Teamcenter Integration for SolidWorks®

Administration Guide

Version 9.2.0

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Contents

Introduction

When first installed, the Teamcenter® Integration for SolidWorks® is configured to satisfy most customers' requirements. As users become more experienced with the product, the Teamcenter Administrator may want to change certain settings to align with customer-specific requirements and processes. The **Configuration** section of this document explains how to change the configuration, and why you may want to do so.

The integration provides a set of utility programs to perform administrative-level functions on SolidWorks data. Those functions, and their usage, are described in the **Utility Functions** section of this document.

Finally, there is a small set of known issues that customers may encounter when working with the product. These are covered in the **Troubleshooting** and **Best Practices** sections.

Technical Support

For technical support, please contact the Global Technical Access Center (GTAC). You can reach GTAC via the World Wide Web at http://support.ugs.com/ or by telephone:

United States and Canada (800) 955-0000

In other countries please see http://support.ugs.com/html/by_phone/ for additional telephone numbers, or call (714) 952-5444.

Configuration

The integration's CAD data management behavior is controlled by settings in four different locations:

- swim.properties
  - This properties file may be controlled at the site level, and also made available to users or groups of users for overriding of selected preferences. It is a text file containing a list of keywords and values. Each keyword is documented within the file, and each has a default value. Together they form a set of preferences which are read when SolidWorks is launched with the integration loaded. If the value for a particular keyword is not modified, then the default value remains in effect. Some preferences in the swim.properties file are best controlled at the site level, while others may be user-specific. The Teamcenter Administrator at each site must decide which settings users are allowed to override.
- swim.xml
  - A set of maps to control integration behavior at the site level. This file is not designed for modification by anyone but the Teamcenter Administrator.
- swim.cfg
  - This file exists only on the integration client workstations. It is not designed for site-level administration, although customers may choose to install the integration in a network-accessible location, and allow users to launch using desktop shortcuts. In

that case, it may effectively become a site-level configuration file. It is a text file with various settings to control the interaction between the integration client, the operating system, and the SolidWorks session, as well as the configuration of certain elements of the Graphical User Interface (GUI). Only a small subset of the included properties is intended for modification by customers.

- Teamcenter preferences
    - There are several Teamcenter preferences that can have an effect on integration behavior. These may be managed as user, group, role or site preferences, although by default most are site preferences. The most commonly encountered scenarios are explained in this guide, but the Teamcenter documentation provides the most complete description of these preferences.

By default, all three of the above files are located in the integration's client installation directory (often called **SWIM_DIR** in various documentation). Both swim.properties and swim.xml may be read from other locations as well. Swim.cfg is only read from the client's installation directory.

Swim.properties

The swim.properties file, located in the installation directory, contains the preferences that can be configured for a site or for an individual user. All of these preferences are optional.

Preference files may be named either swim.properties or .swimrc and may be placed in any of these locations:

- In the working directory where the Teamcenter Integration for SolidWorks is started.
- In the user's home directory.
- In the Teamcenter Integration for SolidWorks installation directory.

When the Integration starts, it reads all swim.properties and .swimrc files in these locations. If several files specify the same preference, the preferences from the working directory take precedence over those from the user's home directory, and preferences from the user's home directory take precedence over those from the installation directory. Preferences that are likely to have the same definition for all users should be specified in the installation directory, while a user's personal customizations may be specified in the home directory or working directory preferences file.

It is recommended that the Teamcenter administrator maintain the installed swim.properties file as a read-only master copy, and make all changes in a separate file, which is included in swim.properties using the preferences.file setting, as in this example:

```
preferences.file = .\\swim_site.properties
```

This technique allows you to override each preference's default value, while preserving the original unchanged value for documentation. It also helps to quickly identify those preferences whose values have been changed by the site.

Special syntax is available for "locking" settings so that users cannot override them, and for resolving values through environment variables (especially useful for settings that point to directory or file locations). This syntax is explained at the top of the swim.properties file, and so it is not repeated here.

Any of the published configuration settings may be overridden in the swim.properties file, but some are more frequently changed than others. The following is a summary of the more commonly modified settings.

General preferences

table.columns.std:	With the default value, the integration displays the most commonly needed attributes of the CAD models. It is possible to display additional or alternative attributes, either by selecting from a predefined set (documented in swim.properties) or by configuring display of custom columns. The attributes included in table.columns.std apply to all integration dialogs, unless overridden by similar dialog-specific preferences, such as table.columns.checkin and table.columns.update.
log.file	The integration automatically writes a log file to the session's startup directory, the location of which may be influenced by several factors including the configuration of desktop shortcuts. Assuming that all users have write access to a consistent location on their client workstations, it is recommended to set log.file to a consistent value for all users, such as c:\\temp\\txdlog.txt.
log.suppress	The default setting for this option is 3, which includes the most important information needed for troubleshooting. In rare cases it may be necessary to increase the level of logging detail. This may be done by setting log.suppress to any value greater than 10000.
summary.table.rows.std	This preference controls the attributes which are displayed in the **Summary** panel on the **Open**, **Open Dependencies**, and **Update** dialogs. The swim.properties file documents a predefined list of keywords that may be used to add or change the set of displayed attributes. In addition, any Teamcenter attribute (out of the box or custom) may be added to the set by first configuring the attribute as a mapped column (see the section on **Column Mapping**, later in this document) and then adding the column map's display_name value to summary.table.rows.std.

Open preferences

filters.checkout.choices	Filters are check boxes that appear on the integration's **Open Dependencies** dialog. They give the SolidWorks designer control over how much CAD data is downloaded from Teamcenter to his client. A subset of the available filters is displayed by default. Filter options can be added or subtracted by modifying this preference.
filters.checkout.default	Once the available filters are configured, it is advisable to set them to the appropriate default true or false values, so that the initial collection of models displayed in the **Open Dependencies** dialog represents the most commonly desired set of dependencies. Users can change and apply filter options in the **Open Dependencies** dialog, but each change requires the integration to recalculate the dependency tree, and this can take significant time for very large structures. Filter options that are included in the filters.checkout.default preference are set to true by default; filters not mentioned in this preference default to false.
checkout.exportdirectory	If this preference is not set, then the initial value of **Work Folder** in the **Open Dependencies** dialog is determined by the SolidWorks session's current working directory. If the user changes his SolidWorks directory, the next export from Teamcenter will be routed to a different location on his client. The result is little or no consistency in the location of managed Teamcenter files. For this reason, it is recommended to set checkout.exportdirectory to a fixed location. The syntax documented in the swim.properties file may be used to read an environment variable as part of the path, for example: `checkout.exportdirectory=${USERPROFILE}\\sw.`
iman.bomrevisionruledefault	When this preference remains unset, the default revision rule that is applied to the operation is determined by the user's Default Revision Rule setting, in the Rich Client's **Options | Product Structure** dialog. Customers may want to configure a different revision rule as the default for SolidWorks users, and this can be done by setting iman.bomrevisionruledefault to the name of any revision rule defined in Teamcenter. It is important to configure a default revision rule that is appropriate for most users, most of the time, because this rule will always be used to configure the selected top-level model during Open. The user will have an opportunity to select a different rule after the **Open Dependencies** dialog is displayed, but applying

the alternative rule requires additional processing on the client and server, which takes additional time.

Save preferences

checkin.after	This setting controls the default value of the **Action on Save** control on the save dialog. Some customers may wish to change this default so that users do not have to remember to do it themselves, during each save operation.
checkin.owner.integration.ignore	This preference controls whether models can be saved to existing Teamcenter Item Revisions. Such item revisions may be mastered by another integration. Ownership of an Item Revision is determined by the presence of an integration-specific dataset. For the SolidWorks integration those datasets are SW* (SWAsm, SWPrt, SWDrw …). The default setting of false prevents saving models to existing Item Revisions that do not already contain a SolidWorks integration dataset. A value of true allows saving models to any Item Revision. It is recommended to maintain the default setting of "false", unless there is a good reason to change it.
eai.jtenabled	When the SolidWorks translator is installed, the integration can call the translator to create JT files during save to Teamcenter, and then automatically upload the resulting JT files into the Teamcenter Bill of Materials. The translation is controlled by a check box on the save dialog, labeled **Save JT Files**. Because client-side JT translation can have a significant impact on save performance, Administrators may wish to disable this function by default, and this is done by simply changing the preference value from true to false.
iman.addonlynewtoselected	By default, all new items are put into a Teamcenter folder. If the user selects a folder in the save dialog, then the items are put into that folder. If he does not explicitly select a folder (this is done by selecting the top-level Teamcenter folder in the folder selection dialog, as shown below)

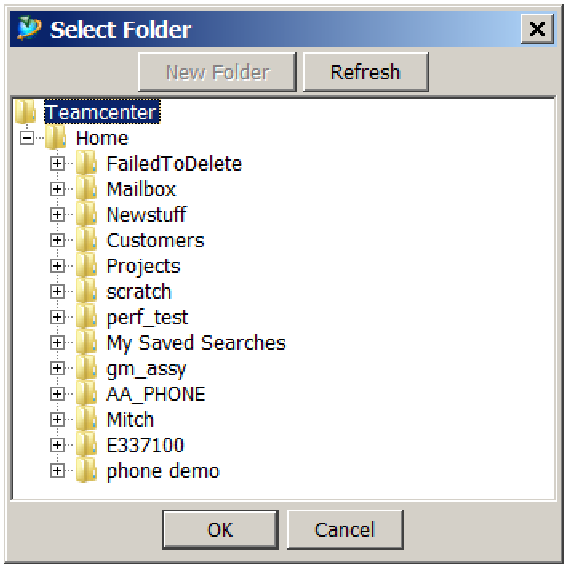


Figure 1 - Selecting the top-level Teamcenter folder

	then the location of the newly-created items is controlled by the WsoInsertNoSelectionsPref preference in Teamcenter. Setting the iman.addonlynewtoselected preference to false tells the integration that not only new items, but also existing items being saved as new versions or new revisions, should be put into the destination folder. This creates links in Teamcenter such that a given item may belong to multiple folders. A typical use case for this is to collect all the components for a large assembly structure into a single folder, so that Teamcenter users can easily find them together.
checkin.ignoremissing	Customers often have CAD assemblies which contain references to other models that are no longer used by the assembly, or that are not available in session. When an assembly is saved to Teamcenter, the integration will find and report these as missing references, and will ask the user whether to continue with the save. Another scenario in which the warning dialog appears is when the assembly has suppressed components that have not already been saved to Teamcenter. Some customers have so much data with this condition that they prefer not to see the warning dialog on each save, and this can be configured by changing the preference from prompt to always.
sw.configurations.default	With the default value of “all”, the integration will attempt to manage all referenced and explicitly added configurations as distinct items and datasets in Teamcenter. Customers who do not have naming conflicts in their CAD

	data can accept the default setting, but many customers who are migrating SolidWorks from other PDM systems, or from disk, where naming conflicts exist and are tolerated, may wish to change this setting to “none”. When set to “none”, the integration will ignore configurations within documents, and will save only the documents to item IDs and datasets in Teamcenter. Configurations are still available within the SolidWorks session, because they are embedded within the .sldasm and .sldprt CAD files managed by Teamcenter. The most significant consequence of choosing to hide configurations in this way is that they are not visible within Teamcenter and therefore cannot appear as assembly components within the Teamcenter Bill of Materials.
sw.configurations.hide	Similar to sw.configurations.default, this setting will prevent Teamcenter from saving configurations as items. However, it is not a global setting but instead is a list of specific configuration names (wildcards may be used) which should be hidden. Customers who use a consistent naming convention for non-product or prototype configurations should use this setting to hide those configurations from Teamcenter.
sw.configurations.keep	The opposite of sw.configurations.hide, this setting will permit configurations to be saved only if their names match a specified pattern (which may be a regular expression). Those not matching the pattern will not be saved as items in Teamcenter.
sw.configurations.masters	SolidWorks itself creates a master configuration, named “Default”, for each document, unless configured otherwise. This setting should contain the names of all master configurations used by the site. For example, a site running in the French locale may have configured SolidWorks to create master configurations as “Défaut”. Such a site should set sw.configurations.masters = Défaut.
sw.configurations.master.document.same	Instead of using a consistent name, or set of names, for master configurations, some customers have named their master configuration the same as the document which contains it. In such cases, the default setting for this preference should be retained. If configurations named the same as the document are not intended to represent the master configuration, then this setting should be changed to “false”, and during save to Teamcenter the users will be

required to rename either the document or the same-named configuration.

Update preferences

update.autoclose	This preference applies only to the **Update Directory** function. The RMB **Update Model** function dialog always closes automatically after the operation is finished. Update.autoclose is set to "true" by default, so that the **Update Directory** dialog automatically closes after the update completes. Some customers may want to change this setting to "false", so that the dialog remains open for further use.

Create preferences

create.enable.manual.itemid	This preference controls whether users are allowed to assign item IDs of their own choosing while creating a new Teamcenter item from within SolidWorks. Changing this preference to "false" will force users to assign item IDs from Teamcenter, typically to satisfy naming rule requirements.

swim.xml

The swim.xml file, located in the installation directory, tells the Integration how to map SolidWorks properties to Teamcenter attributes, which BOMs to create when a model is saved to Teamcenter, whether auxiliary files should be saved with models, which user-defined columns to display, and which Teamcenter dataset and named reference types to use for SolidWorks models. It is an XML file whose format is defined by swim.dtd, also located in the installation directory. You may choose to modify swim.xml, or you can create another map file as long as the map file conforms to the swim.dtd definition. If you choose to create another map file, set the map.file user preference, in swim.properties, to define the path to it, and place a copy of swim.dtd in the same directory as the new map file. The swim.dtd file should not be modified.

Attribute Mapping

When a model is exported from Teamcenter and opened in SolidWorks, attributes assigned to the item, item revision, dataset, or forms may be copied to parameters in the SolidWorks model. When a model is saved from SolidWorks to Teamcenter, its parameter values may be copied back to these objects. The attribute map specifies which attributes should be copied for a particular type of SolidWorks model. Some attributes appear in the properties panel of the **Save** dialog and can be modified. These attributes have create descriptors in Teamcenter. Additional attributes can be added to the properties panel by setting their create descriptors. Learn more by reviewing **Properties display in Save Dialog and Teamcenter New Dialog,** elsewhere in this document.

Within an attribute_map can appear zero or more attribute definitions. Each attribute definition specifies the name of a property in the SolidWorks model (this is the attribute's *CAD name*), the name of the corresponding attribute in Teamcenter (its *PDM name*) and additional information describing its default value, what to do if the attribute does not exist, and so on. The attribute's CAD name and PDM name must be specified, and they must be the first and second tags, respectively, in an attribute definition. The other elements of an attribute definition are optional and may appear in any order.

For example, here is an attribute map for SolidWorks assembly and part models:

```
<attribute_map cad_type="sldasm:sldprt">
  <attribute>
     <cad_name value="PartNumber"/>
     <pdm_name value="item_id"/>
     <missing_attribute_action value="create"/>
     <direction value="pdmtocad"/>
  </attribute>
  <attribute>
     <cad_name value="Weight"/>
     <pdm_name value="ItemRevision Master.wgt"/>
     <default type="Double">1.0</default>
     <direction value="cadtopdm"/>
     <truncate value=”false”>
  </attribute>
</attribute_map>
```

The cad_type defines the type of SolidWorks model to which this map applies. The allowed cad_type values are the six-letter extensions that SolidWorks uses to identify its model files, in this case "sldasm" for assemblies and "sldprt" for part models. If more than one type is specified, use a colon to separate the type names. Additional cad_type values of "slddrw" for drawings and "sldtbx" for toolbox parts may be used.

The example above specifies only two attributes to be mapped between Teamcenter and SolidWorks. One is the part number, which the map says is in a property named PartNumber in SolidWorks assembly and part models, and is the item ID in Teamcenter items. The other is the weight of the assembly or part, which is in a SolidWorks property named Weight and in an attribute named wgt in the item revision master form. The part number is copied only from Teamcenter to SolidWorks (the direction is pdmtocad), while the weight is copied only from SolidWorks to Teamcenter (the direction is cadtopdm). A property for the part number will be created if it does not exist in the SolidWorks model (if the missing_attribute_action is create). If the weight does not exist in the SolidWorks model, a default value of 1.0 will be stored in the form attribute (this is specified by the default tag).

Each attribute tag is a single mapping of SolidWorks property and Teamcenter attribute. A property can be mapped to multiple attributes by adding additional attribute tags for each mapping. Similarly, an attribute can be mapped to multiple properties by adding additional attribute tags. The integration does not validate multiple mappings; each mapping is performed without respect to any rules. It is the site administrator's responsibility to establish a useful set of mappings.

The tags that can be used in an attribute definition are described below. Some of these descriptions refer to *source* and *destination* attributes. When opening a model, the source attributes are those in Teamcenter and the destination attributes are the properties in SolidWorks.

When saving a model, the source attributes are the properties in SolidWorks and the destination attributes are those in Teamcenter.

cad_name

`<cad_name value="`*`name`*`"/>`

The cad_name tag specifies the attribute's name in the SolidWorks model, which should be the name of a custom property in the SolidWorks document. The cad_name tag is required and it must be the first tag in an attribute definition. It has no default.

The following attribute function may be used instead of a property name, but only when direction is "cadtopdm".

is_configuration()	The attribute value is true if the SolidWorks model is a configuration. The value is false if the model is a document.

pdm_name

`<pdm_name value="`*`keyword`*`"|"`*`class[`*`:`*`type]`*`.`*`name`*`"/>`

The pdm_name tag specifies the attribute's name in Teamcenter. This name can be a keyword for certain special attributes, or it can be a more general expression that gives the attribute's name and the class and type of object where the attribute can be found.

