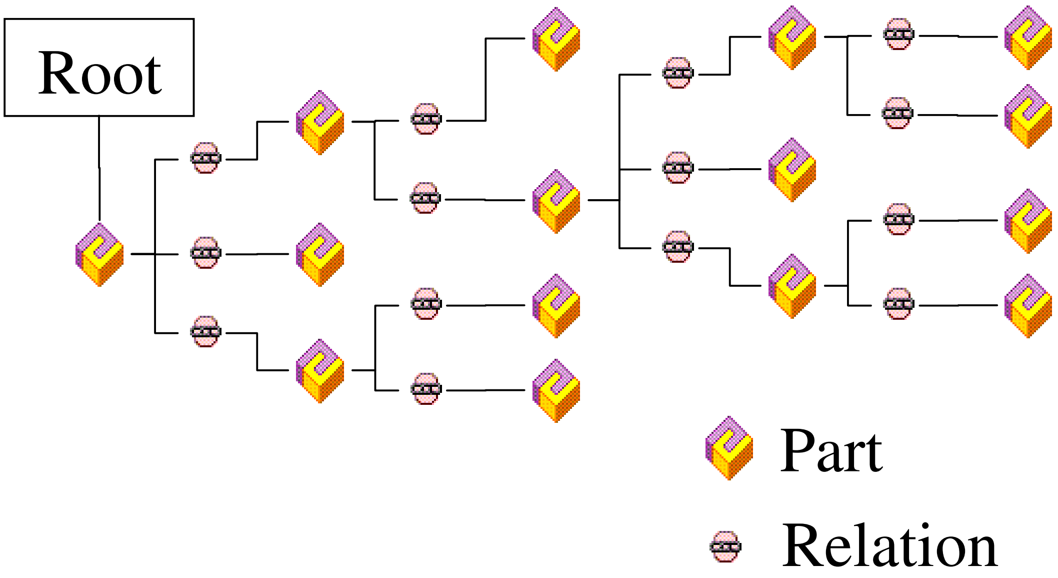


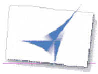
Figure 39: Assembly structure in SAM

In contrast to the flat session of the native CATIA, SAM uses a structured assembly. That means if you change a relation the total sub-assembly will change the position.

It is also possible to use some CATIA models multiple times. If you change the geometry of a multiply used CATIA model, the other models will be resynchronized after a swap to another model.

It is also possible to build in the same sub-assembly several times in one assembly structure. If there is a multiple use of a sub-assembly, the sub-assemblies will be automatically resynchronized.

SAM -> SAM PDM



Start the SAM PDM GUI and bring it into the foreground. If the SAM PDM GUI is already running, then it is brought into the foreground.

Confirm with YES

SAM -> LOAD PDM

Load the structure from the SAM PDM GUI

Confirm with YES

SAM -> SAVE PDM

Read Assembly Items with **SAM > LOAD PDM**

Modify the geometry or position information

Write the modifications back with **SAM > SAVE PDM > ALL**

After your confirmation a list of models that have been modified will come up in CATIA. If some models were active, but the user must not change these models, a status panel will pop up. It is not necessary to confirm this panel.

Select the models you want to update.

The selected models will be updated in the SAM structure.

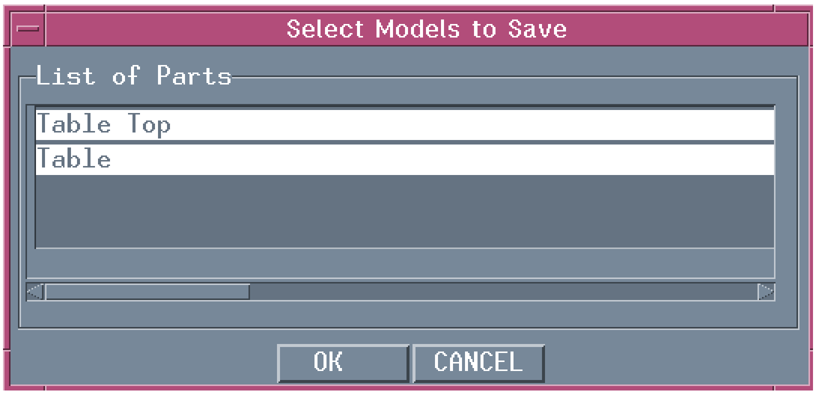


Figure 40: Dialog for SAVE PDM CATIA Models

Figure 41: Status Dialog before SAVE PDM

SAVE PDM > ACTIVE

This menu item updates the geometry information of the active model in CATIA on the disk. As a prerequisite the model must have been loaded from the SAM PDM GUI before.

Read CATIA items with **SAM > LOAD PDM**

Modify the geometry information

Write the modifications back with **SAM > SAVE PDM > ACTIVE**

After your confirmation the geometry information of the active model in CATIA will be updated.

SAVE PDM > POSITION

This menu item updates all position information changed in CATIA in the SAM structure. As a prerequisite the assembly must have been loaded from the SAM PDM GUI before.

Read CATIA items with **SAM > LOAD PDM**

Modify the position information

Write the modifications back with **SAM > SAVE PDM > POSITION**

After your confirmation all position information changed will be updated in the SAM structure.

SAVE PDM > GEOMETRY / MODELS

This menu item updates the geometry information of all activated models in CATIA in the SAM structure. As a prerequisite the models must have been loaded from the SAM PDM GUI before.

Read CATIA items with **SAM > LOAD PDM**

Modify the position information

Write the modifications back with **SAM > SAVE PDM > GEOMETRY**

After your confirmation a list of models that have been modified will come up in CATIA. If some models were active, but the user must not change these models, a status panel will pop up. It is not necessary to confirm this panel.

Select the models you want to update.

The selected models will be updated in the SAM structure.

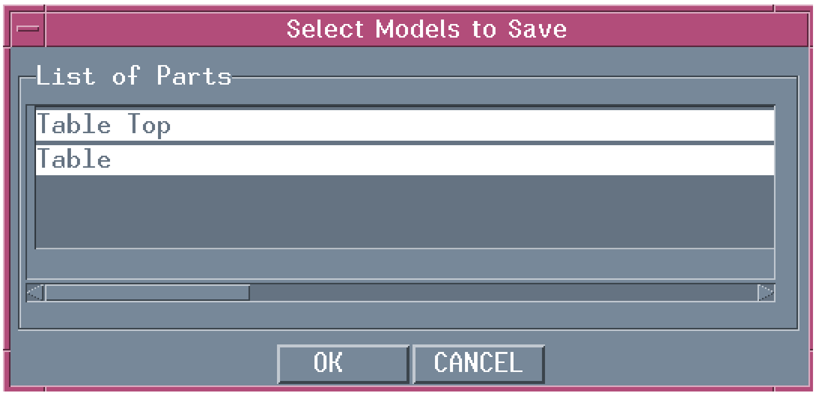


Figure 42: Dialog for Update CATIA Models

Figure 43: Status Dialog before Update

SAM -> RELOAD

This menu item reads CATIA models, CATIA documents and CATIA parts and assemblies placed in the SAM structure.

It works the same way as the SAM > LOAD PDM function, but in difference to it all functions do not delete the current CATIA Session. All SAM > RELOAD functions leave non-SAM models unchanged. This means the designer is able to use models/assemblies from SAM in conjunction with other models, e.g. models loaded using the CATIA File Open… function.

This function is also valuable in cases where you want to expand the assembly already loaded into the SAM PDM GUI in a different way (e.g. de-expand one sub tree and re-expand another one instead). After RELOAD, CATIA will reflect the new situation shown in the SAM PDM GUI.

Generally all SAM > RELOAD functions are much faster than the SAM > LOAD PDM because your current CATIA Session is only updated with the parts of the SAM PDM GUI content not yet loaded to CATIA.

RELOAD > ALL

All unsaved modifications (positions and models) will be updated with the original data from the SAM structure. RELOAD->ALL leaves non-SAM models untouched in the session.

RELOAD > GEOMETRY

RELOAD->GEOMETRY updates the current CATIA Session with the SAM PDM GUI content. In addition RELOAD->GEOMETRY resets all changed and unsaved models to their original condition. RELOAD->GEOMETRY leaves modified positions untouched. RELOAD->GEOMETRY leaves non-SAM models untouched in the session.

RELOAD > POSITION

RELOAD->POSITION updates the current CATIA session with the SAM PDM GUI content. RELOAD->POSITION resets all changed and unsaved positions. RELOAD->POSITION leaves modified models untouched.

RELOAD->POSITION leaves non-SAM models untouched in the session.

RELOAD > RESYNC

RELOAD->RESYNC updates the current CATIA session with the SAM PDM GUI content. Only models which are not already loaded in CATIA will be added to the current CATIA session. In difference to RELOAD->ALL RELOAD->RESYNC leaves your modified models and positions unchanged in your session.

This is the fastest way to synchronize the contents of CATIA with the SAM PDM GUI. RELOAD->RESYNC leaves non-SAM models untouched in the CATIA Session.

SAM -> DOCUMENT

The menu item ‘DOCUMENT’ contains actions to insert the active model in CATIA into the SAM structure.

DOCUMENT > CREATE

This menu item creates a SAM model from the model that is currently active in CATIA. A dialog window in the SAM PDM GUI appears and you can enter additional attributes (see *chapter 2, Creating and Replacing CATIA Files*).

Design or load your local CATIA model

In CATIA select **SAM > DOCUMENT > CREATE**

A SAM PDM GUI dialog window appears.

Fill out the fields and press ‘OK’

A new model will be created in the SAM environment.

> Notice:
> If the active file was opened with File->Open in CATIA, it will be replaced by the new model registered in SAM.
> If the active model came from SAM the new model will be added to the CATIA Session and made the active model.
> So in both cases you can continue to work with the model and later update it in SAM.

DOCUMENT > HL SAM

This menu item marks the selected model in the SAM PDM GUI (Highlight). This provides a link back from CATIA to the SAM PDM GUI.

Choose **DOCUMENT > HL SAM**

Select the model you want to mark in the SAM PDM GUI

After your confirmation the model will be marked in the SAM PDM GUI.

> Notice:
> In some cases, it is possible that CATIA crashes at a highlighting operation.
> So it is recommended to save before a highlighting operation.

SAM -> MODIFY -> MOVE WP

MODIFY > MOVE WP

The work plane defines the level in a sub-tree of a hierarchical assembly relative to which modifications in the hierarchical assembly become effective. The relative location and rotation between a part and one of its sub-parts or sub-structures is defined by a relationship. If there is only one tree structure, then the first part of the tree becomes the root. If there are several trees or other models in CATIA, then a virtual root joins the independent models and sub-trees to one tree.

