Standard Parts V19

User Guide

Contents

Document Scope

This document provides a detailed description of Solid Edge Standard Parts extensions (add-on) for the Solid Edge application. This document should be used by Solid Edge Standard Parts users and administrators.

Solid Edge Standard Parts

Solid Edge Standard Parts is an add-on application for the UGS 3D CAD application, Solid Edge. Solid Edge Standard Parts is a powerful parts management system that allows designers to define, store, select and position commonly used parts – fasteners, bearings, structural members, pipes and fittings – quickly and efficiently, enabling rapid and precise completion of 3D assemblies. With Solid Edge Standard Parts, companies can establish and share their own standards, so designers can concentrate on creative design and not on redundant modeling tasks.

Basic Definitions

For more details about Solid Edge/Insight installation and workflows, refer to the Solid Edge/Insight documentation. Some Standard Parts themes are described in the Solid Edge help file.
- **Standard Parts** – main application. Allows a user to find and place desirable parts from the Standard Parts database. For more details, see its description below.
- **Database Administrator** – application for database administration. For more details see its description below.
- **Database** – all Standard Parts documents information storage. For more details, see the Database and Database Administrator description below.
- **Solid Edge Standard Parts Configuration Wizard** – application for managing Solid Edge Standard Parts global configuration. For more details, see its description below
- **BulkPartAdder** – application for creating Customer Standard Parts from existing Solid Edge documents. BulkPartAdder is a part of Database Administrator.
- **SmartInstaller** – application for installing and updating Standard Parts within working database. SmartInstaller is a part of Database Administrator.
- **Part Editor** – application for changing configuration and adding the Standard and Customer Standard Parts
- **Delivery Database** - *.ddb file containing information on Standard Parts in delivered Standard Parts Library. This information serves mainly to keep the parts registered in the database up-to-date. That means, that if a new version of Standard Parts Libraries will be available, only parts which have changes (compared to older versions) may be updated in working database.
- **Solid Edge Machinery Library** – Library of international norms of standard parts prepared by UGS; contains ANSI, DIN, UNI, GB, JIS, GOST norms in Solid Edge V19.
- **Standard Parts installation folder** – main (usually root) folder of the Standard Parts system. This folder contains subfolders with Library parts, configuration files and others. For more details, see a description of Solid Edge Standard Parts Configuration Wizard
- **Standard Parts folder** – folder with your standard parts. For more details, see the description of Solid Edge Standard Parts Configuration Wizard and Standard Parts.
- **Standard Part** – UGS Library standard part. It can be a Parent part or Single part
- **Customer Standard Part** – Customer specific Standard Part. It can be prepared using BulkPartAdder or Part Editor. For more details see its specification below.

- **Parent part** – UGS Library Standard global file. Contains definitions of all relevant Single parts. It is delivered with Standard Parts Library to a version specific folder.
- **Single part** – generated final part. Derived from Parent part.
- **Part Name** – global part identifier. It should be a full, detailed and unique descriptor for every file in the Standard Parts database. It serves for fast and correct identification of parts in Standard Parts applications.
- **SAC file** (SMAPINIFile.sac) – global definition and settings file.

Solid Edge Standard Parts Scheme

Solid Edge Standard Parts is a simple database oriented document management system for working with standardized parts. The most important user application is Standard Parts. This application allows users to find, select and place (eventually replace or modify) centralized and standardized parts into assemblies. The most common use is to select standard parts from the Solid Edge Standard Parts Library or Customer Standard Parts and place into an assembled project.

Administrator applications are called: Solid Edge Standard Parts Configuration Wizard, Database Administrator and Part Editor. The Solid Edge Standard Parts Configuration Wizard allows system administrators to define a document management structure, paths and databases for all users. The Database Administrator (including integrated SmartInstaller and so called BulkPartAdder) is an application that manages database and database content. The administrator can delete or add desirable standardized parts into a database. The last administrative application is Part Editor; this application allows managing and changing the content (characteristics, properties etc.) of every standard part.
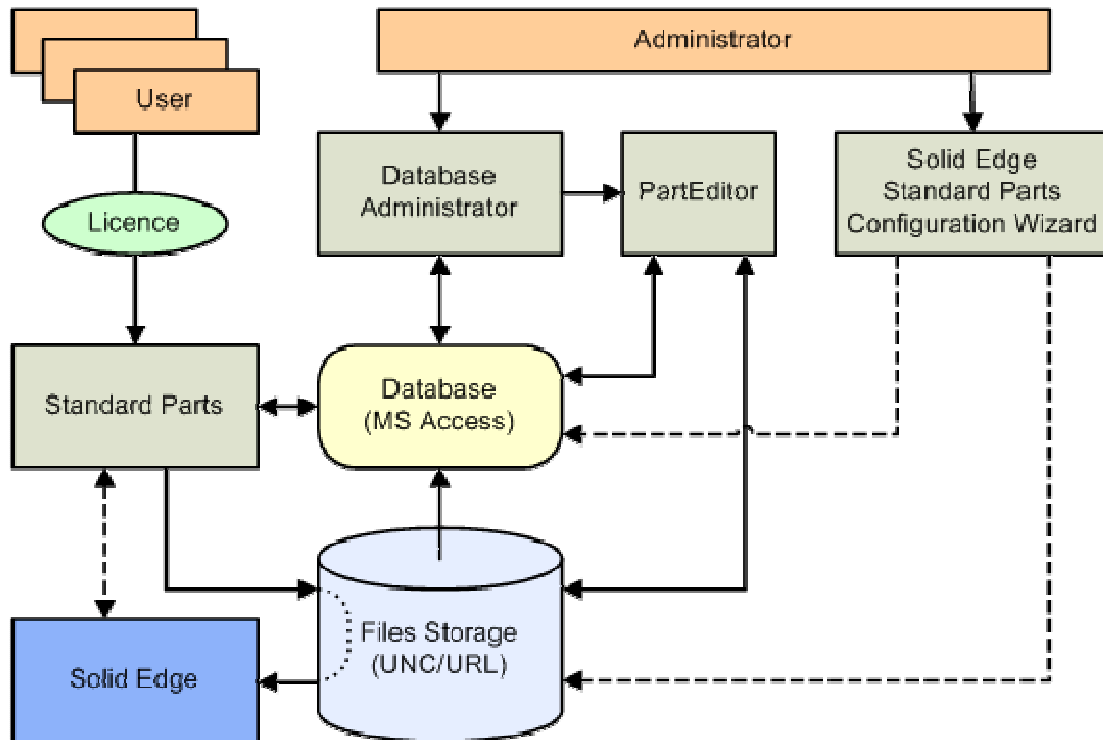
Solid Edge Standard Parts Installation (Local or Network)

Standard Parts Administration Installation

1. Install Solid Edge on all workstations.

2. Install Standard Parts on the workstation used for Standard Parts administration. Change the destination folder to a local/network location accessible from other workstations[1] (e.g. "\\Server\Public\SEStdParts" as shown in figure bellow). Full access permissions are required for the target location.
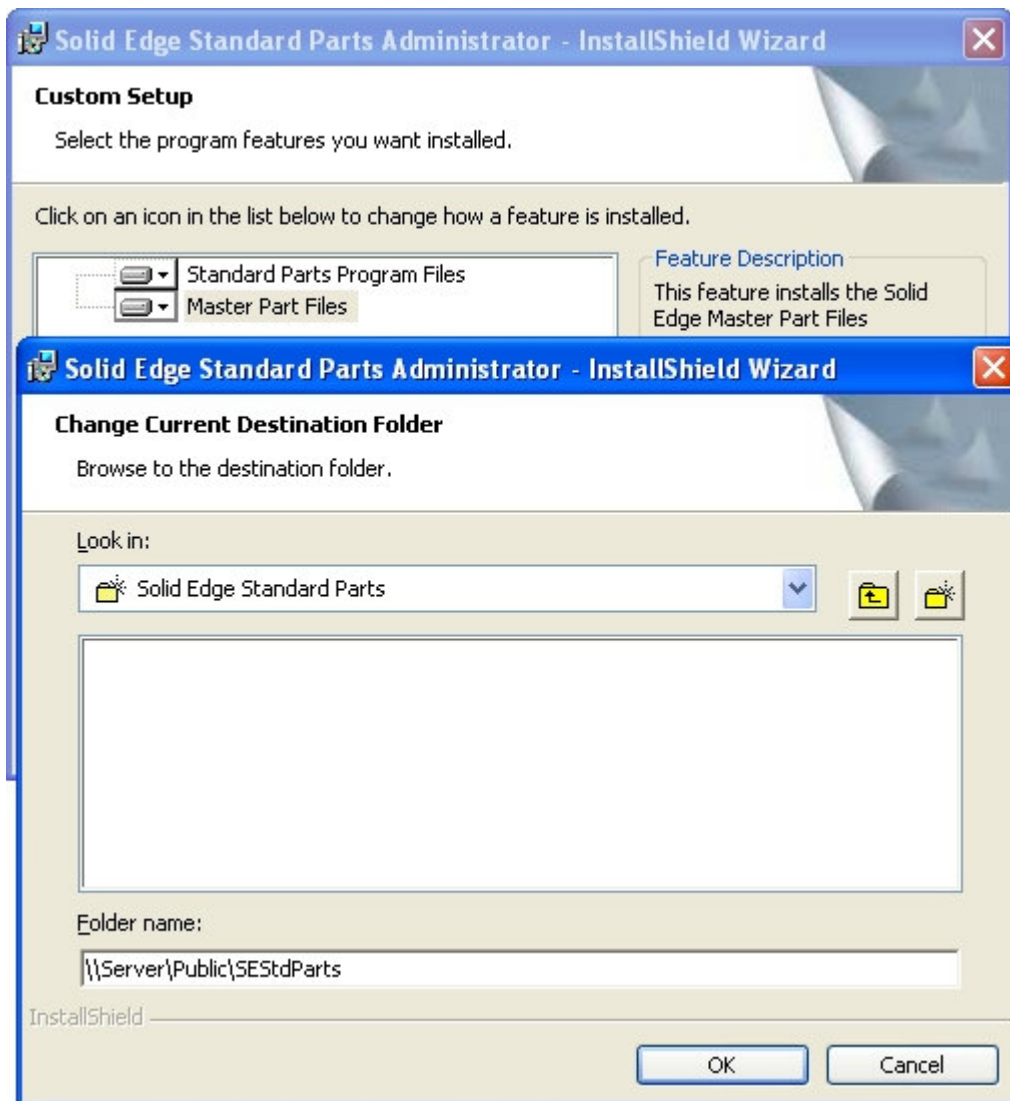
3. Complete the installation.

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[1] It is currently possible and preferred to identify the destination folder by typing the UNC address, rather than by browsing in "Look in"! Nevertheless, local paths or mapped disks are supported.

After completing these steps, the Solid Edge Standard Parts installation folder structure is created (in this case at the location "\\Server\Public\SEStdParts"). For more details about folder structure, see the section "Solid Edge Standard Parts Configuration Wizard". It contains:

- "Database", a folder containing Standard Parts database "StandardParts19" (as default) with Free Content Standard Parts version 7.45 registered.
- "Html" folder
- "Standard Parts" folder containing Free Content parent parts (Standard Parts\3D-Standard_Parts_V7.45\Parent_Parts)
- "Working folder"
- "SMAPINIFile.sac"

Standard Parts Workstation Installation

If the administration workstation is set up; on other workstations just set the Standard Parts SAC file location (Tools – Options – File Locations). The SAC file must be located in the UNC path which has been set during the Standard Parts administration installation (step 2). For the URL location, see the "Insight Installation" section.
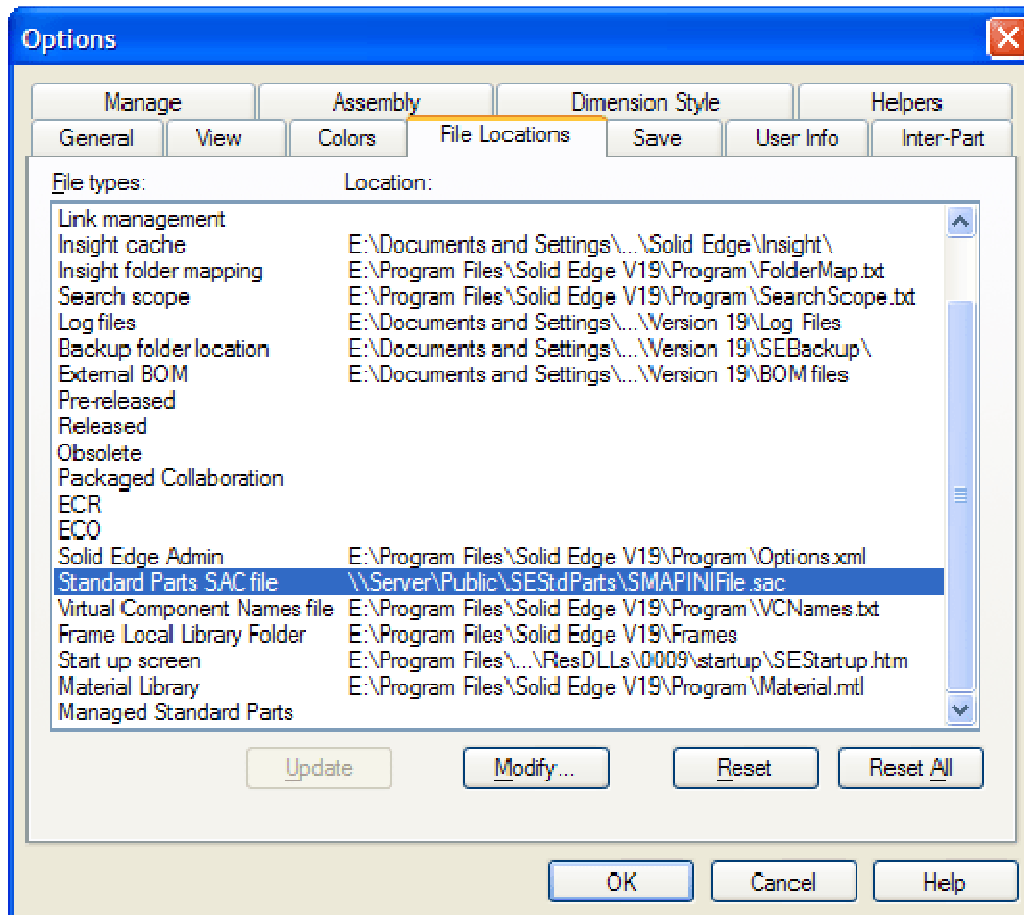
Installing Libraries and Registering Standard Parts

1. If you have purchased Standard Parts libraries, install them directly at the top level of the Solid Edge Standard Parts installation directory. The installation process will automatically copy all standard parts from the CD/DVD to a specific location of the Standard Parts folder. When you are installing the library, it is important to select the top level folder created in step 2 of "Standard Parts administration"!!!
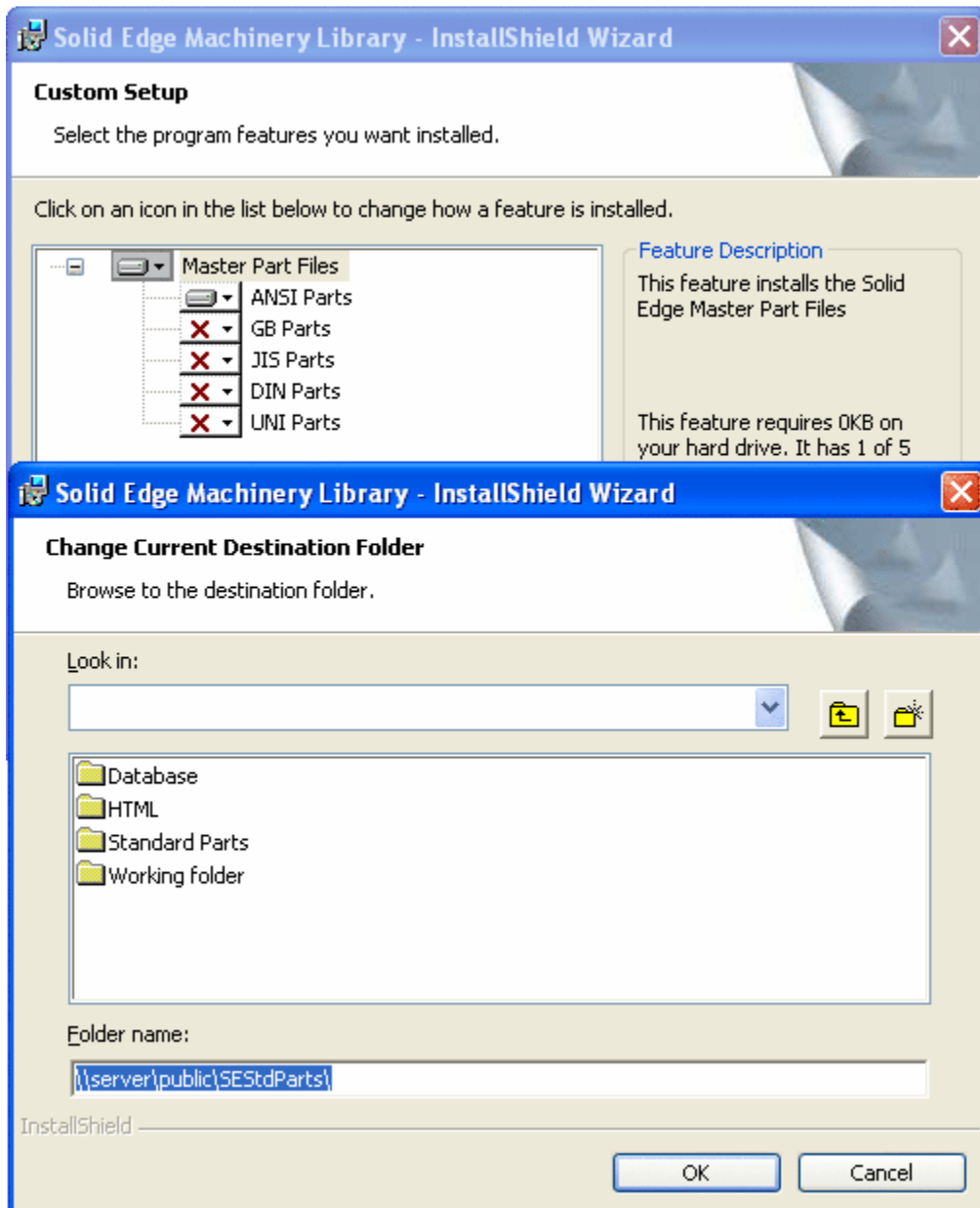
2. When the Library installation is complete, launch Database Administrator and click on "**Add parts**".

Registering Standard Parts in Database via Add Parts Button in Database Administrator

Add Parts Options

1. **Add new part file(s)** – launches BulkPartAdder application for registering user documents as Standard Parts (see Creating Customer Standard Parts chapter for details)
2. **Add from existing single part file** – the same as "Add part" command in version 18 (see Adding Existing Single Part File chapter for details)
3. **Add from existing multiple part files** – the same as "Add parts" command in version 18 (see Adding Existing Part Files chapter for details)
4. **Add from a delivery database** – launches SmartInstaller application. Default option (see Registering Standard Parts Using Delivery Database chapter for details).

Insight Installation

This section describes how to migrate (fully or partially) your local or network system (located at UNC path) to Share Point (URL location). Direct installation/configuration with SharePoint from Standard Parts setup is not possible with this version.

All Insight (server/client) installation requirements must be met prior to performing these migration steps. For more details about installation and working with Insight, see Microsoft and UGS documentation for SharePoint and Insight services (Insight Implementation Guide). The minimum supported role at SharePoint is "Contributor" for all users.

The Solid Edge Standard Parts system has many limitations for Insight localization that are derived from SharePoint and Insight limitations and rules; most important are:

1. Global limitations:
   a. there is a limitation for the filename/path length. Also, some special characters are not supported.
   b. mandatory Insight properties must be added to all Document Libraries using the tool "Workspace Editor" at the server (pass full URLs, i.e. http://server/doclib) prior to importing files to the library.

2. Workstation limitations:
   a. Windows XP only: Web Client service must be disabled.
   b. Proxy must be disabled or bypassed for LAN!

Insight modes

Solid Edge V19 allows use of Insight in two different modes.

The first one – **"All on Insight" mode -** is the same as in Solid Edge V18. In this case the whole Standard Parts folder structure (including the SAC file and excluding the database) must reside on Insight.

The second one – **"Generated parts and working folder on Insight" mode** – uses Insight only as a location for generated parts and working folder.

Standard Parts Administration Installation ("All on Insight" mode)

1. Install Solid Edge on all workstations.

2. Install Standard Parts on the workstation you want to use for remote Standard Parts administration[1]. When asked for the location, change the destination folder to some UNC location that is accessible from this workstation (e.g. "\\Server\Public\SEStdParts"; see the picture bellow).

3. Complete the installation.

4. If you need to install the Standard Parts full version of Libraries, you can do it in the same method described in the section above: "Installing Libraries" (but you don't need to register your parts with the database).

5. Create a target SharePoint document library (http://is19/sites/example/StdPartsV19 in our case). Do not forget to add Insight mandatory properties from Insight Server at this document library.

Standard Parts administration installation (Step 2).