When only a single keyword is given for the value of the pdm_name, it may be one of the following:

dataset_desc	The attribute is the dataset description.
item_desc	The attribute is the item description.
item_name	The attribute is the item name.
item_id	The attribute is the item ID.
item_revision_id	The attribute is the item revision level.
item_revision_desc	The attribute is the revision description.
item_type	The attribute is the item type. Since an item's type cannot change after the item has been created, this is ignored when copying attribute values to an existing item.

To specify attributes other than the special ones listed above, use the notation *class.name* for the pdm_name value, where *class* indicates where the attribute will be found, such as on an item, and *name* is the name of the attribute. The classes of Teamcenter objects currently supported are:

Dataset	The attribute is a dataset property.
Item	The attribute is an item property.
Item Master	The attribute is one of the form properties in the item master. Note that there is a single space in this class name.
Item.Form	The attribute is a property in a form attached to an item. The form must be based on a POM class derived from the Form class. If

	more than one form is attached to an item, the first one that matches the *class* and *type* (see below) is used. Forms that are stored as files are not supported.
ItemRevision	The attribute is an item revision property.
ItemRevision Master	The attribute is one of the form properties in the item revision master. Note that there is a single space in this class name.
ItemRevision.Form	The attribute is a property in a form attached to an item revision. The form must be based on a POM class derived from the Form class. If more than one form is attached to an item revision, the first one that matches the *class* and *type* (see below) is used. Forms that are stored as files are not supported.

For example, to specify the project_id attribute in the item revision master form, the name would be

```
<pdm_name value="ItemRevision Master.project_id"/>
```

An optional type can be specified using the notation *class*:*type*.*name*. An attribute object_desc that will be used only for items of type Part would be named

```
<pdm_name value="Item:Part.object_desc"/>
```

Types may also be included in the compound notation used to specify a general form attribute. For example, suppose items of type Part can have a form of type PartData. An attribute data1 in a PartData form would be named

```
<pdm_name value="Item:Part.Form:PartData.data1"/>
```

When using the *class*.*name* or *class*:*type*.*name* notation, *name* must be the attribute's real property name, not its display name. You can find the real name of an attribute with the Teamcenter BMIDE application. Under POM_application_object, look for ItemMaster and ItemVersionMaster to see the attributes of the Item Master and ItemRevision Master classes; under POM_application_object | WorkspaceObject are the definitions of the Item, ItemRevision, and Dataset classes.

When a keyword or class.name specification exists for the attribute you want to map, <u>and</u> when the attribute mapping applies to all Teamcenter Item types, then you may use the abbreviated syntax. If you want the attribute mapping to apply to only one, or a subset, of the available Teamcenter Item types, then you <u>must</u> use the fully qualified compound notation.

The pdm_name tag is required and it must be the second tag in an attribute definition, following the cad_name tag. It has no default.

The Teamcenter attributes must be present in the swim_policy.xml or custom_policy.xml. If not present in either policy file, then the site administrator should add the attribute to the custom_policy.xml (modification of the swim_policy.xml file is not supported). In the custom_policy.xml, the attributes should be added under the proper object type as shown below for an Item.

```
<ObjectType name="Item">
        <Property name="object_desc"/>
        <Property name="Property2"/>
         :
         :
        <Property name="Propertyn"/>
    </ObjectType>
```

Errors or performance degradation can occur if the attributes referenced as pdm_name are not included in the policy file. Specifically, "assertion" failures mayl appear in the txdlog file if an attribute is missing from the policy file.

A Teamcenter attribute may be mapped to SolidWorks only if SolidWorks supports an equivalent property type. The following Teamcenter types should not be mapped to SolidWorks: Boolean Array, Character Array, Integer Array, Double Array, and Date Array. When saving SolidWorks data to Teamcenter, such attributes may be populated in the save dialog, provided the necessary create descriptors have been attached in the BMIDE.

default

`<default type="Date"|"Double"|"Integer"|"String"|"Boolean">`*data*`</default>`

The default tag specifies a default value to assign to the destination attribute when the source property is missing. The value that is assigned to the destination attribute is *data*. Its *type* may be "Date", "Double", "Integer", "String", or "Boolean" (note that double quotes are required around the type as shown, but should not enclose *data*). Teamcenter types are defined in the template, and so *type* can only specify the SolidWorks type in the cadtopdm direction. When the type is "Date", the *data* must be in a valid date format. When the type is "Boolean", *data* must be either true or false.

The default tag is optional. If the source property is missing and no default is defined, the attribute mapping is skipped.

Teamcenter dates and SolidWorks dates are different. Teamcenter dates include time of day. SolidWorks dates do not include time. If the time is important, then the type for a date should be of type "String" to ensure the date will include the time. If the time is not important the default type should be "Date". If the date is to be modified within SolidWorks, it is best to use the "Date" type, as the integration may not be able to convert the user value to a Teamcenter date format.

When Date is the default type, it is acceptable if the value of the date in the default tag is empty. Use an empty default to avoid incorrect dates in Teamcenter.

missing_attribute_action

```
<missing_attribute_action value="create"|"ignore"/>
```

The missing_attribute_action tag specifies what to do when the mapping direction is cadtopdm, and the destination CAD property is missing. Its value may be one of the following strings (these strings must be enclosed in double quotes as shown).

"create"	The destination property is created if it does not exist. This is the default.
"ignore"	The attribute mapping is skipped if the destination property does not exist.

The missing_attribute_action tag applies only to SolidWorks properties. It is ignored when the destination is a Teamcenter attribute. Thus it is only meaningful when the direction value is “pdmtocad” or “both”.

direction

```
<direction value="both"|"cadtopdm"|"pdmtocad"/>
```

The direction tag indicates whether to copy the attribute only from Teamcenter to SolidWorks, only from SolidWorks to Teamcenter, or in both directions. Its value may be one of the following strings (these strings must be enclosed in double quotes as shown).

"both"	The attribute is copied from Teamcenter to SolidWorks when the model is opened, and it is copied from SolidWorks to Teamcenter when the model is saved. This is the default. The integration does not attempt to modify toolbox parts. If the sldtbx cad type is included in a mapping definition where direction = “both”, the attributes will not be mapped from Teamcenter into the toolbox part.
"cadtopdm"	The attribute is copied only from SolidWorks to Teamcenter when the model is saved.
"pdmtocad"	The attribute is copied only from Teamcenter to SolidWorks when the model is opened. The integration does not attempt to modify toolbox parts. If a sldtbx cad_type is included in a mapping definition where direction = “pdmtocad”, the attributes will not be mapped from Teamcenter into the toolbox part.

Note that two attribute definitions can be provided when different behavior is needed for each direction in which the attribute is copied. One definition may specify the cadtopdm direction and the other definition may specify the pdmtocad direction.

constraint

```
<constraint value="any"|"master"|"non_master"| “use_default_cfg” |
                        "reserved"/>
```

The constraint tag restricts attribute copying to models with certain characteristics. Its value may be one of the following strings (these strings must be enclosed in double quotes as shown).

"any"	There is no restriction on copying the attribute. This is the default.
"master"	The attribute should be copied only if the model “owns” its file, which in the case of SolidWorks means the model is a document. Use this constraint value to prevent the attribute from being copied when the model is a SolidWorks configuration.
"non_master"	The attribute should be copied only if the model does not “own” its file. Use this constraint value to copy the attribute when the model is a SolidWorks configuration, and to prevent the attribute from being copied for SolidWorks documents. This constraint does not apply to “default” configurations (where “default” is defined by sw.configurations.masters preference)
"use_default_cfg"	The attribute should always be copied to or from the configuration-specific tab of the model’s properties. Use this constraint when you want to ensure that all Teamcenter attributes are accessed from the configuration-specific tab, whether the model is a document or a configuration.
"reserved"	The attribute should be copied only if the model is reserved by the user. Use this constraint value to avoid modifying models that cannot be checked in. This constraint applies only to the pdmtocad direction. It is ignored when attributes are copied from SolidWorks to Teamcenter.

The constraint tag may be used more than once. For example, to make sure an attribute is copied only to models that are checked out for modification, and not copied to a SolidWorks configuration, use two constraint tags:

```
<constraint value="reserved"/>
<constraint value="master"/>
```

truncate

```
<truncate value="true"|"false"/>
```

The truncate tag specifies whether an attribute should be truncated when mapping from CAD to PDM. Should a SolidWorks property be of greater length than the corresponding Teamcenter attribute, the attribute can be truncated to the length of the Teamcenter attribute.

"true"	Truncate the attribute prior to writing to Teamcenter.
"false"	Default value. The attribute is not truncated before writing to Teamcenter. If property is too long, an error will occur and the save will be stopped.

If an error occurs, the user should correct the attribute length and re-save all data again, not just the model that had the attribute problem, as some of the data may not have been saved.

Loading the Attribute Mappings into Teamcenter

The attribute mappings defined by the attribute_map tag in the swim.xml must also be defined in the Teamcenter database. Follow the procedure **Loading the Attribute Mappings** defined in the **Post-Installation Configuration** section of the **Installation Guide**. When performing those steps, use the swim.xml file that contains the attribute mappings.

User Defined Columns

User defined columns are used to display attributes in the Integration's dialogs which are not available as keyword column tags in the swim.properties file preferences. Such columns may be associated to the Teamcenter Item, Item Revision, Dataset or Form classes, or to properties of the SolidWorks models. They can be added to any table.columns.* preference in swim.properties. For example, the following setting causes two user defined columns to be displayed for all dialogs which are not overridden by a separate preference:

```
table.columns.std = itemid:itemtype:itemrev:itemname:
UserDefinedCol1:UserDefinedCol1:datasettype:cadname
```

where UserDefinedCol1 and UserDefinedCol2 are defined in the Column Map.

Column Map

Within a column_map definition can appear attribute definitions. Each attribute definition specifies the name of a property in the SolidWorks model (this is the attribute's *CAD name*), or the name of an attribute in Teamcenter (its *PDM name*). The attribute's CAD name or PDM name, but not both, must be specified in a column_map definition.

The column_map tells the integration about columns to be displayed in the dialogs. A column_map contains one or more column tags. Each column definition specifies a tag associated with a preference value defined in the swim.properties file. The display_name tag tells the integration what to display in the column header. The cad_name tag specifies the name of a property in the SolidWorks model (this is the attribute's CAD name) and the pdm_name tag specifies the name of the corresponding attribute in Teamcenter (its PDM name).

For example, here is example of a column map:

```
<column_map>
      <column tag="UserDefinedCol1">
            <display_name value="Item Weight"/>
            <cad_name value="Weight"/>
      </column>
      <column tag="UserDefinedCol2">
            <display_name value="PDM Color"/>
            <pdm_name value="ItemRevision Master.color"/>
      </column>
</column_map>
```

The example above specifies two column maps. The first, named "UserDefinedCol1", is a SolidWorks weight parameter named Weight with column heading of "Item Weight". The other column, "UserDefinedCol2", is a Teamcenter attribute named "color" on the item revision master form with column heading of "PDM Color". The tags that can be used in a column definition are described below.

column tag

```
                        <column tag="name">
```

The column tag specifies the name to be used in the preference value in the swim.properties file. For example, if the name is "UserDefinedCol1" then the name in swim.properties must match and must be unique for the column map. Below is an example:

```
table.columns.std = itemid:itemtype:itemrev:itemname:
UserDefinedCol1:datasettype:cadname
```

display_name

```
                        <display_name value="Column Description"/>
```

The display_name tag specifies the column description.

cad_name

```
                        <cad_name value="name"/>
```

The cad_name tag specifies the property's name in the SolidWorks model, or it may be the name of an attribute function. If the cad_name tag is defined then the pdm_name tag cannot also be defined. If it is the name of a property, it must obey the usual rules for SolidWorks property names.

pdm_name

<pdm_name value="*keyword*"|"*class[*:*type].name*"/>

The pdm_name tag specifies the attribute's name in Teamcenter. If the pdm_name tag is defined then the cad_name tag cannot also be defined. This name can be a keyword for certain special attributes, or it can be a more general expression that gives the attribute's name and the class and type of object where the attribute can be found. Both the keyword and the general expression syntax are the same as documented for the pdm_name tag in the attribute mapping section of this document.

BOM Map

The bom_map tag defines the view type for the BOMs that the Integration must update each time it saves a SolidWorks model in Teamcenter. It has the following format.

```
<bom_map>
  <bom_line parent_type="parent" child_type="child" view_types="type"/>
      .
      .
```

```
        .
</bom_map>
```

BOM View Type

Each bom_line tag specifies the type of BOM view that must be created or updated whenever a SolidWorks model is saved. Both *parent* and *child* must be SolidWorks model types defined by entity tags in the type_map. The *parent* is the type of model for which the BOMs are being created. The *child* is the type of model that will appear on the BOM line. The *parent* is a model that depends on the *child*. The *type* is the type of BOM that should be given a BOM line for the child when the parent model is saved to Teamcenter.

For example, the following bom_map defines three BOM lines, one for assembly-to-assembly relationships, one for assembly-to-part relationships, and one for assembly-to-toolbox part relationships. The first bom_line tag states that, when an assembly model (parent_type) depends on another assembly model (child_type), the item revision containing the parent model should be given a BOM of the view type. The second bom_line tag makes a similar statement about the BOM to add to an assembly item revision that depends on a part model.

```
<bom_map>
  <bom_line parent_type="sldasm" child_type="sldasm" view_types="view">
    <bom_prop cadName="componentReference" pdmName="bl_sequence_no"
    direction="pdmtocad"/>
  </bom_line>
  <bom_line parent_type="sldasm" child_type="sldprt" view_types="view">
    <bom_prop cadName="componentReference" pdmName="bl_sequence_no"
    direction="pdmtocad"/>
  </bom_line>
  <bom_line parent_type="sldasm" child_type="sldtbx" view_types="view">
    <bom_prop cadName="componentReference" pdmName="bl_sequence_no"
    direction="pdmtocad"/>
  </bom_line>
</bom_map>
```

BOM Properties

A bom_line tag may optionally enclose several bom_prop tags, each of which specifies a SolidWorks component property to be synchronized with an occurrence attribute on the BOM line. The bom_prop tag contains a direction for the mapping. The component reference property is the only supported SolidWorks property. This property can be shown in drawing annotations, such as tables or balloon notes. Any Teamcenter BOM line property can be supported. The bom_prop tag can be repeated to establish multiple mappings. One-to-many mapping can be used in the cadtopdm direction, but not in the pdmtocad direction.

Every bom_prop tag has the following structure

```
<bom_prop cadName="ComponentReference" pdmName="bl_sequence_no"
direction="pdmtocad"/>
```

cadName	The name of the SolidWorks property. “ComponentReference” is the only supported value and it is only supported for SolidWorks 2010 and beyond.

pdmName	The name (not display name) of the Teamcenter bomline attribute. For example “bl_sequence_no” is the name of the attribute displayed as “Find No.”. All bomline attributes are supported.
direction	Allowed values are “both”, “pdmtocad”, and “cadtopdm”. If “both”, then the property will be set in Teamcenter during save and in SolidWorks during open. If “pdmtocad”, then the property will be set in SolidWorks during open. If “cadtopdm”, then the property will be set in Teamcenter during save.

When bom_prop tags are set in the swim.xml file, cadtopdm mappings occur on every save. Pdmtocad mappings only occur when the assembly is opened from Teamcenter using the **Synchronize BOM Attributes** checkbox on the **Open** dialog.

If using BOM packing in the Structure Editor, the packing will be affected by pdmtocad mapping of bl_sequence_no. In some cases portions of a bom could become packed or unpacked unexpectedly.

When mapping in the cadtopdm direction, the BOM attributes must be writeable. If not writeable, the save operation will fail resulting in corrupted data in Teamcenter. At a minimum, the BVR will be missing. The integration assumes access to all attributes, as verification of the attribute on every BVR would be a significant performance impact.

String attributes being mapped in the cadtopdm direction will be truncated if the value in cad is greater than the maximum length of the attribute.

Drawing annotations which reference the component reference property will not automatically update if ***Load referenced documents*** *is set to “None”.* ***Load referenced documents*** *is a SolidWorks system option, which can be found on the* ***Tools | Options | System Options | External References*** dialog**.**

If any BOMLine properties are configured as source or destination in the bom_prop tag of swim.xml, those properties should be present in the swim_policy.xml or custom_policy.xml. If not present in either policy file, then the site administrator should add the property to the custom_policy.xml (sites should not modify the swim_policy.xml). In the custom_policy.xml, the properties should be added under ObjectType name=”BOMLine” as shown below.

```
<ObjectType name="BOMLine">
        <Property name="Property1"/>
        <Property name="Property2"/>
         :
         :
        <Property name="Property"/>

</ObjectType>
```

Errors or performance degradation can occur if the BOMLine properties referenced as pdmName are not included in the policy file. Specifically, "assertion failure" errors may appear in the txdlog file if an attribute is missing from the policy file

Auxiliary Files

When a model is saved to Teamcenter, additional local files may be saved with it. These files may also be fetched from Teamcenter when the model is fetched. These additional files are called *auxiliary files*. Examples of auxiliary files are JT files for the Teamcenter Rich Client Viewer, GIF image files, and files of NC machining instructions. The auxiliary_file_map tag tells the Integration about auxiliary files to save or fetch with SolidWorks models.