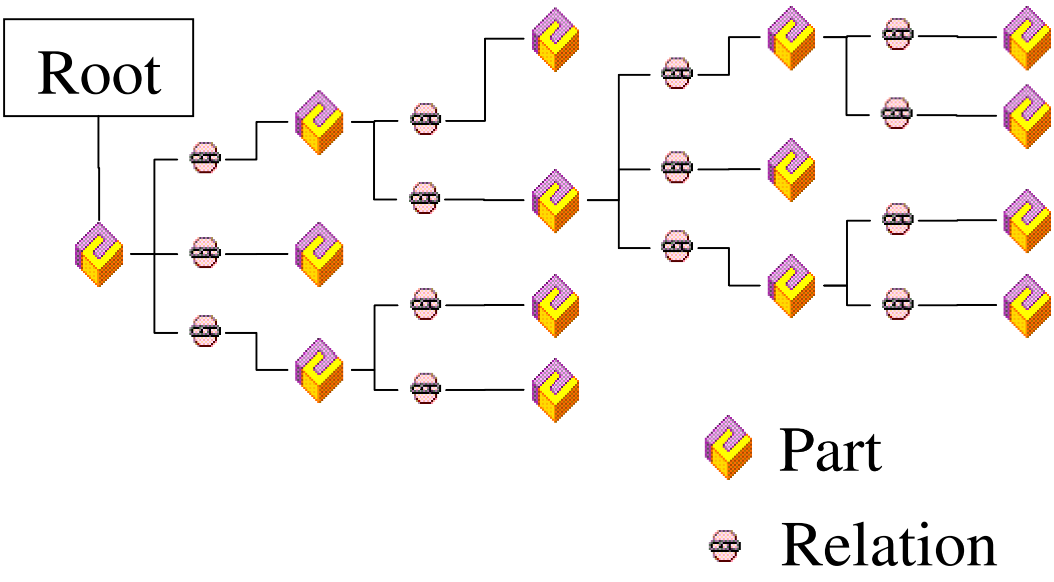


Figure 44: Tree structure with one tree

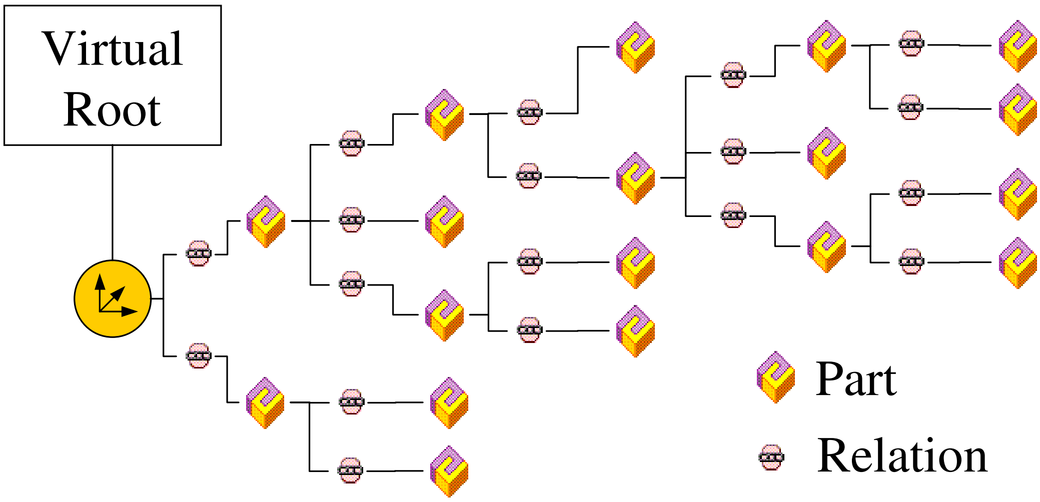


Figure 45: Tree structure with several trees and a virtual root

It is only possible to transform the relationships between the active *workplane* and its direct sub-parts and its sub-structures.

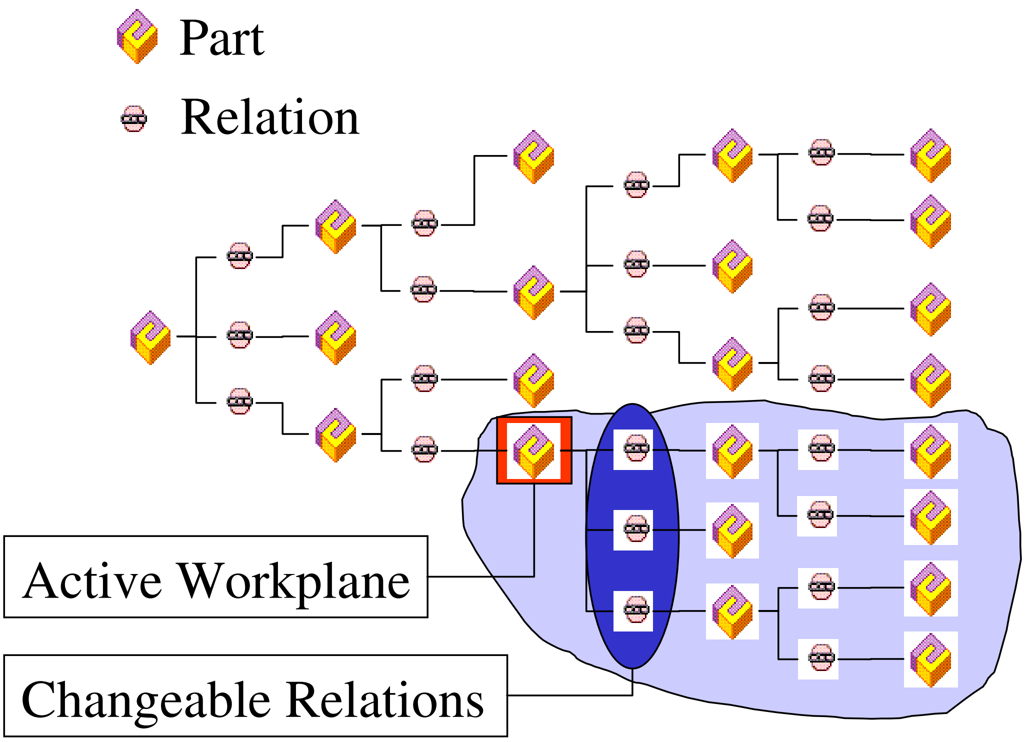


Figure 46: Active workplane in the tree structure

> The active work plane and the active model are different. If you want to edit a model you have to make it active.
>
> If you move or rotate a model in the structure by using the TRANSFOR Function then you do not change the relationship to the root structure element, but the geometry of the active model.

When working with SAM, the standard CATIA functionality to modify model positions in a multi-model structure (CATIA function MODELS) may not be used.

This is very important.

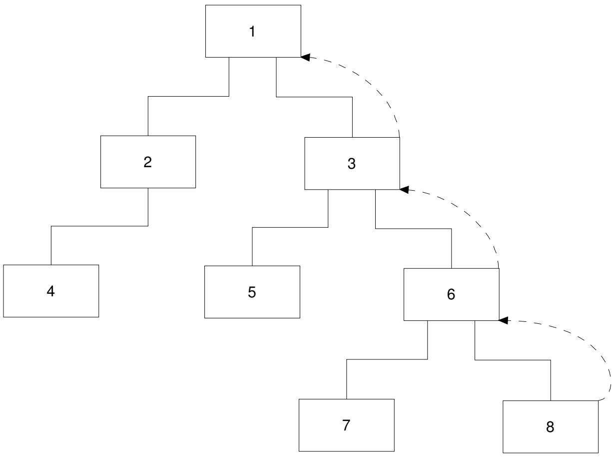
If you modify model positions by using the CATIA function MODELS, then these modifications will be lost as soon as you continue to work with SAM.

There are several possibilities to choose the active work plane:

MOVE WP > UP

CONFIRM with YES

Moving the *Workplane* up means that the *Workplane* will be a part in a higher layer of the structure, see in the figure below. The models below the active workplane are highlighted. If there is no higher layer a message will be displayed.



1.

Figure 47: Moving the Workplane ‘up’.

MOVE WP > DOWN

Moving the workplane down means that the active workplane crosses the tree-structure downwards, see in the figure below. The models below the active workplane are highlighted. If there is no lower workplane a message will be displayed.

CONFIRM with YES

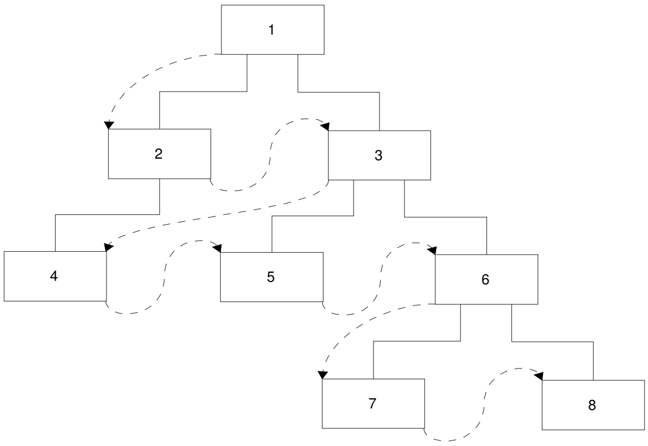


Figure 48: Moving the *Workplane* ‘down’.

MOVE WP > SET

Select Element:

The active workplane and its sub-trees are highlighted.

Or

CONFIRM with YES

Choose the new workplane from the list (the chosen part and its sub-trees are highlighted) and confirm with OK.

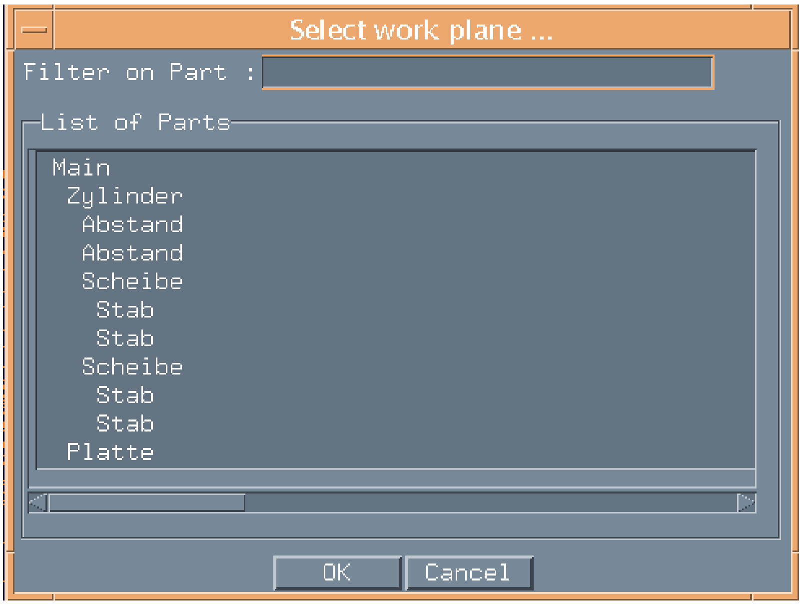


Figure 49: Selection panel for the new workplane

SAM -> DEFAULT

With the menu item "DEFAULT" you can switch the work plane functionality.