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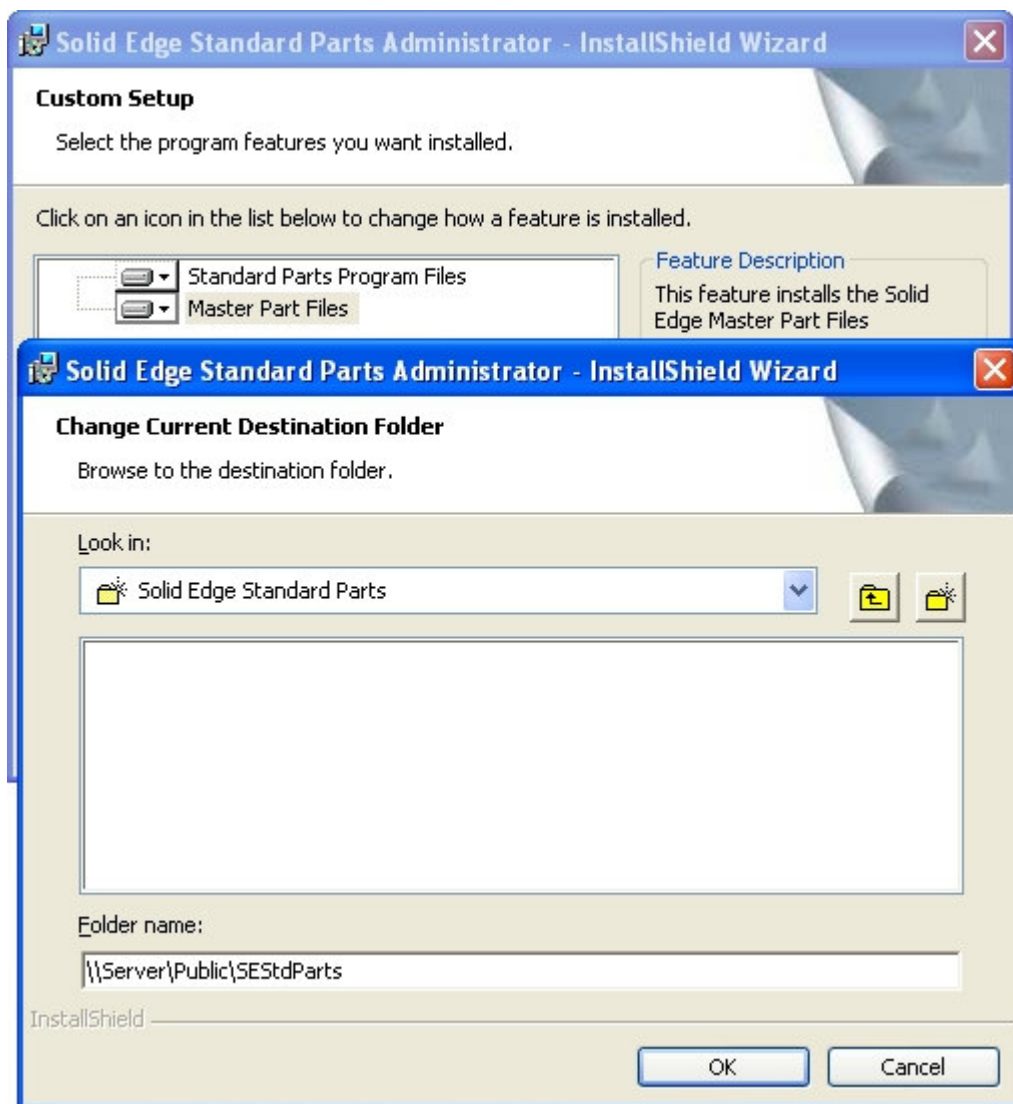
[1] Full Solid Edge software must be installed. Solid Edge Insight Server or Insight Connect installation is not supported!!!
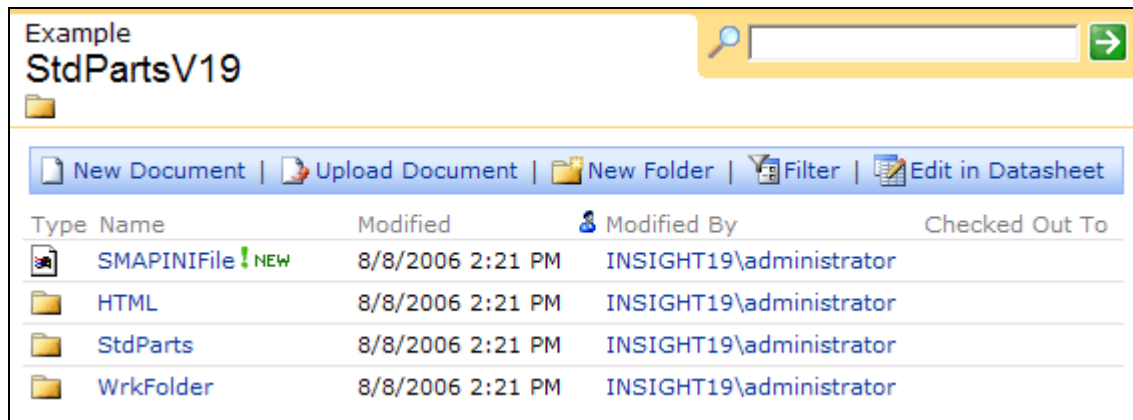
The SAC file and folder structure in Share Point created in Step 5.

6. Add a Network Place on your workstation pointing to the SharePoint URL. This step is necessary for accessibility from some standard dialogs not customized for working with URLs.

7. Launch the Configuration Wizard

   a. Browse to the Network Place in "Step 1" in the configuration wizard (application browse dialog).

   b. Click on "Set Path" (default folder structure will be displayed; see the picture below).

   c. Browse to the database file created during installation in the browse dialog in Step 2 ("\\Server\Public\SEStdParts\Database\StandardParts19Insight.mdb" in this case) or create a new database at some desirable network location (accessible to all users).

   d. Click on "Save". The Solid Edge Standard Parts directory structure will be created at the SharePoint location.
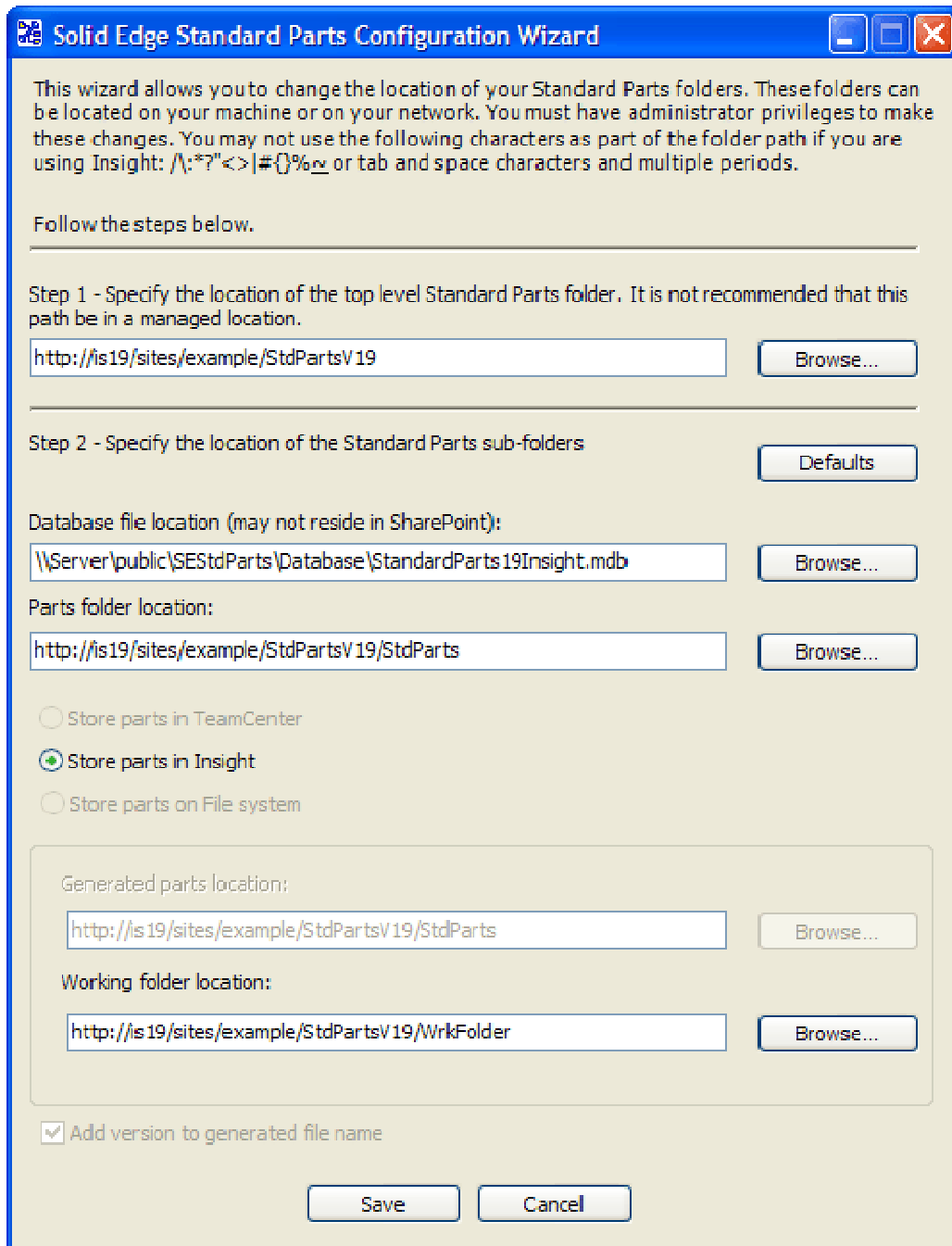
Standard Parts settings in Configuration Wizard (Step 7).

8. Add the Document Library location ("http://is19/sites/example/StdPartsV19" in our case) to the "searchscope.txt" file (typically located at "C:\Program Files\Solid Edge V19\Program").

9. Launch Insight Connect (View and Markup) and add all subfolders of the Standard Parts folder (3D-Standard_Parts_Vx.xx, Previews and Navigation) from a pre-prepared UNC installation ("\\Server\Public\SEStdParts\Standard Parts") to the location of the Standard Parts library ("http://is19/sites/example/StdPartsV19"). This action can take several minutes (depending on the file size and number of files that you are attempting to upload).

Add to Library

Look in:

Standard Parts

3D-Standard_Parts_V6.45
3D-Standard_Parts_V7.45
3D-Standard_Parts_V7.50
Custom_parts
Navigation
Previews

Add   Remove   Remove All   ☑ Include subfolders

Types...

Folders and documents to be added:

\\Server\public\SEStdParts\Standard Parts\3D-Standard_Parts_V7.45
\\Server\public\SEStdParts\Standard Parts\3D-Standard_Parts_V7.50
\\Server\public\SEStdParts\Standard Parts\3D-Standard_Parts_V6.45
\\Server\public\SEStdParts\Standard Parts\Custom_parts
\\Server\public\SEStdParts\Standard Parts\Navigation
\\Server\public\SEStdParts\Standard Parts\Previews

Destination

Library folder:

http://is19/sites/example/StdPartsV19/StdParts

○ Dry run (documents will not be added to library)   ● Add documents to the library

☑ Update status on all documents to "checked-in"

Use default for missing foldermap.txt entries:

OK   Cancel   Help

10. Launch DB Administrator to add your parts into database.

Standard Parts Administration Installation ("Generated parts and working folder on Insight" mode)

1. Install Solid Edge on all workstations.

2. Install Standard Parts on the workstation you want to use for remote Standard Parts administration[1]. When asked for the location, change the destination folder to some UNC location that is accessible from this workstation (e.g. "\\Server\Public\SEStdParts"; see the picture bellow).

3. Complete the installation.

4. If you need to install the Standard Parts full version of Libraries, you can do it using the same method described in the section above: "Installing Libraries" (but you don't need to register your parts with the database).

5. Create a target SharePoint document library (http://is19/Example/DocumentLibrary in our case). Do not forget to add Insight mandatory properties from Insight Server at this document library.
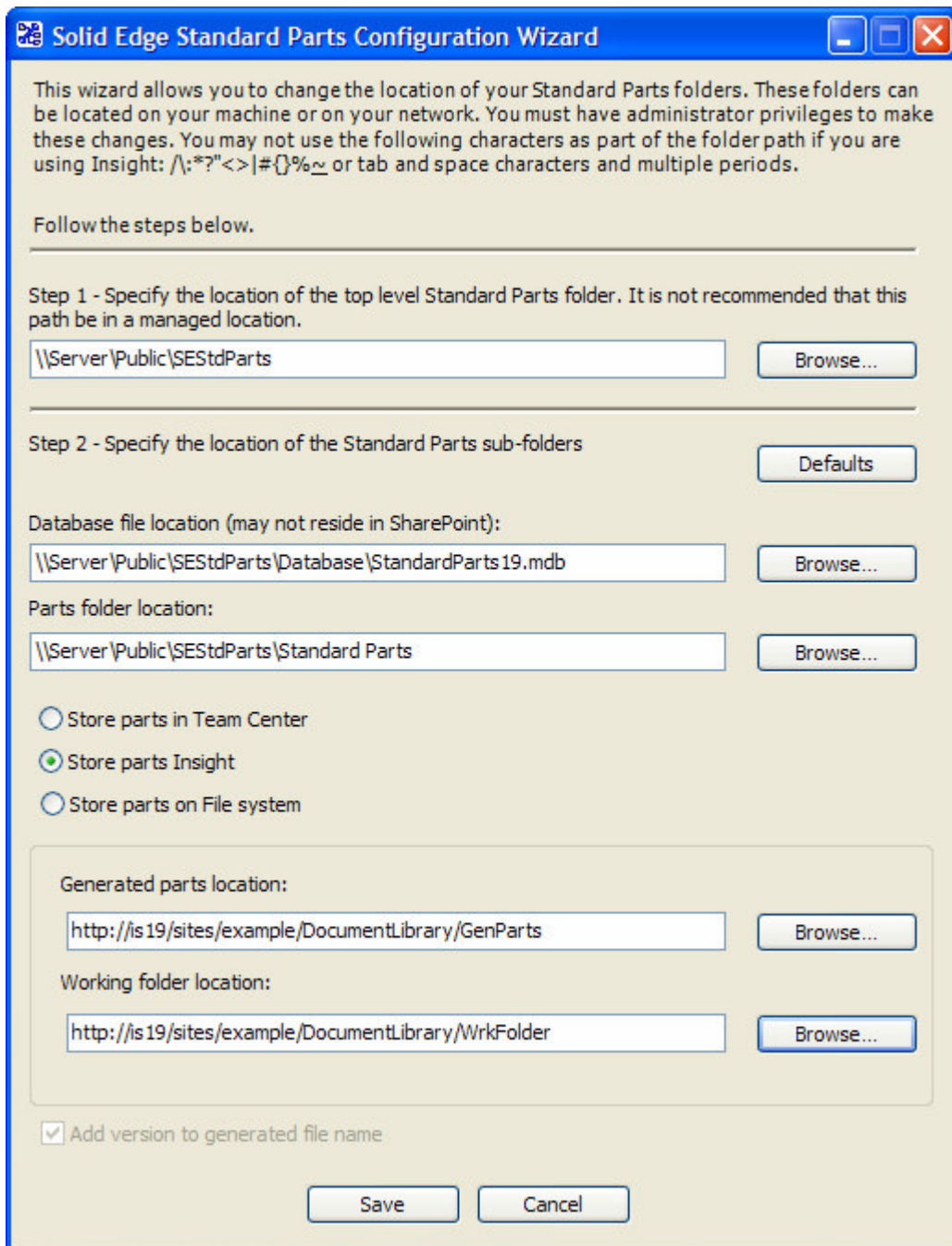
Standard Parts administration installation (Step 2).

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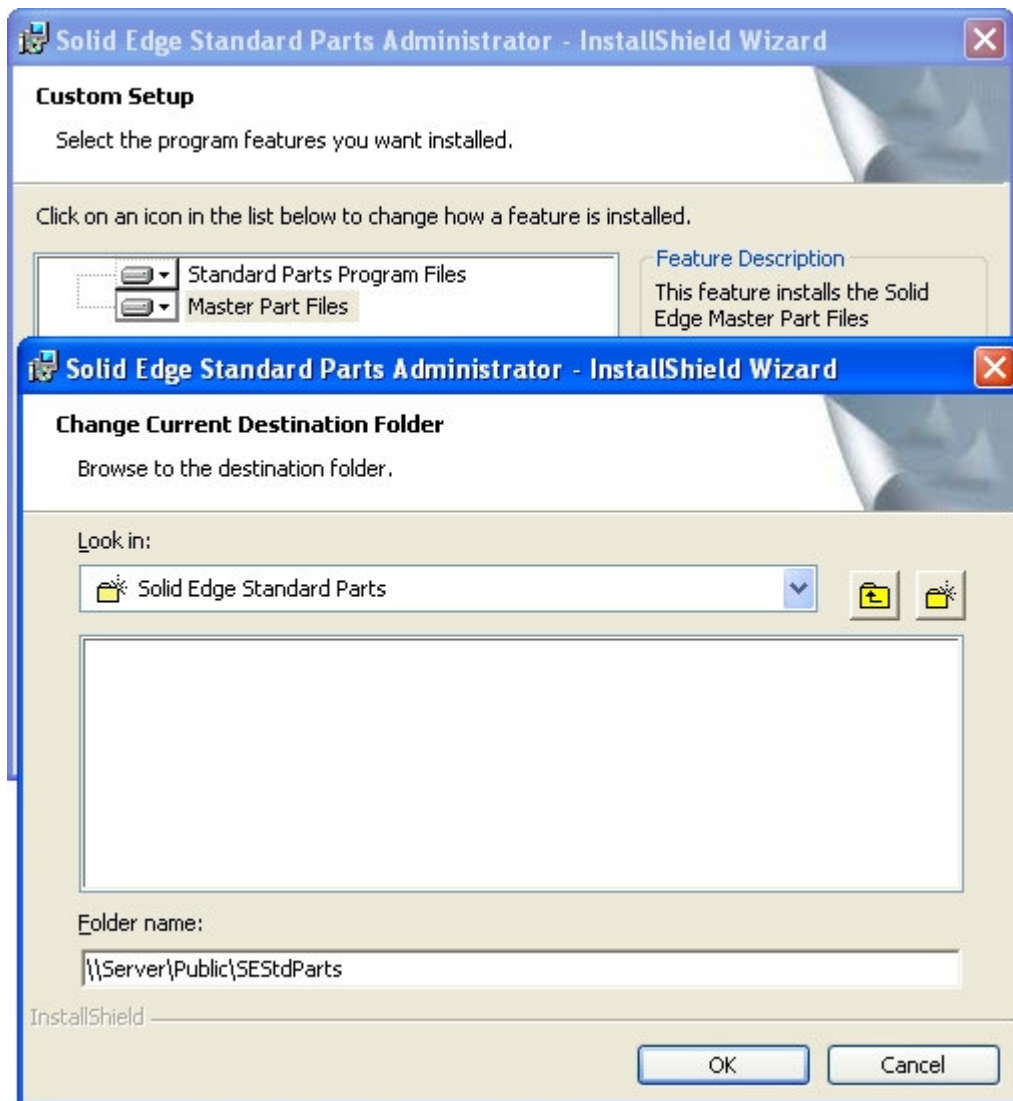
[1] Full Solid Edge software must be installed. Solid Edge Insight Server or Insight Connect installation is not supported!!!
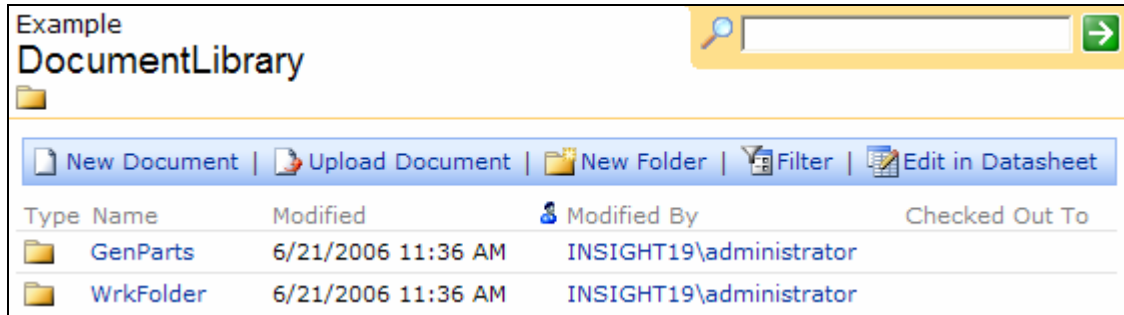
Folders in Share Point created manually in Step 5.

6. Add a Network Place on your workstation pointing to the SharePoint URL. This step is necessary for accessibility from some standard dialogs not customized for working with URLs.

7. Launch the Configuration Wizard

    a. Select **Store parts Insight**.

    b. Select a SharePoint location for **generated parts** (http://is19/sites/example/DocumentLibrary/GenParts in our case).

    c. Select a SharePoint location for the **working folder** (http://is19/sites/example/DocumentLibrary/GenParts in our case).

   d. Click on "Save" to confirm the settings.

Standard Parts settings in Configuration Wizard (Step 7).

8. Add the Document Library location ("http://is19/example/DocumentLibrary" in our case) to the "searchscope.txt" file (typically located at "C:\Program Files\Solid Edge V19\Program").

9. Launch DB Administrator to add your parts into database.

Standard Parts Workstation Installation ("All on Insight" mode)

If the administrative workstation has been set up; on other workstations, set the Standard Parts SAC file location (Tools – Options – File Locations) to the appropriate SharePoint URL (http://is19/sites/example/StdPartsV19/SMAPINIFile.sac in this case). If you are using a different search-scope.txt than the administrative workstation, do not forget to add the target document library to this file ("http://is19/sites/example/StdPartsV19"). Do not forget to configure SharePoint / Insight local settings as described for the administrative workstation (disable web client service, etc.).
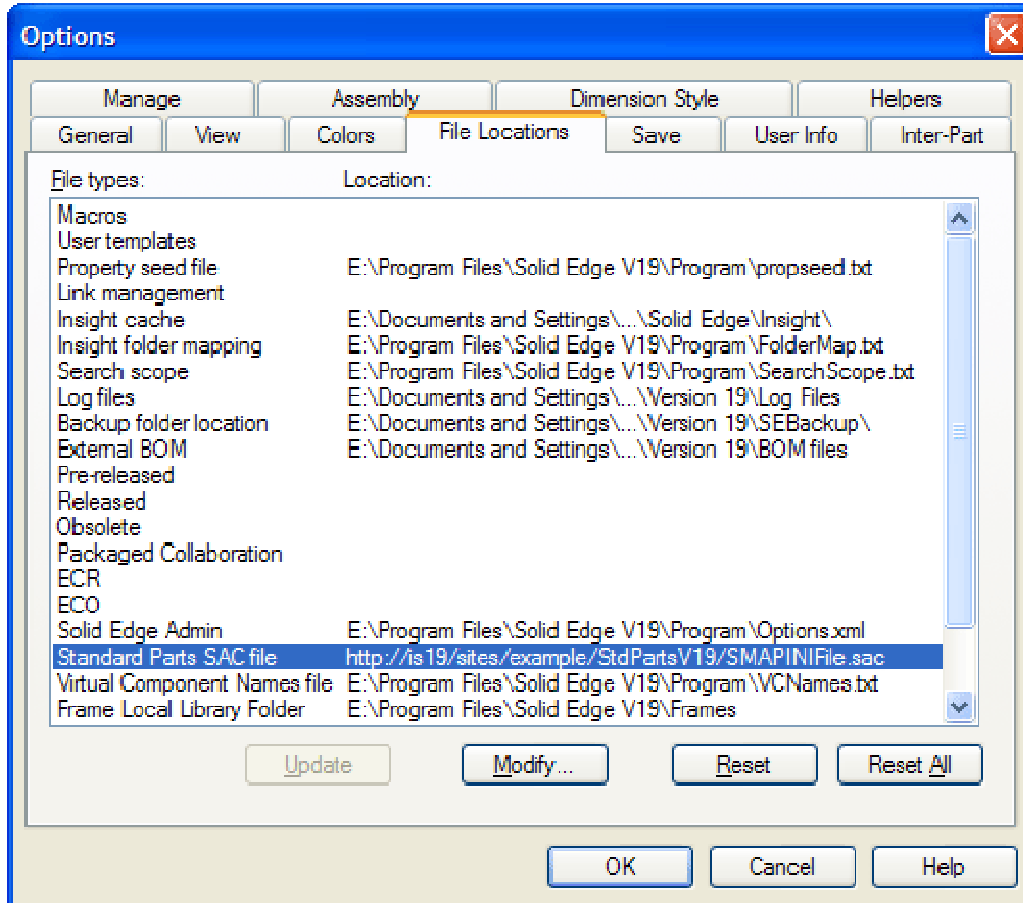
Standard Parts Workstation Installation ("Generated parts and working folder on Insight" mode)