An auxiliary_file_map tag contains one or more auxiliary_file tags. It may also contain jt_file tags, which are described in a later section on JT Files. The general purpose auxiliary_file tag specifies a pattern for the name of a file, the dataset type and named reference type where the file will be stored in Teamcenter, and optional commands for the operating system to execute. It may also specify a checkbox to display in the **Save** dialog for users to choose whether auxiliary files should be saved.

General Auxiliary Files

The auxiliary_file tag supports most kinds of auxiliary files. Each is associated with a particular type of SolidWorks model. When a model of that type is saved to Teamcenter, the Integration will also save any local files that match a specified file name pattern. When the model is fetched from Teamcenter, these files can also be fetched.

For example, here is an auxiliary file map containing a single auxiliary_file tag for JT files.[1] The cad_type="sldprt" attribute means this auxiliary file definition is associated only with part models:

```
<auxiliary_file_map>
  <auxiliary_file cad_type="sldprt">
     <pdm_location named_ref="JTPART" pdm_type="DirectModel"
     relation_type="IMAN_Rendering"/>
     <file_name pattern="{cad_name}.jt"/>
     <os_command cmd="doit {cad_file}"/>
     <cadtopdm_control label="Save JT Files"
     user_preference_name="eai.jtenabled"
     user_preference_default="true"/>
  </auxiliary_file>
</auxiliary_file_map>
```

The pdm_location tag tells the Integration where to put the auxiliary file in Teamcenter. In the example above, the file will be stored in a dataset of type "DirectModel" and the file's named reference will be "JTPART". The dataset is assumed to have the same name as the SolidWorks model's dataset. If the dataset does not exist, it will be created and attached to the model's item revision with an "IMAN_Rendering" relation.

[1] The jt_file tag, described later, is an easier way to handle JT files.

The file_name tag tells the Integration how to find the auxiliary file in the local file system. In the example above, the name of the auxiliary file is composed of the model's lower-case name with a .jt extension. It is not an error if an auxiliary file cannot be found.

An optional operating system command can be specified with the os_command tag. In the example above, the Integration will tell the operating system to execute a command named doit, which takes the path to the model's file as its sole argument. Commands are executed after SolidWorks writes models to the local file system, but before the models or auxiliary files are saved to Teamcenter.

> *If an error occurs while attempting to execute the command, the auxiliary file will not be checked-in to Teamcenter.*

The cadtopdm_control tag defines an optional checkbox to display in the **Save** dialog's **Auxiliary Files** box. In this example, the checkbox label will be **Save JT Files**. This tag also defines a user preference named eai.jtenabled that sets the default state of checkbox. In this case, the checkbox is on by default.

The tags that can be used in a general auxiliary_file definition are described in more detail below.

auxiliary_file

```
<auxiliary_file cad_type="type" [direction ="both"|"cadtopdm"|"pdmtocad"]>
     optional elements
</auxiliary_file>
```

The auxiliary_file tag is the general purpose definition of an auxiliary file. The cad_type defines the type of SolidWorks model to which the auxiliary_file tag applies. The allowed values for *type* are the six-letter extensions that SolidWorks uses to identify its model files. For example, cad_type="sldprt" means the auxiliary file is associated only with part models.

More than one SolidWorks model type may be given for cad_type. Use a colon to separate each six-letter value from the next. For example, to indicate that an auxiliary_file tag applies to parts, assemblies, and drawings, the attribute should be cad_type="sldprt:sldasm:slddrw".

The direction attribute indicates whether to save the file to Teamcenter when the SolidWorks model is saved, fetch it from Teamcenter when the model is fetched, or both. Its value may be one of the following strings (these strings must be enclosed in double quotes as shown).

"both"	The auxiliary file is saved to Teamcenter when the SolidWorks model is saved, and it is fetched from Teamcenter when the model is fetched.
"cadtopdm"	The auxiliary file is saved to Teamcenter when the SolidWorks model is saved, but is not fetched with the model. This is the default.
"pdmtocad"	The auxiliary file is fetched from Teamcenter with the SolidWorks model, but it is not saved when the model is saved.

The *optional elements* that can be included in an auxiliary_file definition are described below.

cadtopdm_control

```
<cadtopdm_control label="label" [user_preference_name="name"]
        [user_preference_default="true"|"false"]/>
```

The cadtopdm_control tag creates a checkbox in the **Save** dialog that allows the user to choose whether to save the auxiliary files specified by the auxiliary_file definition. The text labeling the checkbox is *label*. If the same label is used for several auxiliary_file definitions, a single checkbox can control all of them.

The user_preference_name attribute creates a user preference *name* that can be used in the swim.properties file to set the initial state of the checkbox. When the user preference is not set, the checkbox is initially checked or unchecked according to whether user_preference_default is true or false.

If an auxiliary_file definition has no cadtopdm_control tag, the auxiliary files are always saved.

> *The direction attribute on the `auxiliary_file` definition must be `"cadtopdm"` or `"both"` for the `cadtopdm_control` tag to have any effect.*

create_text_file

```
<create_text_file file_name="path" [separate_files="true"|"false"]
      [header="path"] [body="path"] [footer="path"]
      [phase="in_cad"|"in_directory"|"in_pdm"]/>
```

The create_text_file tag specifies one or more text files to create. By default, the file(s) will be created after SolidWorks has saved the models to the local file system, but before the models or auxiliary files are checked-in to Teamcenter.

> *If an error occurs while attempting to write a text file, the file will not be checked-in to Teamcenter.*

The file to be created is specified by the file_name attribute, where *path* is the path to the file. The path may include keywords described in the section on Substitution Keywords. If separate_files is true, a separate text file will be created for each model saved to Teamcenter. If separate_files is false, a single file will be created, which may contain information about all models saved to Teamcenter.

Each text file is composed from three templates: the header, the body, and the footer. The header and footer appear once in the output file, at its beginning and end, respectively. The body appears between these two sections. If a single file is being created for all models, the body is repeated for each model being saved to Teamcenter. The body appears only once in the output file if a separate output file is created for each model.

Each template is optional, although at least one must be specified. The header attribute specifies the path to a text file that will be used as the header template, the body attribute specifies the path to a text file that will be used as the body template, and the footer attribute specifies the path to a text file that will be used as the footer template. The paths to these files may include keywords from the Substitution Keywords section, and the text within these files may also use these keywords. If separate_files is true, keywords that imply a particular model, such as {cad_name} and {cad_type}, can be used in any template and in any file path. If separate_files is false, keywords that imply a particular model can be used only in the body template and in the path to the body template file. The header and footer, and the file_name path, can use only keywords that do not refer to a specific model when separate_files is false.

The use of these keywords in the paths to the template files makes it possible to use separate body templates for parts, assemblies, and drawings, or to use separate templates for SolidWorks configurations and documents. For example, suppose the C:\text_templates directory contains a header template file named header.txt with the following contents:

> "These models are being checked-in to Teamcenter…"

and two body template files named sldprt.txt and sldasm.txt, in which sldprt.txt contains:

> "A part named {cad_name} is being checked-in to Teamcenter."

and sldasm.txt contains:

> "An assembly named {cad_name} is being checked-in to Teamcenter."

A create_text_file tag that uses these templates to create a single text file summarizing the parts and assemblies selected for check-in to Teamcenter might look like this:

```
<create_text_file file_name="summary.txt"
  header="C:\text_templates\header.txt"
  body="C:\text_templates\{cad_type}.txt"/>
```

When an assembly named Coupling and two parts, Pin and Shaft, are saved to Teamcenter, the contents of summary.txt would be:

```
These models are being checked-in to Teamcenter…
A part named Pin is being checked-in to Teamcenter.
A part named Shaft is being checked-in to Teamcenter.
An assembly named Coupling is being checked-in to Teamcenter.
```

The {cad_type} pattern in the path to the body template would have been replaced with sldprt or sldasm, depending on the model type, and the {cad_name} pattern in the body templates would have been replaced by the model name.

You can also specify a phase in which the text file(s) will be created. Refer to the os_command tag for more information concerning the phase attribute. By default, text files are created during the in_directory phase.

ets_request

```
<ets_request translator="service" [provider="name"] [priority="0"|"1"|"2"|"3"]
        [request_per_model="true"|"false"]>
     <translator_option name="translator_option">value</translator_option>
</ets_request>
```

The ets_request tag submits translation requests to Teamcenter's Dispatcher after all models have been checked in to Teamcenter. The name of the translator that will handle the request, also known as the *service*, is given by the required translator attribute. The default provider is "SIEMENS", but other providers can be specified with the optional provider attribute. The translation request can be given a priority of 1, 2 or 3, corresponding to high, medium or low priority, using the priority attribute. If the priority is set to 0, no translation request is submitted.

By default, the ets_request tag submits a separate translation request for each SolidWorks model of the specified type that is saved to Teamcenter. However, if the translator can handle more than one model at a time, you can set the request_per_model attribute to "false", which will cause one translation request to be submitted containing all models that have been saved to Teamcenter. For example, if four parts are saved to Teamcenter and an auxiliary_file definition with a cad_type of "sldprt" specifies an ets_request, four separate translation requests will be submitted by default or if request_per_model is explicitly "true". If request_per_model is "false", a single translation request will be submitted containing all four parts[2].

If an auxiliary file definition uses ets_request, the Dispatcher will create the auxiliary file and import it into Teamcenter, so there is no need to specify a file or a PDM location for it. For example, an auxiliary file definition that submits a request for translating drawings to DXF files might look like this:

```
<auxiliary_file cad_type="slddrw">
  <cadtopdm_control label="Save DXF Files (ETS)"/>
  <ets_request translator="swtodxf"/>
</auxiliary_file>
```

In this example, the cadtopdm_control tag adds a checkbox to the **Save** dialog, giving the user control over whether the DXF file will be created.

If the translator accepts additional options, these can be specified with one or more translator_option sub-tags. Refer to the documentation for your translator to determine whether additional translator options are supported, and to find out the names and allowed values of these options. The name attribute specifies the name of a translator option, and the string to be assigned to that option is given by *value*. When request_per_model is "true", *value* may include any of the keywords that are described later in the Substitution Keywords section. When request_per_model is "false", only substitution keywords that do not refer to a specific model should be used. Substitution keywords in *value* are replaced when the request is submitted to the Dispatcher, not when the translation occurs.

The ets_request tag always submits Dispatcher requests during the in_pdm phase. An auxiliary_file tag should not have more than one action per phase, so do not use other in_pdm actions within the same auxiliary_file tag as your ets_request tag. See the section on the os_command tag for more information on phases.

file_name

```
<file_name pattern="pattern" [version="all"|"latest"]/>
```

The file_name tag identifies the auxiliary file by defining a pattern that matches the file's name. The pattern may give an absolute path to the file, or the path may be relative to the directory where the SolidWorks model file is saved. The * wildcard character is allowed in the simple file name part of the pattern, but wildcards are not allowed in directory names. For example, "myfile.*" and "images/*.gif" are acceptable patterns, but "image*/myfile.*" is not allowed because a wildcard is used in a directory name.

[2] The OOTB translators SWToJT and SWToDXF do not accept more than a single model per translation request; therefore the request_per_model attribute only applies to alternative or custom translators.

The pattern may also include certain keywords that the Integration will replace with data from the model. Keywords can be used anywhere in a pattern, both in the file name part and the directory parts of a path. The recognized keywords are described later in the section on Substitution Keywords.

The optional version attribute indicates whether all auxiliary files matching the pattern should be saved to Teamcenter, or only the file with the latest modification time. The default is "all", which saves all auxiliary files that match the pattern.

The file_name tag is optional. If it is not included in an auxiliary_file tag, the Integration will not attempt to find and save an auxiliary file when it saves the corresponding SolidWorks model, nor will it attempt to execute any operating system commands specified by the os_command tag.

os_command

```
<os_command [cmd="command"] [pre_cmd="command"] [post_cmd="command"]
     [phase="in_cad"|"in_directory"|"in_pdm"]
     [ignore_status="true"|"false"]/>
```

The os_command tag specifies one, two, or three optional commands for the operating system to execute. By default, these commands are executed after SolidWorks has saved the models to the local file system, but before the models or auxiliary files are checked-in to Teamcenter.

If an error occurs while attempting to execute one of these commands, the auxiliary files affected by that particular auxiliary_file tag will not be checked-in to Teamcenter.

The string in cmd="*command*" is an operating system command that is executed for each SolidWorks model saved to Teamcenter. The command string may include the same keywords allowed in the file_name pattern, which are listed in the section on Substitution Keywords.

The strings in pre_cmd="*command*" and post_cmd="*command*" are also operating system commands. The pre_cmd string is executed once, before any cmd strings from any auxiliary_file tag are executed ("once" means just once during a Save, Save As, Save All Checkouts, or Save All command, regardless of the number of models being saved). The post_cmd string is also executed only once, after all cmd strings from all auxiliary_file tags have been executed. The pre_cmd and post_cmd strings may also include some of the keywords described in the Substitution Keywords section, but not every keyword is meaningful in these command strings. Since pre_cmd and post_cmd are not associated with one specific model, keywords that refer to a model's name, directory, item ID, or other model-specific data, are not applicable in these strings.

If you provide a pre_cmd, cmd, or post_cmd, the Integration requires the command to execute successfully before it saves the auxiliary file to Teamcenter. If pre_cmd, cmd, or post_cmd fails, the auxiliary file matched by the file_name pattern will not be saved. Also, if pre_cmd fails, neither cmd nor post_cmd will be executed.

If an operating system command exits with a non-zero status, the Integration usually assumes the command has failed. However, if ignore_status="true" is specified, the Integration ignores exit status and assumes commands always succeed when executed.

You can also specify the *phase* in which these commands will execute. When the Integration saves models from SolidWorks to Teamcenter, the models pass through three phases corresponding to the values that can be assigned to the phase attribute:

in_cad	The models are in the SolidWorks session and have not yet been saved to the local file system.
in_directory	The models have been saved from SolidWorks to the local file system, but neither the models nor any auxiliary files have been checked in to Teamcenter.
in_pdm	All models and auxiliary files have been checked in to Teamcenter.

By default, the commands specified by an os_command tag are executed during the in_directory phase, which is usually the phase in which you would want to run any translators that read a SolidWorks file to create an auxiliary file. If you have any batch processes that must run after models have been checked in to Teamcenter, you might want to use an os_command tag to launch them during the in_pdm phase. As a point of interest, the ets_request tag submits translation requests during the in_pdm phase.

You can think of the os_command, create_text_file, and ets_request tags as actions within an auxiliary_file definition. Each auxiliary_file definition may have one such action per phase, which means that as many as three os_command tags can appear in an auxiliary_file definition if each one specifies a different phase. Alternatively, an auxiliary_file definition could have one create_text_file tag for the in_directory phase, an os_command tag for the in_pdm phase, and so on.

Every attribute in an os_command tag is optional.

preview_command

```
<preview_command [cmd="command"] [pre_cmd="command"] [post_cmd="command"]
      [phase="in_cad"|"in_directory"|"in_pdm"]
      [ignore_status="true"|"false"]/>
```

The preview_command tag specifies one, two, or three optional commands for SolidWorks to execute. By default, these commands are executed after SolidWorks has saved the models to the local file system, but before the models or auxiliary files are checked into Teamcenter. The preview_command tag is similar to the os_command tag except the keyword "built-in *file_name_tag*" is added to invoke added functionality by the Integration. You may notice that by default, the swim.xml file already has these tags:

```
<auxiliary_file_map>
     <auxiliary_file cad_type="sldprt" direction="cadtopdm">
         <!-- Uncomment the below if you want to give the user choice of
         saving the preview files -->
         <!-- cadtopdm_control label="Save PNG Files"/-->
         <pdm_location named_ref="PNG" pdm_type="SWPrt"/>
         <file_name pattern="{temp_dir}\{cad_name}p.PNG"/>
         <preview_command cmd="built_in {temp_dir}\{cad_name}p.PNG"
        ignore_status="true" phase="in_directory"/>
     </auxiliary_file>
     <auxiliary_file cad_type="sldasm" direction="cadtopdm">
         <pdm_location named_ref="PNG" pdm_type="SWAsm"/>
         <file_name pattern="{temp_dir}\{cad_name}a.PNG"/>
```

```
            <preview_command cmd="built_in {temp_dir}\{cad_name}a.PNG"
            ignore_status="true" phase="in_directory"/>
        </auxiliary_file>
        <auxiliary_file cad_type="slddrw" direction="cadtopdm">
            <pdm_location named_ref="PNG" pdm_type="SWDrw"/>
            <file_name pattern="{temp_dir}\{cad_name}d.PNG"/>
            <preview_command cmd="built_in {temp_dir}\{cad_name}d.PNG"
            ignore_status="true" phase="in_directory"/>
        </auxiliary_file>
   </auxiliary_file_map>
```

generate_command

```
   <generate_command [cmd="command"] [pre_cmd="command"] [post_cmd="command"]
         [phase="in_cad"|"in_directory"|"in_pdm"]
         cad_type="SolidWorks Save as type"
         [ignore_status="true"|"false"]/>
```

The generate_command tag specifies one, two, or three optional commands for SolidWorks to execute. By default, these commands are executed after SolidWorks has saved the models to the local file system, but before the models or auxiliary files are checked into to Teamcenter. The generate_command tag is similar to the os_command tag except for the keyword “built_in *file_name_tag*”, which is added to the cmd tag to invoke added functionality by the Integration, and *“cad_type”*. Below is an example of using these tags to generate PDF files drawings:

```
<auxiliary_file_map>
    <auxiliary_file cad_type="slddrw" direction="cadtopdm">
        <pdm_location named_ref="PDF_Reference"
        relation_type="IMAN_rendering" pdm_type="PDF"/>
        <file_name pattern="{temp_dir}\{cad_name}d.PDF"/>
        <generate_command cmd="built_in {temp_dir}\{cad_name}d.PDF"
        cad_type="PDF" ignore_status="true" phase="in_directory"/>
    </auxiliary_file>
</auxiliary_file_map>
```

PDF files cannot be generated for individual part or assembly configurations. The PDF is generated for the active configuration. PDF generation is intended for drawings.