> If you do geometry transformations with the **MOD POS >** ... function then it is necessary to have the right work plane active
>
> This panel allows you to select the way in which the work plane is activated.

The default selection of the panel is Assembly.

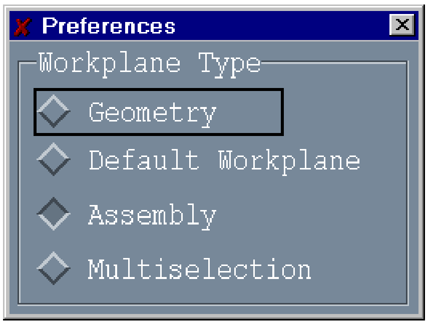


Figure 50: Default panel

Geometry

If you choose "Geometry", the MODIFY > MOD POS functions will change the position of the selected geometry (Model) instead of the Assembly. This option is only available if Geometry positions are supported in your customization of SAM.

Default Workplane

If "Default Workplane" is chosen, the work plane is automatically set to the next higher assembly in the assembly tree when you select a model.

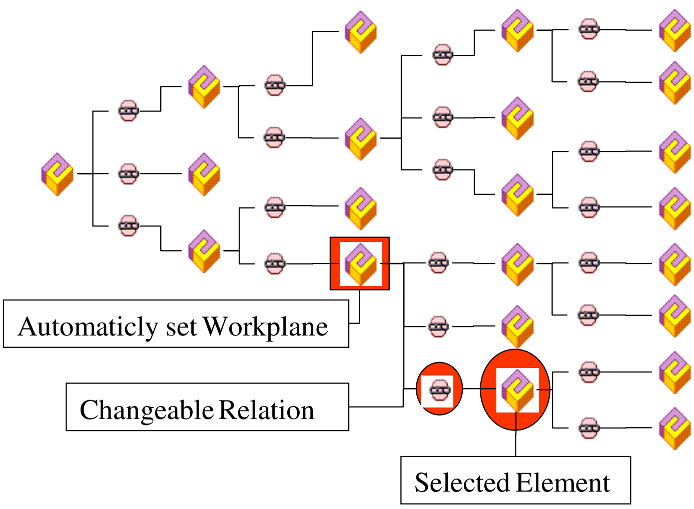


Figure 51: Automatically set default work plane

Assembly

If you choose "Assembly" you can set the work plane with the MODIFY > SET WP commands.

Multiselection

Choose Multiselection to apply MODIFY > MOD POS positioning to multiple Parts at once. The Parts to be positioned can be selected in a panel.
Note that only the leaves of an assembly are displayed for selection. Also a part that occurs in the assembly both as a leaf and with subassemblies is not displayed for selection.

To-do:

Check **SAM > DEFAULT** Multiselection

SAM> MODIFY > MOD POS > ...

Confirm to get a list of the selectable elements

Select the elements to be transformed (The selected elements are highlighted red) and confirm with OK.

In the figure below is possible to select part 1. It is impossible to select part 2, because it is not allowed to multiselect parts with sub-structures. If you drop all the sub-trees of part 2 out of CATIA then you also can multiselect part 2. After updating the changes you can RELOAD the dropped sub-structures.

Do the transformation as described in **SAM > MODIFY > MOD POS >** ...

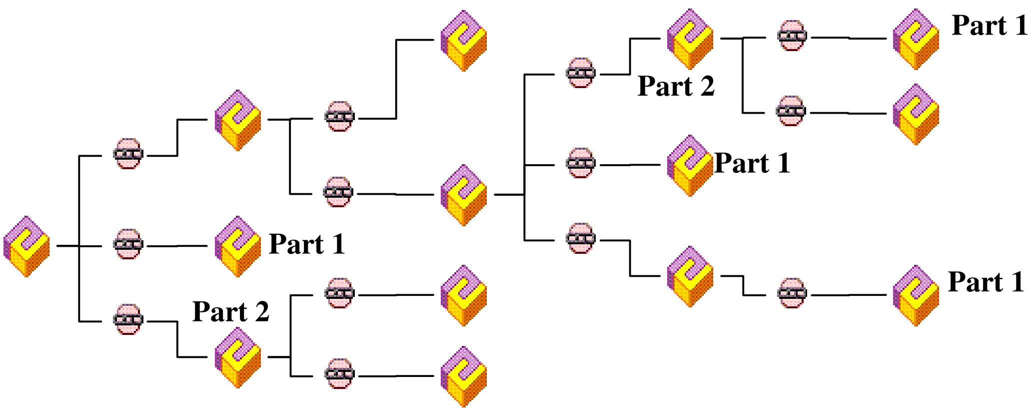


Figure 52: Example for Multiselection

SAM -> MODIFY -> MOD POS

MODIFY > MOD POS > TRANSLAT

This menu item (Translate) translates the selected part in the direction indicated by the selected line, axis or from point to point.

Read an assembly as described in chapter **SAM > LOAD PDM**

Select workplane as described in chapter **MODIFY > MOVE WP**

Select**: MODIFY > MOD POS > TRANSLAT**

Select a part in a sub-tree of the current workplane (the selected tree will be highlighted red).

Select a vector or line (the selected line/vector will be highlighted red). You can also select a first and second point (a red highlighted vector between the 2 points will be created).

Enter a distance or select the second point.

Click at the vector or press the ENTER Button in order to change the transformation direction.

YES NO Confirm with YES.
The selected part will be translated.

MODIFY > MOD POS > ROTATE

This menu item rotates the selected part around an axis.

Read an assembly as described in chapter **SAM > LOAD PDM**

Select workplane as described in chapter **MODIFY > MOVE WP**

Select: **MODIFY > MOD POS > ROTATE**

Select a part in a sub-tree of the current workplane (the selected tree will be highlighted red).

Select an vector/line (the selected vector/line will be highlighted red). In addition a rotation angle occurs to visualize the rotation direction.

Enter an angle or
Select a point/line/plane and a 2nd point/line/plane.
A rotation angle will be calculated and displayed.

YES NO Confirm with YES.
The selected part will be rotated.

MODIFY > MOD POS > MOVE > AXIS

This menu item moves the selected part from one coordinate system to another one.

Read an assembly as described in chapter **SAM > LOAD PDM**

Select workplane as described in chapter **MODIFY > MOVE WP**

Select:
MODIFY > MOD POS > MOVE > AXIS

Select a part in a sub-tree of the current workplane (the selected tree will be red highlighted)

Or

Confirm with YES

Select part in list

Select first axis

or

create your axis by:

Select a point to create the base

Select another point, line, plane to create the x-axis (change direction by ENTER)

Select another point, line or plane to create the y-axis (change direction by ENTER)

Confirm with YES to create the z-axis

Select second axis

or

create your axis by:

Select a point to create the base

Select another point, line or plane to create the x-axis (change direction by ENTER)

Select another point, line or plane to create the y-axis (change direction by ENTER)

Confirm with YES to create the z-axis

Click at the vector or press the ENTER Button in order to change the transformation direction.

Confirm with YES.
The selected part will be moved to the second coordinate system.

MODIFY > MOD POS > MOVE > SYMMETRY

This menu item mirrors the selected part in relation to a 2D plane.

Read an assembly as described in chapter **SAM > LOAD PDM**

Select workplane as described in chapter **MODIFY > MOVE WP**

Select:
MODIFY > MOD POS > MOVE > SYMMETRY

Select a part in a sub-tree of the current workplane (the selected tree will be highlighted red).

Select a plane (the selected plane will be highlighted red).

Confirm with YES.
The selected part will be mirrored.

MODIFY > MOD POS > MOVE > LINE

This menu item moves a part. The relative position between the first selected line and the selected part is identical to the relative position between the second selected line and the part after the action is completed.

Read an assembly as described in chapter **SAM > LOAD PDM**

Select workplane as described in chapter **MODIFY > MOVE WP**

Select:
MODIFY > MOD POS > MOVE > LINE

Select a part in a sub-tree of the current workplane (the selected tree will be highlighted red).

Select the first and second line (the selected lines will be highlighted red).

Confirm with YES.
The selected part will be moved.

MODIFY > MOD POS > TRAFO

The TRAFO functionality is to facilitate the work with all transformation routines of the SAM CATIA module.
This function gives the user the ability to reuse a transformation once defined multiple times. To work with this function at least one transformation must have been performed before.

MODIFY > MOD POS > TRAFO > STORE

This menu item stores the last transformation done (MODIFY > MOD POS> TRANSLATE / ROTATE / MOVE). The stored transformation can be used on any other part (TRAFO > REUSE).

Change the position of an assembly: **SAM > MODIFY> MOD POS > MOVE > ...**

Select:
MOD POS > TRAFO > STORE

Confirm with YES.
The last transformation part will be stored.

MODIFY > MOD POS > TRAFO > REUSE

After you have stored a transformation, you can reuse it on any part you want. This menu item reuses the last stored transformation.

Select work plane as described in chapter **SAM > MODIFY > MOVE WP**

Select a part in a sub-tree of the current workplane (the selected tree will be highlighted red).

Select:
MOD POS > TRAFO > REUSE

Confirm with YES.
The last transformation part will be reused on the highlighted part.

HELP

To get help for SAM you have 2 possibilities:

Click on SAM -> HELP, and the browser which was specified in the customization will appear in order to show the complete SAM user's manual.

To get help for a specific menu point of the SAM CATIA module do the following:

Activate the menu point for which you want to get the help.

Select in the Help Drop Down menu “SAM Context Help”

The browser which was specified in the customization will appear in order to show the Help chapter for the active menu point.