If the administration workstation is set up; on other workstations just set the Standard Parts SAC file location (Tools – Options – File Locations), the same as for a local or network installation. The SAC file must be located in the same UNC folder that was set during the Standard Parts administration installation. The document library containing the folder for generated parts and the working folder must be set in the "searchscope.txt" file.
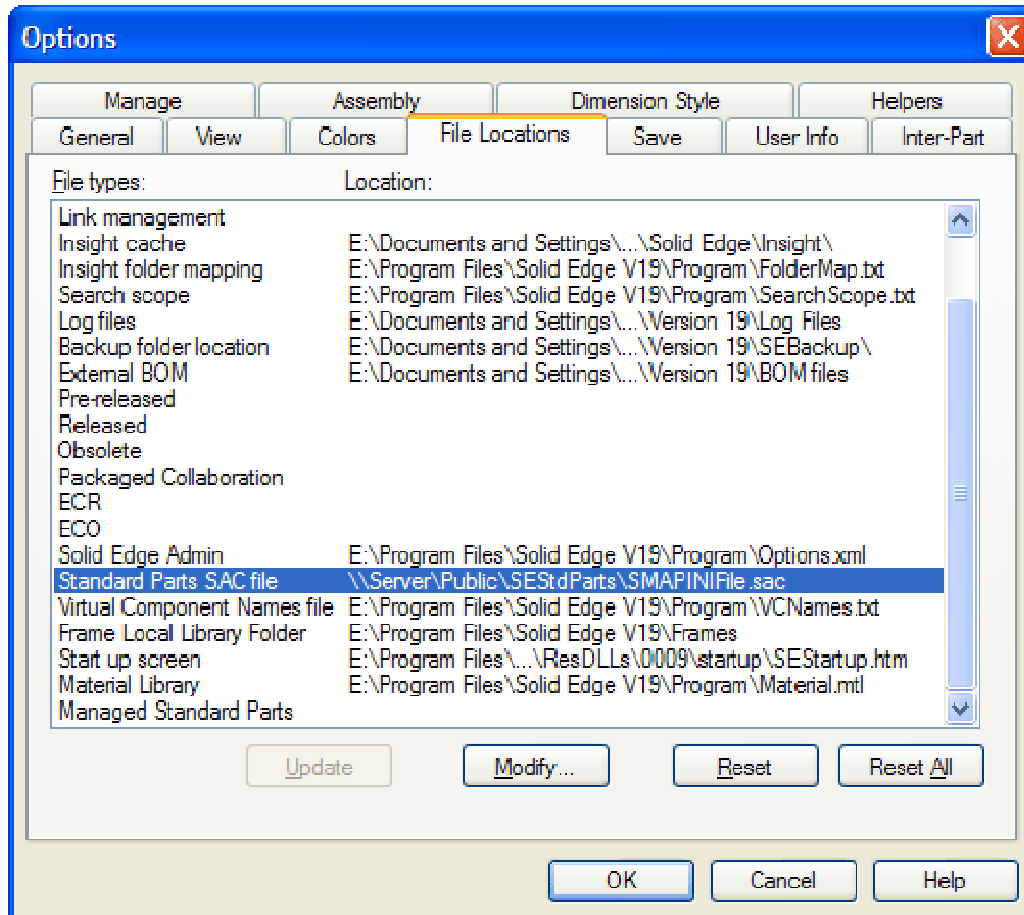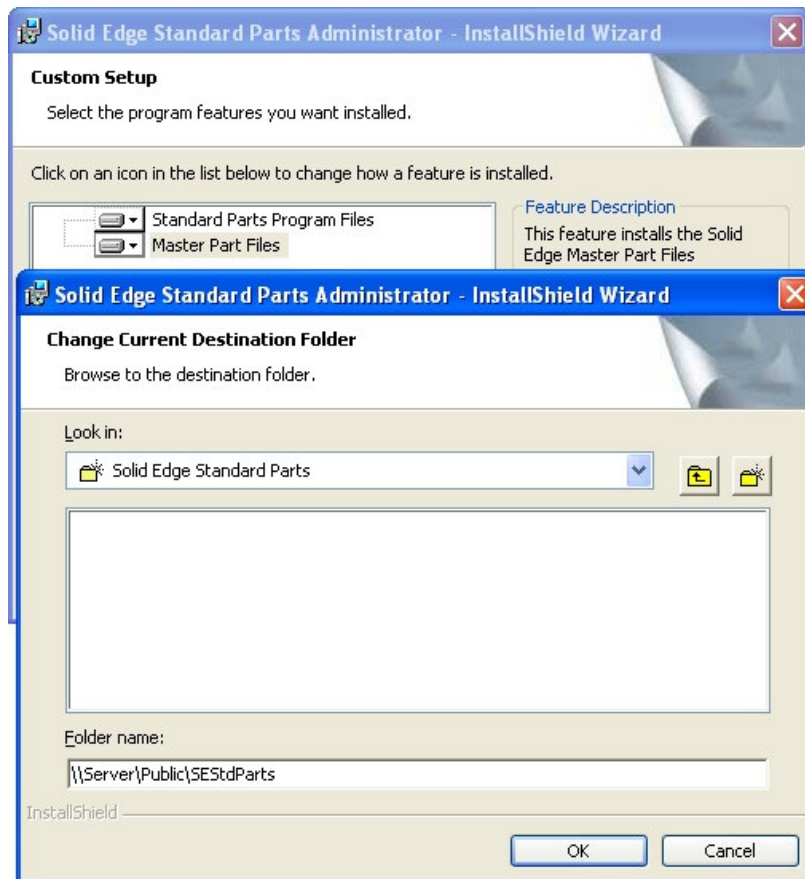
Teamcenter Installation

This section describes how to set your local or network system (located on a UNC path) to store generated parts and newly created documents in Teamcenter.
All Teamcenter (server/client) installation requirements must be met prior to performing these installation steps. For more details about installation and working with Teamcenter, see corresponding documentation.

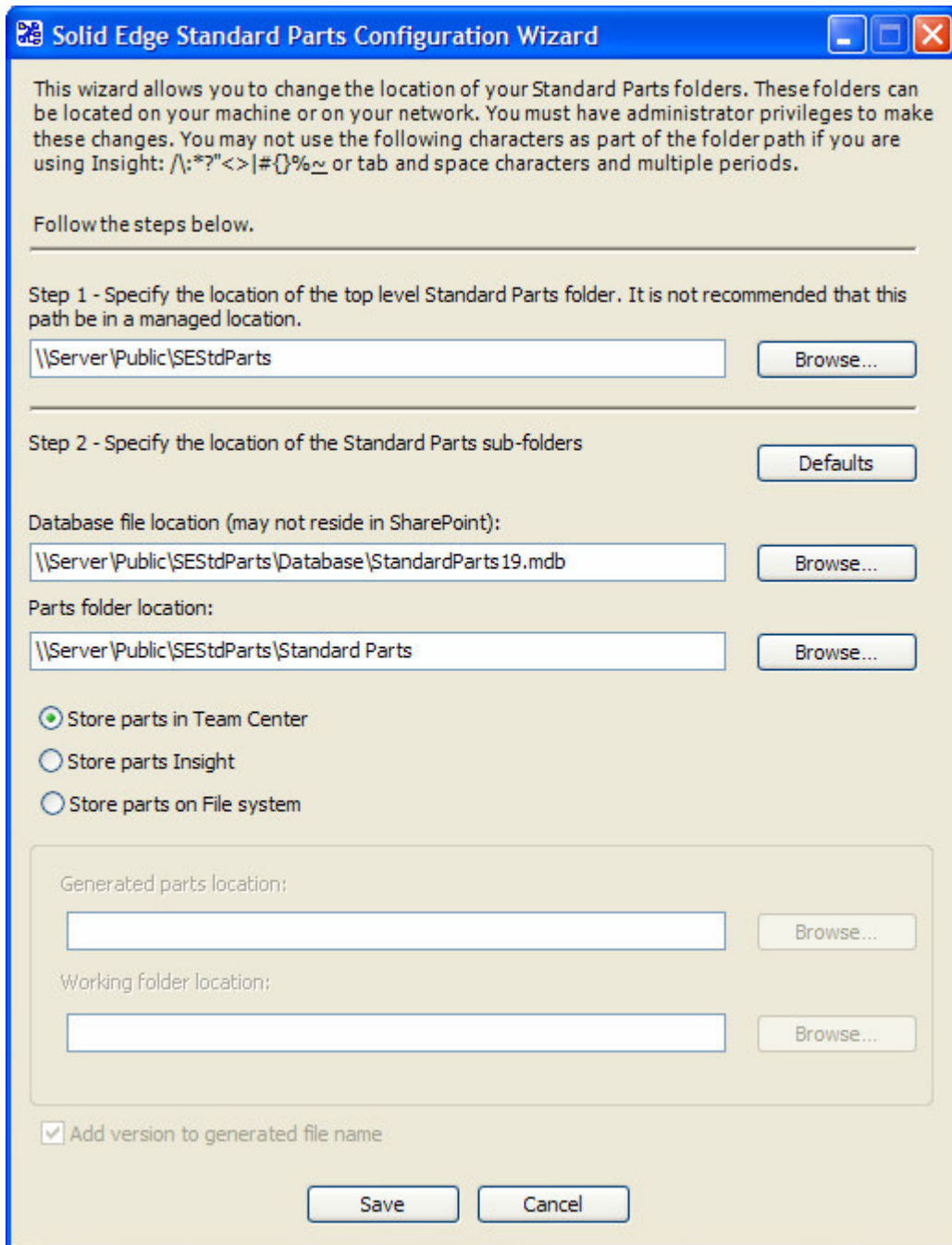## Standard Parts Administration Installation (Teamcenter)

1. Install Teamcenter Express Client, Solid Edge and Solid Edge Embedded Client on all workstations.

2. Install Standard Parts on the workstation you want to use for remote Standard Parts administration[1]. When asked for the location, change the destination folder to a UNC folder that is accessible from this workstation (e.g. "\\Server\Public\SEStdParts"; see the picture below).

3. Complete the installation.

4. If you need to install the full version of the Standard Parts libraries, you can do it using the same method described in the section "Installing Libraries" (but you do not need to register your parts into database).

Standard Parts administration installation (Step 2).

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[1] Full Solid Edge software must be installed. Solid Edge Insight Server or Insight Connect installation is not supported!!!

5. Launch the Configuration Wizard
   a. Select **Store parts in Teamcenter**.
   b. Click on "Save" to confirm the settings.

Standard Parts settings in Configuration Wizard

6. Launch DB Administrator, create the database and add your parts into database.

Standard Parts Workstation Installation (Teamcenter)

If the administration workstation is set up; on other workstations just set the Standard Parts SAC file location (Tools – Options – File Locations), the same as for a local or network installation. The SAC file must be located in the same UNC folder that was set during the Standard Parts administration installation.
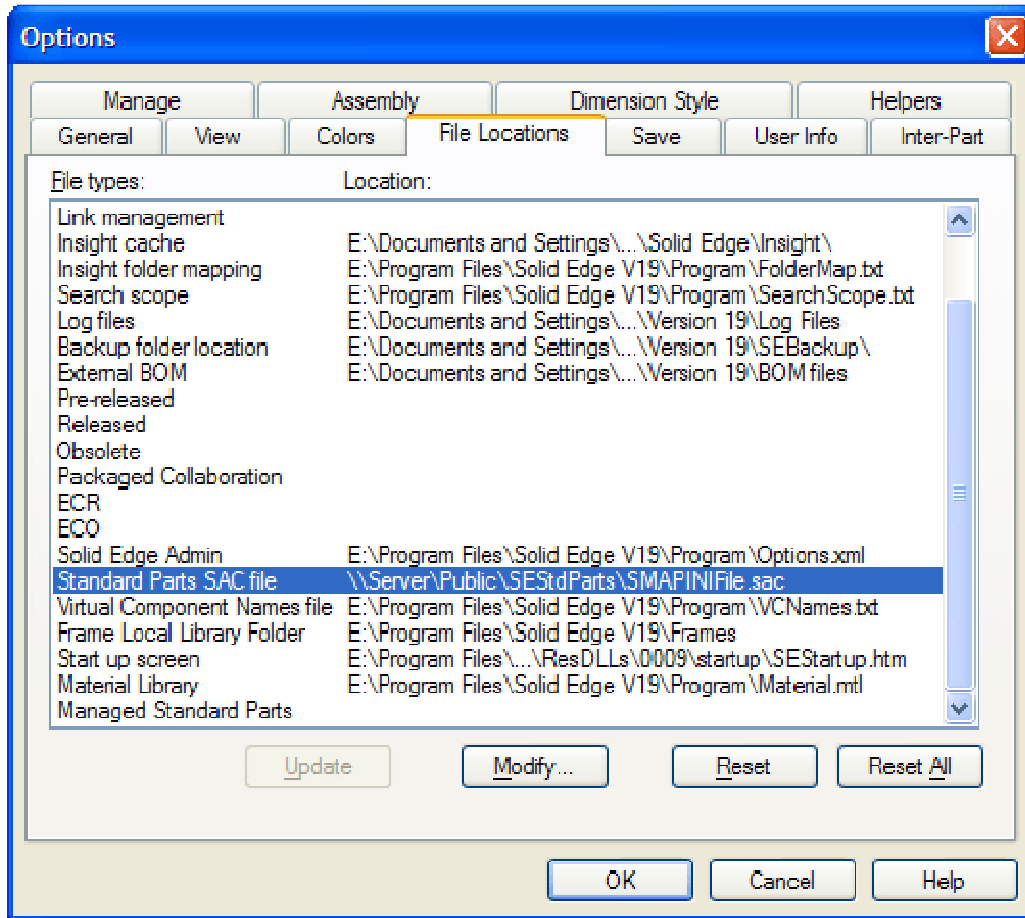
Standard Part Files Definition

A Standard part in Solid Edge Standard Parts can be any Solid Edge native file (assembly, part, sheet metal, draft and weldment). An advantage of all standard parts is an additional information stream; a special property set called "Standard Parts Stream" (SMAP Stream). This stream is always included in the file. When the file is registered in the database, then the information from stream is also contained there. All Standard Parts Stream information can be changed or managed via the Part Editor.

Parent Part

The Parent Part is a global definition file. It contains a global 3D model and definition of all part variants (parent part filename suffix is "_Var_Gen.par"). A parent part is a global definition for a kind of standard (e.g. "*Bearing_DIN_ISO_3547_Bush_A_Var_Gen.par*"). When this file is registered by Database Administrator, all included single parts are registered in the database as virtual files. *Note: Parent parts are contained in your purchased Standard Parts Library.*
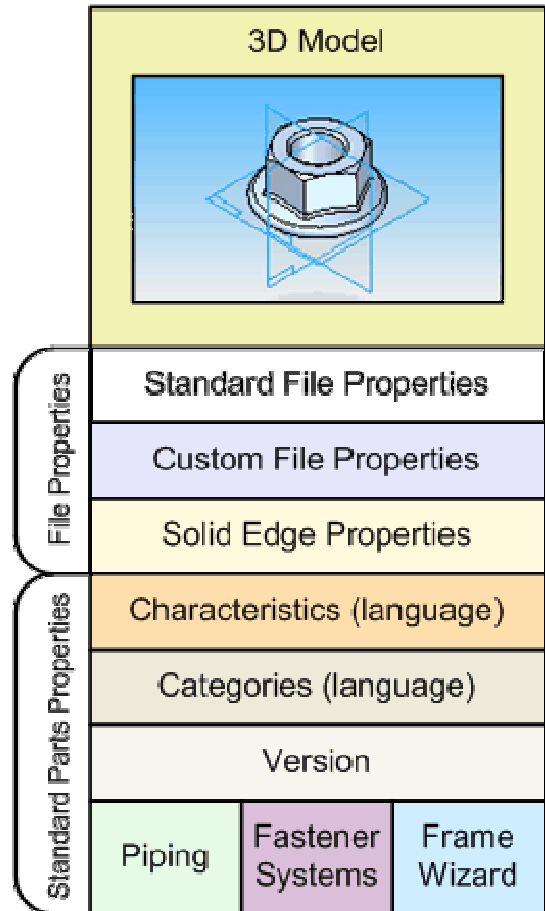
Single Part

The Single Part file represents a single variant of a component included in a Parent part. A Single part can be included in your assembly (filename suffix *_vX.XX.par" where vX.XX is the version of file; e.g. "*Bush_ISO_3547_4_A_5.5x4_v6.51.par*").
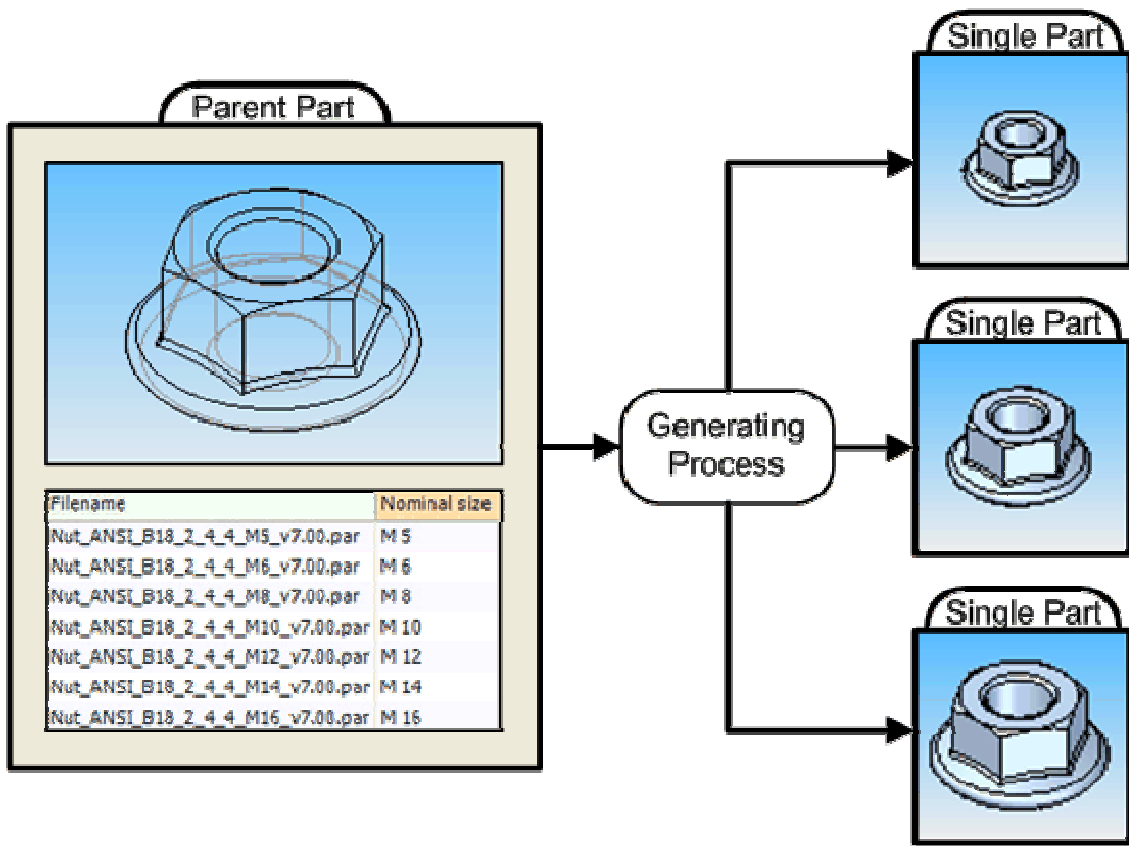
Non-generated Single Part

The Non-generated Single Part is a variant included in a Parent part which does not exist as an individual file. It is registered in the database (with 'Not generated' status). If a non-generated single part is needed in Solid Edge (e.g. it is to be placed from Standard Parts or selected by a Piping command to build a piping path) it must be generated first and then included in the assembly as a generated file. Single parts are generated automatically when they are placed for the first time. They can also be generated manually using Database Administrator.

Generated Single Part

Sometimes a single part is called a "generated part" or "generated variant". This file is created ("generated") from appropriate Parent part. This is the final file that is linked in your assembly.[1]

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[1] Customer standard parts are managed in a similar way as generated single parts.

Parent Part

Filename	Nominal size
Nut_ANSI_B18_2_4_4_M5_v7.00.par	M 5
Nut_ANSI_B18_2_4_4_M6_v7.00.par	M 6
Nut_ANSI_B18_2_4_4_M8_v7.00.par	M 8
Nut_ANSI_B18_2_4_4_M10_v7.08.par	M 10
Nut_ANSI_B18_2_4_4_M12_v7.08.par	M 12
Nut_ANSI_B18_2_4_4_M14_v7.08.par	M 14
Nut_ANSI_B18_2_4_4_M16_v7.08.par	M 16

Generating Process

Single Part

Single Part

Single Part

Standard Parts Folder Structure and File Locations

The Standard Parts system uses various files. For correct operation all these files must be found in specific locations.

Top Level Standard Parts Folder

The top level standard parts folder is the folder where "SMAPINIFile.sac" is saved. This file contains global Standard Parts settings and can be changed only using Solid Edge Standard Parts Configuration Wizard.
The default folder structure is shown to the right.[1]
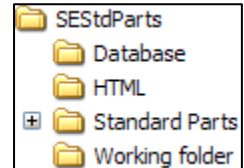
Database File Location

The database file location is where the Standard Parts database is stored. When you are using Insight, the database must be stored on a network drive.
Default database file location is the "Database" folder which is a sub-folder of the top level Standard Parts folder.

Note: Databases created for different platforms (file system, Insight, Teamcenter) differ!!!

Parts Folder Location

The parts folder location is where the Standard Parts files are stored. The structure is shown to the right.
"3D-Standard_Parts_V7.45" and "3D-Standard_Parts_V7.50" are sub-folders with standard parts version 7.45 (Free Content) and 7.50.
Parent parts files are stored in a sub-folder "Parent_Parts".
"Navigation" folder stores information on visual navigation which can be used in the Standard Parts application as one of the ways to find a required part.
"Previews" folder contains supplementary GIF previews, which serve

mainly to clarify the meaning of particular characteristics of a part selected in the Standard Parts application.
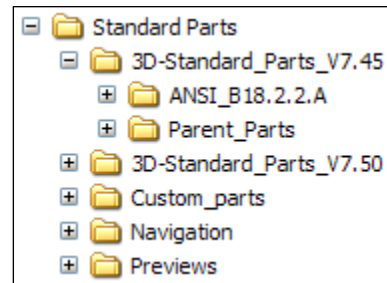
Generated Parts Location

If a part is being used (opened or placed) for the first time, it must be generated. In this case a new single part file is created. This file is stored in a location which depends on the platform used – file system, Insight or Teamcenter.
If the file system is being used, single parts are generated in folders which are on the same level as the "Parent_Parts" folder. The folder names correspond with the end categories to which these single parts belong.

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[1] Default definition is different for URL (Insight) location because of path/filename lengths and character limitations

For an Insight installation, you can set a generated parts folder yourself. Single parts will be generated into its sub-folders corresponding with end categories as for the file system.
Teamcenter manages generated parts automatically (if it is set as the generated parts location).