The generate_command tag may be applied to any file type which can be exported from a SolidWorks session, and which also has a defined named reference type in Teamcenter. To determine the list of eligible file types for a given CAD type, review the list of file extensions displayed by the **File | Save** As function in SolidWorks.

pdm_location

```
   <pdm_location named_ref="ref" [pdm_type="dataset"] [relation_type="relation"]/>
```

The pdm_location tag specifies where an auxiliary file should be saved in Teamcenter. Auxiliary files associated with a particular model are always saved in the same item revision as the model, but they can be stored in the model’s dataset or in a dataset of their own.

The named_ref attribute must always be specified, where *ref* is the type of named reference to be used for the auxiliary file. The pdm_type attribute is the type of dataset in which to store the auxiliary file. The dataset is assumed to have the same name as the dataset holding the SolidWorks model. If the pdm_type is not specified, the auxiliary file is stored in the model's dataset.

If a dataset must be created for the auxiliary file, it will be attached to the item revision using the type of relation specified by relation_type. If not specified, the default relation type is "IMAN_Rendering".

The named_ref attribute is required in a pdm_location definition. The pdm_type and relation_type attributes are optional.

Substitution Keywords

Patterns of the form {*keyword*} can be used in the file_name pattern, and in the os_command and create_text_file tags. The Integration replaces these patterns with appropriate values. For example, if a model is named widget, the pattern "{cad_name}.gif" will become "widget.gif" when widget is saved to Teamcenter.

The modifier upper or lower may be appended to any keyword. Use a comma to separate the modifier from the keyword, without spaces. The {*keyword*,upper} pattern will be replaced with a result that is all upper case. The {*keyword*,lower} pattern converts the result to lower case. For example, the pattern "{cad_name,upper}.gif" would produce "WIDGET.gif" when the model's name is widget.

The escape modifier can also be appended to any keyword. This modifier causes every backslash in the result to be replaced with two backslashes. For example, if the {cad_directory} pattern produces the result "C:\work\latest", the pattern {cad_directory,escape} will produce "C:\\work\\latest".

The recognized keywords are:

{cad_dataset_uid}	The UID for the model's Teamcenter dataset, or an empty string if the dataset does not exist at the time this pattern is replaced. When this pattern is used in an os_command string, for example, the dataset might not exist during the in_cad and in_directory phases.
{cad_directory}	The absolute path to the directory containing the model file[3].
{cad_file}	The absolute path to the model file.
{cad_name}	The model's name.
{cad_type}	The SolidWorks six-letter model type, such as sldasm or sldprt.
{document_cad_name}	The name of the model's SolidWorks document. If the model is a document, this is the same as {cad_name}.
{group}	The user's current Teamcenter group.

[3] This keyword can be used only where a single model is implied, such as in the file_name pattern or in the os_command cmd string. It is not recognized in the os_command pre_cmd or post_cmd strings because these commands are not executed for each model.

{item_id}	The Teamcenter item ID for the model.
{item_name}	The Teamcenter item name for the model.
{item_revision_id}	The Teamcenter item revision ID for the model.
{item_revision_uid}	The UID for the model's Teamcenter item revision, or an empty string if the item revision does not exist at the time this pattern is replaced. When this pattern is used in an os_command string, for example, the item revision might not exist during the in_cad and in_directory phases.
{item_uid}	The UID for the model's Teamcenter item, or an empty string if the item does not exist at the time this pattern is replaced. When this pattern is used in an os_command string, for example, the item might not exist during the in_cad and in_directory phases.
{named_ref}	The named reference for the auxiliary file.
{pdm_type}	The Teamcenter dataset type for the auxiliary file. For example, if the auxiliary file is saved in the same dataset as a SolidWorks assembly model, the dataset type is SWAsm. For JT files, the dataset type is DirectModel.
{swim_dir}	The absolute path to the directory where the Teamcenter Integration for SolidWorks is installed.
{user_id}	The user's Teamcenter user ID.

JT Files

The jt_file tag is simpler than the auxiliary_file tag for configuring the Integration to save JT files.[4] The jt_file element shown below in the auxiliary_file_map makes this happen:

```
<auxiliary_file_map>
    <jt_file action="translate" cad_type="sldprt:sldtbx"
    eai_dir="D:\apps\translators\swjt"/>
</auxiliary_file_map>
```

This tells the Integration to save a JT file with each SolidWorks part or Toolbox model saved to Teamcenter (cad_type="sldprt:sldtbx"). For non-Toolbox parts, the JT file must have the same name as the part and must be in the same directory as the model's file. To apply this to both parts and assemblies, use `"sldprt:sldasm"` for the cad_type value.

The file_name tag, described previously for general auxiliary files, may also be used within a jt_file element if the JT file's name will not have the same name as the part or will be located in a different directory.

> *The presence of a jt_file tag in the auxiliary_file_map makes the* ***Save JT Files*** *checkbox visible in the Save dialog. When this checkbox is on, the jt_file tag is enabled and the Integration saves JT files as directed by the action type. When this checkbox is off, the jt_file tag is disabled as if action is "none" and save is "false".*

[4] Siemens's SolidWorksToJT translator must be installed and licensed in order to generate JT files.

The default state of this checkbox can be set with the eai.jtenabled user preference. This option and others are described in detail in the swim.properties file, found in the directory where the Teamcenter Integration for SolidWorks is installed.

Template Map

The template_map tag specifies template models, also known as seed models or start models, that can be copied when a new model must be created. A template will usually be needed when the Open command attempts to fetch a model from Teamcenter and the model's dataset is empty. When that happens, a template model of the required type will be copied and placed in the destination directory, and the copy is given the name that the missing model should have had.

The template_map is only used when opening seed parts from Teamcenter. It is not used by the SolidWorks ***File | New*** *command, or by the* ***Teamcenter | Teamcenter New*** *command.*

A template_map contains zero or more template tags:

```
<template_map>
  <template cad_type="type" location/>
      .
      .
      .
</template_map>
```

Each template tag specifies a location where a template can be found, either on the local file system or in Teamcenter. Each template tag also has a cad_type attribute that specifies the type of SolidWorks model to be created from the template. The allowed values for type are "sldasm", "slddrw", and "sldprt".

If the template tag specifies a location in Teamcenter, more than one SolidWorks model type may be given for cad_type. Use a colon to separate each three-letter value from the next. For example, to indicate that an item in Teamcenter contains templates for parts, assemblies, and drawings, the attribute should be cad_type="sldprt:sldasm:slddrw".

The cad_type attribute should specify only one type when the location is on the local file system.

The models used as templates should not depend on other models. When a model specified in the template_map is copied to create a new model, that model alone is copied. No other models are copied with it.

Note that, when a SolidWorks document is copied to create a new model, the document is copied in its entirety. The resulting new document will have the same configurations, with the same names, as in the original.

A SolidWorks configuration cannot be used as a template.

Using Templates from a Directory

A template model that is located in a directory is specified by the path to its file:

```
<template cad_type="type" path="path"/>
```

The file must exist, and *path* cannot contain any wildcards, but *path* may include keywords described in the **Substitution Keywords** section for general auxiliary files, if the keywords do not imply a specific model. For example, a SolidWorks part model could be included in the template_map as follows.

```
<template cad_type="sldprt" path="C:\SW\data\templates\Part.sldprt"/>
```

Using Templates from Teamcenter

A template model located in Teamcenter is specified by its item ID, revision ID and model name:

```
<template cad_type="type"
  item_id="item" [item_revision_id="revision"] [model_name="name"]/>
```

The revision and model name are optional, and if not specified, all revisions and datasets in the item are searched for templates of the appropriate type. Wildcards may also be used in *item*, *revision* and *name*, as if conducting a simple search in the Open dialog. If more than one revision of a model is found for a given type, the latest one is used. If several models with different names are found for a given type, one is arbitrarily selected. For example, suppose an item Seeds is created in Teamcenter to hold the templates for parts and assemblies. This item can be referenced using a single template tag:

```
<template cad_type="sldprt:sldasm" item_id="Seeds"/>
```

If Seeds has a part named mm_part and an assembly named mm_asm, the document for mm_part will be used to create parts and the document for mm_asm will be used to create assemblies. If Seeds has two revisions, the documents from the later revision will be used. However, if Seeds also has a part model named in_part, either mm_part or in_part will be arbitrarily selected as the template to use when creating a new part. In a situation such as this, it may be helpful to also specify the model_name attribute so only one of these models will be selected:

```
<template cad_type="sldprt:sldasm" item_id="Seeds" model_name="mm_*"/>
```

Dataset Map

Each type of SolidWorks CAD model maps to a unique dataset type and named reference in Teamcenter. The type_map and dataset_map tag define these one-to-one mappings. The type_map is not intended for customer modification and so it is not covered in this document.

A typical dataset map is shown below. None of these mappings are optional.

```
<dataset_map>
   <dataset allowed_item_types="Item:Part:Document" allowed_workflows="Quick
      Release:Design Review" named_ref="AsmFile" pdm_type="SWAsm"/>
   <dataset allowed_item_types="Part:Item:Document" allowed_workflows="Quick
      Release:Design Review" named_ref="PrtFile" pdm_type="SWPrt"/>
   <dataset allowed_item_types="Item:Part:Document" allowed_workflows="Quick
      Release:Design Review" named_ref="DrwFile" pdm_type="SWDrw"/>
   <dataset allowed_item_types="Item:Part:Document" allowed_workflows="Quick
      Release:Design Review" named_ref="TbxFile" pdm_type="SW2Tbx"/>

</dataset_map>
```

For each mapping between a SolidWorks CAD type and a Teamcenter dataset type, the allowable named reference, item types, and workflows are defined by an entity tag. The pdm_type identifies the dataset type, and named_ref identifies the named reference for the SolidWorks model. These tags should not be changed by customers.

Allowed Item Types

The mandatory allowed_item_types attribute is a colon separated list that defines the list of item types to appear in the drop-down list on the **Save** dialog. The first item type in the list is the default item type for the specified CAD type[5].

> *Customers must configure each dataset type's "allowed_item_types" to include the same item type for CAD types which may be saved together to the same item ID, such as a drawing and its 3D model. This allows a dataset such as a SWDrw to be saved under the 3D model revision. Otherwise an error will occur.*
>
> *Assigning a singular and different value to the "allowed_item_type" preference for each dataset type will prevent more than one dataset type from being saved under the same item revision.*

Submit to Workflow

The integration can be configured to submit Item Revisions to a workflow process by adding "workflow" to the table.columns.checkin preference in swim.properties.

The optional allowed_workflows tag is a colon separated list that defines the list of Teamcenter workflows to appear in the workflow column drop-down list on the **Save** dialog. The first workflow in the list is the default for the specified CAD type.

> *Customers must configure the same default workflow for CAD types which can be saved together to the same item ID, such as a drawing and its 3D model. Assigning different default workflows to such CAD types can result in the same item revision being submitted to multiple workflows simultaneously, which is not the desired result.*

The integration determines if an object is checked in, checked out, or released by evaluating the status of the dataset. Workflows that apply status to the objects should set the status on the Item Revision and the SolidWorks dataset. Failure to set the status on the dataset will enable the user to continue to work on the object. Teamcenter does not provide a workflow that will simultaneously release both the Item Revision and the Dataset. See the Teamcenter documentation for more information on how to create and configure workflows.

[5] Note that the ordering of the allowed_item_types list in the dataset map replaces the single iman.itemtypedefault swim.properties preference that was used in earlier releases, providing more configuration flexibility.