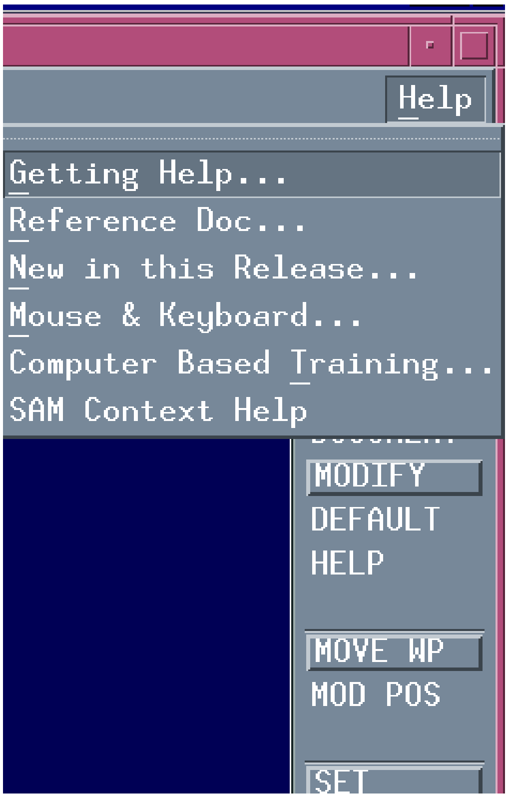


Figure 53: SAM Context Help

Please refer to the installation guide how to customize your default help viewer launching.

The Messages in CATIA

In addition to the online help, it is also possible to get more information to the messages inside the CATIA session on the CATIA command line as following:

/help

Following the list of message numbers and their meaning

Number	Panel output	Description
20501	MODEL LOAD OK	Your model has been loaded successfully.
20502	MODEL SAVE FINISHED	Your model has been saved successfully.
20503	MODEL UPDATE FINISHED	Your model has been updated successfully.
20504	LOAD OK	Your assembly is now loaded.
20505	TEMPORARY ASSEMBLY LOAD OK	Your assembly is now loaded temporary.
20506	ASSEMBLY UPDATE FINISHED	Your assembly has been successfully updated.
20507	ASSEMBLY DROP FINISHED	Your DROP action has been successfully finished.
20508	NO MODEL HAS BEEN CHANGED	No model or assembly has been changed since loading.
20509	MODEL IS ALREADY IN MAP	The model you want to create is already in exchange map. So a overwriting would be performed and could cause lost of model data.
20510	NO POSITION HAS BEEN CHANGED	No position has been changed since loading.
20600	INIT PDM MODULE	This is just a hint that the init process was started
20605	INIT OK	This is just a hint that the init process was ok
20606	REQUEST REFUSED	The request was refused because an EDM process is always pending. Please end the current action and try again.
20610	CAN NOT ALLOCATE PANELFILES	The panel files can not be found, please consult your CATIA system administrator
20615	ERROR IN PERSONAL ENVIRONMENT SCRIPT	Your private environment settings script is corrupt. Please contact your system administrator.
20616	ERROR IN GENERAL ENVIRONMENT SCRIPT	The general environment settings script is corrupt. Please contact your system administrator.
20620	FAILED TO OPEN THE LOGFILE	You have specified a log file in your setting. This log file cannot be opened.
20625	TEMPORARY PART	You have selected a temporary part, which cannot be transformed in its position.
20626	TEMPORARY MODEL	Your active model is a part of a temporary added document. Such models can't be created or updated. Please swap back to the model loaded active if you wish to save it.
20630	NO ASSEMBLY	You have selected a normal model that isn't a part of an assembly.
20631	DROP ERROR	An error occurred while you tried to drop a model. This may happen, if a drop to an active model was performed. To fix it, please change your work plane before DROP.
20632	NO ASSEMBLY LOADED	Only one model is loaded!
20633	NO MODEL LOADED	No model was loaded to perform this action.
20634	SYMMETRY NOT ALLOWED	Symmetry is allowed only for geometry position changing. Change the current working mode to geometry.
20635	ACTIVE MODEL WRONG	The active model is inside of the subassembly selected for modification, Please swap to a model outside of this subassembly.
20636	ALL MODELS IN READ ONLY ACCESS	Your active model is a temporary empty model; it is only used internally. Please swap to the model your selected action is related to.
20637	ACTIVE MODEL WRONG	You tried to replace an assembly outside of your work plane. Please change your work plane.
20638	NO MODEL SELECTED	No model was selected. Please change your work plane.
20639	ACTIVE MODEL HAS NO FRAME	Your currently selected active model is not your first model below the root. Only the first root model contains valid drawing frame attributes! No plot sheet will be generated.
20640	MODELNAME IS MISSING	You tried to save a model without a title. Please setup a title for this model.
20641	SOME MODEL(S) ARE NOT READABLE	CATIA was unable to read the following model(s):

20642	FAILED TO RENAME MODEL	The model RENAME function failed probably because of problems with your exchange map.
20643	MODEL IS NOT READABLE	CATIA was unable to read the model C01 located in C02
20644	FAILED TO SAVE MODEL	CATIA cannot save the active model in the PDM system. Check your access rights of this model in the PDM system.
20645	FAILED TO UPDATE MODEL	CATIA cannot update the active model in the PDM system.
20646	FAILED TO LOAD MODEL	CATIA cannot load the specified model.
20648	MODEL UPDATE NOT POSSIBLE	You can't update a model that isn't loaded from PDM system. Please use CREATE to insert the model in PDM.
20650	NO CORRECT MODEL FOUND	CATIA cannot handle this model in the PDM system.
20653	NO WRITE ACCESS TO MODEL	You tried to write a model in read only mode. You are not the owner of this model.
20654	NO ROOT MODEL	The assembly you have loaded has no root or an empty root.
20655	INCORRECT ASSEMBLY	Something is wrong with your assembly.
20656	MODIFICATION NOT ALOWED	It isn't allowed to modify the selected model; this model is a part of root.
20760	ACTION CANCELED	The action performed was canceled by EDM system.
20761	MODEL HAS WRONG PROJECT	Some Models have incompatible project environment settings to your CATIA project. Check your project environment.
20762	ASSEMBLY PARTIALLY UPDATED	Your assembly has been successfully updated, For more information refer your EDM system.
20763	MODEL PARTIALLY UPDATED	Some models of the splitted document couldn't be updated.
20764	COMMUNICATION FAILED	Is your workbench open and initialized?
20765	ACTION FAILED	The action performed was ended with an error.
20800	PANEL ERROR	The panel function failed, check if the desired panel definition exists in the panel install directory.
20801	NO INFO RETURNED	No info available for this model
20810	NO VALID PLOT FORMAT ERROR	The ditto in current view isn't holding a valid plot format. Please check the ditto.
20811	PLOT FINISHED	The plot is finished.
20812	PLOT IN PROGRESS	The plot is in progress.
20900	PLOT FORMAT TYPE ERROR	The selected plot format is invalid.
20910	CANNOT MOVE WORKPLANE	The work plane can't be moved in this direction. The deepest or highest level in assembly structure has been reached.
20911	FAILED TO CHANGE WORKPLANE	No valid document found to set the WP.
20920	MODEL IN EXCHANGE MAP ALWAYS EXISTS	A model with such a name always exists in your exchange map. Please rename your model or check in the model in your exchange map first and retry.
20921	NO MODEL HAS BEEN LOADED	Before something can be added temporary, a model must be loaded with the read function.
20922	WARNING: MAP NOT FOUND IN DECL SERIES	The map path searched for is not defined yet in your declaration series. A new map will be created temporary.
20923	NO MAP NAME FOR EXCHANGE MAP FOUND	There is no MAP entry in your environment. Please contact your system administrator.
20924	NO VALID MAP PATH OR DLNAME FOUND	Couldn't find a valid entry in declaration series For the given search criteria. Maybe the map you want to read from is invalid Please consult your CATIA system administrator.
20930	SAME POINT SELECTED	Please select 3 different points.
20931	WRONG VECTOR SELECTED	Please select the transformation vector to change the direction of current transformation.
20932	SAME AXIS SELECTED	Please select 2 different axes.
20940	SELECTED MODEL IS NOT IN ASSEMBLY	The selected model belongs not to an assembly loaded, so it can't be managed here. Please use the standard CATIA functions
20941	EXCHANNG MAP PATH DOES NOT EXIST	The specified exchange map does not exist Please contact your system administrator.
20942	CHANGES TO RO MODELS NOT ALLOWED	Update a read only model.
20943	MODEL DOES NOT EXIST	The model does not further exist in CATIA memory. The model was deleted.
20944	MODEL WRITE WAS NOT SUCCESSFULL	The model could not be correctly written to Map. Maybe disk storage is low or NFS is poor. Please contact your system administrator.
20950	WARNING: CHANGES IN SUBASS	There are changes in the subassembly selected for DROP. All model and relation changes have been made in this subassembly will be lost if you continue. If you are unsure save the assembly before DROP. No model or assembly to update. The structure you have loaded contains no model.
20960	NOTHING TO SAVE/UPDATE	No model or assembly to update. The structure you have loaded contains no model.

20965	NO TRANSFORMATION STORED	Before you are able to use this function you have to store the last transformation you have done.
20966	NO VALID TRANSFORMATION FOUND	Unable to found a valid transformation for reuse. First a position must be changed, to be able to store a transformation.
20967	NO TRANSFORMATION DONE YET	First a position must be changed, to be able to store a transformation.
20970	NO HELP SYSTEM DEFINED	A help system (Netscape or mosaic) must be defined. Please contact your system administrator.
20971	CAN NOT CONNECT TO HELP SYSTEM	Problems to connect to help system. Please contact your system administrator.
20972	NO HELP FILE FOUND	No or a wrong help file path was defined in your environment. Please contact your system administrator.
20973	MODEL IS NOT PART OF WORKPLANE	The model you've tried to change is not a part of a subassembly defined by the work plane. Please reset your work plane.
20974	FAILED TO READ STORAGE	Failed to read session data from application data. Please contact your system administrator.
20975	MODEL NEED TO BE WRITTEN	Before you can add a new model to an assembly you have to save this model on file system.
20976	CATIA SESSION HAS BEEN CHANGED	Current CATIA session has been changed. Please redo your selections.
20980	NO LICENSE NUMBER SET	No license number is set in environment. Please contact your system administrator.
20981	LICENSE INIT FAILED	Could not init license process. Please contact your system administrator.
20982	NO MORE FREE LICENSES	All available licenses are currently in use. Retry later or contact your system administrator.
20983	LICENSE EXPIRED	Your license has been expired. Please contact your system administrator.
20984	NO LICENSE FOR MODULE	No license installed for this module. Please contact your system administrator.
20985	DIFFERENT MACHINE TIMES	The LLD machine time differs from the GLD machine time. Please contact your system administrator.
20986	NO CONNECTION TO LLD	No connection to local license daemon. Please contact your system administrator.
20987	NO CONNECTION TO GLD	No connection to global license daemon. Please contact your system administrator.
20988	UNKNOWN LICENSE ERROR	Something is wrong with your license environment. Please contact your system administrator.
20989	UNABLE TO START LLD	Please check your license environment or contact your system administrator.
20990	NO MODEL SELECTED	No model was selected.
20991	NO MODEL FOUND	No model found.
20992	NO POINT SELECTED	No point was selected.
20993	SAME POINT SELECTED	Please select 3 different points.
20994	POINTS ARE LYING ON ONE LINE	Please select at least one different point.
20995	POINTS ARE IDENTICAL	Distance between two points is too small.
20996	INVALID ANGLE VALUE	Input angle is less than angular tolerance. Invalid value for defining rotation.