Working Folder Location

Standard parts are normally generated into the locations described above. These paths are stored as links in the assemblies where the parts are placed. By activating the checkbox "Copy to working folder" (in the Standard Parts application) this default setting can be changed. In this case, the part is generated into its default folder and a copy of the part is created in the Working folder. This copy is then referenced in the assembly. This option is useful if you know you want to make an individual user part from some standard part (e.g. to bore a hole into a bolt). For parts with variable characteristics (such as profiles) the part is always copied into the Working folder because every part is unique (e.g. beams with different lengths created using one profile). The location of the Working folder can be either set to the default (subfolder of the Solid Edge Standard Parts folder) or set as the folder where the active assembly resides. The setting can be changed in the Standard Parts dialog (Tools – Options – checkbox "Use assembly folder as working folder"). A general definition of this folder is: the folder where the parts are copied to when the option "Copy to working folder" is set in Standard Parts dialog (if the "Use assembly folder as working folder" option is not set).

Note: Variable parts (parts with at least one variable characteristic - structural shapes and tubes typically) are always copied to working folder.

HTLM Folder Location

The HTML folder is a folder containing additional files associated with end categories. These files can be displayed using a special function from the Standard Parts application. HTML folder is created automatically as a subfolder of the top level Standard Parts folder.

Database

The Standard Parts document management system uses a Microsoft Access database. Users can manage the Solid Edge Standard Parts system via the Database Administrator application; this application can Add – Update – Register New – Delete CAD Standard Parts files within the database.

Document Version Schema

Standard Parts can be modified (e.g. norm or model changes). However, every standard part must be unique in the Standard Parts document management system. So, some parts must be included in the system (database) as the same part as an existing part, but with higher version. Solid Edge Standard Parts system can be managed with a Standard Part version. This means that all standard parts documents have their own information (property) about their version. The version is recognized and processed on every Database Administrator operation depending on the current settings. The most important usage is with the Add/Update Parts function: a user can easily keep his database up-to-date with the latest version of standards or can keep all parts with all older versions in the current database.

Solid Edge Standard Parts Configuration Wizard

The Solid Edge Standard Parts Configuration Wizard is a tool that helps you set the necessary paths (described in Standard Parts Folder Structure and File Locations section) needed for using Solid Edge Standard Parts. This tool allows creation of the entire directory structure needed for organization of the system and files. If you install Solid Edge Standard Parts on a local machine or network you do not need to launch this wizard (paths are set automatically), but there is a possibility to change settings. In the case of an Insight installation or just storing parts on Insight or Teamcenter, using this wizard is always necessary.

In "Step 1" the **top level Standard Parts folder** should be set. If you set this to an existing top level Standard Parts folder here, all settings in the Configuration Wizard will be set automatically according to the appropriate SAC file. Top level Standard Parts can be set to a file system folder or an Insight folder (Insight is not recommended but possible).

In "Step 2" the rest of necessary folders can be set. You can use the default values easily by clicking on "Defaults" button.

Database file location can only be set to a file system folder. The "Defaults" button sets the file name to "StandardParts19.mdb" in a "Database" folder, which is created as a sub-folder of the top level Standard Parts folder.

Default **parts** and **working folder locations** are "Standard Parts" and "Working folder" (sub-folders of the top level Standard Parts folder) if the "Store parts on file system" option is set. When using "Store parts in Insight" the parts and working folder must be an Insight location. Teamcenter manages these locations automatically (these cannot be modified). Generated parts and working folder must always be set on the same platform.

Single parts are usually generated with a version suffix in the file name (e.g. Pipe_DIN_2440_1x3.25_v7.50.par). If you require generation of single parts without this suffix (e.g. Pipe_DIN_2440_1x3.25.par), you can uncheck the "Add version to generated file name" box. This checkbox is available only when storing parts on the file system.

Database Administrator

The Database Administrator is an application designed for managing Standard Parts database/s. It allows you to create, delete and update a database, switch between databases, register parts into a database, generate parts, define part priorities[1], etc.
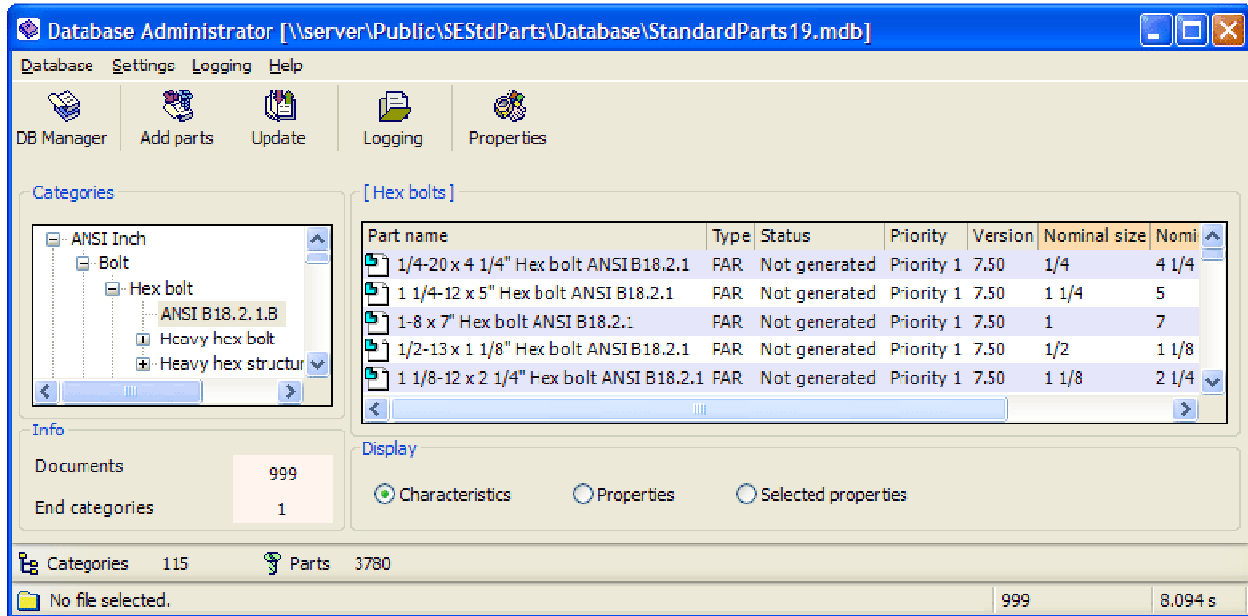
Database Administrator Main Form

In the Database Administrator main form displays the categories and documents which are registered in the current working database.

The "Categories" section shows a complete tree structure of all categories in the database. Using the context menu, you can perform various operations. You can generate all parts in a selected category (including subcategories). You can also delete selected categories from the database (files are not removed from the hard drive) and set priority for a selected category. Priority can be used for filtering parts in the Standard Parts dialog where you can select the lowest priority that should be visible. In this case the "Use priority filter" option must be set (in menu Settings – Priorities).
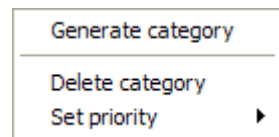
In the "Documents" section there are parts (variants) displayed which are included in the selected end category. For each document, Part name (or Path), Type, Status, Priority and Version are always shown. You can choose from 3 types of views in the "Display" section. The "Characteristics" view is the default view. If it is selected the characteristics of the parts are shown (all parts included in the end category must have the same set of characteristics). If you select the "Properties" view, then Standard file properties (Author, Comments, Keywords, Subject, Title, Category, Company, Manager) and Solid Edge properties (Material, Document Number, Revision Number, Project Name) will be displayed. Choosing "Selected properties" allows you to set your own type of view. In the properties dialog you can select the properties to be displayed.

Using the context menu in the documents section you can perform some operations in addition to those that can be performed on the categories. The "Edit properties" command starts PartEditor. Using it you

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[1] Priorities are only used with Customer Standard Parts and UGS Library standard generated parts

can change properties, characteristics, and categorization etc. of the selected document. This is recommended only for customer standard parts.

With the "open" command you can open selected parts in Solid Edge. It is active only for generated parts. You can also generate non-generated parts using the "Generate" command. The "Update" command updates the properties of the selected document (the properties are reread from the file).

You can also remove selected documents from the database and even delete the appropriate file (active only for generated documents).

There is also the possibility to set priority for each document individually.

Edit properties

Open	F3
Generate	
Update	
Remove from database	Del
Remove from database and delete file	Shift+Del
Set priority	▶

In the "Info" section there are document and end category counts displayed for the currently selected category. In the row below the total number of categories and documents registered in the database is displayed.

In the bottom-most row of the Database Administrator form the full path of the selected document is displayed. This can also be displayed instead of "Part name" in the documents section. This can be set by checking "Show full path" in "Selected properties" dialog.

Database Manager

In the Database Manager you can create a database, set a database as a working database or delete a database. The working database cannot be deleted. If you want to delete it, you should set another database as the working database first.

In the "Databases" section there is a list of databases available in the location set in the "Path" box. The "Working database" (its file name with full path) is displayed in the box below the "Path" box.
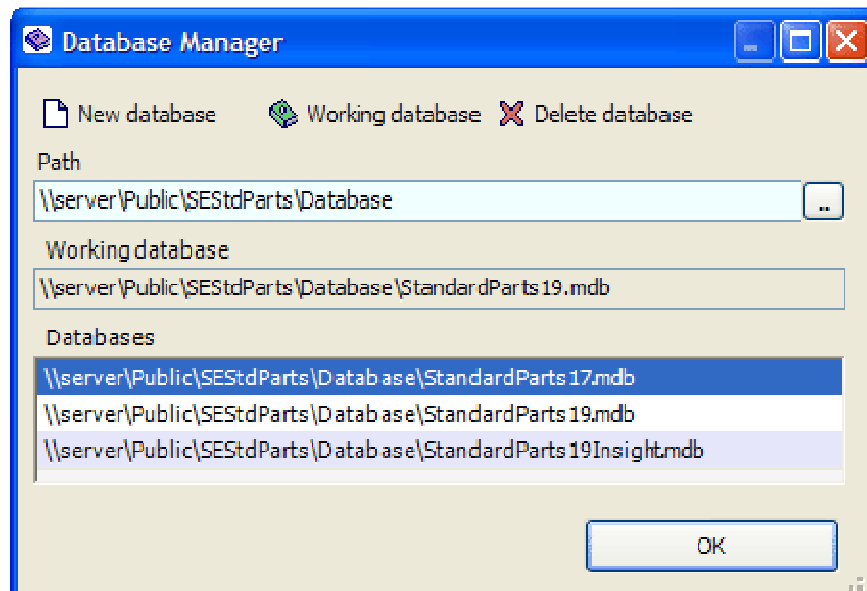
Change Path Command

The Change path command is useful when you need to migrate your documents to another location. The Change path command solves (changes) links at the database – not in the file! Migration in this case means moving e.g. your all Solid Edge Standard Parts Directory or Solid Edge Standard parts directory to other machine/disk, even if you are deciding to change your local system to a shared network system.

If, for example, the name of a server and therefore the path to the standard parts has changed, you can correct the path via the "Change paths" function. Usage is straightforward. Enter the old path (which you want to change). You can also select the path from list box. The path listed is the previously used path from the "Add parts" dialog. Then enter or browse the new path and click "OK" to start the process.

To check whether the path of the selected Standard parts has been changed correctly, click 'Edit properties' from the context menu. If it has not been changed correctly, you will receive an error message that the file was not found.

Note: The "Change paths" function works with text replacement, not with real file system paths (pay attention to the ending backslash). This function does not change any links to standard parts in Solid Edge assemblies! This function is not acceptable when you are migrating your system from UNC path to URL path (Insight)! This function can change your database in an irregular way – this option is recommended only for administrators.
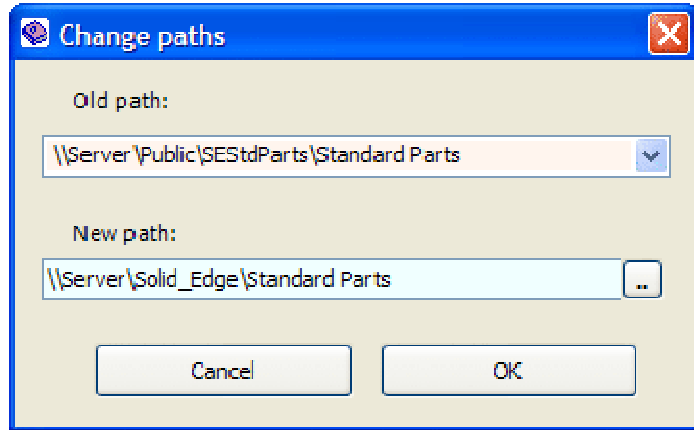
Add Parts Command

The most important function of the Database Administrator application is to register parts into the working database. After launching the Add Parts command the Add Purchased Standard Parts dialog appears. It has four options to add parts from various locations and in various ways:

1. **Add new part file(s)** option serves for registering user documents as Standard Parts.
2. **Add from existing single part file** allows registration of one single part or parent part in the database.
3. **Add from existing multiple part files** option allows registration of multiple standard parts from specified location/s
4. **Add from a delivery database** option uses the Delivery database for registering or updating standard parts in the database.
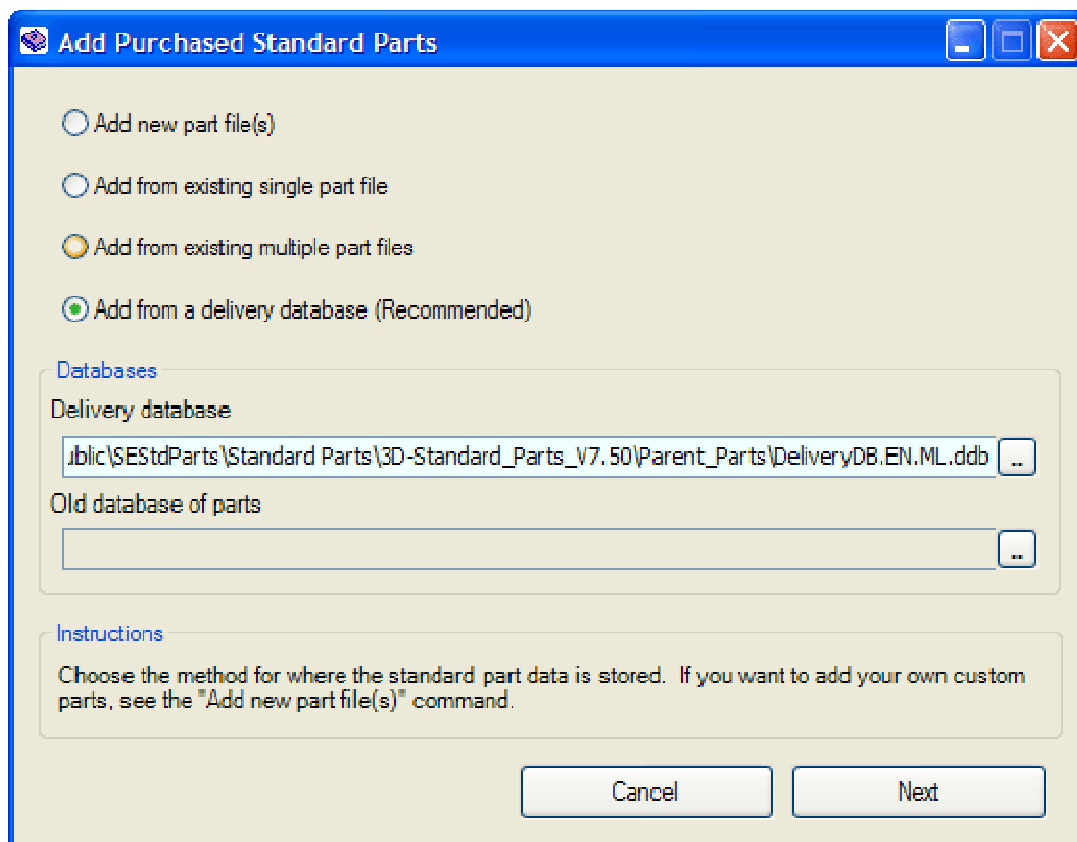
Update Command

Updating the database is the process of rereading and updating information from previously registered parts in the database. During this operation no new documents are imported except when the 'Scan links' option is checked. In this case, Database Administrator scans the links of all documents (registered in database) and checks whether the linked documents are also registered in the database. If they are not, Database Administrator registers them with the database.

To update the whole database, use the "Update all" button. If you want to update only certain categories, check them in the list and use the "Update selected" button.
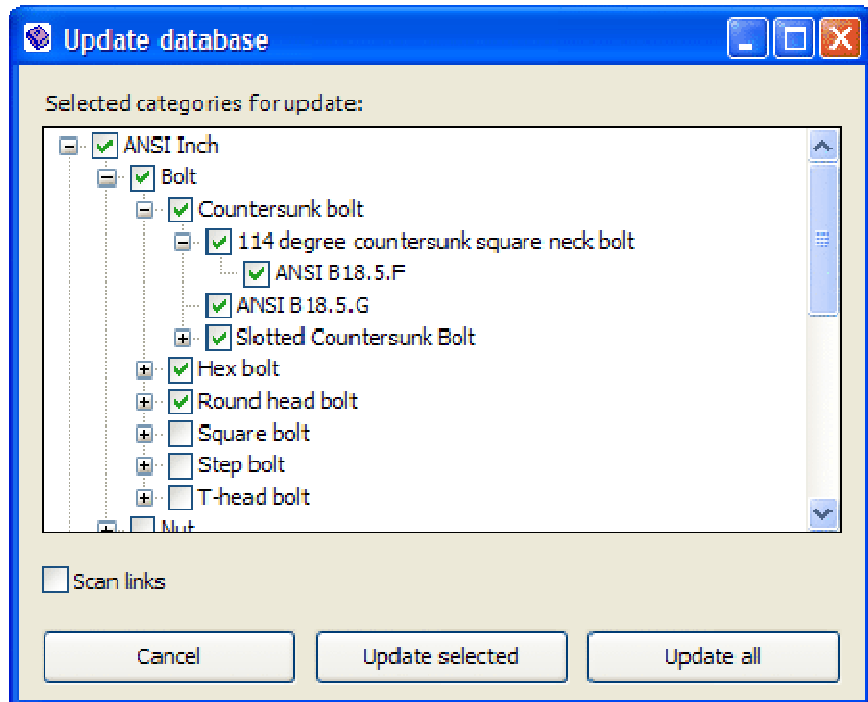
Log File

The log file saves information about Database Administrator processes (registering parts and updating the database). The log file contains dates, times and descriptions of the processes, file names (with full paths) of the documents which were processed and the status of these processes (OK, Aborted, etc.).

With a click on the "Logging" button in the toolbar, the "Logging" dialog appears. Here you can set the log file (you can browse the location and enter file name). You can also enable or disable logging via the "Use logging" checkbox.
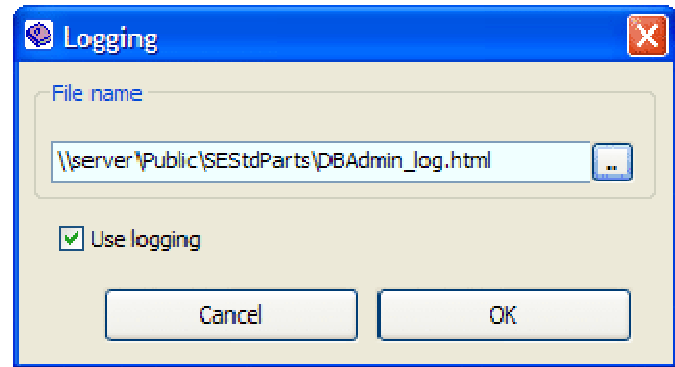
Creating Customer Standard Parts

In Standard Parts version 19 there is a special tool for creating customer standard parts. This tool is integrated in the Database Administrator. This wizard is called BulkPartAdder. You can launch it by selecting "Add new part file(s)" in the Add Purchased Standard Parts dialog (Add parts command) and then clicking on the "Next" button.

In the first step you can select one of three methods to add your custom parts (these are dependent on your source file):

- Add from Family Part Master – you can add all the Family of Parts members at once.
- Add from active Microsoft Excel sheet – you can use an Excel table (either the whole table or just a part of it) which contains information on the parts you want to add.
- Add from manual data entry – you can manually select the part file(s) which you want to add to the database as standard part(s).

Creating Customer Standard Parts from Family Part Master

This method allows easy registration of an entire Family of Parts using its Part Master.

Step 1

Select "Add from Family Part Master" option and click on "Next".
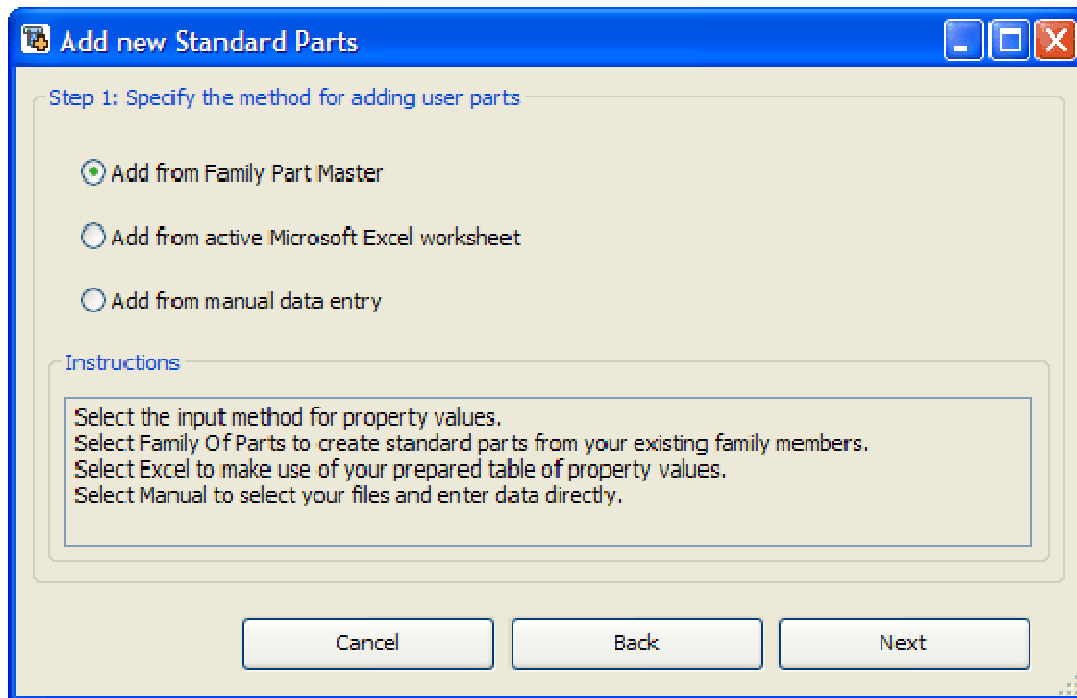
Step 2

Browse (via the browse button) to the required Family Master Part. Members of the family of parts will appear with full path, status and all variables which have values set in the Family of Parts. Family members must be created and up-to-date.

Step 3

Settings in this step are optional. If you want your custom part to be used as a fitting (or tube) by Solid Edge Piping you can check "Add to Piping" and set a number of coordinate systems pairs. To create a custom part to be used in Fastener Systems, check "Add to Fastener System" and select appropriate Fastener type (bolt, screw, washer or nut).