Example Map File

The contents of a complete map file are shown below. It illustrates the use of the attribute map, the template map, and the column map, as well as the default maps which are configured at time of installation.

```
<?xml version="1.0" encoding="UTF-8"?>
<!DOCTYPE data_def SYSTEM "swim.dtd">
<data_def>
    <type_map>
        <entity cad_type="sldasm" pdm_type="SWAsm"/>
        <entity cad_type="sldprt" pdm_type="SWPrt"/>
        <entity cad_type="slddrw" pdm_type="SWDrw"/>
        <entity cad_type="sldtbx" pdm_type="SW2Tbx"/>
    </type_map>

    <dataset_map>
        <dataset allowed_item_types="Item:Document" allowed_workflows=”TCM
             Release” named_ref="AsmFile" pdm_type="SWAsm"/>
        <dataset allowed_item_types=" Item:Document" allowed_workflows=”TCM
             Release” named_ref="PrtFile" pdm_type="SWPrt"/>
        <dataset allowed_item_types=" Item:Document" allowed_workflows=”TCM
             Release” named_ref="DrwFile" pdm_type="SWDrw"/>
        <dataset allowed_item_types=" Item:Document" allowed_workflows=”TCM
             Release” named_ref="TbxFile" pdm_type="SW2Tbx"/>
    </dataset_map>

    <template_map>
        <template cad_type="sldprt" path="c:\SOLIDW~2\templates\collar.sldprt"/>
        <template cad_type="sldasm" path="c:\SOLIDW~2\templates\Assembly.sldasm"/>
        <template cad_type="slddrw" path="c:\SOLIDW~2\templates\Drawing.slddrw"/>
    </template_map>

    <attribute_map cad_type="sldasm:sldprt">
        <!-- Attribute functions -->
        <attribute>
            <cad_name value="is_configuration()"/>
            <pdm_name value="ItemRevision:Pa4CADItemRevision.Form:
             Pa4CADItemRevisionMaster.pa4is_family_member"/>
            <direction value="cadtopdm"/>
        </attribute>
    </attribute_map>
    <attribute_map cad_type="sldasm:sldprt:slddrw">
           <!-- Keyword attributes -->
        <attribute>
            <cad_name value="DatasetComment"/>
            <pdm_name value="dataset_desc"/>
            <missing_attribute_action value="create"/>
            <default type="String">Test Dataset Description</default>
            <direction value="both"/>
        </attribute>
        <attribute>
            <cad_name value="ItemComment"/>
            <pdm_name value="item_desc"/>
            <missing_attribute_action value="create"/>
            <default type="String">Test Item Description</default>
            <direction value="both"/>
        </attribute>
        <attribute>
            <cad_name value="RevComment"/>
            <pdm_name value="item_revision_desc"/>
            <missing_attribute_action value="create"/>
            <default type="String">Test Revision comment</default>
            <direction value="both"/>
        </attribute>
        <attribute>
            <cad_name value="Description"/>
            <pdm_name value="item_name"/>
            <direction value="cadtopdm"/>
             <constraint value="use_default_cfg"/>
        </attribute>
        <attribute>
            <cad_name value="Description"/>
            <pdm_name value="ItemRevision.object_name"/>
            <direction value="both"/>
        </attribute>
        <attribute>
            <cad_name value="PartNumber"/>
            <pdm_name value="item_id"/>
            <missing_attribute_action value="create"/>
            <direction value="pdmtocad"/>
             <constraint value="use_default_cfg"/>
        </attribute>
        <attribute>
            <cad_name value="Revision"/>
            <pdm_name value="item_revision_id"/>
            <missing_attribute_action value="create"/>
            <direction value="pdmtocad"/>
             <constraint value="use_default_cfg"/>
        </attribute>
        <attribute>
            <cad_name value="ItemType"/>
            <pdm_name value="item_type"/>
            <missing_attribute_action value="create"/>
            <direction value="pdmtocad"/>
             <constraint value="use_default_cfg"/>
        </attribute>
        <!-- Attributes for miscellaneous Teamcenter classes -->
        <attribute>
            <cad_name value="UnitOfMeasure"/>
            <pdm_name value="Item.uom_tag"/>
            <missing_attribute_action value="create"/>
            <direction value="pdmtocad"/>
        </attribute>
        <attribute>
            <cad_name value="Status"/>
            <pdm_name value="ItemRevision.release_status_list"/>
            <missing_attribute_action value="create"/>
            <direction value="pdmtocad"/>
        </attribute>
        <!-- Attributes for custom form -->
        <attribute>
            <cad_name value="Size"/>
            <pdm_name value="ItemRevision:Pa4CADItemRevision.Form:
             Pa4CustomForm.pa4S"/>
            <prompt value="none"/>
            <default type="String">Test size</default>
            <missing_attribute_action value="create"/>
            <direction value="both"/>
        </attribute>
        <attribute>
            <cad_name value="DIAMETER"/>
            <pdm_name value="ItemRevision:Pa4CADItemRevision.Form:
             Pa4CustomForm.pa4D"/>
            <prompt value="none"/>
```

```
        <default type="Double">6.666</default>
        <missing_attribute_action value="create"/>
        <direction value="both"/>
    </attribute>
    <attribute>
        <cad_name value="LATEST"/>
        <pdm_name value="ItemRevision:Pa4CADItemRevision.Form:
         Pa4CustomForm.pa4L"/>
        <prompt value="none"/>
        <default type="Boolean">true</default>
        <missing_attribute_action value="create"/>
        <direction value="both"/>
    </attribute>
    <attribute>
        <cad_name value="Date"/>
        <pdm_name value="ItemRevision:Pa4CADItemRevision.Form:
         Pa4CustomForm.pa4DT"/>
        <prompt value="none"/>
        <default type="String">03-Sep-2010</default>
        <missing_attribute_action value="create"/>
        <direction value="both"/>
    </attribute>
    <attribute>
        <cad_name value="ID"/>
        <pdm_name value="ItemRevision:Pa4CADItemRevision.Form:
         Pa4CustomForm.pa4I"/>
        <prompt value="none"/>
        <default type="Integer">2010</default>
        <missing_attribute_action value="create"/>
         <truncate value="true"/>
        <direction value="both"/>
    </attribute>
</attribute_map>

<!-- Column Mapping -->
<column_map>
    <column tag="COLUMN1">
        <display_name value="Column1"/>
        <cad_name value="MODELED_BY"/>
    </column>
    <column tag="COLUMN2">
        <display_name value="Column2"/>
        <cad_name value="DESCRIPTION"/>
    </column>
    <column tag="MAXIMUM">
        <display_name value="MAXIMUM!!"/>
        <pdm_name value="ItemRevision:Pa4CADItemRevision.Form:
         Pa4CustomForm.pa4I"/>
    </column>
    <column tag="WHICH_SIZE">
        <display_name value="SIZE?"/>
        <pdm_name value="ItemRevision:Pa4CADItemRevision.Form:
         Pa4CustomForm.pa4S"/>
    </column>
    <column tag="USER_DATA">
        <display_name value="User Data"/>
        <pdm_name value="Item Master.user_data_2"/>
    </column>
</column_map>

<auxiliary_file_map>
    <auxiliary_file cad_type="sldprt" direction="cadtopdm">
        <!-- Uncomment the below if you want to give the user choice of saving the
```

```
                preview files -->
        <!-- cadtopdm_control label="Save Preview Files"/ -->
        <pdm_location named_ref="PNG" pdm_type="SWPrt"/>
        <file_name pattern="{temp_dir}\{cad_name}p.PNG"/>
        <preview_command cmd="built_in {temp_dir}\{cad_name}p.PNG"
                ignore_status="true" phase="in_directory"/>
    </auxiliary_file>
    <auxiliary_file cad_type="sldtbx" direction="cadtopdm">
        <pdm_location named_ref="PNG" pdm_type="SW2Tbx"/>
        <file_name pattern="{temp_dir}\{cad_name}t.PNG"/>
        <preview_command cmd="built_in {temp_dir}\{cad_name}t.PNG"
                ignore_status="true" phase="in_directory"/>
    </auxiliary_file>
    <auxiliary_file cad_type="sldasm" direction="cadtopdm">
        <pdm_location named_ref="PNG" pdm_type="SWAsm"/>
        <file_name pattern="{temp_dir}\{cad_name}a.PNG"/>
        <preview_command cmd="built_in {temp_dir}\{cad_name}a.PNG"
                ignore_status="true" phase="in_directory"/>
    </auxiliary_file>
    <auxiliary_file cad_type="slddrw" direction="cadtopdm">
        <pdm_location named_ref="PNG" pdm_type="SWDrw"/>
        <file_name pattern="{temp_dir}\{cad_name}d.PNG"/>
        <preview_command cmd="built_in {temp_dir}\{cad_name}d.PNG"
                ignore_status="true" phase="in_directory"/>
    </auxiliary_file>
    <jt_file action="translate" cad_type="sldprt:sldtbx"
                eai_dir="D:\apps\translators\swjt"/>
    <auxiliary_file cad_type="slddrw" direction="cadtopdm">
         <cadtopdm_control label="Save PDF Files"/>
         <pdm_location named_ref="PDF_Reference"
         relation_type="IMAN_specification"
         pdm_type="PDF"/>
         <file_name pattern="{temp_dir}\{cad_name}d.PDF"/>
         <generate_command cmd="built_in {temp_dir}\{cad_name}d.PDF"
         cad_type="PDF"
         ignore_status="true" phase="in_directory"/>
     </auxiliary_file>
</auxiliary_file_map>

<bom_map>
    <bom_line child_type="sldasm" parent_type="sldasm" view_types="view">
        <!-- Uncomment the line below to set the componentReference -->
        <bom_prop cadName="componentReference" pdmName="bl_sequence_no"
                direction="pdmtocad"/>
    </bom_line>
    <bom_line child_type="sldprt" parent_type="sldasm" view_types="view">
        <!-- Uncomment the line below to set the componentReference -->
        <bom_prop cadName="componentReference" pdmName="bl_sequence_no"
                direction="pdmtocad"/>
    </bom_line>
    <bom_line child_type="sldtbx" parent_type="sldasm" view_types="view">
        <!-- Uncomment the line below to set the componentReference -->
        <bom_prop cadName="componentReference" pdmName="bl_sequence_no"
                direction="pdmtocad"/>
    </bom_line>
</bom_map>
<transfer_ownership>
    <set_preference session_name="PORTAL_SESSION_IMAN_ownership_export">TRUE
    </set_preference>
    <set_preference name="IMAN_run_ie_in_background"
     session_name="PORTAL_SESSION_IMAN_run_ie_in_background">FALSE
    </set_preference>
    <set_preference name="IMAN_include_folder_contents"
```

```
 session_name="PORTAL_SESSION_IMAN_include_folder_contents">FALSE
</set_preference>
<set_preference name="IMAN_item_latest_rev_export"
 session_name="PORTAL_SESSION_IMAN_item_latest_rev_export">FALSE
</set_preference>
<set_preference name="IMAN_item_latest_working_rev_export"
 session_name="PORTAL_SESSION_IMAN_item_latest_working_rev_export">
 FALSE</set_preference>
<set_preference name="IMAN_item_latest_working_or_any_rs_export"
 session_name="PORTAL_SESSION_IMAN_item_latest_working_or_any_rs_export">
 FALSE</set_preference>
<set_preference name="IMAN_item_latest_released_rev_export"
 session_name="PORTAL_SESSION_IMAN_item_latest_released_rev_export">
 FALSE</set_preference>
<set_preference name="IMAN_item_selected_revs_export"
 session_name="PORTAL_SESSION_IMAN_item_selected_revs_export">FALSE
</set_preference>
<set_preference name="IMAN_modified_only_export"
 session_name="PORTAL_SESSION_IMAN_modified_only_export">FALSE
</set_preference>
<set_preference name="IMAN_protected_object_export"
 session_name="PORTAL_SESSION_IMAN_protected_object_export">FALSE
</set_preference>
<set_preference name="IMAN_bom_level_export"
 session_name="PORTAL_SESSION_IMAN_bom_level_export">0
</set_preference>
<set_preference name="IMAN_assy_xfer_top_only"
 session_name="PORTAL_SESSION_IMAN_assy_xfer_top_only">FALSE
</set_preference>
<set_preference name="IMAN_assy_no_xfer_comp"
 session_name="PORTAL_SESSION_IMAN_assy_no_xfer_comp">FALSE
</set_preference>
<set_preference name="IMAN_assy_no_export_comp"
 session_name="PORTAL_SESSION_IMAN_assy_no_export_comp">FALSE
</set_preference>
<set_preference name="IMAN_assy_retrieve_distrib_comp"
 session_name="PORTAL_SESSION_IMAN_assy_retrieve_distrib_comp">
 FALSE</set_preference>
<set_preference name="IMAN_item_specific_status_export"
 session_name="PORTAL_SESSION_IMAN_item_specific_status_export">
 FALSE</set_preference>
<set_preference name="IMAN_ie_continue_on_error"
 session_name="PORTAL_SESSION_IMAN_ie_continue_on_error">FALSE
</set_preference>
<set_preference name="IMAN_sync_auto_synchronize"
 session_name="PORTAL_SESSION_IMAN_sync_auto_synchronize">FALSE
</set_preference>
<set_preference name="IMAN_sync_batch_synchronize"
 session_name="PORTAL_SESSION_IMAN_sync_batch_synchronize">FALSE
</set_preference>
<set_preference name="IMAN_sync_notify"
 session_name="PORTAL_SESSION_IMAN_sync_notify">FALSE</set_preference>
<set_preference name="IMAN_bomchange_export"
 session_name="PORTAL_SESSION_IMAN_bomchange_export">FALSE
</set_preference>
<set_preference name="IMAN_supercedure_export"
 session_name="PORTAL_SESSION_IMAN_supercedure_export">FALSE
</set_preference>
<set_preference name="IMAN_item_revs_export"
 session_name="PORTAL_SESSION_IMAN_item_revs_export">TRUE
</set_preference>
<set_preference name="IMAN_dataset_vers_export"
 session_name="PORTAL_SESSION_IMAN_dataset_vers_export">TRUE
```

```
        </set_preference>
        <set_preference name="IMAN_dataset_refs_export"
         session_name="PORTAL_SESSION_IMAN_dataset_refs_export">TRUE
        </set_preference>
        <add_preference_list name="IMAN_relation_export"
         session_name="PORTAL_SESSION_IMAN_relation_export">
            <preference_list_value>IMAN_master_form</preference_list_value>
            <preference_list_value>IMAN_Rendering</preference_list_value>
            <preference_list_value>IMAN_requirement</preference_list_value>
            <preference_list_value>IMAN_specification</preference_list_value>
            <preference_list_value>IMAN_RES_audit</preference_list_value>
            <preference_list_value>TXD_long_name_relation</preference_list_value>
        </add_preference_list>
        <remove_preference_list name="IMAN_relation_export"
         session_name="PORTAL_SESSION_IMAN_relation_export">
            <preference_list_value>IPEM_dependency</preference_list_value>
            <preference_list_value>SWIM_dependency</preference_list_value>
            <preference_list_value>SWIM_master_dependency</preference_list_value>
            <preference_list_value>SWIM_suppressed_dependency</preference_list_value>
            <preference_list_value>SW2_bom_exclusion</preference_list_value>
            <preference_list_value>SW2_configuration</preference_list_value>
        </remove_preference_list>
    </transfer_ownership>
</data_def>
```

swim.cfg

Many customers will not find a reason to deal with this file at all. There are only three settings that may require modification, under normal circumstances:

sw.toolbox.dirs	The integration's installation program asks the user whether SolidWorks Toolbox should be configured, and if so, updates this preference with the specified path(s) to the Toolbox directories. If new Toolbox directories are added after the initial installation, it may be easier to extend the set of locations by editing this text file, instead of re-running the installation process. It is a simple list of directory paths, separated by the ';' character.
sw.manager.node.text	In the Integration's default configuration, CAD models are displayed in the Teamcenter task pane with a predefined set of key properties and attributes. The default set of values, or the order in which they appear, may be changed by modifying this setting. You may use properties of the CAD models, or Teamcenter attributes, or both together, to display the specific model information that is most meaningful for your CAD users. For example, in the following figure the Teamcenter Item ID/Revision appear first, followed by the SolidWorks Description, which is mapped from the Teamcenter Item Name:

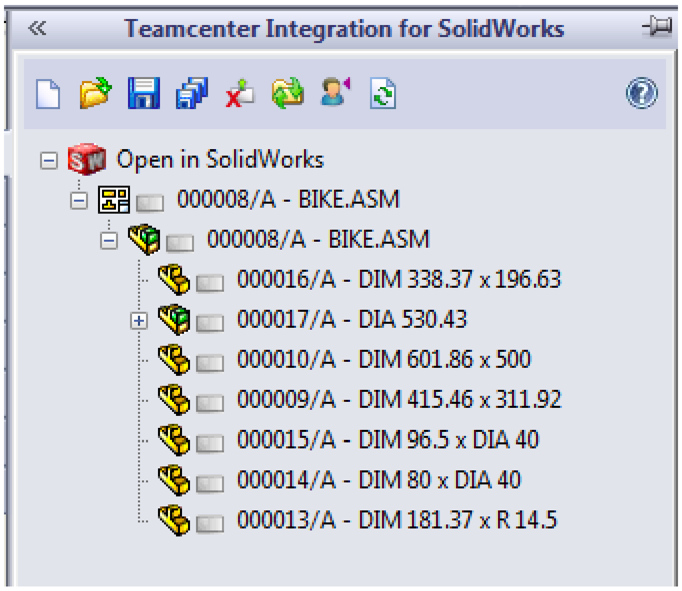

Teamcenter Integration for SolidWorks
Open in SolidWorks
000008/A - BIKE.ASM
000008/A - BIKE.ASM
000016/A - DIM 338.37 x 196.63
000017/A - DIA 530.43
000010/A - DIM 601.86 x 500
000009/A - DIM 415.46 x 311.92
000015/A - DIM 96.5 x DIA 40
000014/A - DIM 80 x DIA 40
000013/A - DIM 181.37 x R 14.5

Figure 2 Teamcenter task recommended display values

sw.debugPrints	When a more detailed client_socket.log file is needed for troubleshooting, set this preference to any value greater than 0.

Teamcenter Preferences

TC_Allow_Longer_ID_Names:	If the Teamcenter preference TC_Allow_Longer_ID_Names is set to true, Item Ids and Item Revision names up to 132 characters, and Dataset names up to 125 characters, are allowed. The Dataset names are limited to 125 characters because of the limit of named references to 132 characters including extensions (7 characters are used for the extension, such as .sldprt).
WsoInsertNoSelectionsPref:	This preference controls where newly-created items are placed in Teamcenter, when the user has not explicitly selected a destination folder. This occurs when the user selects the top-level Teamcenter folder in the **Save** dialog. The default for this preference is ‘1’, which means that new items will be added to the Newstuff folder. Other integer values cause the item to be added to either the user’s Home folder, or no folder at all.
TC_relation_required_on_export:	When SolidWorks data is transferred from one Teamcenter site to another, the Integration’s CAD-specific relationship must be traversed to find and include all dependencies which are not already part of the assembly BOM. The Teamcenter preference must be extended to include the following relations:

- SWIM_dependency
- SWIM_master_dependency
- SWIM_suppressed_dependency
- SW2_configuration

EPM_multiple_processes_targets:	The integration can be configured to submit Item Revisions to a workflow process by adding “workflow” to the table.columns.checkin preference in swim.properties. The integration does not prevent users from submitting objects to multiple workflows. Sites which want to prevent this should set the Teamcenter preference EPM_multiple_process_targets to “OFF

Utility Functions

In addition to using the interactive Integration in your SolidWorks session, you can also use several command line utilities to work with SolidWorks models in Teamcenter. These utilities are found in the same directory where the Teamcenter Integration for SolidWorks is installed. These are:

- The swimexport utility for exporting SolidWorks models from Teamcenter.
- The swimimport utility for saving SolidWorks models to Teamcenter.
- The swimrename utility for renaming SolidWorks models in Teamcenter.

If the Integration’s installation directory is in your path, you can run any of these utilities by opening a shell or command window for your operating system, and typing the utility’s name with appropriate arguments on the command line. Arguments are shown in detail in the following sections under Usage. Arguments enclosed in square brackets [] are optional, a vertical bar | separates alternate values that can be used for an argument, and text shown in italics is to be replaced by an appropriate value when you use the command. For example,

swimexport -u *username* **-p** *password* [**-export** true|false]

shows the swimexport utility’s –export argument is optional because it is enclosed in brackets, and that its value can be one of the literal words true or false, while *username* and *password* should be replaced by your Teamcenter user name and password.

Exporting SolidWorks Models from Teamcenter with swimexport

The swimexport utility exports SolidWorks models from Teamcenter. Datasets may also be checked out, i.e. reserved, when exporting models. Models that are checked out by swimexport may be checked back into Teamcenter with the swimimport utility.

Usage

```
swimexport [-login] [-u username -p password
[-g group]]
[-help]
[-project project_id]
```

```
[-export_dir path]
[-change_id id] [-comment string]
[-export true|false] [-export_related true|false]
[-checkout true|false] [-checkout_related true|false]
[-config rule] [-precise_is_imprecise]
[-precise_relation_is_imprecise true|false]
[-precise_configuration_is_imprecise true|false]
[-no_filters] [-filter name] [-levels limit]
[-item_id id] [-item_revision_id rev] [-model_name datasetname]
[-model_type type] [-dry_run] [-output instructionfile]
[-log logfile]
[-incomplete_operation_fatal | -incomplete_operation_ok]
[-serverhost host] [-servername servername]
[-marker markername]
 file1 file2 ...
```

Description

The swimexport utility will export a SolidWorks model, and the models on which it depends, from Teamcenter to a local directory. The model to export may be identified with the –item_id, –item_revision_id, –model_name and –model_type options on the command line, or several models may be specified in *instruction files*, given by arguments *file1*, *file2*, etc., following any command line options. Instruction files are text files that list models to export, one model per line.

Some options affect only *explicitly specified* models identified on the command line or in instruction files. Other options affect *implicitly specified* models that are not identified on the command line or in instruction files, but which are also exported because explicitly specified models depend on them.

The default action is to export all explicitly specified models and the models on which they depend, but this can be changed using the –export option for explicitly specified models and the –export_related option for implicitly specified models. To reserve models, the –checkout option reserves explicitly specified models and –checkout_related reserves implicitly specified models. Models that have been reserved may be checked back into Teamcenter using the swimimport utility.

The swimexport utility uses the same mapping file and user preferences used by the Teamcenter Integration for SolidWorks.

One significant difference between the swimexport utility and the Teamcenter Integration for SolidWorks is that swimexport does not copy attribute values from Teamcenter to properties in the SolidWorks models.

The swimexport options are described below.

-change_id *id*

Specifies a change ID to assign to all datasets that are checked out (optional).