CHAPTER 4

The SAM CATIA V5 Module

This chapter describes the functionality of the CATIA V5 module SAM together with the SAM PDM GUI.

SAM Toolbar

The following figure shows the toolbar of the SAM CATIA module. Descriptions of the toolbar and the command icons follow the diagram.

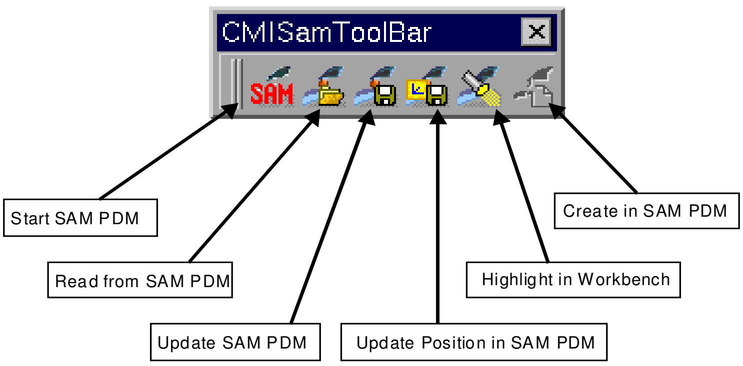
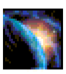


Figure 54: SAM Toolbar in CATIA V5

The icon ‘Create in SAM PDM’ is only active if you are working with a CATPart in a CATPart-specific Workbench or with a CATDrawing in a CATDrawing-specific Workbench.

SAM toolbar: Start SAM PDM



Click on the **‘Start SAM PDM’** icon .

The SAM PDM GUI is started and brought into the foreground. If the SAM PDM GUI is already running, then it is brought into the foreground.

SAM toolbar: Read from SAM PDM

Click on **'Read from SAM PDM'** icon .

The structure displayed and expanded in the SAM PDM GUI is loaded into CATIA.

When a PDM structure is loaded into CATIA, the part structure is mapped to a CATProduct structure.

CATProducts that are already existing in the assembly structure in the SAM PDM GUI are loaded into CATIA. For parts that do not yet have a CATProduct, a CATProduct is created on the fly and loaded in CATIA. The CATParts referred by the CATProducts are also loaded into CATIA. If the PDM structure also contains CATIA V4 models, they are also loaded into the structure in CATIA V5. So, the resulting CATIA V5 structure can contain both CATParts and CATIA V4 models. Position information is taken from the PDM structure. The transformation matrices in the CATProducts are updated accordingly.

In the GDG view, only the CATProducts that are attached to "GDG Composition" objects are CATProduct files. All other CATProducts except the top-level CATProduct are CATIA "components".

In the Smaragd view, all CATProducts are CATProduct files.

The CATIA files that are to be loaded into CATIA are at first copied into the SAM "exchange map" directory and then loaded from there into CATIA.

> In the CATIA V5 options (Tools/Options/General/General, Referenced Documents), the "Load referenced documents" checkbox has to be activated.

> In the CATIA V5 options (Tools/Options/General/Document, Linked Document Localization), the "folder of the pointing document" has to be high priority.
>
> Thus, CATIA will be able to load the files following the links contained in the CATProduct and CATDrawing files, even if the links have full path names corresponding to the directory where they were saved before importing them with SAM.
>
> The pure file name components in the path names are not changed by SAM.

If a CATProduct loaded from the SAM PDM GUI contains a link (instance) to a part (CATPart or CATProduct) that does not correspond to the PDM structure, then CATIA will try to resolve this link in the CATProduct at load-time, will not find the missing part, and will represent the missing part with a broken-link icon (greyed out – meaning hidden, see CATIA Hide/Show operation). In the case that the part is existing in the PDM structure to be loaded but the instance name contained in the CATProduct does not correspond to the PDM structure, this instance of the part will also be represented by a greyed out icon (hidden).

Some of the PDM attributes are mapped to CATIA V5 properties (Part Number, Revision, Nomenclature and Instance Name) for the CATProducts and CATParts.

If the assembly structure in the SAM PDM GUI also contains CATDrawings, they are also loaded into CATIA. Each CATDrawing is displayed in a separate CATIA Drafting workbench window.

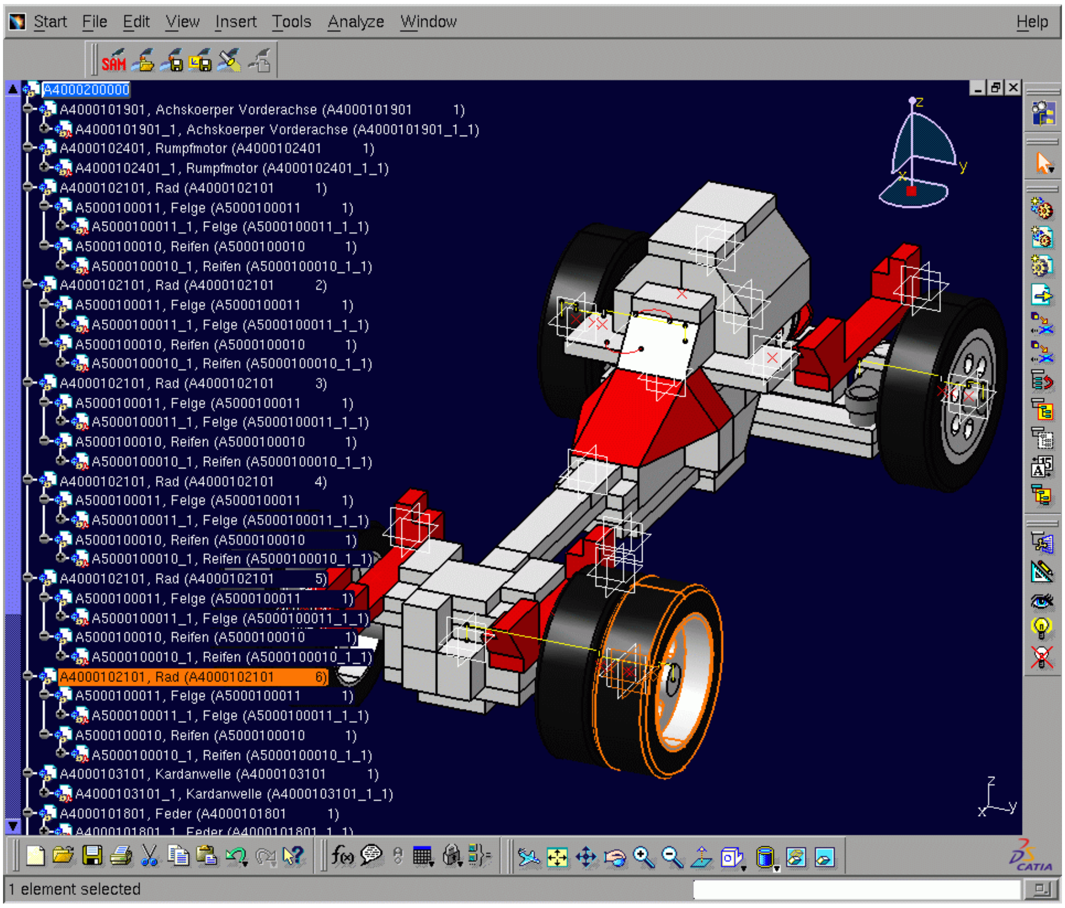


Figure 55: Sample structure loaded by SAM

Special option: Add SAM structure to an already loaded structure

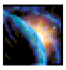
In case of the environment variable setting "CMI_ENABLE_USEROOTASCMIWORKBENCH=ON" there may be in the CATIA V5 options (Tools/Options/General/Compatibility/CMI) an additional checkbox "Use existing Product as temporary CMI Workbench Product". If you activate this checkbox (if existing) and the checkbox "Use one temporary product window for all CMI Workbench contents", then SAM will add the structure from the SAM PDM GUI to the already existing product structure in CATIA, if there is already a CATIA product structure existing that was not loaded by SAM.

Warning:
If these checkboxes are activated, the functionalities for saving a structure from CATIA to SAM PDM like “Update SAM PDM”, “Update Position in SAM PDM” or “Update Part in SAM PDM” (see below) are not supported.

Special option: Single part modus

In case of the environment variable setting "CMI_ENABLE_SINGLEPARTMODUS_READ=ON" there will be in the CATIA V5 options (Tools/Options/General/Compatibility/CMI) an additional checkbox "Single Part Modus". If you activate this checkbox, then SAM will load each CATPart from the SAM PDM GUI in a single CATIA Part Design workbench window (without CATProduct structure).

SAM toolbar: Update SAM PDM



Modify the contents in CATIA and the positions in the CATIA product structure.

Click on **'Update SAM PDM'** icon  to write the modifications back.

The CATIA files loaded by SAM (except temporary CATProducts if used) are saved in the "exchange map" directory and in their original directories.

The positional information in the current CATIA product structure (transformation matrices and CATProduct files - if used) is updated in the SAM PDM GUI.