Step 4

In this step you should set a category for your custom part(s). Each category should have at least one characteristic assigned to it. If you want to add your custom parts to an existing category, you have to use the same characteristic set or change (typically increase) the category version. You can create two types of characteristics – constant or variable. For variable characteristics a variable must be set in the appropriate section of the Properties grid (this variable should correspond with an existing variable in the selected Family Master Part). For each characteristic you should set a unit class and the appropriate unit. In the Properties section there is also the possibility to add (or remove) custom properties. You can also enter a specification and set a standard for the created end category.

Step 5

In step 5 the values of the characteristics (set in the previous step) should be set for each part (family member). If you click on the "(select source)" field (below the characteristic name) a list of variables (from the Family of Parts) will appear. Selecting one of them will cause its values to be filled out in the appropriate column. All values for all characteristics must be set.

If you check the "Show additional properties" box, then the file properties (standard, Solid Edge and custom) will be also displayed in the grid. Filling in these values is optional.

| Full path name	Part Name	Priority	Nominal size	Ring diameter
			Dimension 339	(select sourc ∨
\\Server\Public\SEStd Parts\Working folder\Ring_20x15.par	Ring_20x15	1	15	(select source)
\\Server\Public\SEStd Parts\Working folder\Ring_20x10.par	Ring_20x10	1	10	Dimension 339
\\Server\Public\SEStd Parts\Working folder\Ring_20x5.par	Ring_20x5	1	5	Dimension 374
\\Server\Public\SEStd Parts\Working folder\Ring_20x20.par	Ring_20x20	1	20	20

☐ Show additional properties

Instructions

Fill in all requested property values. Only file properties are optional here.

| Cancel	Back	Next

Step 6

In this step you are required to create a Visual Navigation category for your newly created category. Creation of a Visual Navigation category structure requires the following steps:

1. Click on the "Add node" 🔄 button to create new visual navigation node.

2. Confirm new visual navigation code.

3. Set a name for new node.

4. Click on the "Change thumbnail" 🖼 button and select an image to replace the visual navigation code on the visual category's thumbnail.

5. Change the category thumbnail for the end category if necessary (preview of the part is selected by default).

6. Click on "OK" to confirm your newly created visual navigation category.
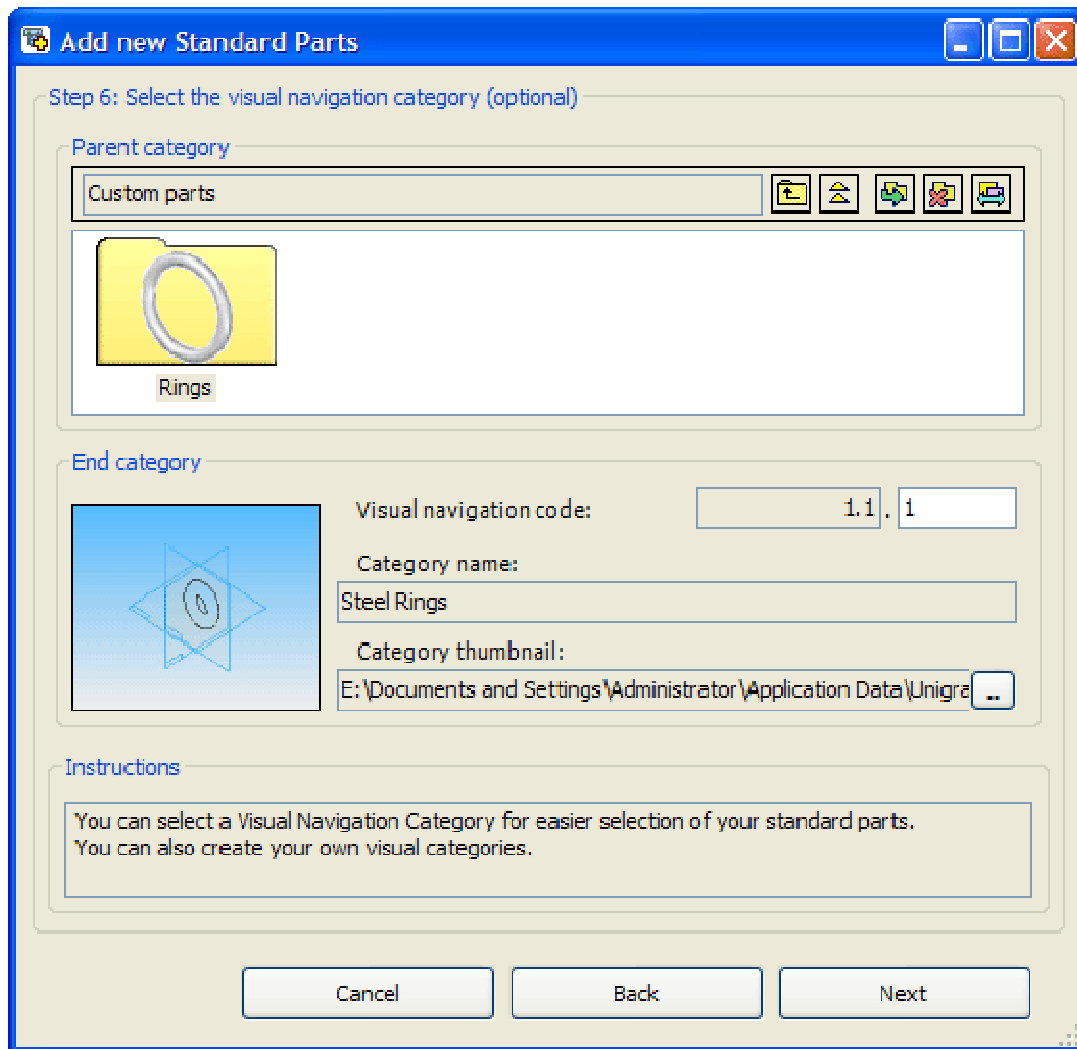
Step 7

In this step the necessary part information is created automatically at first. If you click on "Finish", the required parts will be copied to the Custom_parts folder (in the generated parts file location) and registered in the database.
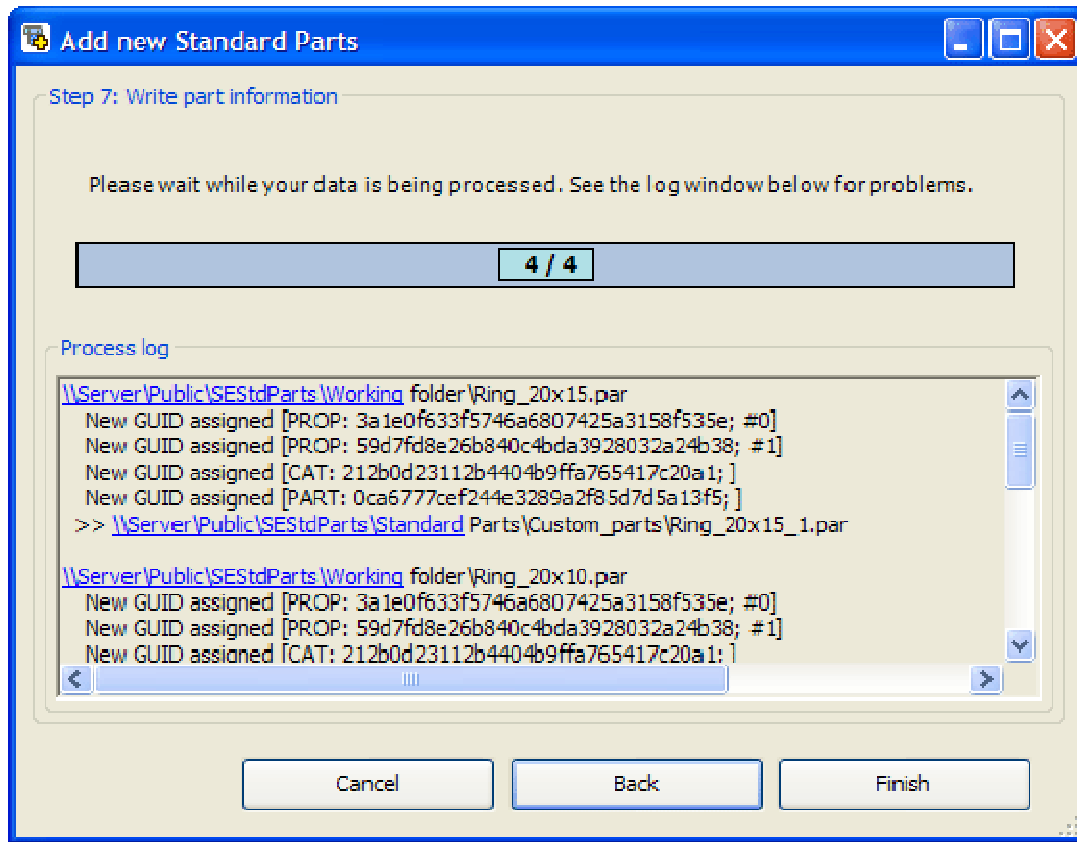
Creating Customer Standard Parts from Excel worksheet

This method may be suitable when you have created an Excel table which contains information about parts you want to register as customer standard parts (see example below).

	A	B	C	D
1	Path	Diameter	Height	Part name
2	\\server\Public\SEStdParts\Working folder\Cylinder20x40.par	20	40	Cylinder 20x40
3	\\server\Public\SEStdParts\Working folder\Cylinder30x40.par	30	40	Cylinder 30x40
4	\\server\Public\SEStdParts\Working folder\Cylinder30x50.par	30	50	Cylinder 30x50
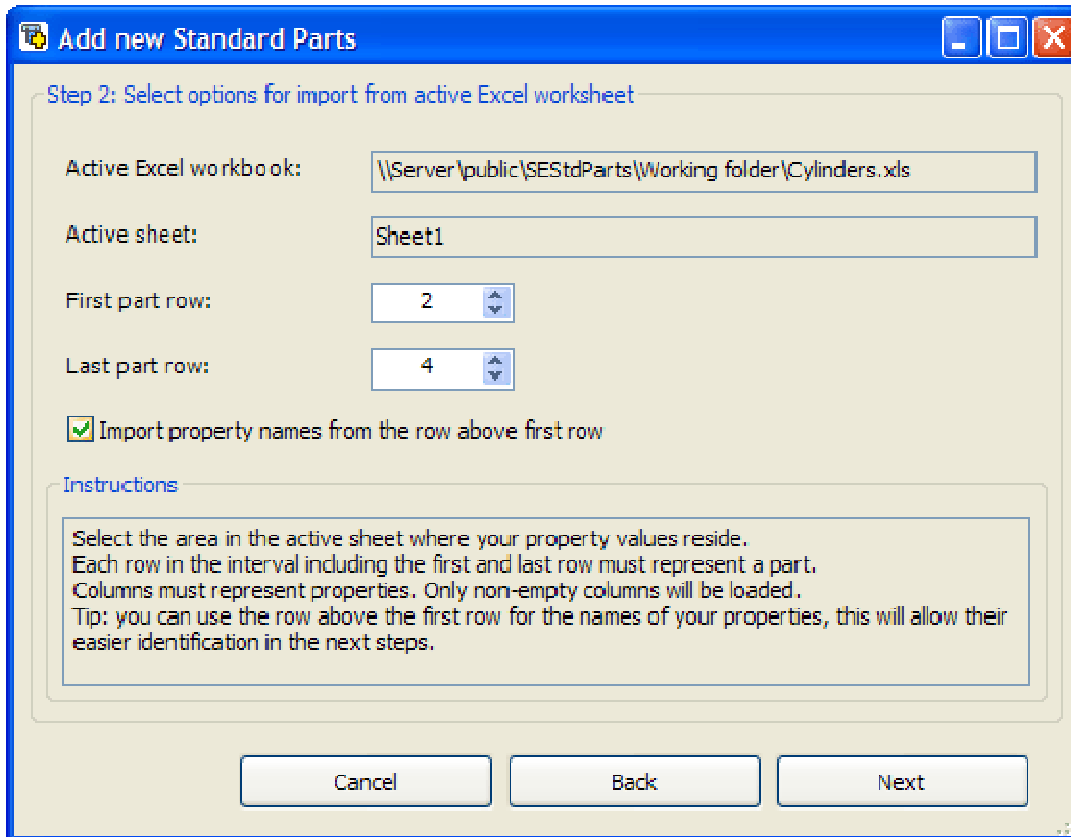
Step 1

Select "Add from active Microsoft Excel worksheet" option and click on "Next" (required Excel worksheet must be opened).

Step 2

In this step you should select the rows (from the active sheet) which will be processed. These are rows 2 to 4 (in our case). By checking a box you can also import names for each column in the sheet. It could serve for better identification of values in step 5.

Step 3

Settings in this step are optional. If you want your custom part to be used as a fitting (or tube) by Solid Edge Piping you can check "Add to Piping" and set a number of coordinate systems pairs. To create a custom part to be used in Fastener Systems, check "Add to Fastener System" and select appropriate Fastener type (bolt, screw, washer or nut).

Step 4

In this step you should set a category for your custom part(s). Each category should have at least one characteristic assigned to it. If you want to add your custom parts to an existing category, you have to use the same characteristic set or change (typically increase) the category version. You can create two types of characteristics – constant or variable. For variable characteristics a variable must be set in the appropriate section of the Properties grid (this variable should correspond with an existing variable in the selected Family Master Part). For each characteristic you should set a unit class and the appropriate unit. In the Properties section there is also the possibility to add (or remove) custom properties. You can also enter a specification and set a standard for the created end category.

Step 5

This step requires, at a minimum, values for Full path, Part Name and characteristics set in previous step to be entered. For each column there is a drop down list "(select source)" containing the labels of columns in the active Excel worksheet. It allows mapping of values from the active Excel worksheet to the appropriate columns that are displayed in this step. You can also set (or edit) values manually.

If you check the "Show additional properties" box, then file properties (standard, Solid Edge and custom) will also be displayed in the grid. Filling out these values is optional.

Step 6

In this step it is possible to create a Visual Navigation category for your newly created category. This is the same as for creating customer standard parts from a family part master.

Note: If you are registering your custom parts into an existing end category, this step will be skipped automatically.

Step 7

In this step the necessary part information is created automatically at first. If you click on "Finish", the required parts will be copied to the Custom_parts folder (in the generated parts file location) and registered in the database.

Creating Customer Standard Parts from Manual Data Entries

This is a method for common user parts. You can just manually select required parts (or part locations) and complete the wizard.

Step 1

Select "Add from manual data entry" and click on "Next".
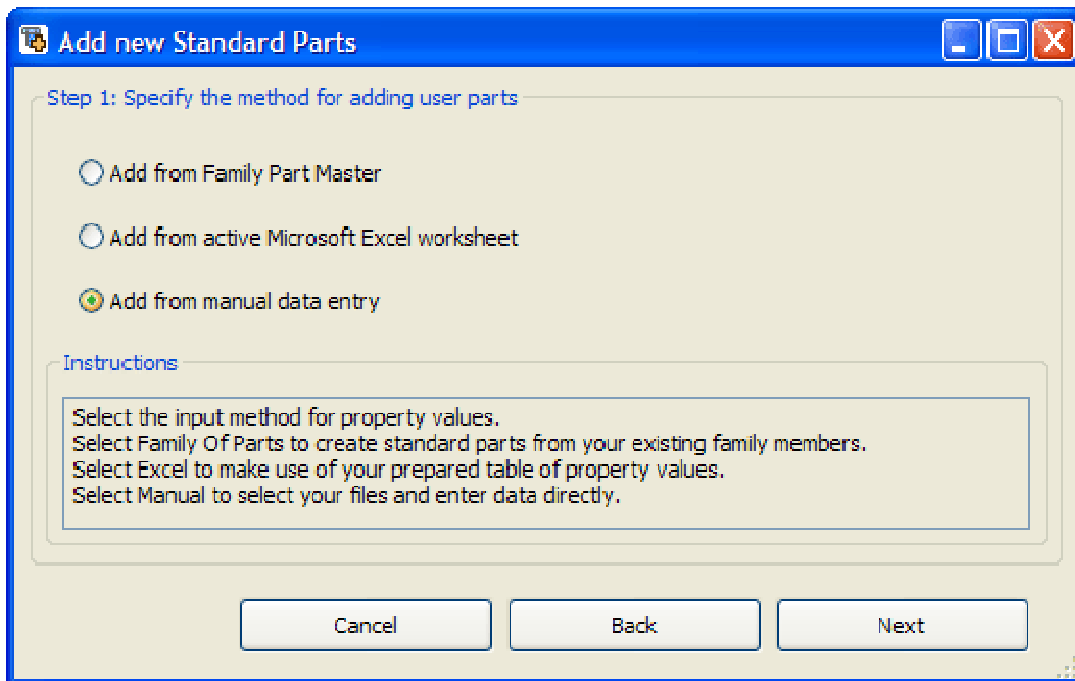
Step 2

In this step you should select the parts you want to register as customer standard parts. You can add documents from a specified path in bulk (Add path) or add particular files (Add files). You can even remove files from the list (Files to add) via the "Remove selected" button.

Step 3

Settings in this step are optional. If you want your custom part to be used as a fitting (or tube) by Solid Edge Piping you can check "Add to Piping" and set a number of coordinate systems pairs. To create a custom part to be used in Fastener Systems, check "Add to Fastener System" and select appropriate Fastener type (bolt, screw, washer or nut).

Step 4

In this step you should set a category for your custom part(s). Each category should have at least one characteristic assigned to it. If you want to add your custom parts to an existing category, you have to use the same characteristic set or change (typically increase) the category version. You can create two types of characteristics – constant or variable. For variable characteristics a variable must be set in the appropriate section of the Properties grid (this variable should correspond with an existing variable in the selected Family Master Part). For each characteristic you should set a unit class and the appropriate unit. In the Properties section there is also the possibility to add (or remove) custom properties. You can also enter a specification and set a standard for the created end category.

Step 5

This step requires, at a minimum, values for characteristics set in previous step to be entered. Full path, Part name and Priority are set automatically, but Part name and Priority can be modified manually.
If you check the "Show additional properties" box, then file properties (standard, Solid Edge and custom) will also be displayed in the grid. Filling out these values is optional.

Step 6

In this step it is possible to create a Visual Navigation category for your newly created category. This is the same as for creating customer standard parts from a family part master or using an Excel worksheet.

Note: If you are registering your custom parts into an existing end category, this step will be skipped automatically.

Step 7

In this step the necessary part information is created automatically at first. If you click on "Finish", the required parts will be copied to the Custom_parts folder (in the generated parts file location) and registered in the database.

Adding an Existing Single Part File

The "Add from existing single part file" option in Database Administrator V19 has the same function as the "Add part" command in previous versions.
In the Add part dialog you can browse (in the Parts or Generated parts location only) to a single file. It can be Single part, Parent part or any custom Solid Edge document. If the Scan links option is checked, links found in the selected part will be checked and linked documents will also be registered.
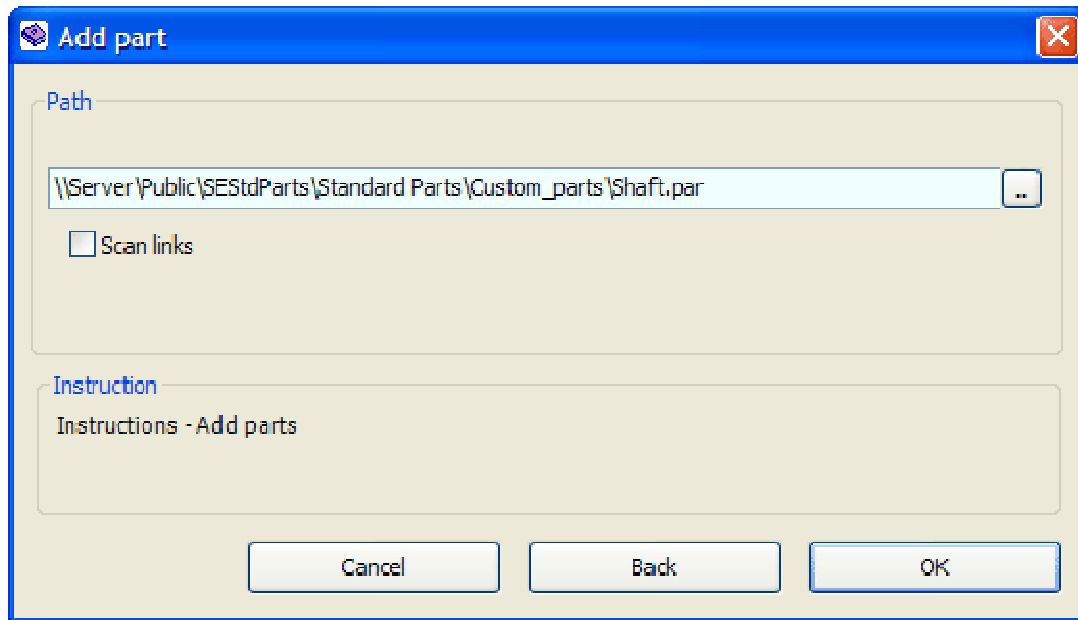
After clicking on "OK" the selected part will be registered.

Registering Non-standard Parts

If the selected part contains no standard parts information (category, characteristics etc.), the following dialog will appear:
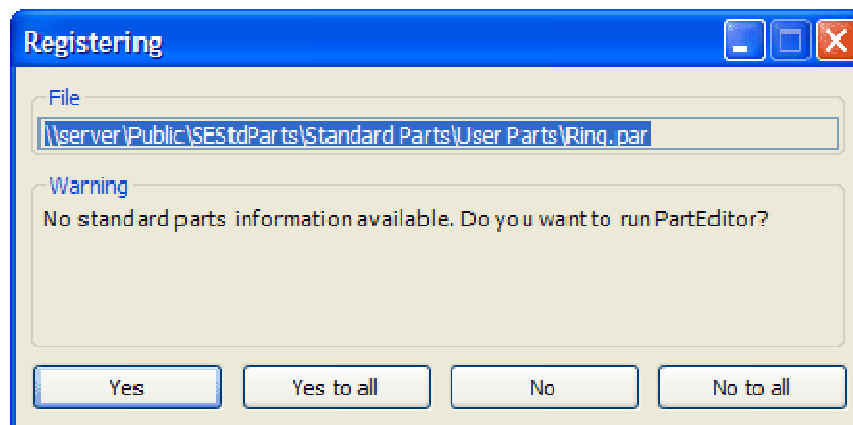
1. Click on "Yes" to launch PartEditor.