`-checkout` *true | false*

True specifies that explicitly specified models should be reserved (optional). The default is false, which means that these models will not be reserved. Note that if –export is false, this option can be used to reserve a dataset without exporting the model file in it. This option does not affect implicitly specified models. To reserve implicitly specified models, use the –checkout_related option.

`-checkout_related` *true | false*

True specifies that implicitly specified models should be reserved (optional). The default is false, which means that these models will not be reserved. Note that if –export_related is false, this option can be used to reserve datasets without exporting the model files in them.

`-comment` *string*

Specifies a comment to assign to all datasets that are checked out (optional).

`-config` *rule*

Specifies the name of a configuration rule (optional). The default is to use the rule specified by the iman.bomrevisionruledefault user preference or the default Teamcenter rule.

`-dry_run` The utility goes through most of the actions it would ordinarily execute, but it does not export any files or reserve datasets (optional). When used in conjunction with the –output option, an instruction file can be generated for use in a subsequent run of this utility.

`-export` *true | false*

True specifies that explicitly specified models should be exported, which is the default (optional). False means that these models will not be exported. Note that –checkout may be true even if –export is false. This option does not affect implicitly specified models. To export implicitly specified models, use the –export_related option.

`-export_dir` *path*

Specifies the path to the export directory (optional). The default is to export models to the default working directory as specified during product installation. The default working directory can be overridden by specifying checkout.exportdirectory preference in swim.properties.

`-export_related` *true | false*

True specifies that implicitly specified models should be exported, which is the default (optional). False means that these models will not be exported. Note that the –checkout_related option may be true even when -export_related is false.

`-filter` *name*

Specifies a filter to use when collecting implicitly specified models (optional). Each use of this option adds another filter to any filters previously specified. Later filters can override the actions of previous filters. The default filters are those specified by the filters.checkout.default user preference, but note that the -no_filters option can erase those prior filters. See the filters.checkout.choices user preference for the names of filters that can be used with this option. User preferences are described in the swim.properties file located in the directory where the Teamcenter Integration for SolidWorks is installed.

`-g` *groupname*
: Specifies the group name to use when connecting to Teamcenter (optional).   Ignored when -login is specified.

`-help` Prints usage information and exits.

`-incomplete_operation_fatal`
: Causes the program to exit when an operation on a model cannot be completed due to an error (optional). The default is -incomplete_operation_ok.

`-incomplete_operation_ok`
: Causes the program to continue when an operation on a model cannot be completed due to an error (optional). This is the default.

`-item_id` *id*
: Specifies the Teamcenter item ID of a model to export (optional). This option may be used only once on the command line, but wildcards are allowed.

`-item_revision_id` *rev*
: Specifies the Teamcenter revision of a model to export (optional). This option may be used only once on the command line, but wildcards are allowed.

`-levels` *limit*
: Specifies the number of levels to include in the hierarchy of dependencies when collecting implicitly specified models (optional). If not specified, or if *limit* is a negative number, the number of implicitly specified models is unlimited. A positive number sets the maximum number of levels of implicitly specified models that can be collected. This option has the same purpose as the limit.levels user preference, described in the swim.properties file located in the directory where the Teamcenter Integration for SolidWorks is installed, although swimexport ignores limit.levels. The limit can only be set using this command line argument.

`-log` *logfile* Defines the name of the log file that will be created (optional). If not specified, the log file will be swimexport.txt in the current directory. Logging cannot be disabled and warning messages cannot be suppressed, but otherwise the logging preferences in swim.properties have their usual effect.

`-login` Uses the Teamcenter login dialog. When this option is specified, the -u, -p, -g, and -marker options are ignored.

`-marker` *markername*

The Teamcenter server marker name or URI. If not specified, the default marker is that of the first server in the client_specific.properties file, found in the Teamcenter Rich Client installation directory. If URI includes "services/PLMGatewayService"; remove it. For example, "http://serverrname:7001/tc/services/PLMGatewayService" should be "http://servername:7001/tc/". Ignored when –login is specified

`-model_name` *datasetname*

Specifies the name of a SolidWorks model, which may be either the name of a SolidWorks document or the name of a SolidWorks configuration. The model name is the same as the dataset name.

`-model_type` *type*

Specifies the dataset type or the six-letter SolidWorks model type. For example, models stored in the files widget.prt and gadget.sldprt both have type sldprt, and their datasets are of type SWPrt, so *type* for a SolidWorks part model may be either sldprt or SWPrt.

`-no_filters`

Causes any previous list of filters to be erased, such that no filters will be used when collecting implicitly specified models. However, a new list of filters will be created if this option is followed by one or more -filter options.

`-output` *instructionfile*

Defines the name of an instruction file to be written (optional). This file lists each model exported or reserved from Teamcenter.

`-p` *password* Specifies the password to use when connecting to Teamcenter.   Ignored when -login is specified.

`-precise_is_imprecise`

Causes precise relations to be treated as if they are imprecise, which means the rule given by the –config option will be used to select item revisions (optional). This option is a shortcut for the two options:

-precise_relation_is_imprecise true
-precise_configuration_is_imprecise true

`-precise_configuration_is_imprecise` *true | false*

True causes a precise relation between a configuration and its SolidWorks document to be treated as if it is imprecise, which means the rule given by the –config option will be used to select the document's item revision (optional).

`-precise_relation_is_imprecise` *true | false*

True causes precise general relations to be treated as if they are imprecise, which means the rule given by the –config option will be used to select

	item revisions (optional). This option does not affect the treatment of relations between configurations and their SolidWorks document.
`-project` *project_id*	Specifies the project to use when connection to Teamcenter (optional).
`-u` *username*	Specifies the user name to use when connecting to Teamcenter. Ignored when -login is specified.

Configurations

Every SolidWorks model in Teamcenter is represented by a dataset, even models that are configurations. However, in the case of a configuration, the dataset does not contain the model's file. Instead, configurations depend on their document, and it is in the dataset for the document that the SolidWorks model file is found.

If you "export" a configuration, swimexport will actually export the model file from the document's dataset. If you reserve a configuration, swimexport will also reserve the document.

Toolbox Parts

The swimexport utility does not export files for toolbox parts stored as sw2Tbx datasets. An assembly exported by swimexport must be opened with a SolidWorks installation having access to the toolbox. For more information on managing Toolbox parts, see the **Best Practices** section in this document.

Multiple Model Versions

Only one version of any given SolidWorks model will be exported in one execution of the swimexport utility. For example, if several revisions of a model are specified on the command line or in instruction files, only the first one encountered will actually be exported.

Instruction Files

An instruction file is a text file that lists SolidWorks models. Each line in the file should identify one model. Blank lines and comments are also allowed. The comment character is #. Any text between the comment character and the end of the line will be ignored.

A model is identified by several *keyword*=*value* pairs. A value containing spaces must be enclosed in double quotes, and *keyword* is one of the following:

checkout	This is the same as the –checkout command line option, but it applies only to the model specified on the line where this keyword appears. The value may be either true or false. If this keyword is not specified, the value of the –checkout command line option applies.
export	This is the same as the –export command line option, but it applies only to the model specified on the line where this keyword appears. The value may be either true or false. If this keyword is not specified, the value of the –export command line option applies.
item_id	The value is the item ID.

item_revision_id
: The value is the revision.

model_name
: The value is the name of the SolidWorks model, which may be either the name of a SolidWorks document or the name of a SolidWorks configuration.. The model name is the same as the dataset name.

model_type
: The value is the dataset type or the three-letter file extension that identifies the SolidWorks model type. For example, models stored in the files widget.prt and gadget.sldprt both have type sldprt, and their datasets are of type SWPrt, so model type for a SolidWorks part model may be either sldprt or SWPrt

If a value contains spaces, it must be enclosed in double quotes.

A simple instruction file is shown below.

```
# Sample swimexport instruction file
item_id=702283 item_revision_id=B model_name=Bolt_A model_type=sldprt
item_id=702284 item_revision_id=A export=false checkout=false
item_id=702285 item_revision_id=A
```

The first line is a comment. The second line identifies a particular model, the SolidWorks part Bolt_A in revision B of item 702283. No checkout or export keyword is given on this line, so the command line options –checkout and –export will be used to decide whether to export and/or reserve this model. By default, swimexport will export the model without reserving it.

The third line specifies an item ID and revision, but the model name and type are not given. If revision A of item 702284 has several SolidWorks datasets, this line affects all of them. For example, if revision A contains both a part and a drawing, neither will be exported or checked out because export and checkout are both false on this line.

The fourth line also identifies a model using only its item ID and revision. Since no export or checkout keyword is included on this line, the model will be exported or reserved according to the command line options –export and –checkout.

Examples

The following example shows a simple use of this export utility.

```
swimexport -u infodba -p infodba –item_id 702283 –item_revision_id=B
```

This command will export the SolidWorks model(s) in revision B of item 702283, plus all models on which it depends. The model files will be placed in the current working directory. None of these models will be reserved. If the models in 702283 should be reserved, the command becomes:

```
swimexport -u infodba -p infodba –item_id 702283 –item_revision_id=B \
      -checkout true –comment "Changes per 4283"
```

This reserves only the model in revision B of 702283, but the models on which it depends are still exported. If the SolidWorks model in revision B should be reserved without being exported, and the models on which it depends also should not be exported, the command would be:

```
swimexport -u infodba -p infodba –item_id 702283 –item_revision_id=B \
      -checkout true -export false –export_related false \
      –comment "Changes per 4283"
```

If 702283 is a configuration, the command above will also reserve the document regardless of how –export_related is set.

Suppose an instruction file named exports.txt specifies the models to export. In this case, the command would be:

```
swimexport -u infodba -p infodba exports.txt
```

To find out what will really be exported or reserved prior to executing the command above, use the –dry_run option and ask swimexport to generate an instruction file:

```
swimexport -u infodba -p infodba -dry_run –output out.txt exports.txt
```

Importing SolidWorks Models into Teamcenter with swimimport

The swimimport utility imports SolidWorks models into Teamcenter. This non-interactive utility saves models to Teamcenter after opening them in SolidWorks. During the import process, items, item revisions, and datasets may be created, parameter values may be copied to attributes in Teamcenter, and BOMs may be updated.

Usage

swimimport [**-login**] [**-u** *username* **-p** *password* [**-g** *group*]]
[**-help**] [**-dry_run**] **[-project** project_id]
[**-output** *instructionfile*] [**-log** *logfile*]
[**-folder** *folder*] [**-overwrite** | **-revise**] [**-all_configurations**]
[**-marker** *markername*] *file1 file2 ...*

Description

The arguments *file1*, *file2*, ..., are SolidWorks models to import, directories containing SolidWorks models, or *instruction files*. An instruction file is a text file in which each line is either the path to a SolidWorks model file or a directory containing SolidWorks models. This utility opens all of the models specified by these arguments in a SolidWorks session, obtains information concerning parent-child relationships, parameter values, configurations, and so on, and then imports the models into Teamcenter.

Each argument *file1*, *file2*, ..., must be an absolute path.

Models are imported in two steps. In the first step, Teamcenter objects (items, item revisions, and datasets) are created as needed and the model files are imported into the datasets. In the second step, parameter values are copied from SolidWorks to attributes of the Teamcenter objects.

When importing a model into an existing dataset, you must either use the –overwrite option or check out the dataset prior to running swimimport. Models will not be imported into datasets that are reserved by another user. Also, if you reserve a dataset using check out with export, the model can only be imported from the export directory that was specified at the time you reserved the dataset, and the model will be deleted from that directory after it is imported.

The other arguments for swimimport are described below.

`-all_configurations`

Saves all configurations within a document as distinct items in Teamcenter (optional). This option overrides the sw.configurations.default user preference to save all configurations except those matching the sw.configurations.masters user preference. If not given, the default is to create items only for configurations that are explicitly identified in an instruction file or implicitly allowed by the sw.configurations.default user preference. User preferences are described in the swim.properties file found in the Teamcenter Integration for SolidWorks installation directory.

`-dry_run`

The utility goes through most of the actions it would ordinarily execute, but it does not import any files or other data to Teamcenter (optional). When used in conjunction with the –output option, an instruction file can be generated for use in a subsequent run of this utility.

`-folder` *folder*

Specifies the name of a Teamcenter folder, belonging to the user, into which new items will be placed (optional).

`-help` Prints usage information and exits.

`-log` *logfile* The absolute path to the log file to be written (optional). If not specified, the log file will be swimimport.txt in the directory where the Teamcenter Integration for SolidWorks is installed. Logging cannot be disabled and warning messages cannot be suppressed, but otherwise the logging preferences in swim.properties have their usual effect.

`-marker` *markername*

The Teamcenter server marker name or URI. If not specified, the default marker is that of the first server in the client_specific.properties file, found in the Teamcenter Rich Client installation directory. If URI includes "services/PLMGatewayService"; remove it. For example, "http://serverrname:7001/tc/services/PLMGatewayService" should be "http://servername:7001/tc/".

`-output` *instructionfile*

The absolute path to an instruction file to be written (optional). This file lists all of the models imported into Teamcenter.

`-overwrite`

Causes existing model files already in Teamcenter to be replaced if a new file is imported (optional). The default is to skip models that already exist in the latest revision.

`-p` *password* Specifies the password to use when connecting to Teamcenter. Ignored when -login is specified.

`-project` *project_id*

	Specifies the project to use when connection to Teamcenter (optional).
`-revise`	Causes a new revision to be created for each model imported into Teamcenter (optional). The default is to import into the latest revision, skipping any model if the latest revision already contains a version of the model.
`-u` *username*	Specifies the user name to use when connecting to Teamcenter. Ignored when -login is specified.

Item IDs, Names, Types, and Revisions

A model's item ID, name, type, and revision can be specified in an instruction file or by mapping parameters from the SolidWorks model to these attributes in Teamcenter. See the section on Instruction Files below for more information on how to do this in an instruction file, or refer to the section on Attribute Mapping for more information on setting up an attribute map. If the model's item ID, name, type, or revision is not specified, the following defaults apply.

- The default item ID is assumed to be the same as the model name. The model name is also used as the default item name. For example, if a part file widget.sldprt is imported into Teamcenter, the item ID and item name will both be widget.
- The default item type is determined by a combination of the model's CAD type and the first item type to appear in the allowed_item_types list in the dataset map within the swim.xml file. See the section on the map file for more details.
- The default revision is the latest revision of an item unless the -revise option is specified.

When several SolidWorks models have the same item ID, they will be imported into the same item. For example, if widget.slddrw contains a drawing of widget.sldprt and no other item ID is specified for either model, both will be stored in the item whose ID is widget.

Attributes

This utility will copy parameter values from SolidWorks models to attributes in Teamcenter. The attribute values that are copied are specified by the Teamcenter Integration for SolidWorks mapping file. The default mapping file is swim.xml in the Teamcenter Integration for SolidWorks installation directory.

If a model was skipped during the file import step because its item revision already has a dataset for the model, it will also be skipped during the step that copies attribute values. Specifying the –overwrite, or -revise option will ensure no models are skipped.

BOMs

BOMs will be created or updated by this utility. The types of models that are added to the BOMs, and the BOMs that are affected, are specified by the Teamcenter Integration for SolidWorks mapping file. The default mapping file is swim.xml in the Teamcenter Integration for SolidWorks installation directory.

BOMs are precise by default. This can be changed by setting the iman.precisebom preference, described in the swim.properties file in the Teamcenter Integration for SolidWorks installation directory.

If a model was skipped during the file import step because its item revision already has a dataset for the model, it will also be skipped during the step that updates BOMs. Specifying the -overwrite, or -revise option will ensure no models are skipped.

Configurations and Design Tables

By default, Teamcenter items, item revisions, and datasets are created only for SolidWorks documents, but this behavior can be changed to save the individual configurations found within a document, too. The sw.configurations.default option, described in the swim.properties file in the Teamcenter Integration for SolidWorks installation directory, specifies whether configurations should or should not be saved as separate entities in Teamcenter. Specific configurations may also be saved if their names are entered in instruction files (described below).

This behavior can be changed with the –all_configurations option. When specified, swimimport creates separate items for all configurations, except for those excluded by the sw.configurations.masters, sw.configurations.master.document.same, and sw.configurations.hide user preferences.

A dataset that represents a SolidWorks configuration is given the same name as the configuration. The dataset for a SolidWorks document is given the same name as the document.

Configurations created by a SolidWorks design table are handled in the same way as other configurations within a document. If the configurations are saved separately, by means of the sw.configurations.default option in swim.properties or by adding their names to instruction files, each design table member is treated as a model in its own right. An item, item revision, and dataset will be created for it, although the SolidWorks file will be stored only in the dataset for the document.

Toolbox Parts

The swimimport utility must be executed on a system with SolidWorks Toolbox installed and configured, in order for the sw2Tbx datasets to be created for the Toolbox parts. Toolbox parts cannot be imported directly, as top-level objects. They are only imported when used as components within an assembly, and that assembly is imported.

Auxiliary Files

The swimimport utility will create and import auxiliary files when importing models if an auxiliary file map exists. For more information on auxiliary files and the map file, refer to the section on The Map File.

If you do not want to import models, but only auxiliary files, consider using the import_file utility, which is included with Teamcenter. Note that the Teamcenter Integration for SolidWorks can be configured, via the auxiliary file map, to provide the item ID, revision, dataset name, dataset type, and named reference for each auxiliary file required when a model is saved to Teamcenter. This information can be used later to import auxiliary files with import_file.