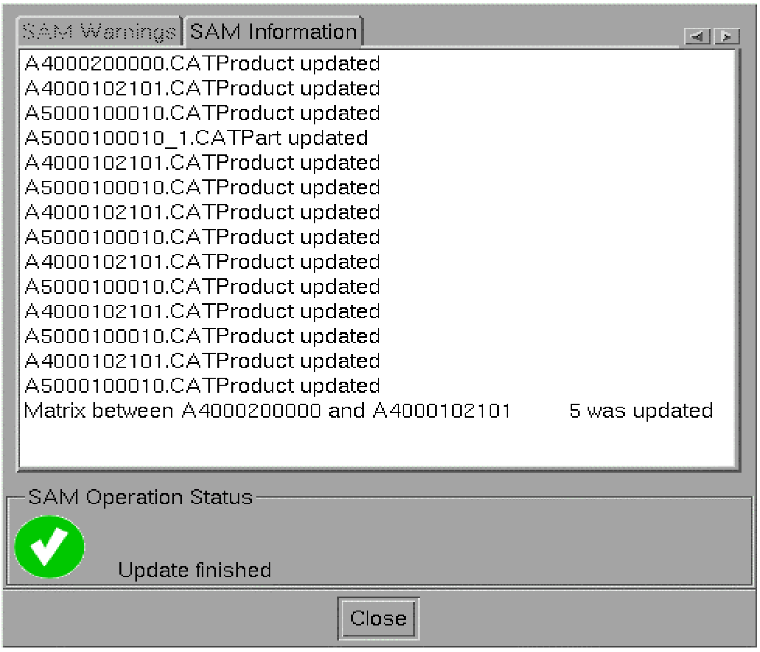


Figure 56: Information messages at "Update SAM PDM"

The current CATProduct structure is scanned and checked whether it is consistent with the structure in the SAM PDM GUI. If it contains CATProduct sub-assemblies or CATParts that don't correspond to the SAM PDM assembly structure, these changes are rejected.

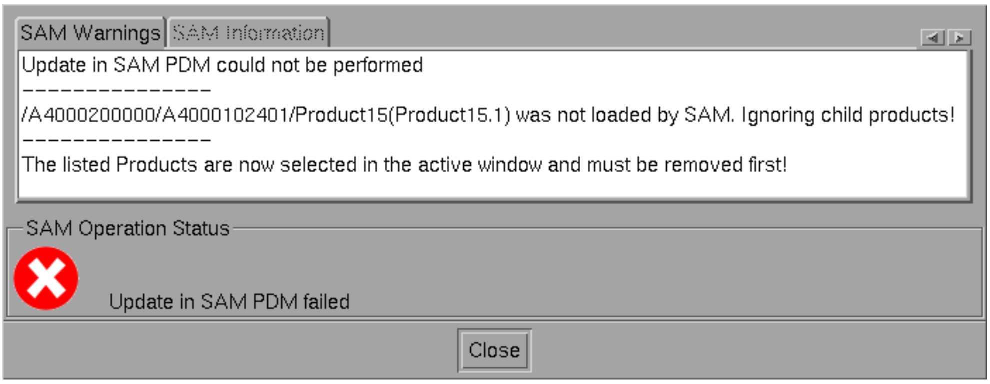


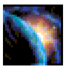
Figure 57: Error messages at "Update SAM PDM" if structure is not consistent

Special option: Select modified transformation matrices

In case of the environment variable setting "CMI_ENABLE_UPDATEPOSITIONDIALOG=ON" you can select at the "Update SAM PDM" operation which of the modified transformation matrices are to be saved.

If this environment variable is not set, then all modified transformation matrices will be saved at the "Update SAM PDM" operation.

SAM toolbar: Update Position in SAM PDM



Modify the positions in the CATIA product structure.

Click on **'Update Position in SAM PDM'** icon  to write the position information back**.**

The positional information in the current CATIA product structure (transformation matrices) is updated in the SAM PDM GUI.

The difference to the "Update SAM PDM" operation described above is that the CATIA files are not saved.

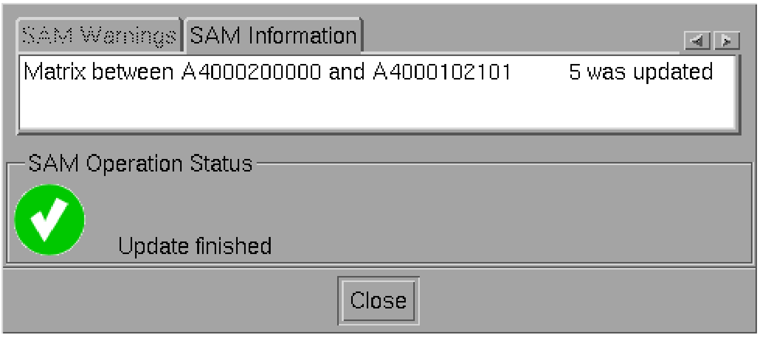
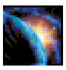


Figure 58: Information messages at "Update Position in SAM PDM"

SAM toolbar: Highlight in Workbench



In CATIA, select a CATPart or CATIA V4 model that was loaded by SAM (click on the geometry or on the CATPart/model instance in the CATIA specification tree).

Click on **'Highlight in Workbench'**  icon.

The selected CATPart/model instance will be highlighted in the SAM PDM GUI.

SAM toolbar: Create in SAM PDM

With this toolbar button a CATPart or CATDrawing currently set active in CATIA V5 can be created in SAM.

Design or load your CATPart or CATDrawing.

Click on **'Create in SAM PDM'** icon .

In the SAM PDM GUI, a create dialog window appears to enable the user to set additional attributes (see *chapter 2, Creating and Replacing CATIA Files*).

The CATPart or CATDrawing will be saved in the specified directory and it will be included in the SAM PDM structure at the specified node.

> The "Create in SAM PDM" operation is available only if the CATPart or CATDrawing has been opened in a separate Part Design or Drafting Workbench window in CATIA V5.

After the "Create in SAM PDM" operation, you can close the Part Design or Drafting Workbench window.

To load the updated structure with the new CATPart or CATDrawing from the SAM PDM GUI into CATIA, click on the "Read from SAM PDM" icon.

SAM toolbar: Synchronize SAM

In case of the environment variable setting "SAM_ENABLE_SYNCHRONIZECMD=ON" the “Synchronize/Update“ functionality is active. The SAM toolbar displays the “Synchronize SAM“ icon in addition.

Figure 59: SAM Toolbar with “Synchronize SAM”

The “Synchronize/Update“ functionality is available in the Smaragd mode only.

With “Synchronize/Update” you can take over a CATProduct structure from CATIA V5 into a PDM structure in the SAM PDM GUI. The dialog has 2 phases: In phase 1 “Validate” you select for each CATProduct the type of the respective part to be created in the SAM PDM structure. In phase 2 “Synchronize” the CATProduct structure with the CATIA product properties is taken over into the SAM PDM structure step by step, allowing you to edit the attributes of the parts to be created. The CATIA files with their CATIA product properties are renamed accordingly so that the CATIA structure remains consistent.

Example: The small assembly listed below is given as a CATProduct structure in CATIA V5 and is to be taken over into a SAM PDM structure using “Synchronize/Update”.

Step 1: Load CATProduct structure in CATIA.

Step 2: Start SAM PDM GUI. The “new file location” in SAM PDM must point to a valid directory as, during the synchronize dialog, the CATIA files in the structure are copied into this directory.

Step 3: Phase “Validate”. Click on the “Synchronize SAM” icon in the SAM toolbar. The “Synchronize/Update” dialog is started, beginning with the CATIA window “SAM Synchronize Structure”. Select a CATProduct in the window and click on “Edit PDM-Part category”. For each CATProduct you have to select the type (“category”) of the part to be created.

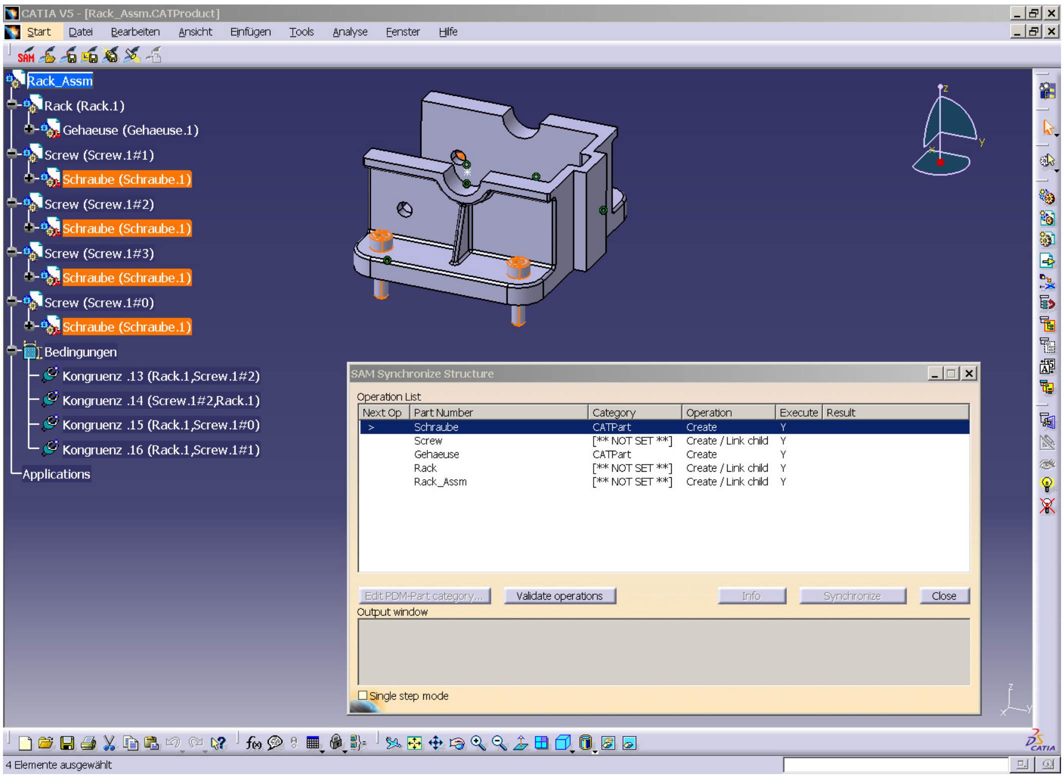


Figure 60: SAM Synchronize step 3 before selecting the part category

Step 4: Phase “Validate”. When you have selected the “category” for all CATProducts, confirm by clicking on “Validate operations”.

Step 5: Phase “Synchronize”. Click on “Synchronize” to start the 2nd dialog phase. The CATIA structure is traversed bottom up step by step. Here it starts with a CATPart. For this, an additional SAM PDM window pops up where you can edit the CATPart attributes for SAM PDM. The CATIA product properties are used as default values for Part Number and Description.

Step 6: Phase “Synchronize”. If the parent CATProduct has several child CATParts, then the dialog continues with one of these CATParts (does not occur in this example). Otherwise the dialog continues with the parent CATProduct. For this, an additional SAM PDM window pops up where you can edit the part attributes for SAM PDM. The CATPart Part Number edited before is used as default value for the Part Number of the part. After this the CATIA structure traversing is continued until the top-level CATProduct is reached. The structure tree in the SAM PDM GUI is continuously updated.
Once you confirm with “Always OK” in the additional SAM PDM window, the rest of the CATIA structure will be automatically traversed.