2. In the PartEditor main form click on the "Category" button 🧩 in the main menu to create a new category for your part.

   a. Click in the free space in the categories list with right mouse button to show the context menu.

   b. Click on "Add" to create new category (e.g. "Company Standards").

   c. Click with the right mouse button on the newly created category to add a subcategory (e.g. "Rings").

   d. Enter a specification for this "end category" (e.g. "Steel rings").

   e. Click on "OK"
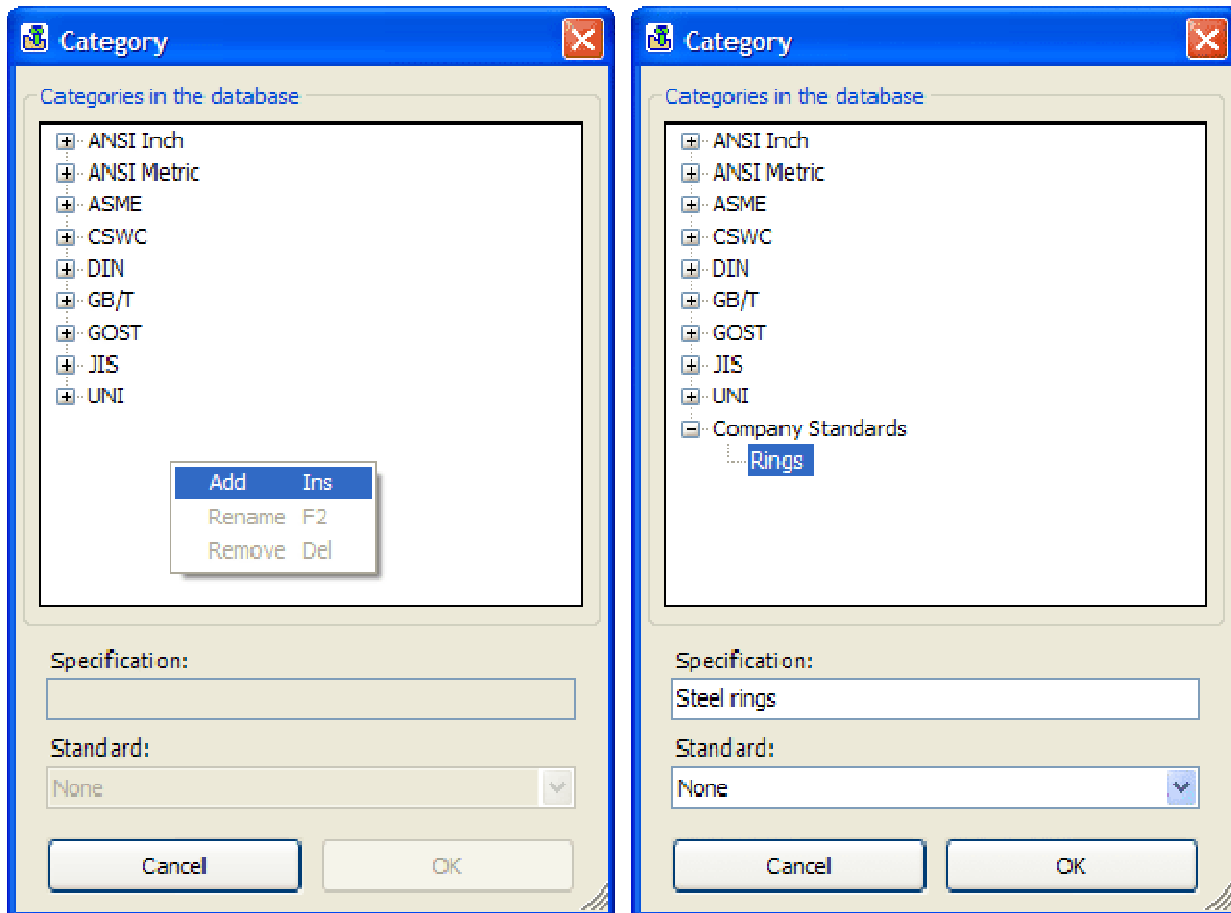
3. You can also create a visual navigation category structure (optional). In the PartEditor main form click on the "Visual navigation category" button in the main menu to create a new visual navigation category for your part.

   a. Click on "Add node" 🔧 to create new visual navigation node.

   b. Confirm the new visual navigation code.

   c. Set a name for the new node.

   d. Click on the "Change thumbnail" 🖨 button and select an image to replace the visual navigation code in the visual category's thumbnail.

e. Change the category thumbnail for the end category if necessary (preview of the part is selected by default).

f. Click on "OK" to confirm your newly created visual navigation category.

4. Every Solid Edge Standard Part must have at least one characteristic[1]. These are shown in the bottom section of PartEditor's Category Info tab. For example, for defining a ring we need two characteristics (two diameters).

a. Click on "Add characteristic".

b. Specify names and values for all characteristics. Select unit class ("Length" in this case) and unit ("mm").

c. Enter "Part name" (e.g. "Steel ring 50/10") and click "OK".
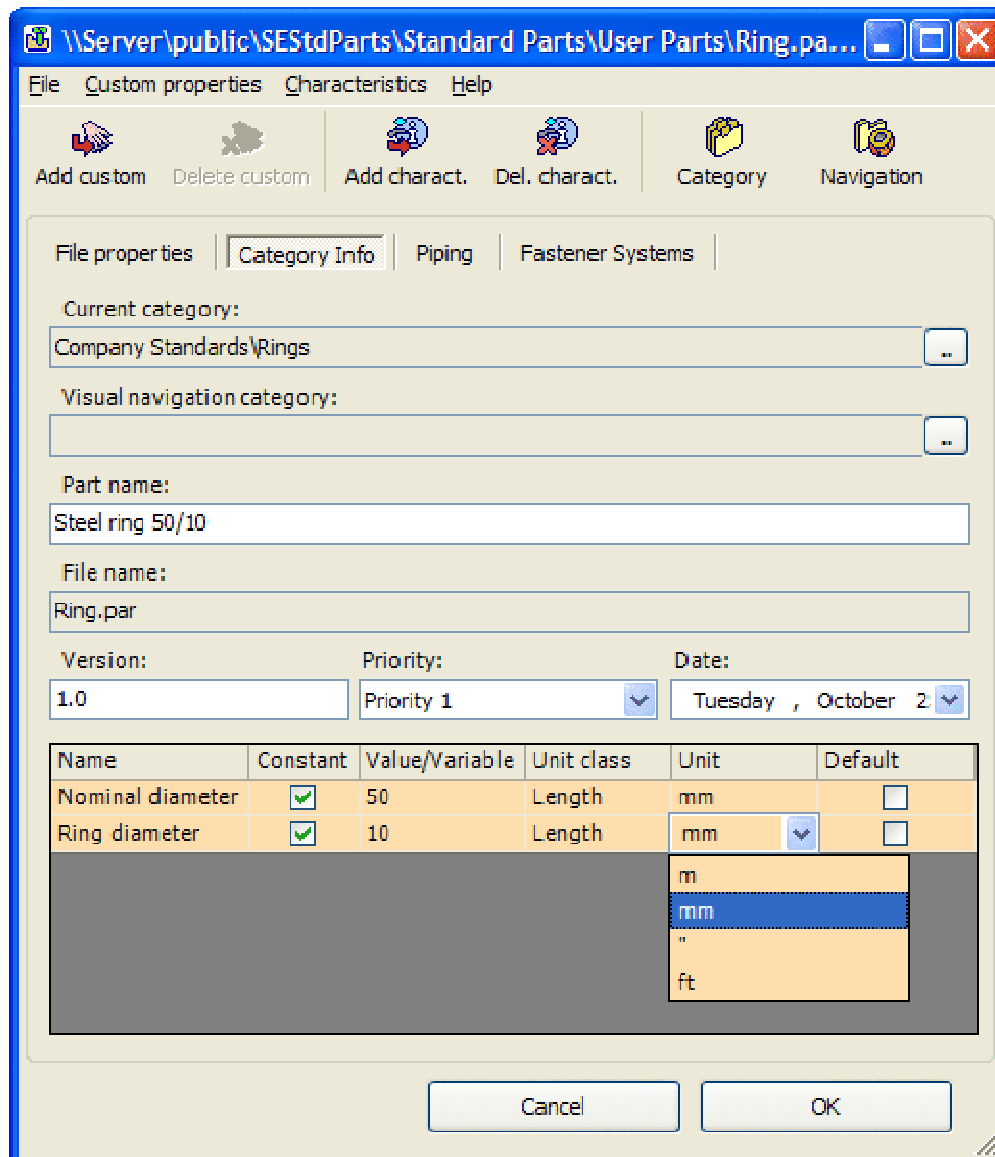
5. The user document "Ring.par" is now fully categorized as a standard part.

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[1]Characteristic: key and probably unique property that belongs to described part (object)

Adding Existing Part Files

The "Add from existing multiple part files" option in Database Administrator V19 has the same function as "Add parts" command in previous versions.
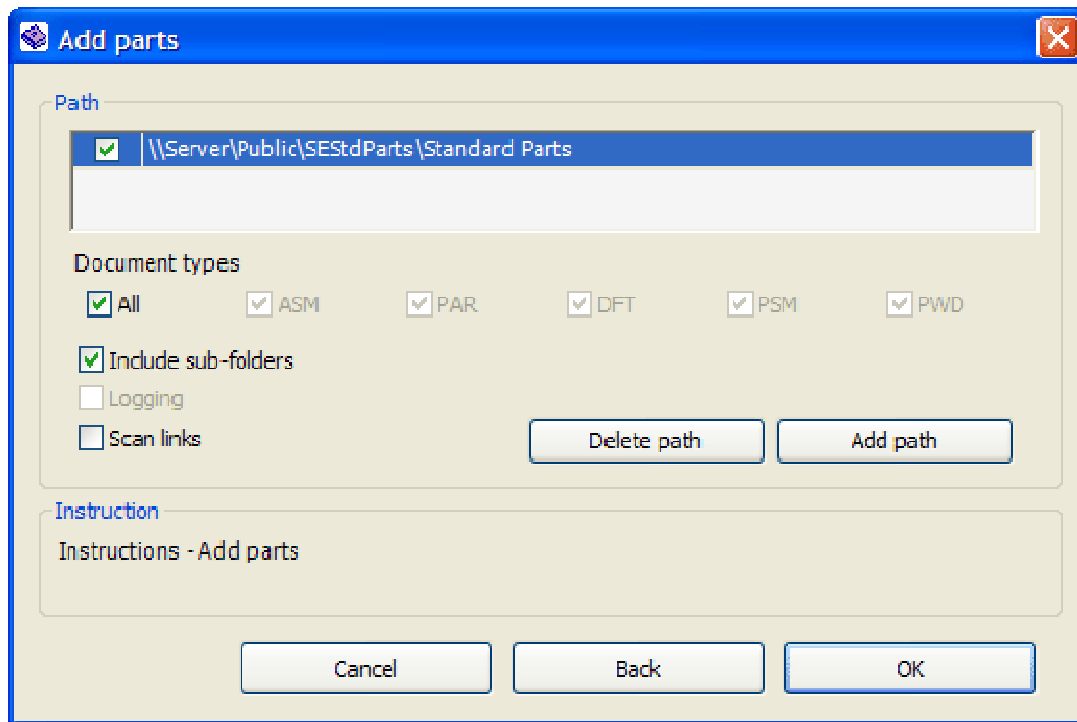

This option registers all Standard Parts (installed Libraries only) from the selected location/s. You can use it for registering the Solid Edge Standard Parts libraries.

For registering new parts check the required path or use the "Add path" button if it is not present in the list of paths (upper dialog section). Then click "OK" to launch the process.

There are also some additional options. You can select the document types that should be registered (all, ASM, PAR, DFT, PSM, PWD). If you check the "Include sub-folders" option the parts from the selected path/s and also from their subfolders will be registered. The "Logging" option enables logging information from the process to a log file, the log file must already exist and logging must be enabled. If the "Scan links" option is checked the links in parts will be checked during registration and linked parts will also be registered in the database.

If a part from the selected path/s contains no standard parts information (category, characteristics etc.), then the "Registering" dialog will appear (see Adding an Existing Single Part chapter for more information).

Registering Standard Parts Using Delivery Database

Using a Delivery Database (Add from a delivery database) is the default method for registering standard parts contained in the Standard Parts Libraries. This technology has several benefits:

- A move from an older version of Solid Edge (V17 or lower) to V19 is greatly simplified. It is possible to create a new database containing the same parts (as the original database) with updated versions.
- It is possible to select, from a clearly presented list (in SmartInstaller), the parts you want to install. The benefit being that the parts you will not be using will not be registered in database. This can improve the performance of the Standard Parts application and shorten the time consuming process of registering standard parts in the database.
- If you obtain a new version of Standard Parts Library, then only the parts with changes (relative to the version you have installed) will be registered in the database.
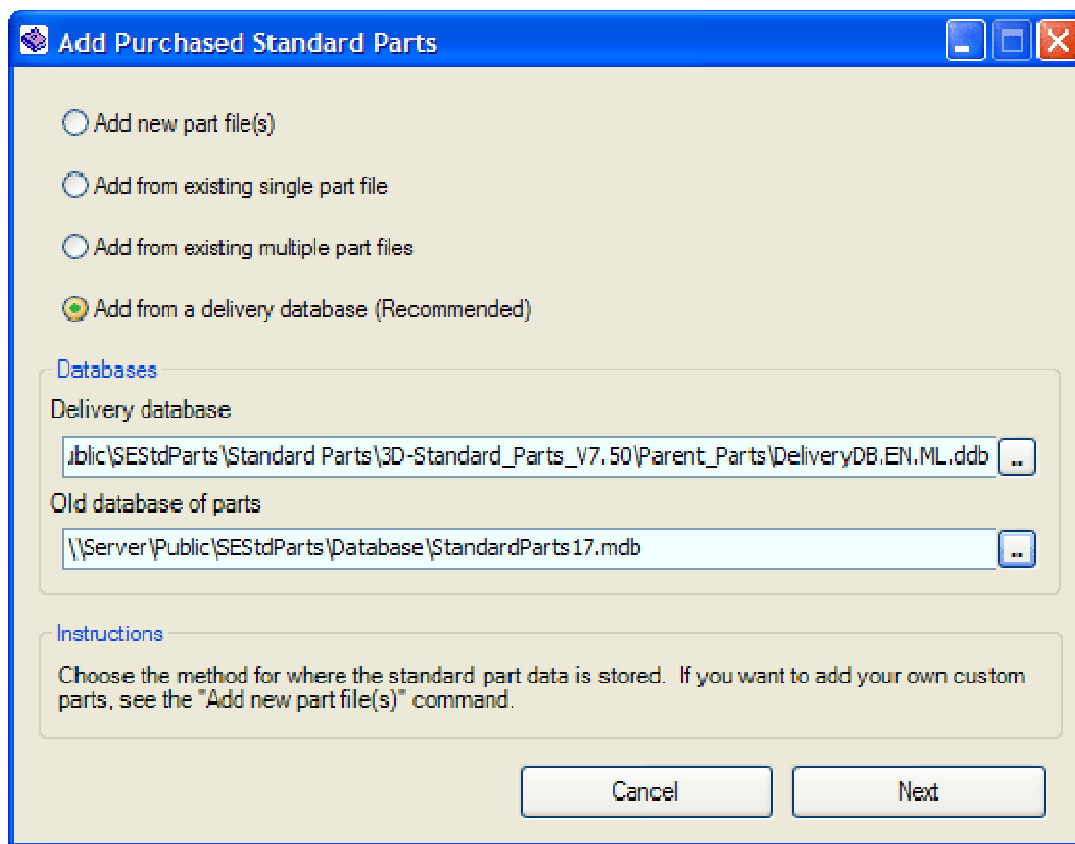
In the Add Purchased Standard Parts dialog there is the Database section where it is possible to set the input databases.

In the **Delivery database** field the delivery database which will be used for registering/updating parts in the database should be selected. If there is a Delivery database located in the parts folder (but not in "Previews" or "Navigation" subfolder) it is selected automatically (Delivery databases are copied to the "Parent_Parts" folder automatically during the Standard Parts Libraries installation). If there are multiple Delivery databases, the one with the highest version is selected automatically.

Note: Delivery databases for Machinery and Piping Library are delivered separately. If you have purchased both of these libraries you should register/update in two steps and select one of the delivery databases manually. Delivery databases can be distinguished via their file name, which contains "ML" for Machinery Library or "PL" for Piping Library (e.g. DeliveryDB.EN.ML.ddb and DeliveryDB.EN.PL.ddb).

The **Old database of parts** field serves to set a reference database. It has various purposes for various operations:

- **"Moving" from an older Solid Edge / Standard Parts version** – Suppose that you have newly installed Solid Edge and Standard Parts version 19. However, you want to continue using your existing Standard Parts system created with a lower version. Given this scenario you can use an existing Top Level Standard Parts folder with one restriction: you should create new database (if the previous version was V18 there is no need to change it). If you change the database via the Solid Edge Standard Parts Configuration Wizard (and save the changes), the Add Purchased Standard Parts dialog will pre-select the previously used database (lower version) as the "Old database of parts".
- **Updating a database with the new Standard Parts Library** – If you obtain a new version of the Standard Parts Library (typically with a Service pack) and try to add these parts via the Add parts dialog, then no "Old database of parts" will be displayed. In this case your current working database will be taken as reference database.
- **Registering parts in a new database** – There is also the case, that you would like to create a new database containing the same parts as an existing database, but you want to create the new database with up-to-date parts. In this case you can set your reference database (Old database of parts) manually.
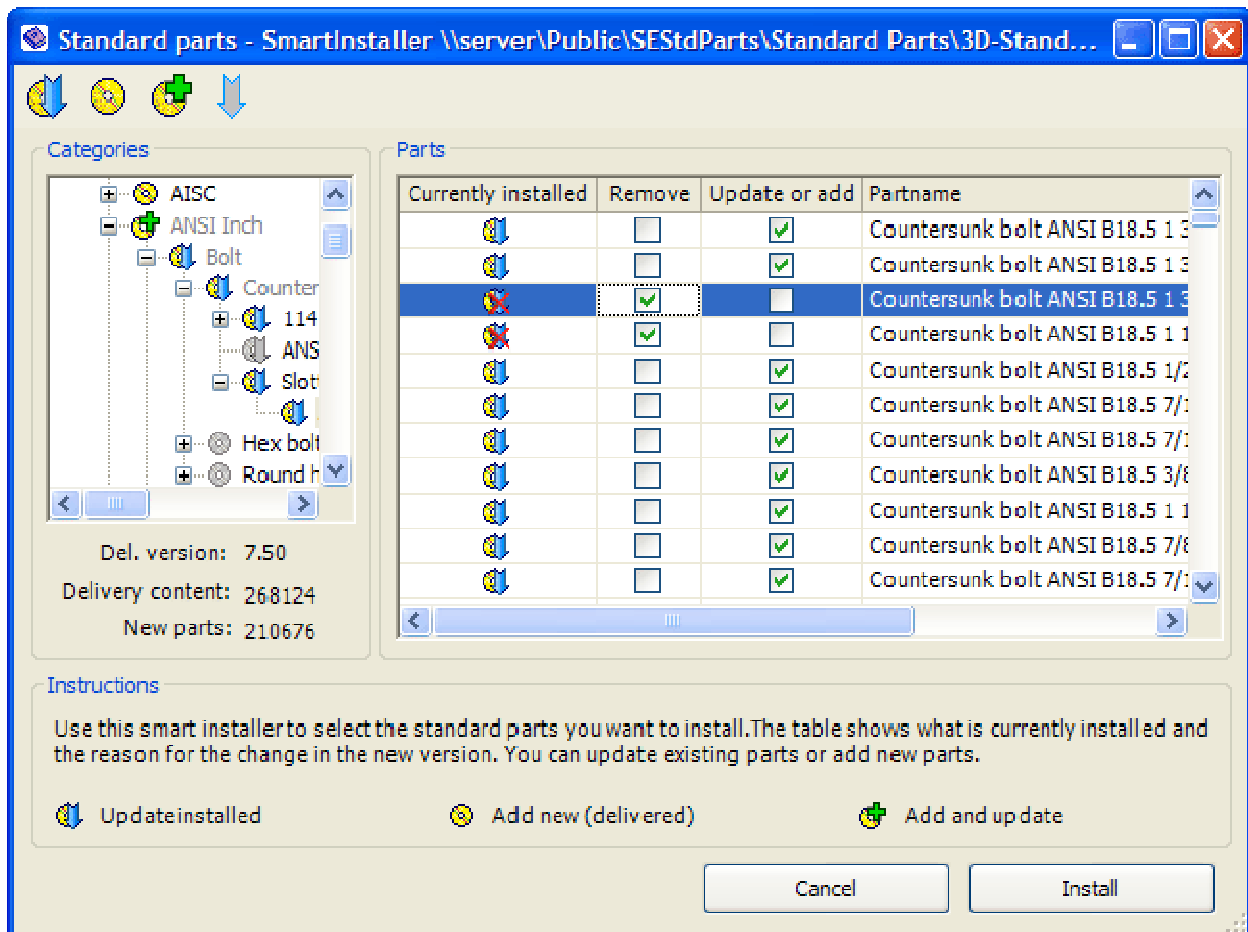
SmartInstaller Main Form

The SmartInstaller main form consists of a Categories and a Part section.
The **Categories** section displays all the categories contained in the Delivery and "Old" database. For each category there is a symbol describing the operation to be performed during installation:

- Category will be newly registered
- Category will not be registered
- End category is installed and will not be updated
- Category is installed and will not be updated (not delivered or no change in delivery database)
- Category is installed and will be updated (status is "changed" in the delivery database)
- Category will be newly registered, existing category will be updated

These operations can be modified via the toolbar buttons **Update**, **Add**, **Add and update** and **Don't change**.

Under the category tree structure there is additional information displayed: Delivery version (version of delivered parts and Delivery database), Delivery content (total number of parts in Delivery database) and New parts (new in the Delivery version)

In the **Parts** section single parts contained in selected end category are listed. There are six columns displayed in the grid:

- The "Currently installed" column displays information on a single parts installation status:
  - Part will be newly registered
  - Part will not be registered
  - Part is installed and will be updated
  - Part is installed (and is not included in Delivery database)
  - Part will be removed from database
  - Part will be removed from database (and is not included in Delivery database)

- If you check the "Remove" box, the selected part will be removed from the database
- The "Update or add" checkbox allows updating or adding of a single part (available only when the selected part is contained in the Delivery database)
- "Part name" serves for part identification
- In the "Reason for change" column you can find information on changes made in the current version. Possible changes are:
  - Geometry – some changes have been made in the part geometry
  - File Name – file name of the parent and/or single part(s) has been changed
  - Part Name – part name has been changed
  - Classification – there were some changes made in the category information/structure
  - Fastener System – some information needed for Fastener Systems has been changed
  - Piping – some information needed for Solid Edge Piping has been changed
  - Invalid – part is no longer valid and will be removed from the database
  - Other changes – some unspecified changes have been made
- The "Installed version" column displays the version of the part that is currently registered in the database.

By clicking on the "Install" button all changes set in the SmartInstaller main form will be performed on the current working database.

PartEditor

PartEditor allows you to edit the properties of documents registered in database. However there are several significant limitations:

- Non-generated parts cannot be edited. You can just view their properties.
- Generated standard parts cannot be edited because they are given a file status read-only. You can remove this status to enable editing, but it is not recommended to edit Solid Edge Standard Parts.
- If you want to add, remove or edit characteristics you must change the version (it is recommended to raise the version) or category of an edited part. The same applies for visual navigation.

From this you can deduce that PartEditor is mainly designed for creating and maintaining new customer standard parts.

There are four tabs in the main form: File properties, Category Info, Piping and Fastener Systems.

File Properties

In this tab the standard file properties (Author, Comments, Keywords, Subject, Title, Category, Company, and Manager), Solid Edge properties (Material, Document Number, Revision Number, Project Name) and custom properties of the current document are displayed. You can edit the value of any property and you can even add or delete any custom property using the appropriate buttons (Add custom and Delete custom) in the toolbar.