Instruction Files

An instruction file is a text file that lists SolidWorks files or directories containing SolidWorks models. Each line in the file should contain the path to one file or directory. Blank lines and comments are also allowed. The comment character is #. Any text between the comment character and the end of the line will be ignored.

If the path to a file or directory contains spaces, it must be enclosed in double quotes. For example, the following path does not have to be quoted:

```
D:\Models\widget.sldprt
```

but this path must be quoted:

```
"D:\Sw  Models\widget.sldprt"
```

The path to a SolidWorks model may be followed by several optional arguments on the same line. Each argument has the form *keyword=value*, where *keyword* is one of the following:

configuration	The value is the name of a SolidWorks configuration. If this option is specified, this line in the instruction file indicates the configuration is to be treated as a separate model in Teamcenter, distinct from its document. Without this option, this line in the instruction file refers to the document.
dataset_desc	The value is the dataset description. This is ignored if the dataset already exists.
document	The value is the name of a SolidWorks document. This is the same as the base name of the file, without any extension or path. For example, the document name for the file widget.sldprt is widget. This option is redundant if the file name is given. It cannot be used to rename the model.
item_desc	The value is the item description. This is ignored if the item already exists.
item_name	The value is the item name. This is ignored if the item already exists.
item_id	The value is the item ID.
item_revision_id	The value is the revision level.
item_type	The value is the item type. This is ignored if the item already exists.
model_type	The value is the six-letter extension that identifies the SolidWorks model type. For example, the type of a model stored in the file widget.sldprt or widget.prt is sldprt. This option is redundant if the file name is given. It cannot be used to change the type of the model.
property[class[:type].name]	The property keyword is used to specify Teamcenter properties other than the keyword properties listed above. The *class[:type].name* string specifies the Teamcenter property. Refer pdm_name section under **Attribute Mapping** for details on *class[:type].name* syntax. If the property is an array, the value must be included in curly braces {} and comma separated. If required properties have been specified in the

BMIDE then the property key word may be required to use swimimport. If a value contains spaces, it must be enclosed in double quotes.

The SolidWorks file name does not have to be given if the document and model_type options are included on the line. Although these two options are not sufficient to identify the file to load, they can be used to refer to a model that is loaded implicitly or explicitly by a file specified elsewhere in the arguments given to swimimport. For example, suppose three different revisions of widget.sldprt are stored in directories c:\reva, c:\revb, and c:\revc, and a single instruction file c:\widget.txt has been created to specify the item ID to use for the model:

```
  document=widget model_type=sldprt item_id=184786
```

The swimimport utility can then be executed three times to load the revisions:

```
swimimport –u infodba –p infodba –revise c:\widget.txt c:\reva\widget.sldprt
swimimport –u infodba –p infodba –revise c:\widget.txt c:\revb\widget.sldprt
swimimport –u infodba –p infodba –revise c:\widget.txt c:\revc\widget.sldprt
```

In each case, the path to the model's file is specified on the command line. The model identified in the instruction file depends on the file that the utility loads.

Another simple instruction file is shown below.

```
# Sample swimimport instruction file
D:\jjs\models
D:\jjs\bolts\bolt.sldprt item_id=702283 item_desc="Bolt design table"
  document=bolt configuration=BOLT_A model_type=sldprt item_id=702284
  document=bolt configuration=BOLT_B model_type=sldprt item_id=702285
  document=bolt configuration=BOLT_C model_type=sldprt item_id=702286
```

The first line is a comment. The second line specifies a directory D:\jjs\models. The swimimport utility will load all SolidWorks models that it finds in that directory. The third line specifies a SolidWorks part file bolt.sldprt. Two optional arguments appear on this line to specify the model's item ID and, if the item has to be created, a description to include with the item.

In this example, suppose bolt.sldprt contains a design table. The user wants to specify the item ID where each configuration is to be stored, which is what the last three lines accomplish. Since the file name alone cannot identify the individual configurations, the last three lines use the document, configuration and model_type keywords to identify these models.

The instruction file below shows the property keyword.

```
D:\Work_Dir\A2.sldasm property[Item:Ipm9_AllPropItem.ipm9_dateArray]="{15-
Oct-2012 08:55,12-Jan-2012 05:14,25-Mar-2012 06:20}" "property[ItemRevision
Master:Ipm9_AllPropItemRevisionMaster.user_data_1]"="data1" "property[Item
Master.user_data_2]"="data2"
property[ItemRevision:Ipm9_AllPropItemRevision.ipm9_cascadingLOV]="Detroit"
D:\Work_Dir\P3.sldasm property[Item:Ipm9_AllPropItem.ipm9_dateArray]="{15-
Oct-2012 12:32,12-Jan-2012 05:14,25-Mar-2012 06:20}" "property[ItemRevision
Master:Ipm9_AllPropItemRevisionMaster.user_data_1]"="data1" "property[Item
Master.user_data_2]"="data2"
property[ItemRevision:Ipm9_AllPropItemRevision.ipm9_cascadingLOV]="Boston"
D:\Work_Dir\P4.sldasm property[Item:Ipm9_AllPropItem.ipm9_dateArray]="{15-
Oct-2012 14:55,12-Jan-2012 05:14,25-Mar-2012 06:20}" "property[ItemRevision
Master:Ipm9_AllPropItemRevisionMaster.user_data_1]"="data1" "property[Item
Master.user_data_2]"="data2"
property[ItemRevision:Ipm9_AllPropItemRevision.ipm9_cascadingLOV]="Mumbai"
```

Examples

The following example shows a simple use of this import utility.

```
swimimport -u infodba -p infodba c:\SwMdls
```

This command will load the latest version of each SolidWorks model from the c:\SwMdls directory into Teamcenter, but it will skip any model that already has a dataset in Teamcenter. For the models that are imported, new items, item revisions, and datasets will be created as needed, attribute values will be copied to Teamcenter and BOMs will be created or updated.

To force all models to be imported, use the –overwrite option:

```
swimimport -u infodba -p infodba –overwrite c:\SwMdls
```

For items that already exist, the models will be imported into each item's latest revision, replacing SolidWorks model files that may have been imported previously.

If a new item revision should be created for those items that already exist, the import utility could be used this way:

```
swimimport -u infodba -p infodba -revise c:\SwMdls
```

This command carries out the same operations as in the previous example, including creation of new items for models that have not been imported previously, but new revisions are created for existing items.

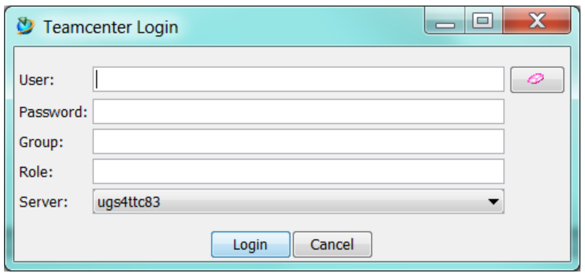
Importing SolidWorks Models into Teamcenter with bulk import

The swimbulkimport utility saves SolidWorks models into Teamcenter, directly from a disk location. This interactive utility is a simplified version of the **Save** dialog, so users will find it intuitive. The bulk import utility is intended to simplify the import of relatively small and self-contained sets of data. Items, Item Revisions, and Datasets may be created, property values may be copied to attributes in Teamcenter, and BOMs may be updated.

Usage

To start the bulk importer, double click on swimbulkimport.bat in the SWIM installation directory.

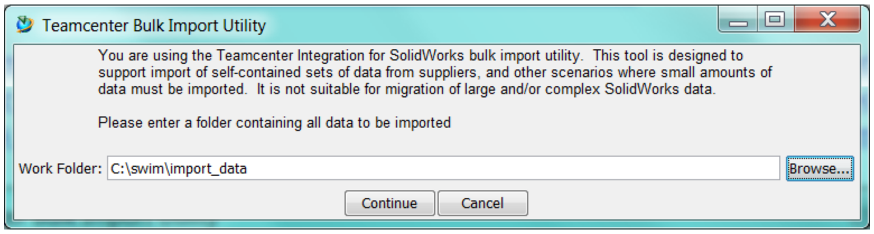
When the **login** dialog appears, enter user credentials and other information as desired.



Server: ugs4ttc83

Figure 3 - Login dialog for bulk importer

Next, the user must select the work folder containing all of the required SolidWorks models. All models to be importered must be in the selected folder. The only exception is toolbox parts. The toolbox files must be available from the folders defined by sw.toolbox.dirs preference in swim.cfg.


Teamcenter Bulk Import Utility

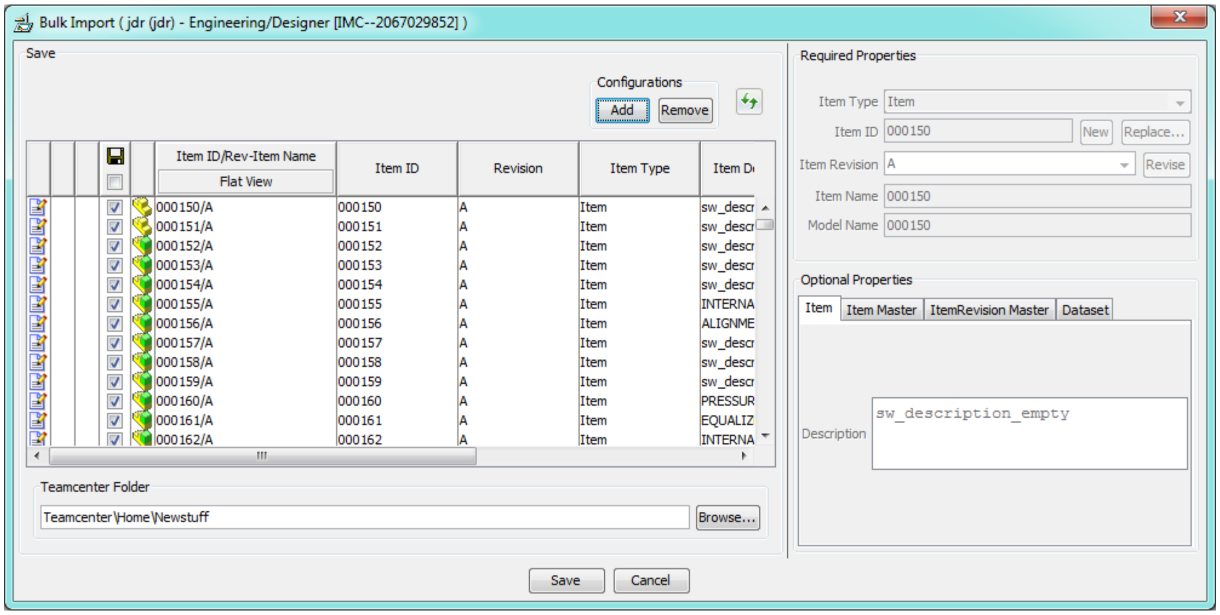
You are using the Teamcenter Integration for SolidWorks bulk import utility. This tool is designed to support import of self-contained sets of data from suppliers, and other scenarios where small amounts of data must be imported. It is not suitable for migration of large and/or complex SolidWorks data.

Please enter a folder containing all data to be imported

Work Folder: C:\swim\import_data

Figure 4 - Folder selection dialog for bulk importer

At this point, the **Bulk Import** dialog appears. The dialog is similar to the **Save** dialog in presentation and function. It is possible to assign Item IDs, set revisions, itentify models that already exist in Teamcenter, and assign required and optional property values. For more information on how to use the dialog, review the **Save dialog** documentation in the **User Guide**.


Bulk Import (jdr (jdr) - Engineering/Designer [IMC--2067029852])

Item ID/Rev-Item Name Flat View	Item ID	Revision	Item Type	Item D
000150/A	000150	A	Item	sw_descr
000151/A	000151	A	Item	sw_descr
000152/A	000152	A	Item	sw_descr
000153/A	000153	A	Item	sw_descr
000154/A	000154	A	Item	sw_descr
000155/A	000155	A	Item	INTERNA
000156/A	000156	A	Item	ALIGNME
000157/A	000157	A	Item	sw_descr
000158/A	000158	A	Item	sw_descr
000159/A	000159	A	Item	sw_descr
000160/A	000160	A	Item	PRESSUR
000161/A	000161	A	Item	EQUALIZ
000162/A	000162	A	Item	INTERNA

Teamcenter Folder: Teamcenter\Home\Newstuff

Required Properties

Item Type: Item
Item ID: 000150
Item Revision: A
Item Name: 000150
Model Name: 000150

Optional Properties

Description: sw_description_empty

Figure 5 - Bulk import dialog

Selecting the Save button will import all models into Teamcenter.

Configuration

The SWIM bulk importer works like the save operation. The configuration settings for the **Save** dialog apply to the bulk importer. Review save (checkin) settings in swim.xml, swim.cfg, and swim.properties.

Limitations

The SWIM bulk importer does not use a session of SolidWorks. Instead it uses the SolidWorks document manager API to rapidly interrogate and modify files. Consequently there are some limitations with the bulk importer:

- All files must be in the selected directory. The only exception is toolbox parts, which must be in the directory specified by swim.cfg configuration sw.toolbox.dirs.
- SolidWorks 2004 and earlier files cannot be imported – These old files cannot be read. The old files are detected and user will be alerted to open and save models with newer version of SolidWorks.
- The import process modifies files in such a way that imported files cannot be reused by another bulk import session. It is best to start with a fresh directory of files for each bulk import.
- The bulk importer does not store length units of measure in the Teamcenter BOM – The SolidWorks document manager API does not support unit of measure access. As a result, assembly visualization in the Teamcenter **Structure Manager** may be incorrect until the assembly is saved interactively. SPR459029 has been raised with SolidWorks.
- Assembly configurations which are renamed by the document manager API as part of the import process must be opened and saved to disk, from SolidWorks, before they can be imported again. The most common scenarios in which this limitation may be encountered are:
    - The user renames an assembly configuration (via the **New** or **Replace** buttons) within the bulk import dialog, and then decides to cancel the operation. He must open the renamed models in SolidWorks and save them back to disk, before attempting to import them again.
    - The user renames an assembly configuration as described in the last bullet, as part of a complete import operation. He then exports the model from Teamcenter to a disk location, without opening it in a SolidWorks session. He then attempts to bulk-import the exported model under a new Item ID, as part of a clone operation.

  The first step could also occur another application based on the document manager modified the files. The bulk importer detects the renamed configurations and alerts the user to open and save models interactively from within a SolidWorks session.

Renaming SolidWorks Models in Teamcenter with swimrename

The swimrename utility renames SolidWorks models and their datasets in Teamcenter. This non-interactive utility will check out a model from Teamcenter, open it in SolidWorks, rename the model and save it back to Teamcenter. It also checks out, updates and saves all of the SolidWorks models that use the renamed model, and optionally changes the item ID and the names of other objects to match the new name.

Usage

swimrename [**-login**] [**-u** *username* **-p** *password* [**-g** *group*]] [**-help**]

```
[-project project_id]
-item itemid [-rev revision] -type type
-old_name oldname -new_name newname -work_dir directory
[-no_autorename]
[-autorename_item_id true|false]
[-autorename_item_name true|false]
[-autorename_other true|false]
[-autorename_other_exact_length true|false]
[-marker markername]
[-progress_file outputfile] [inputfile]
```

Description

Before running the swimrename utility, all models that will be renamed and all of the models that use them must be checked-in to Teamcenter. The swimrename utility will stop with an error if it finds that a model to be renamed, or a model that uses a renamed model, is already reserved.

A log file named swimrename.txt is always produced containing information about the actions carried out during the renaming process. This log is located in the Teamcenter Integration for SolidWorks installation directory.

The arguments for swimrename are described below.

-autorename_item_id *true | false*
: Specifies whether the item ID should be changed to match the model name (optional). If true, the item ID is changed to the new model name if the old item ID matches the old model name, ignoring case. If false, the item ID is not changed. Specifying -no_autorename also sets this option to false. This option can follow -no_autorename on the command line to override -no_autorename. If not specified, the default is given by the iman.autorename.itemid user preference (see the swim.properties file in the installation directory for a description of user preferences).

-autorename_item_name *true | false*
: Specifies whether the item name should be changed to match the model name (optional). If true, the item name and item revision name are changed to the new model name if their old names match the old model name, ignoring case. If false, the names are not changed. Specifying -no_autorename also sets this option to false. This option can follow -no_autorename on the command line to override -no_autorename. If not specified, the default is given by the iman.autorename.itemname user preference (see the swim.properties file in the installation directory for a description of user preferences).

-autorename_other *true | false*
: Specifies whether the names of other objects within the item or item revisions should be changed to match the model name (optional). Examples of “other” objects include forms, BOM views, and non-SolidWorks datasets and their named references, i.e. any Teamcenter

object in an item or item revision that is not a SolidWorks dataset. When true, these other objects are renamed to the new model name if their old names match the old model name, ignoring case (the -autorename_other_exact_length option determines how the names are matched). When false, the names are not changed. Specifying -no_autorename also sets this option to false. This option can follow -no_autorename on the command line to override -no_autorename. If not specified, the default is given by the iman.autorename.other user preference (see the swim.properties file in the installation directory for a description of user preferences).

`-autorename_other_exact_length` *true | false*
Specifies how to match names when deciding whether to rename other objects in the item or item revisions (optional). When true, the name of a Teamcenter object matches the SolidWorks model name if both names match character-for-character, ignoring case, and are exactly the same length (for named references, the file name without extension is tested). When false, the names match if the object's name starts with the model name, ignoring case. For example, suppose the model's name is WIDGET. If -autorename_other_exact_length is true, an item master form named WIDGET will match the model name, but not an item revision master form named WIDGET/A. If -autorename_other_exact_length is false, both forms match the model's name. This option only affects the "other" objects in the item and item revisions when -autorename_other is true. Exact length is always used when matching the model name to the item ID, item name, or item revision name. If not specified, the default is given by the iman.autorename.other.exactlength user preference (see the swim.properties file in the installation directory for a description of user preferences).