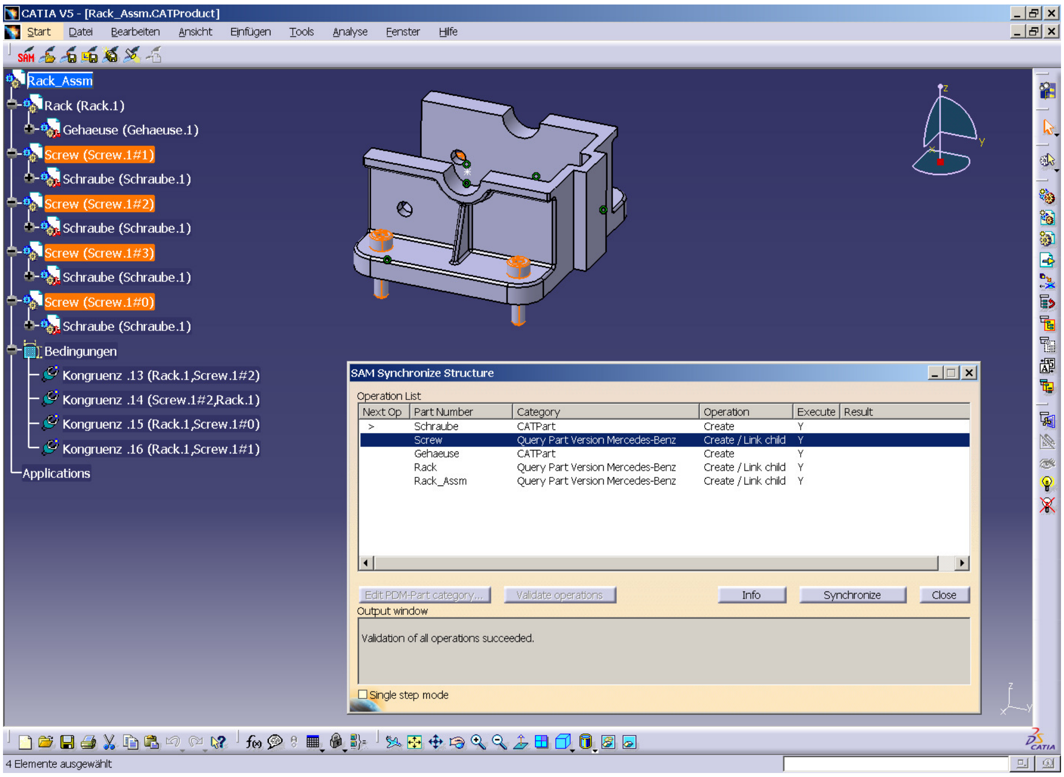


Figure 61: SAM Synchronize step 4 after "Validate operations"

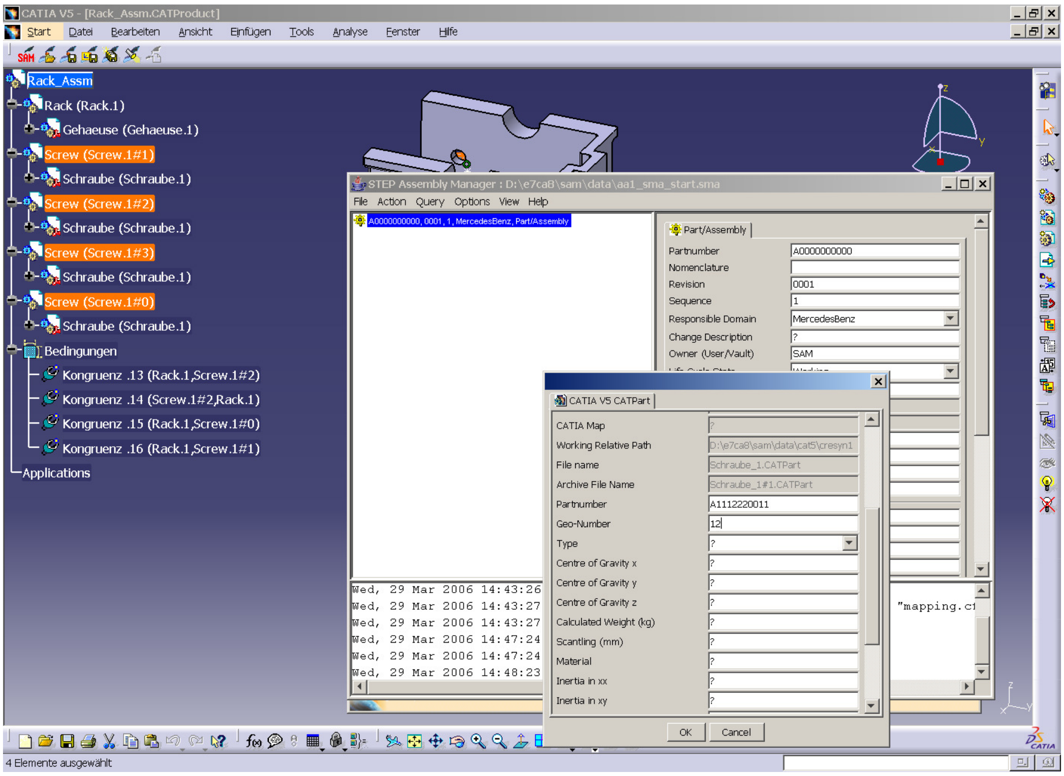


Figure 62: SAM Synchronize step 5: edit CATPart attributes

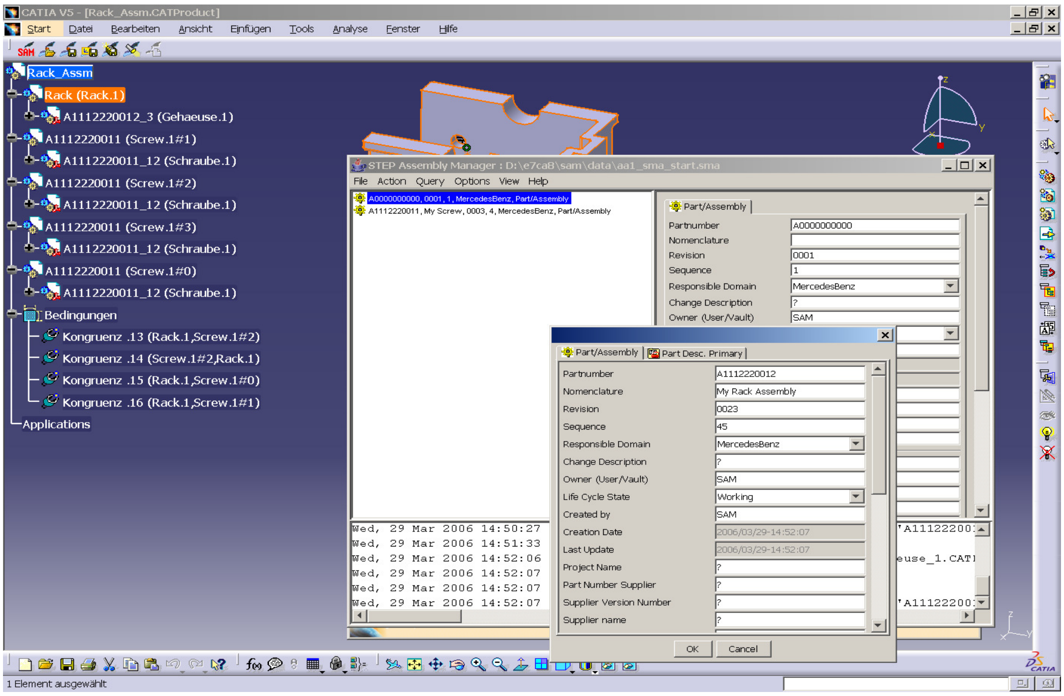


Figure 63: SAM Synchronize step 6: edit part attributes

Step 7: At the end of the synchronize dialog the top-level part is displayed in the SAM PDM GUI. You can expand it to display the complete structure. Before you go on working with the structure, you should first save it (save / save as) and load it again (open) in the SAM PDM GUI. By this, the CATIA files that were saved in the "new file location" are renamed once more so that they get version dependent file names according to the SAM Smaragd mode. Then you can go on working with the created structure in SAM as normal.

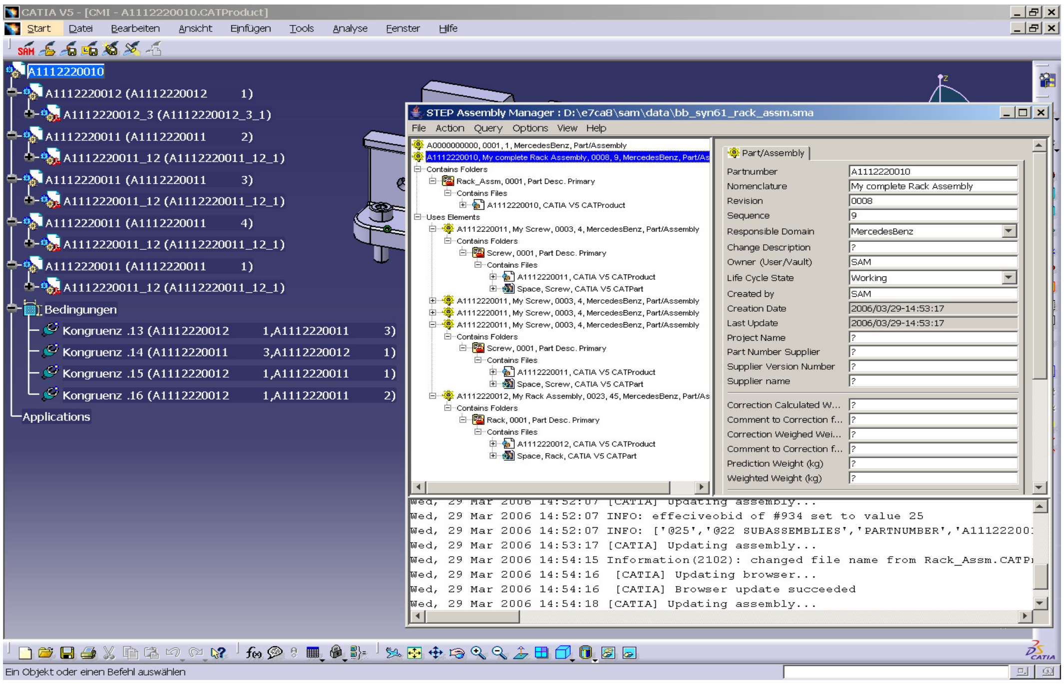


Figure 64: SAM Synchronize step 7: created structure loaded again from SAM

Step 8: You can load the created structure from SAM into CATIA as usual. You can change the structure in CATIA with normal CATIA means, e. g. add, move or delete an instance or modify a geometrical position. Then you can update the changed structure into SAM-PDM by repeating the “Synchronize/Update” dialog from step 3.