If you want to use your customer standard parts with Solid Edge Piping, Fastener Systems or Frame Design you should add a custom property "PartType" with the appropriate value. You can find the list of correct values in the Appendix.
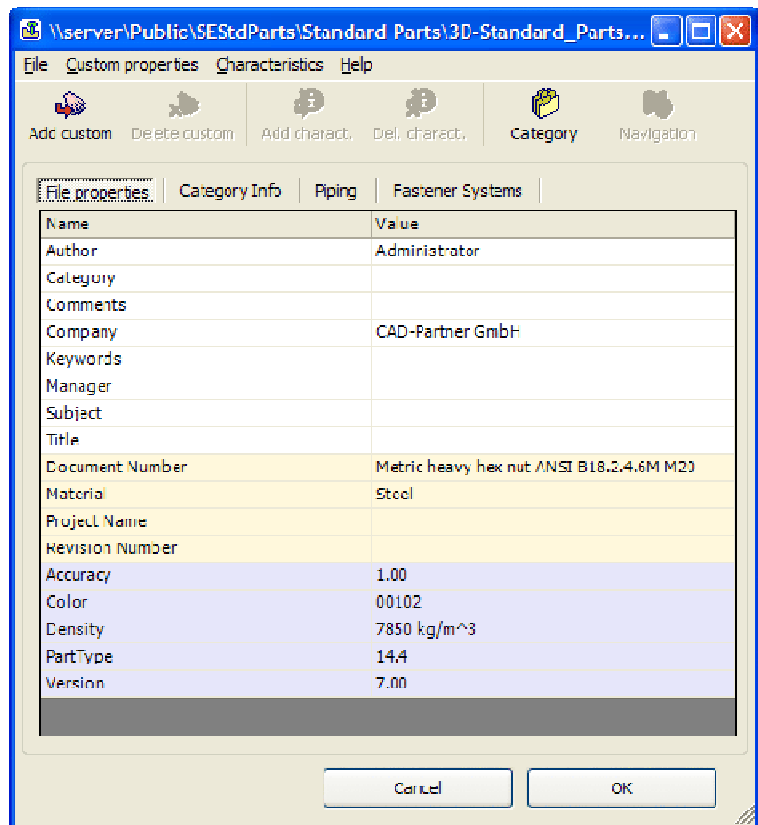
Category Info

The "Current category" box shows the full category structure (topmost category and subcategories including end category) of the current document. With the "Category" button or browse button you can call out the "Category" dialog and change the current category.

"Visual navigation category" displays a category structure for visual navigation. You can change it via the "Navigation" or browse buttons.

Part name is a global, language dependent identifier. It is displayed in the Database Administrator, in the Standard Parts dialog and it is also written in a Solid Edge property "Document Number" (not in Customer standard Parts).

Version is what lets you keep your customer standard parts up-to-date. If you change the version you can edit the characteristics set (even add or delete characteristics) and also you can edit the visual navigation category. Characteristics and visual navigation must be the same for all parts with the same category and version.

Priority can be used for filtering documents in Standard Parts dialog. It must be enabled in Database Administrator.

The date that the customer standard part was created can be set for informational purposes.
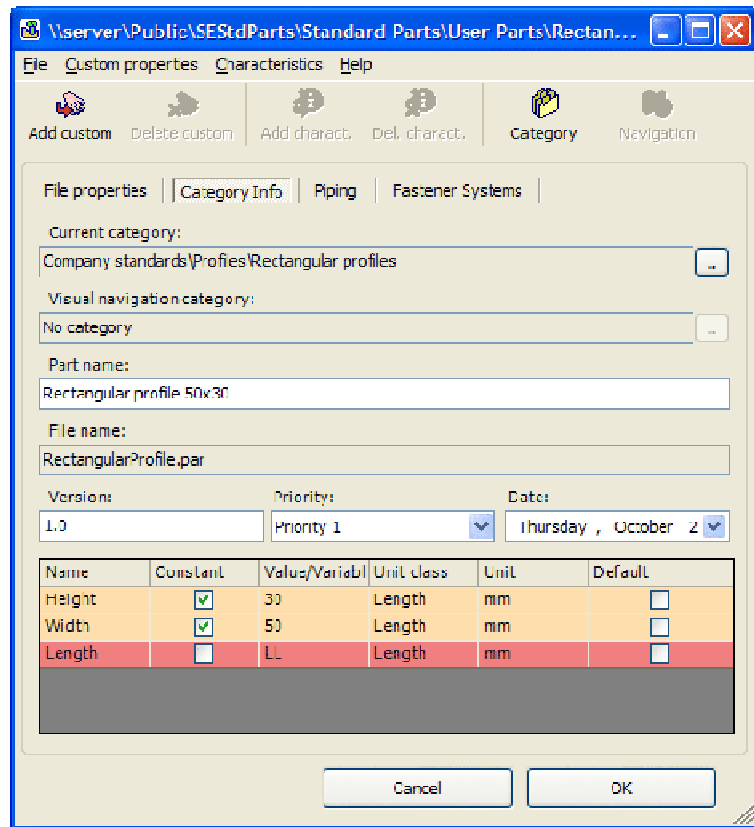
Piping

This tab displays the special property set that is needed for Solid Edge Piping. Here you can set the number of coordinate system pairs; you must then fill out the values in the table for each coordinate system.

For detailed information about Coordinate System parameters see Appendix.
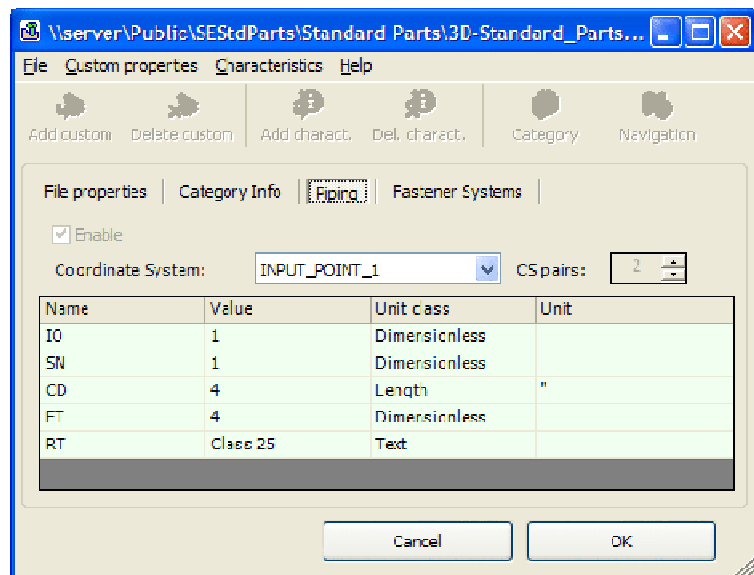
Fastener Systems

In this tab you should set the special properties which are required for Solid Edge Fastener Systems. If you choose the fastener type (bolt, screw, washer or nut) the table will be filled in with the appropriate properties. You should then enter only the values.

For detailed information about Fastener Systems parameters see Appendix.
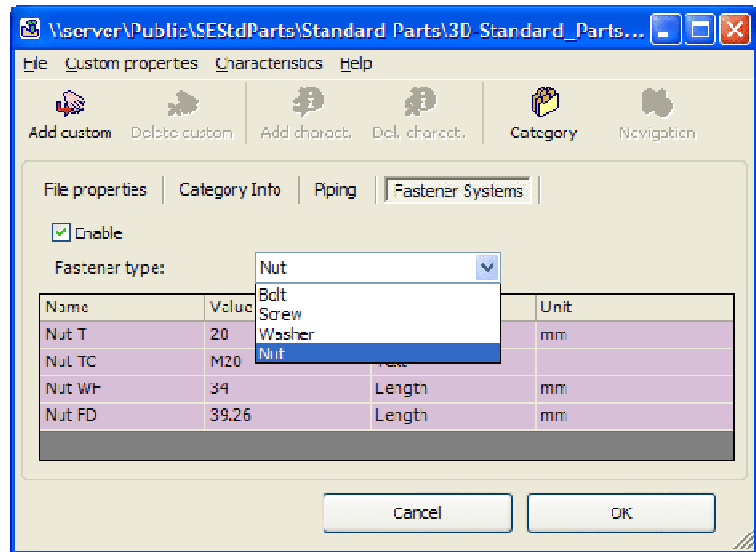
Standard Parts

The Standard Parts dialog is a tool designed for easy selection of standard parts (and customer standard parts) which you want to place, replace or modify in Solid Edge assemblies. You can adjust the Standard Parts environment to best fit your needs. There are several possibilities to change the layout of the Standard Parts application form. You can simply switch between normal and vertical (similar to the Edge Bar) views using the F2 key. You can show or hide other panels such as the preview of the part (F3) and the parts properties (F4). See the following sections for a more detailed description.
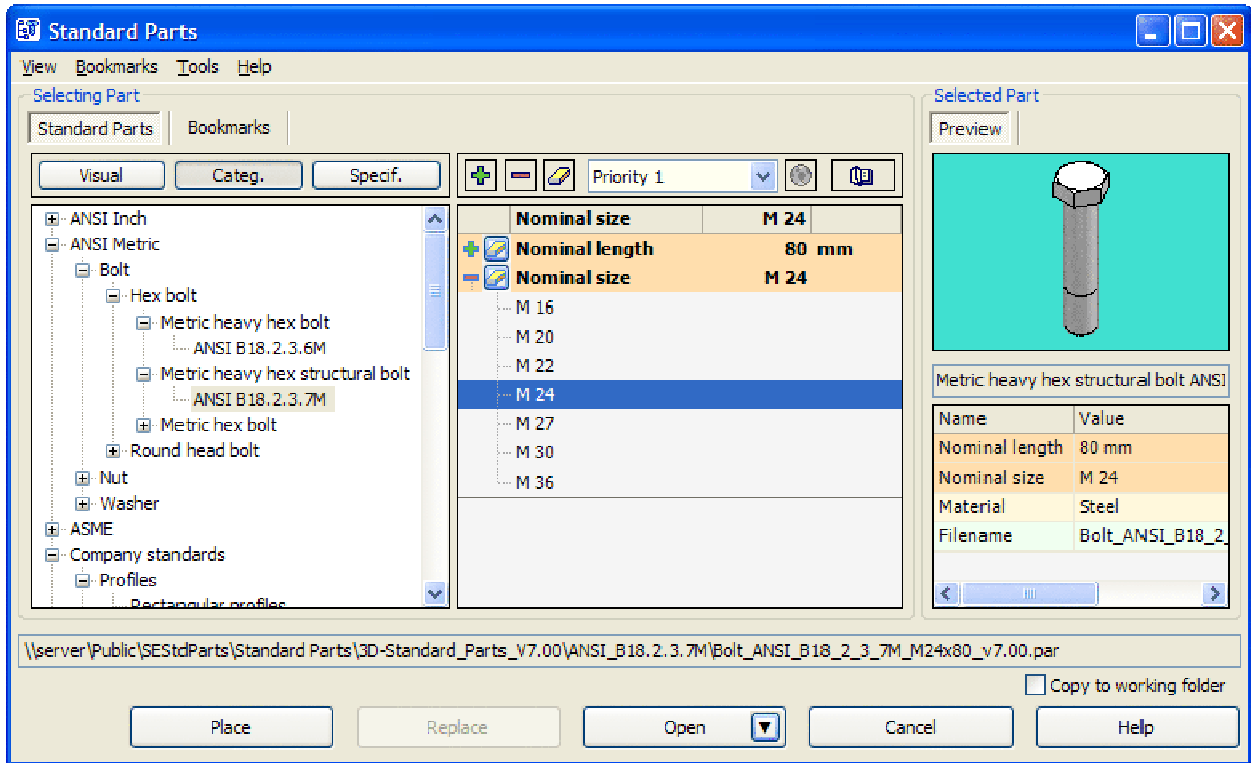
Selecting Part

There are three ways to find a desired part. You can use visual navigation, the categories tree view or the specification view. You can also add selected parts as bookmarks for easier searching in the future.

Visual Navigation

Visual navigation makes it possible to find the desired part easily with pictograms which are specific to the type of the parts.
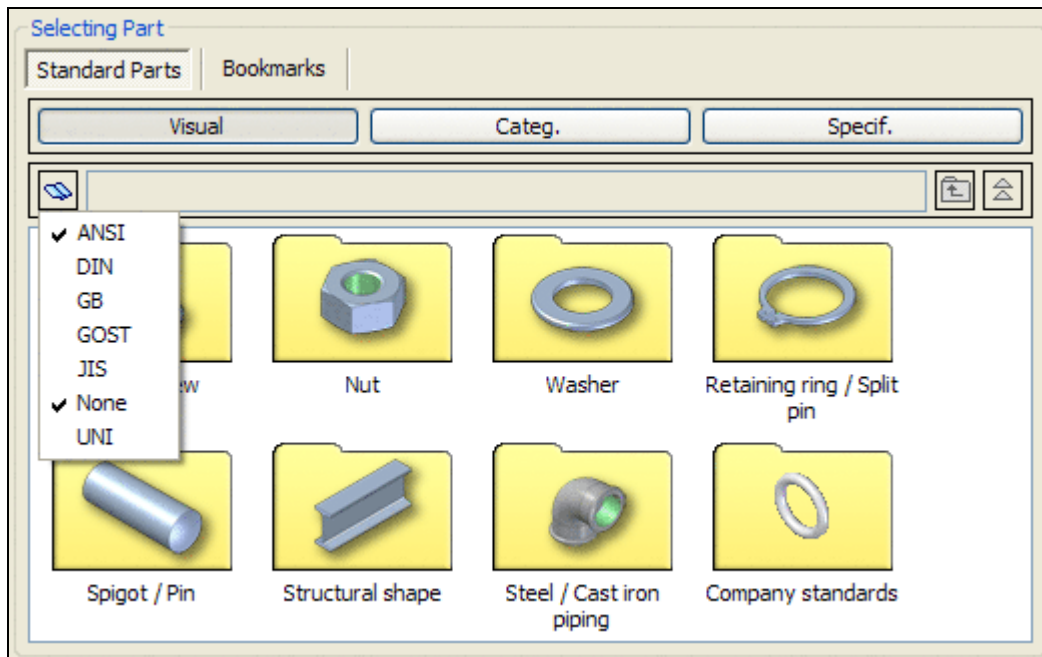
Visual navigation is active when the "Visual" button in the navigation selection bar is pressed. Visual categories are then shown in a section below. In the above image there are top level categories displayed. Therefore the "Level Up" and "Top Level" buttons (on the right above the categories display) are disabled. With the "Change selection of standards" button you can reduce the number of international standards to be displayed.

A double click on any category you will display its content. Categories which contain subcategories are displayed with yellow background (a folder image).

The end category has got no background (see image below). When you click on it the characteristics for the variants contained within will be displayed and you can select the desired variant.
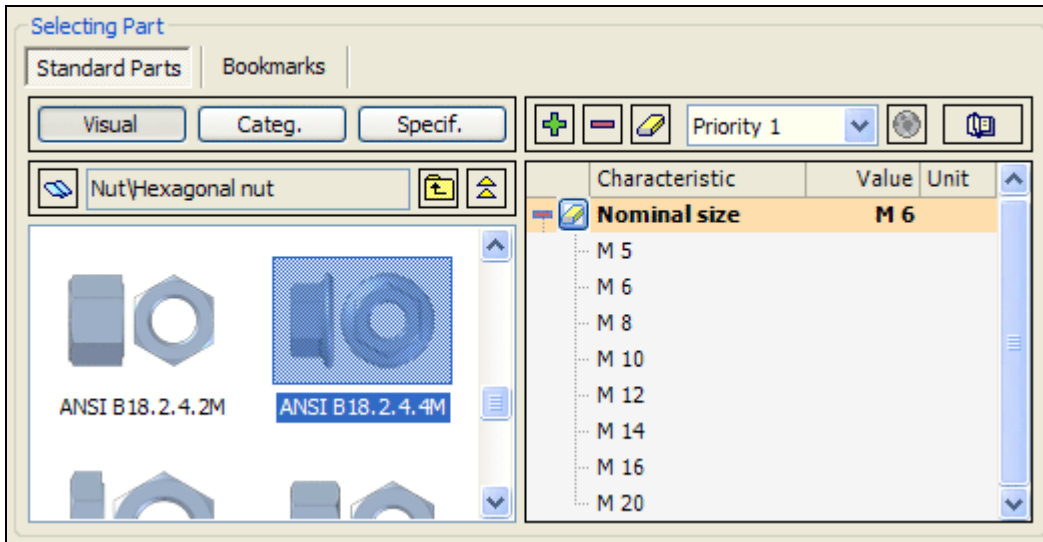
Categories Tree View

The concept of this view is very similar to visual navigation but the nodes are not displayed as images. A tree view is displayed with description of all categories. The key identifier (top-level node in the tree) is the international standard (e.g. "DIN").
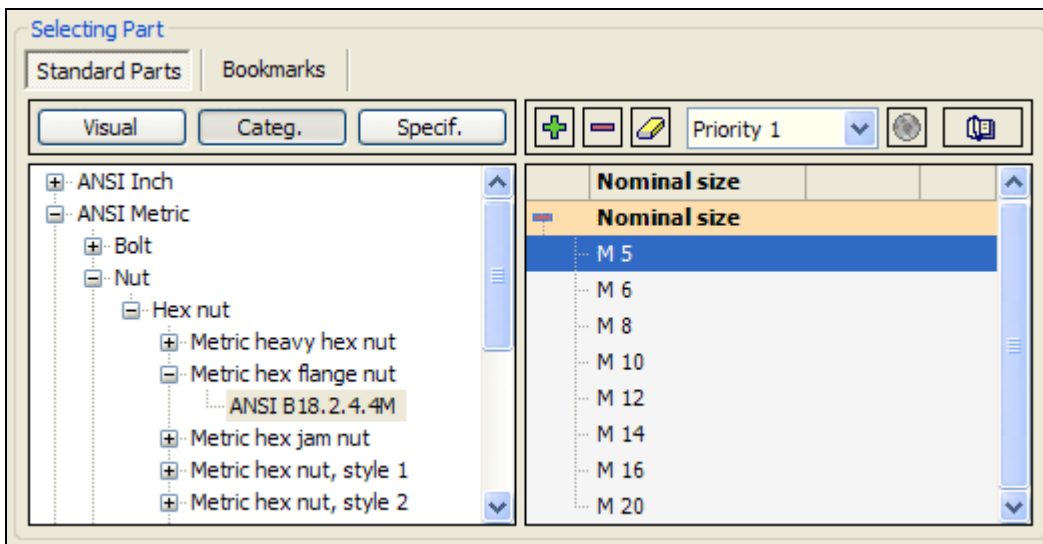
When you click on an end category (a category without any subcategory) the characteristics for the variants contained within will be displayed and you can select the desired variant.

Specification View

If you select the specification view there is just a list of end categories (standards) with versions and specifications displayed. You can select the category which fits best to your needs. If you click on a category the characteristics for the variants contained within will be displayed, and you can then select the desired variant.
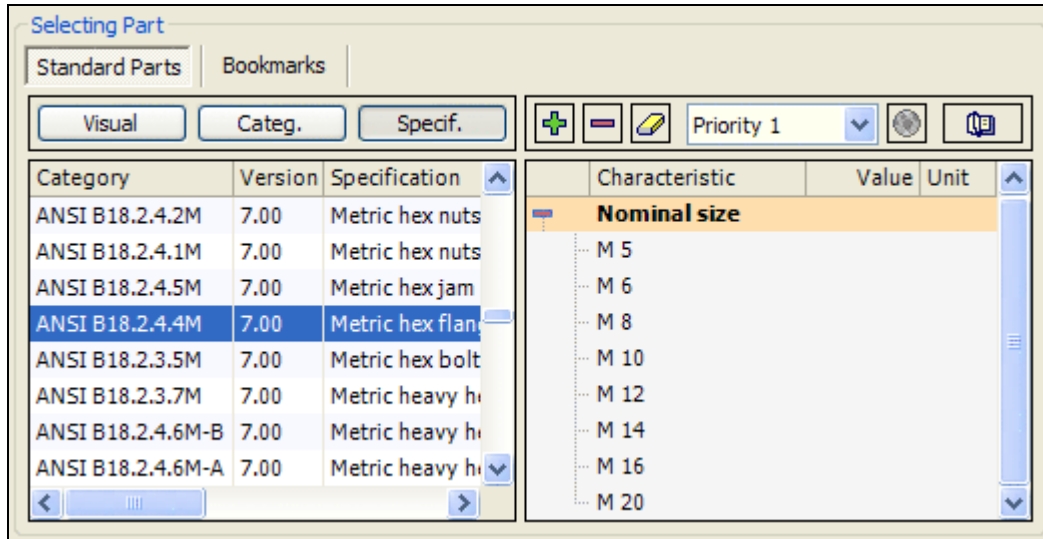
Characteristics

This section is shown in visual navigation, categories tree view and the specification view. It appears when an end category is selected and allows selection of a particular variant from the selected end category (see the right section of the image above).

All the characteristics for the currently selected end category and all their possible values are displayed. If you click on a value, it will be selected and the appropriate characteristic will collapse. The values of the other characteristics that are no longer available (in combination with selected value) are then disabled.

Selection is completed when values for all characteristics have been selected (all characteristics are collapsed with the selected values displayed). The selected part is then displayed in the 'Selected part' section and can be used for the desired operation.

There are several tools to aid selection:
- Expand all characteristics ✚ expands all values for each characteristic. The current selection is kept.
- Collapse all characteristics ▬ collapses all values for each characteristic. No new values are selected in the process.
- Delete complete selection 🖉 cancels the selection. Values of all characteristics are expanded.
- Priority filter allows selection of only those parts with priority higher or equal to the priority set in combo box.
- The open associated HTML or PDF file 🌐 command displays a HTML or PDF file, typically containing some additional information about selected end category.
  There are two different ways to associate files with a category:
    a) Associated files must be located in HTML folder which is a subfolder of the Solid Edge Standard Parts Installation folder (e.g. "\\Server\Public\SEStdParts\HTML"). The file name must correspond with appropriate end category and must have 'htm', 'html' or 'pdf' extension, e.g. "ANSI B18.2.2.D.htm" (for the end category ANSI B18.2.2.D).

b) It is also possible to add a "html.txt" file to the Solid Edge Standard Parts Installation folder. Associated file/s should be specified by name/s in row record/s with the following syntax: "last subcategory;file name".
Example: ANSI B18.2.2.D;\\server\documents\ANSI_nuts.pdf (for end category ANSI B18.2.2.D the ANSI_nuts.pdf file located in \\server\documents\ will be displayed)

- Show standard parts list 📖 changes the type of view (see next chapter).

Standard Parts List

This is an alternative possibility for displaying variants in an end category. This view replaces the default characteristic tree-list view for the selected standard (end category). The standard parts list does not show all the possible values of all characteristics, instead it just displays the available combinations. This option only makes sense for end categories with at least two characteristics.