You can repeat the sequence from step 3 to step 8 several times. Please note that, after having changed the structure in CATIA, you have to do “Synchronize/Update” before you change the structure in SAM-PDM. On the other hand, after having changed the structure in SAM-PDM, you first have to load from SAM to CATIA before you change the structure in CATIA. If you exit the SAM PDM GUI during a running CATIA session, you first have to start it again, open the saved structure and load it to CATIA before you continue your work in CATIA.

Special option for CATIA catalogues

If the environment variable "SAM_SYNC_CATALOG_DIR" points to a CATIA catalogue directory, the catalogue parts will be handled in a special way at “Synchronize/Update”. Then each part in the CATIA structure will be checked whether it exists under the same name in the catalogue directory. If yes, it will be created in SAM-PDM as read-only (lock symbol) and deactivated (pale).

How to define a default part class

You can define a default part class for the parts to be created in the Smaragd view by “Synchronize/Update” so that you don’t have to select the type (“category”) of the part to be created for each CATProduct in the CATIA window “SAM Synchronize Structure” in the “Synchronize/Update” dialog (see step 3 above).

In the configuration file defined by the environment variable “CMI_CONFIGURATION_FILE” (see installation manual), edit the line that contains "`PartCategories Default`". Replace “NOTSET” by the short name of the respective part class, and replace “ON” by “OFF.

Example: Original line:

```
<PartCategories Default="NOTSET" UserMustSelectNonDefault="ON">
```

Edited line:

```
<PartCategories Default="QRY_J0PRTVER_MB" UserMustSelectNonDefault="OFF">
```

This defines “Query Part Version Mercedes-Benz ” as default part class.

SAM toolbar: Creates/Attaches an Archive

In case of the environment variable setting "SAM_ENABLE_ATTARCHIVECMD=ON" the “Creates/Attaches an Archive“ functionality is active. The SAM toolbar displays this icon in addition.

Figure 65: SAM Toolbar with “Creates/Attaches an Archive”

The “Creates/Attaches an Archive“ functionality is available in the GDG mode only.

This operation enables to create in SAM a “CMI archive”. This is a data object in the SAM PDM structure that contains a CATIA product structure.

Precondition: First you load from SAM PDM in the GDG mode a structure which contains an empty composition. The empty composition is represented as a CATProduct in the CATIA product structure. Using CATIA standard functionality, you can add in the CATIA product structure below this “composition CATProduct” a component (CATProduct, CATPart etc.) which may contain further components.

Now you can pack this structure below the “composition CATProduct” into a CMI archive, using the “Creates/Attaches an Archive“ functionality. To do this, you select in the CATIA product structure the “composition CATProduct” as the current structure level (blue background, see following figure) and the component immediately below the “composition CATProduct” as the active element (orange background, see following figure). Then you click on the “Creates/Attaches an Archive“ icon in the SAM toolbar.

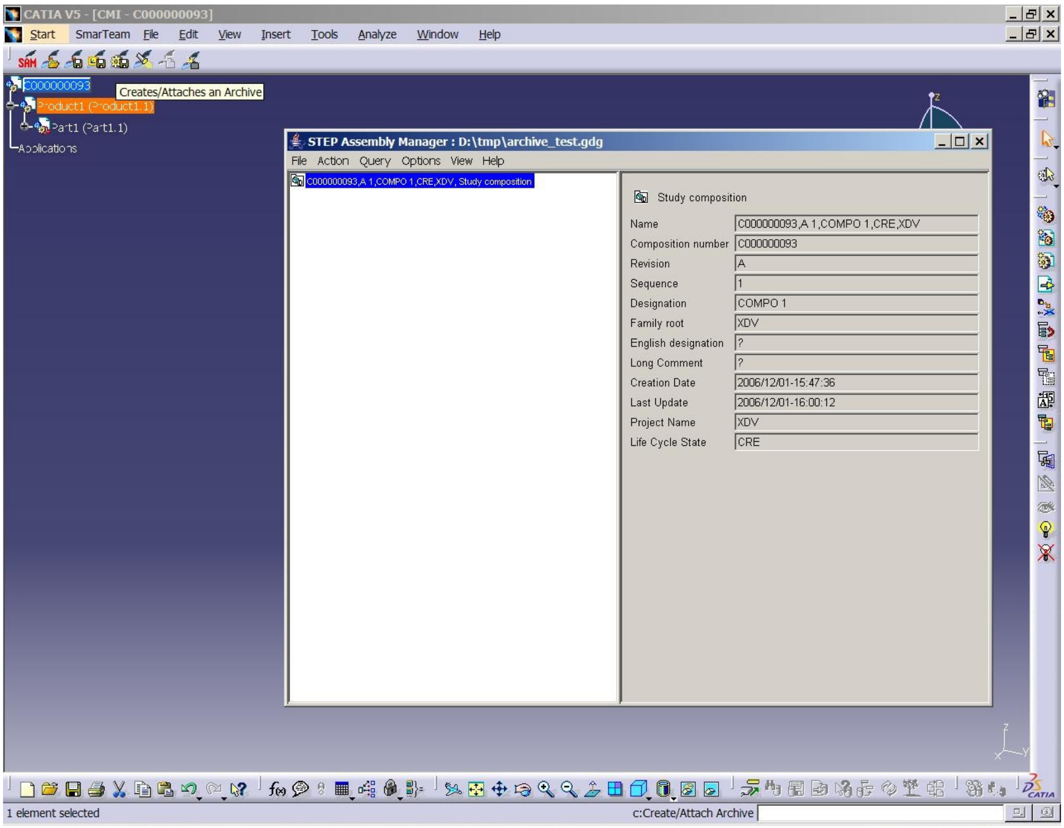


Figure 66: Create CMI archive

Then you select in the SAM PDM GUI the directory where to save the CMI archive file. Then, as the result, the CMI archive appears in the SAM PDM structure.

When you load the SAM PDM structure with the CMI archive back to CATIA, the CMI archive will automatically be unpacked.

SAM toolbar: Update Part in SAM PDM

In case of the environment variable setting "SAM_ENABLE_UPDATEGEO=ON" the “Update Part in SAM PDM“ functionality is active. The SAM toolbar displays this icon in addition.

Figure 67: SAM Toolbar with “Update Part in SAM PDM”

This operation enables to save modified CATPart files.

The difference to the "Update SAM PDM" operation described above is that no modifications in the CATIA product structure (e. g. position information – transformation matrices) will be saved.

SAM toolbar: Delete PDM Context

In case of the environment variable setting "SAM_ENABLE_DELETEPDMCONTEXT=ON" the “Delete PDM Context“ functionality is active. The SAM toolbar displays this icon in addition.

Figure 68: SAM Toolbar with “Delete PDM Context”

This function allows the user to handle data loaded by SAM as new data or as data loaded from a local store. This function causes the loss of any PDM meta information of any loaded file.

This function is useful in combination with “SAM Synchronize” if you want to make CATIA “forget” that a “SAM Synchronize” operation has already been done, e. g. in the following scenario:
First use “SAM Synchronize” to save a CATIA structure in SAM PDM. Then use “Delete PDM Context“ to make CATIA “forget” that the CATIA structure already has been synchronized. Then modify the CATIA structure. Then use “SAM Synchronize” again to save the modified CATIA structure in another SAM PDM structure.

CATIA model directories

The following figure shows the CATIA model directories in SAM for CATIA V5.

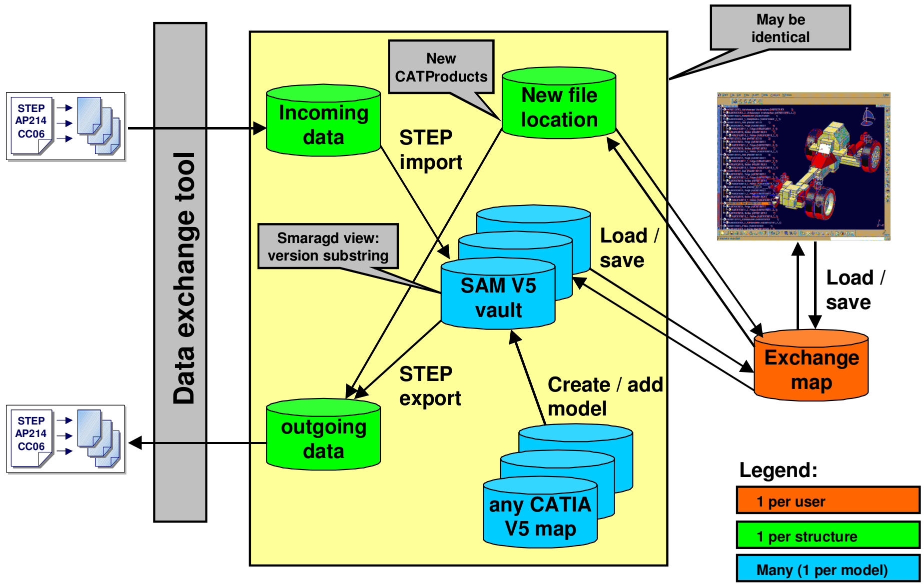


Figure 69: CATIA model directories in SAM for CATIA V5

Glossary

Administrator

Person who configures the system, inserts users, manages users permissions and maintains the database.

AP214

Application Protocol 214 "Core Data for Automotive Mechanical Design Processes", part of *STEP*. SAM uses STEP AP214 as the data exchange format for assembly structures.

Browser

Window that displays *icons* representing *objects*.

Dialog Window

Window in which the user enters information.

GII

Graphics Interactive Interface. The GII is a powerful programming tool, which completes the Open System Access to the CATIA environment.

GUI

Graphical User Interface. User interface of a software application that operates with a graphics screen and a mouse.

Icon

Graphical representation of an *object*.

List View

Style of *browser* window in which *objects* are viewed as small *icons* in a list with attribute values in columns.

Object

An item or relationship.

PDM

Product data management. PDM systems typically handle product data like assembly structures, organizational data etc.

Pop-Up Menu

The menu that appears when the user points an *icon* and holds the right mouse button pressed.

STEP

Standard for the exchange of product model data. International Standard ISO 10303.

Tree View

Style of *browser* window in which *objects* are displayed as relationships in horizontal branches.