You can change the display type by clicking on the book symbol ◆ in the top right of the "Selecting Part" section.
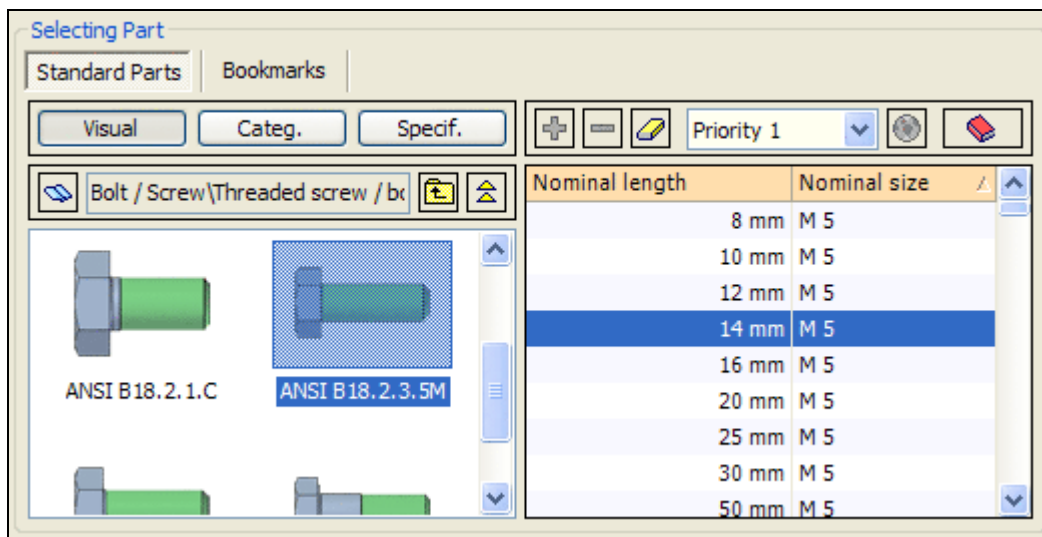
Some characteristics are always shown, but you can display other properties[1]. To do this, select "Display of parts list" in the "Tools" menu and then in the "Properties to display" dialog check the properties you want to add.

Bookmarks

Using bookmarks is advantageous if you use certain parts over and over again. Adding a bookmark is very easy: when the desired part is selected just use the "Add Bookmark" function from the "Bookmarks menu.

---

[1] Meaning a property contained in the current database. If the property has no value for the selected part then it is displayed without a value.

The properties list displayed in the "Bookmarks" tab is the same as for the "Standard parts list". You can also change them in the same way (see "Standard Parts List section").

Selected Part

In this section of Standard Parts dialog you can see information about the selected part. The specifications, characteristics, selected properties and a preview are displayed.

Preview

At the top of this section there is a preview of the selected part displayed. You can select the "CAD" preview (Preview) or "GIF" preview (e.g. Preview 1) if available.

In the Standard Parts options (Tools) you can set the type of CAD preview. It may be 3D (you can move and rotate the preview like in Solid Edge) or bitmap (the image contained in the part file).

3D preview can have two different background colors. Cyan (see figure) signifies a non-generated part. Dark blue is for generated parts.

Displaying the 3D preview may take time for more complicated parts or assemblies. If you want to increase performance, it is better to switch to the bitmap preview or disable the preview completely.

GIF preview usually shows the meaning of the different characteristics of the selected part. GIF previews are an advantage of Solid Edge Standard Parts Libraries.
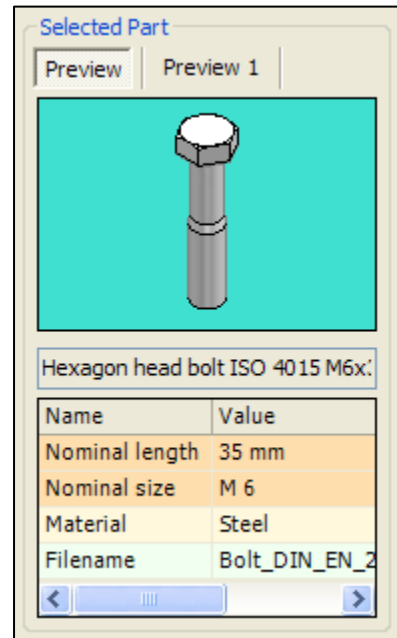
Specification

In the row below the preview the specification is displayed. This is a brief description of the selected part. The specification is sometimes called "Part name".

Displayed Properties

The bottom of the Selected Part section displays the characteristics and file properties of the selected part. Characteristics (with their values) are always shown, displaying of properties is optional. By clicking the right mouse button over this section you can open the "Properties to Display" dialog (see figure). You can then check the properties which you want to be displayed for each selected part.

Note: Only properties with non-empty values will be displayed.
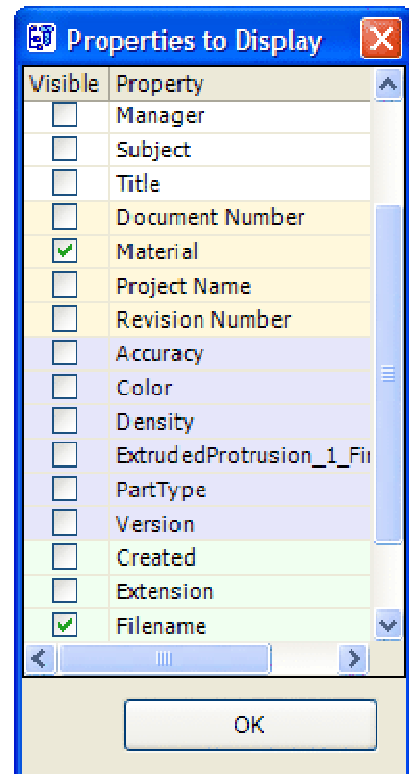
Interaction with Solid Edge (CAD operations)

There are three possible operations you can perform with the selected part. These are "**Place**", "**Replace**" and "**Modify**".

Place

This command allows you to place selected part into the active Solid Edge assembly. When no assembly is open, the place command is disabled.[1]

If you want to place a part:
1. Open the assembly where you want to place the part. If you are creating a new one – save it (e.g. to the Solid Edge Standard Parts working folder)
2. Launch the Standard Parts dialog from the Edge Bar / Part Library.
3. Select the part that you want to place.
4. Click on "Place".
5. Set appropriate relationships in the Solid Edge assembly (if necessary).

Note: If there were the same occurrences with different versions placed previously on the same assembly level, the user is given the chance (after finishing the place operation) to replace all the same occurrences with the current version.

Replace

If you want to replace a part, assembly, weldment or sheetmetal part in the Assembly environment with a standard (or customer standard) part you can use the "Replace" command. If there are multiple identical occurrences, Standard Parts asks you whether you want to replace all the same occurrences. If you have several components selected (see step 2), then you will be asked for each component separately.
If you want to replace a part:
1. Open the assembly where you want to replace a component.
2. Select the component (or components) that you want to replace.
3. Launch the Standard Parts dialog from the Edge Bar / Part Library.
4. Select the part with which you want to replace the original component.
5. Click on "Replace". (If it is disabled, press "F5" to synchronize with Solid Edge first.)

Modify

This command only supports variable parts. The modify command is not the same as replacing/placing the other parts in an assembly; the current part only has a variable characteristic changed. The modify operation is useful for parts such as profiles, pipes or pins – the user can change the value of a variable characteristic (e.g. "Length") over and over again. A typical situation is adjusting the length of a profile.

If you want to modify a part:
1. Open the assembly containing the part you want to modify.
2. Select the variable part you want to modify (it must be currently registered in the database and contain at least one variable characteristic[2]!).
3. Launch the Standard Parts dialog.
4. Press "F5" to synchronize with Solid Edge.
5. Click on "Modify".
6. Set new values for the variable characteristic/s and click on "OK".

---

[1] Some operations are also disabled when the user has no Standard Parts Library license.
[2] Variable characteristics are displayed in red in the Standard Parts dialog.

Standard Parts Options

In Standard Parts Options dialog you can control the behavior of the Standard Parts dialog and the properties of the part which is being viewed or placed.

"**Recall the last used standard on next program start**" sets whether the last selected end category will be selected again the next time Standard Parts is started.

If the "**Use assembly folder as working folder**" option is checked then parts are copied to the location where the assembly is saved, instead of the working folder. This option is only used if the "Copy to working folder" option is set in the Standard Parts dialog, or for variable parts which are always copied to the working folder.

Selection of the "**Preview**" type controls the CAD preview of the selected part in the Standard Parts dialog (see the Selected Parts section for more details).

Checking the "**Place simplified parts**" can increase performance, especially for large assemblies.

With the "**Standard Parts after action**" switch you can control what the Standard Parts dialog should do after finishing an action (place, replace, modify). It may be closed, be shown again or be minimized.
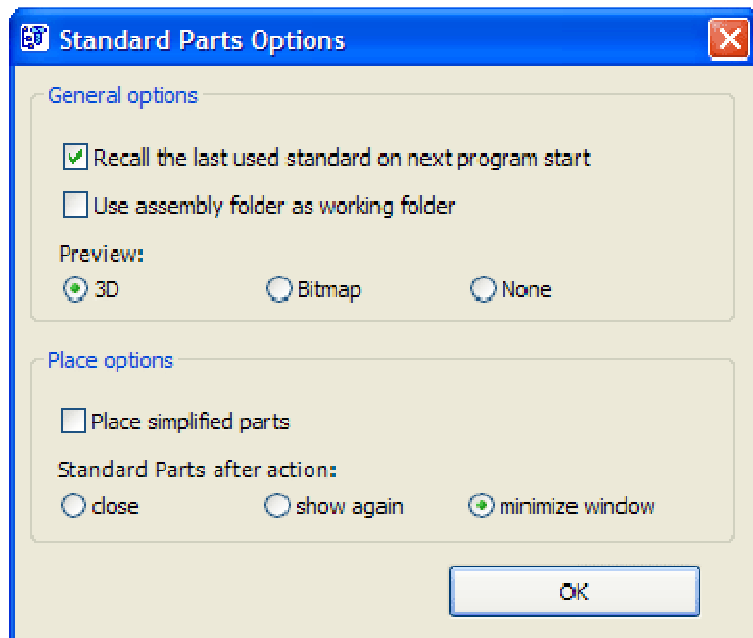
Appendix

Coordinate System Parameters

Name	Description	Value or Special Instructions
IO	Type of CS (Coordinate System)	1 = input cs 2 = output cs
SN	Sequence Number	Pipes have two pairs of CS with SN = 1 and 2. The sequence numbers can be assigned to the pair INPUT/OUTPUT_POINT_1 or INPUT/OUTPUT_POINT_2 arbitrarily.
		In fittings with two openings, SN = 1 is the pair at the opening with bigger diameter, SN = 2 is the opposite pair. If the openings are the same size, SN = 1 is the pair at the opening with male connection and SN = 2 is the pair at the opening with female connection. If the fitting has no male/female connections, the sequence numbers are arbitrary.
		Laterals have SN = 3, the pair at the lateral opening, SN = 1 is the pair at the opening near lateral opening and SN = 2 is the pair opposite to pair with SN = 1.
		T's have SN = 3 for the side opening pair, SN = 1 is opening with bigger diameter and SN = 2 is the pair opposite to pair with SN = 1.
		Y's have SN = 1 for the pair at the opening opposite to both branches which get SN = 2 and SN = 3.
		Crosses have SN = 1 is the pair at the opening with biggest diameter, SN = 2 is the pair opposite to pair with SN = 1, SN = 3 is the pair at the opening with bigger diameter from the remaining openings and SN = 4 is the last pair.
CD	Connecting (nominal) diameter of the opening	
ET	End Treatment	0 = Not specified 1 = Threaded 2 = Socket Weld 3 = Butt Weld 4 = Flanged 5 = Welded Neck 6 = Blind 7 = Slip on 8 = Lap joint 9 = Socket
RT	Rating of the part	

Fastener Systems Parameters

Bolt/Screw

Name	Abbrev	Description
Bolt/Screw SD	BS_SD	Bolt/Screw Shank Diameter
Bolt/Screw L	BS_L	Bolt/Screw Length
Bolt/Screw CA	BS_CA	Bolt/Screw Countersink Angle
Bolt/Screw TX	BS_TX	Bolt/Screw Thread Extent
Bolt/Screw TC	BS_TC	Bolt/Screw Thread Count*
Bolt/Screw HH	BS_HH	Bolt/Screw Head Height
Bolt HWF	B_HWF	Bolt Head Width Across Flats
Bolt HWC	B_HWC	Bolt Head Width Across Corners
Screw HD	S_HD	Screw Head Diameter

* *thread name listed in Holes.txt*
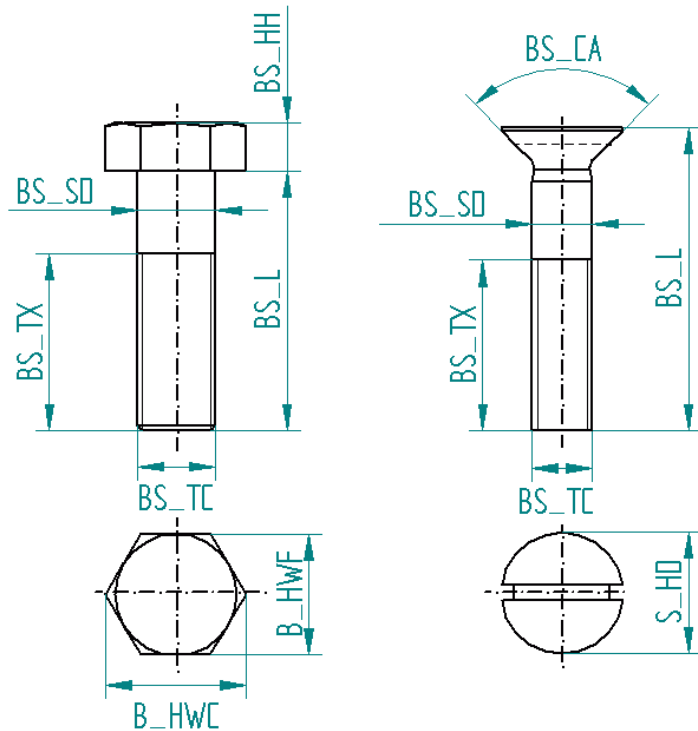
Nut

Name	Abbrev	Description
Nut T	N_T	Nut Thickness
Nut TC	N_TC	Nut Thread Count*
Nut WF	N_WF	Nut Width Across Flats
Nut FD	N_FD	Nut Flange Diameter

* thread name listed in Holes.txt
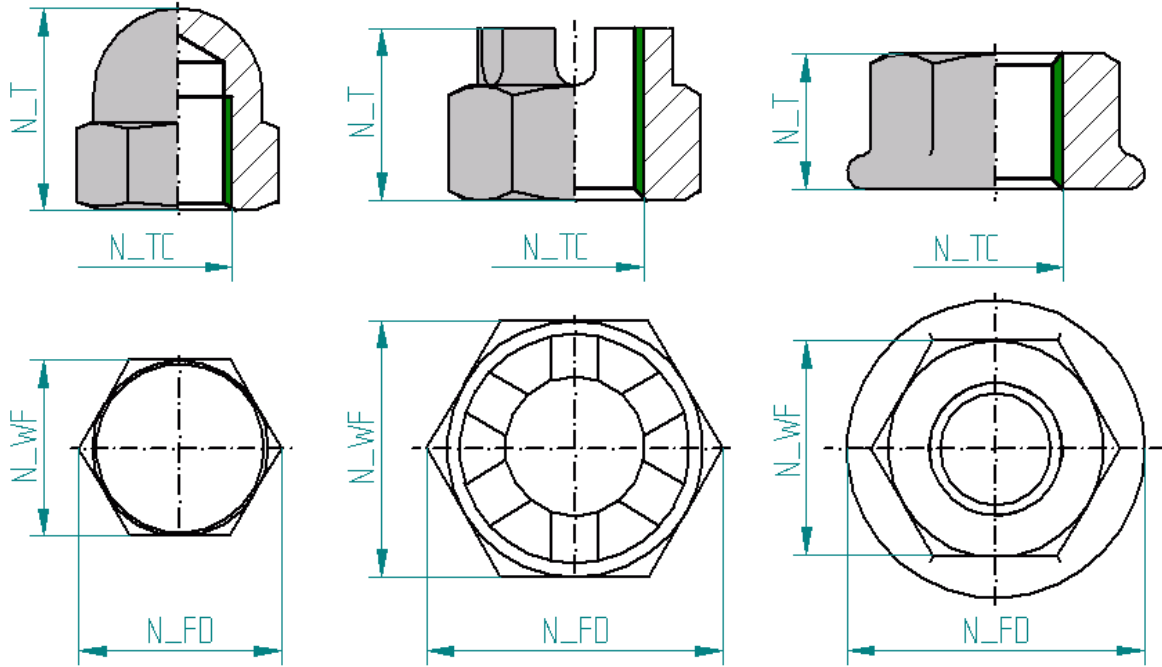
Washer

Name	Abbrev	Description
Washer ID	W_ID	Washer Inner Diameter
Washer T	W_T	Washer Thickness
Washer OD	W_OD	Washer Outer Diameter
Washer CA	W_CA	Washer Countersink Angle
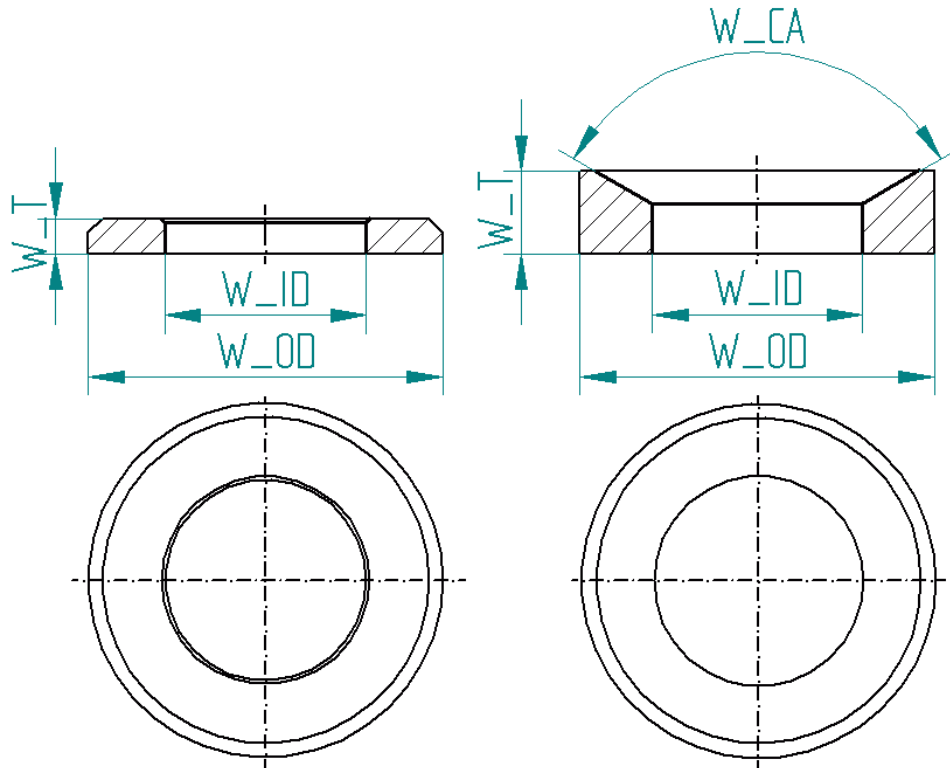
Values of custom property PartType

Piping Components

1	Pipes	
2	Fittings	
	2.1	Elbows
		2.1.1 = 5 5/8° Elbows 2.1.2 = 11 1/4° Elbows 2.1.3 = 22 1/2° Elbows 2.1.4 = 30° Elbows 2.1.5 = 45° Elbows 2.1.6 = 60° Elbows 2.1.7 = 90° Elbows 2.1.8 = Side-outlet 90° Elbows 2.1.9 = Double-branch Elbows
	2.2	Bends
		2.2.1 = 11 1/4° Bends 2.2.2 = 22 1/2° Bends 2.2.3 = 45° Bends 2.2.4 = 90° Bends 2.2.5 = Return Bends 2.2.6 = 5 5/8° Bends
	2.3	Returns (180°)
	2.4	Y's
		2.4.1 = True Y's (90°) 2.4.2 = Straight Y branch
	2.5	Tees
		2.5.1 = Straight Tee 2.5.2 = Side Outlet
	2.6	Crosses
	2.7	Laterals
		2.7.1 = 45° Laterals
	2.8	Reducers
		2.8.1 = Concentric Reducer 2.8.2 = Eccentric Reducer
	2.9	Couplings
	2.10	Plugs
	2.11	Ends
	2.12	Caps
	2.13	Bushings
	2.14	Long Screws
	2.15	Nipples
	2.16	Sockets
	2.17	Collars
3	Flanges	
4	Unions	
5	Accessories	
6	Seals	

Fasteners

Codes for Element (All Fasteners)	
11	Bolts*
12	Screws
13	Washers**
14	Nuts
15	Retaining elements
16	Rivets

PartType for Screws/Bolts is composed of four fields delimited by a period:
<Element>.<Contact face>.<Head Type>.<Screw drive type>
*** PartType for Washers is composed of two fields delimited by a period:*
<Element>.<Contact face>

Screw/Bolt Contact Face	
1	None
2	Plain
3	82° countersunk
4	90° countersunk
5	100° countersunk
6	114° countersunk
Screw/Bolt Head Type	
1	None
2	Binding head
3	Binding head undercut
4	Button head
5	Eye head
6	Fillister head
7	Flat head
8	Flat head undercut
9	Hexagon head
10	Hexagon flanged head
11	Hexagon washer head
12	Cheese head
13	Indented hexagon head
14	Indented hexagon flanged head
15	Indented hexagon washer head
16	Knurled
17	Oval head
18	Oval head undercut
19	Pan head
20	Ring head
21	Round head
22	Square head
23	Square shoulder head
24	T head
25	Triangle head
26	Truss head
27	Socket head
Screw/Bolt Drive Type	
1	None
2	Hexagon socket
3	One-way

4	Phillips
5	Phillips square combo
6	Pozidriv
7	Slotted
8	Slotted Phillips combo
9	Spanner
10	Square socket
11	Torq-Set
12	Torx
13	Triangle socket
14	Tri-Wing
15	Four lobe
16	Six lobe

Washer Contact Face	
1	Plain symmetrical
2	Asymmetrical (spring, locks)
3	120° countersunk
4	90° countersunk

Nuts	
14.1	Cap
14.2	Castle nut
14.3	Hexagon flange nut
14.4	Hexagon nut
14.5	Octagon nut
14.6	Ring nut
14.7	Round knurled nut
14.8	Square nut
14.9	Lifting eye nut
14.10	Lock nut
14.11	Slotted round nut
14.12	Self locking counter nut
14.13	Pipe nut
14.14	Hex slotted hex

Profiles

Solid Shapes	
21.1	Square steel
21.2	Rectangular steel
21.3	Round steel
21.4	Half-round steel
21.5	Half-oval steel
21.6	Hexagon steel

Tubes	
22.1	Square tube
22.2	Rectangular tube
22.3	Round tube (pipe)

Structural Sections	
23.1	Equal leg angle
23.2	Unequal leg angle
23.3	Z-section
23.4	T-section
23.5	Channel section
23.6	I-section
23.7	Lipped channel section
23.8	Hat section
23.9	Lipped Z-section
23.10	Unequal channel section