`-g` *group* The Teamcenter group (optional). Ignored when -login is specified.

`-help` Prints usage information and quits (optional).

`-item` *itemid*
The item ID of the model to be renamed.

`-login` Uses the Teamcenter login dialog. When this option is specified, the -u, -p, -g, and -marker options are ignored.

`-marker` *markername*
The Teamcenter server marker name or URI. If not specified, the default marker is that of the first server in the client_specific.properties file, found in the Teamcenter Rich Client installation directory. If URI includes "services/PLMGatewayService"; remove it. For example, "http://serverrname:7001/tc/services/PLMGatewayService" should be "http://servername:7001/tc/". Ignored when –login is specified.

-new_name *newname*
The new name for the SolidWorks model. Do not include the file extension.

-no_autorename
Turns off automatic renaming of the item, item revisions, and other objects (optional). This is the same as setting -autorename_item_id, -autorename_item_name, and -autorename_other to false.

-old_name *oldname*
The old name for the SolidWorks model. Do not include the file extension.

-p *password* The Teamcenter password. Ignored when -login is specified.

-project *project_id*

Specifies the project to use when connection to Teamcenter (optional).

-progress_file *outputfile*
The absolute path to a text file that this program will create to record its progress (optional). If an error occurs, the program can be restarted using the file as the *inputfile* argument.

-rev *revision*
The item revision ID of the model to be renamed (optional). If not specified, all revisions of the model will be renamed. Use of this option should be avoided, except in unusual circumstances. In most cases, all revisions of a model should be renamed. This is particularly true if the model occurs in any imprecise BOMs.

-type *type* The six-letter SolidWorks file extension that identifies the type of the model, such as sldasm. Alternatively, this may also be the dataset type, such as SWAsm.

-u *username* The user's Teamcenter login name. Ignored when -login is specified.

-work_dir *directory*
The absolute path to a temporary working directory. This directory will be created if it does not exist. If the directory does exist, it must be empty. You will usually want to use a directory that is not the same as the directory where you start the swimrename utility.

Examples

Suppose an assembly named "cylinder" is stored in an item whose item ID is 0000753, and it is decided to rename the model to match the item ID. The command to execute this change might look something like this:

```
swimrename -u infodba -p infodba -item 0000753 \
   -type sldprt -old_name cylinder -new_name 0000753 \
   -work_dir d:\tmp -progress_file c:\restart1.txt
```

The –rev option is not specified, so all revisions of cylinder in item 0000753 are renamed to 0000753. This is recommended if there is a possibility that item 0000753 occurs in the imprecise BOMs of other assemblies. A directory d:\tmp is used as a temporary space for SolidWorks to work on these models, and a file c:\restart1.txt will be created in case the command aborts before all of the models are renamed.

Suppose an assembly that uses cylinder is currently checked out by someone else. The swimrename utility will stop with an error because it will not be able to check out that other assembly, but this may occur after swimrename has already renamed and saved several revisions. Since the –progress_file option is used in the example above, swimrename can be restarted once the other assembly becomes available. The command to execute swimrename is the same as before, but the output file created with the –progress_file option above is now given as an input to swimrename:

```
swimrename –u infodba –p infodba –item 0000753 \
   –type sldprt –old_name CYLINDER –new_name 0000753 \
   –work_dir d:\tmp –progress_file c:\restart2.txt  c:\restart1.txt
```

Note that this command also creates a new text file, c:\restart2.txt, which can be used to restart swimrename again if another error occurs.

Suppose it is later decided to rename part 0000753 to 5000753 after a drawing named 0000753 has been added to the same item. The part and the drawing should both be renamed, but you must run swimrename twice to do this. Either could be renamed first, but it may be desirable to avoid changing the item ID until both models are renamed. The two runs of swimrename might look like this:

```
swimrename –u infodba –p infodba –item 0000753 \
   –type sldprt –old_name 0000753 –new_name 5000753 –no_autorename \
   –work_dir d:\tmp –progress_file c:\restart.txt

swimrename –u infodba –p infodba –item 0000753 \
   –type slddrw –old_name 0000753 –new_name 5000753 \
   –work_dir d:\tmp –progress_file c:\restart.txt
```

Notice that -no_autorename is used in the first run, when renaming the part. This prevents the item ID from changing and allows the second run to use almost the same arguments to rename the drawing, except for changing the -type argument and dropping -no_autorename. At the end of the second run, swimrename will change the item ID to 5000753.

Best Practice Recommendations

Properties display in Save Dialog and Teamcenter New Dialog

The **Save** and **Teamcenter New** dialogs have two panels of properties, **Required Properties** and **Optional Properties**. By default, the **Required Properties** panel contains only the system properties required by the SolidWorks integration. These system properties are Item Type, Item ID, Item Revision and Item Name. On the other hand, the **Optional Properties** panel's tabs are populated with properties that have Teamcenter *create descriptors*. By default only a few properties have *create descriptors*.

By the use of Teamcenter *create descriptors*, sites can change the contents of panels. Using BMIDE to specify *create descriptors* on additional properties will add those properties to the

Optional Properties tabs. Setting a *create descriptor* to required in BMIDE will cause the property to appear in the **Required Properties** panel. All of the required properties must have a value before the model can be saved for the first time.

Additional Notes:

- *Create descriptors* can differ between Item Types
- A property with a required create descriptor will only appear in the **Required Properties** panel.
- Models already existing in Teamcenter without values for required properties can be saved without the user setting the values – the required *create descriptors* are only enforced upon first-time creation of an item.
- Overuse of required *create descriptors* could cause users to defer saving work in progress, and so they should be used selectively.

Configurations

Integration preferences related to configurations

Every SolidWorks part or assembly has one or more configurations. Not all configurations are used the same way. The integration provides several preferences intended to help customers manage and resolve various types of conflicts and inconsistencies in their SolidWorks data:

- Set sw.configuration.default = all. The idea being that all active or referenced configurations are intended to be managed.
- Populate sw.configurations.hide with configuration names that should never be managed. Establish site practices for designers to use unmanaged configuration names when they create representations, this will prevent unwanted configurations from being saved into Teamcenter. It can be difficult to remove data from Teamcenter, so it is best to establish practices that avoid creating the data in the first place. Alternatively, consider using sw.configurations.keep to control which configurations may be managed in Teamcenter. The choice of which preference to use depends largely upon how easily identifiable the SolidWorks data is, by naming convention.
- Set sw.configurations.master.document.same = true, to handle configurations with the same name with the document. Data such as this often comes from Toolbox (see Toolbox documentation elsewhere in this document) or suppliers. In such cases the document usually contains only the single configuration and no other “Master Configurations”, thus managing the configuration with the document is desired.

SolidWorks External References

SolidWorks creates an external reference between documents when the parent document depends on the feature definition of the child document. When the child document is modified, the parent document is modified too. The integration detects external references when saving the data to Teamcenter. However, if a user renames the child in SolidWorks or the integration **Save** dialog the external reference will not be updated. The relationship will be broken. To “save-as” data with external references, use the SolidWorks **Pack and Go** utility, then import the renamed data into Teamcenter via interactive **Save** or **Bulk Import**. Another option is to disable external references by selecting “Do not create references external to the model” from SolidWorks **Tools, Options, System Options External References** dialog.

SolidWorks Toolbox

SolidWorks toolbox is a library of fasteners, nuts, bolts, screws, bearings, and more. The toolbox is highly integrated into SolidWorks. SolidWorks enables the sizing of the components at the time of instantiation into SolidWorks assemblies. The toolbox is a powerful tool, and its files are best not managed by Teamcenter, but left to SolidWorks to manage. On the other hand, Teamcenter requires Items to exist for the toolbox parts, for life cycle management within Teamcenter.

To enable Teamcenter knowledge of the toolbox parts, the SolidWorks integration to Teamcenter creates Items, Item revisions, and Datasets for toolbox configurations, but does not store the files. A sw2Tbx dataset type is used to represent the toolbox parts. The associated items appear in the BVR of parent assemblies. Toolbox parts are represented by the icon in the integration dialogs.

SolidWorks SpeedPak

Speedpak creates a simplified configuration of an assembly to improve assembly performance. Another benefit of SpeedPak is assembly sharing. The integration does not fully support SpeedPak configurations. With this release, it is best if SpeedPak configurations are not managed with Teamcenter. One means to avoid saving SpeedPak configurations is to set sw.configuration.hide = *speedpak. When using this setting, if a SpeedPak configuration is active at the time of save, all references to child components will be lost.

SolidWorks Virtual Components

Virtual components are internal to the SolidWorks assembly file instead of in separate subassembly or part files. The integration hides virtual components from the user. The child relationships of a virtual component are assumed by the parent assembly. JT files cannot be generated for virtual components. When a virtual component is made external the integration will manage the external copy. When an external component is made virtual the integration will hide the internal copy. Once a component is managed by the integration the recommendation is to not virtualize the component as changes to the virtual component will not be reflected in Teamcenter.

Multisite

Upgrading the Teamcenter database for multi-site

Upgrading the database is only required for sites that have saved data with integration version 9.1.0 or earlier. **Sites that have only deployed version 9.2.0 and later do not need to run the utility**. The upgrade adds SW2_configuration relations into the database. These relations are required to enable complete multi-site transfer of a document and all related configurations. Starting with integration version 9.2.0 of the integration, the SW2_configuration relations are created by the integration operations.

Usage

```
swim_upgrade_database -u user -p password [-g group]
  -mode=SINGlE|FULL [-input_file=inputfilename]
  [-statistics]
  [-dryrun]
  [-batch_size=number of records to process]
  [-debug=filename]
  [-log=filename]
  [-help]
```

Description

The upgrade utility creates SW2_configuration GRM relations in the database where they do not already exist. The SW2_configuration relations are most important to customers who transfer SolidWorks data via Teamcenter multi-site. Failure to run the utility could result in duplicated configuration items between sites.

The utility cannot create SW2_configuration GRM relations for all cases. For instance, if there is more than one dataset as the secondary of a SWIM_master_dependency relation, the utility will report the relation and not create a SW2_configuration relation. The skipped relation will be reported and mode=SINGLE can be used to fix to add the relation.

The swim_upgrade_database utility must be run on the server and must be run by a database administrator.

-batch_size=*number of record to process*
: Optional argument that specifies the number of relations to process at a time. The default is 5000. Too large a value could result in memory failures.

-debug=*filename*
: An optional argument to specify the filename and path to record debug information.

-dryrun
: When specified, the utility does not create the relations. All other utility actions are performed. Run in dryrun mode to determine if there are going to be problems.

-g=*group* Optional argument to specify the administrators group.

-help Prints usage information and quits (optional).

-input_file=*inputfilename*

This argument is required only if the user has specified –mode=SINGLE. The input file consists of lines that contain the following fields, separated by spaces. If a field itself contains a space, enclose it with quotes (The error statements in the log file from FULL mode may be used as a template for the property syntax and formatting):

```
Document_itemid document_revid document_dataset document_dataset_type config_itemid
config_revid config_dataset config_dataset_type
```

-log=*filename*

An optional argument to specify the filename and path where the utility records relations that it failed to process.

-mode=*SINGLE|FULL*

A required argument. SINGLE is to process a list of relations, and FULL is to process the entire database.

-p=*password* The Teamcenter password.

-statistics

An optional argument that outputs the count of each GRM relation the utility queries. No other processing will occur when this option is specified.

-u *username* The user's Teamcenter login name.

Configuring Teamcenter Multi-Site to Export SolidWorks Models

If you are using Teamcenter Multi-Site to export items containing SolidWorks datasets, you must include the following relation types in the references to be exported:

```
IMAN_master_form
IMAN_Rendering
IMAN_specification
TXD_long_name_relation
```

You should add these types to your TC_relation_required_on_export site preference. These additional types should be included when exporting models with the other SolidWorks models on which they depend:

```
SWIM_dependency
SWIM_mandatory_dependency
SWIM_master_dependency
SWIM_suppressed_dependency
SW2_configuration
```

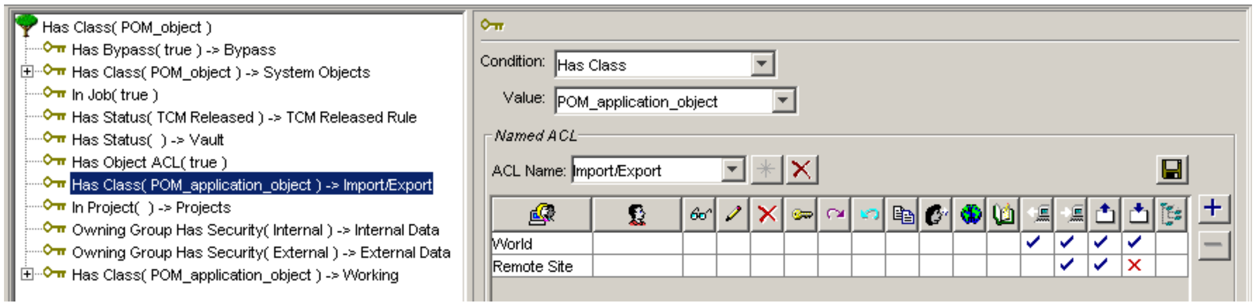
When exporting items using the **Tools | Export | Objects** command in the Teamcenter Portal's Navigator application, or when importing items with the **Tools | Import | Remote** command, remember to include the relation types above in the references shown in the **Advanced** tab of the **Export Preferences** dialog or **Import Remote Options** dialog, respectively.

Configuring Teamcenter Multi-Site for Transferring Ownership of SolidWorks Models

If users are allowed to transfer ownership of SolidWorks models from remote sites to your site, first configure Multi-Site to replicate items containing SolidWorks models, remembering when exporting and importing items to include the relation types described in the previous section.

At the site that owns the items you want to transfer, make sure your site is included in the list of sites defined by the IDSM_permitted_transfer_sites Teamcenter preference.

The site that owns the items must also grant the TRANSFER_OUT privilege for objects of the POM_application_object class. For example, in the Teamcenter Rich Client's **Access Manager**, this privilege might be granted by modifying the default Import/Export ACL as shown in Figure 10.



```
Has Class( POM_object )
  Has Bypass( true ) -> Bypass
  Has Class( POM_object ) -> System Objects
  In Job( true )
  Has Status( TCM Released ) -> TCM Released Rule
  Has Status( ) -> Vault
  Has Object ACL( true )
  Has Class( POM_application_object ) -> Import/Export
  In Project( ) -> Projects
  Owning Group Has Security( Internal ) -> Internal Data
  Owning Group Has Security( External ) -> External Data
  Has Class( POM_application_object ) -> Working
```

Condition: Has Class

Value: POM_application_object

Named ACL

ACL Name: Import/Export

World												✓	✓	✓	✓	
Remote Site													✓	✓	✗	

Figure 6 ACL Granting TRANSFER_OUT Privilege

Before starting SolidWorks, you may want to add a column to the Teamcenter Integration for SolidWorks dialogs so users can see the name of the site that owns each SolidWorks model. Do this by adding the site keyword to the list of columns defined by the table.columns.std user preference in the swim.properties file. This user preference and others are described in the swim.properties file, which can be found in the installation directory.

If you chose to add the **Transfer Ownership** command during installation, the command will appear in the shortcut menu when the user right-clicks on a model in the Teamcenter Integration for SolidWorks dialogs. If this command does not appear in the shortcut menu, run the installer again, choosing a Custom installation with only the Configure Client for Multi-Site option selected. Refer to the **User Guide's** section on Using the Shortcut Menu for more information on the **Transfer Ownership** command.

Configuring the Teamcenter Integration for SolidWorks for Performance

You may want to consider the following configuration changes to improve the Integration's performance.

- Set the iman.allrelations user preference to false in the swim.properties file, located in the installation directory. If inaccessible dependencies can be ignored, Open operations may be faster. However, you should also set checkin.ignoremissing to prompt or never to prevent models from being saved when some of their dependencies in Teamcenter are inaccessible. For example, you may want to add these lines to your swim.properties file:

  ```
  iman.allrelations = false
  checkin.ignoremissing = prompt
  ```

- If using Teamcenter 9.1 and later, set the iman.metacache.enabled user preference to true in the swim.properties file located in the installation directory. When set to true the integration will automatically check if the cache is up to date at time of login. If not, the cache will be updated. To enable the integration to use the Teamcenter metacache, you may add the following line to your swim.properties file:

  ```
  iman.metacache.enabled = true
  ```

In addition to the changes described above, best performance is usually obtained when using a two-tier installation of the Teamcenter Rich Client. Performance with four-tier installations will be slower than with a two-tier installation, when the clients and the server are on the same LAN. If the clients and the server are separated by a WAN with significant network latency, then four-tier is likely to be faster and more scalable than two-tier.

Restricted access to data

While opening CAD data from Teamcenter, and while saving CAD data to Teamcenter, the integration identifies all models which are related to the top-level structured model (drawing, assembly, and configuration). During Open and Save, Teamcenter relationships are expanded and traversed. During Save, the CAD session is also interrogated to read its network of dependency relations.

While expanding and traversing these relations, Teamcenter may encounter one or more components which are inaccessible in one way or another. Common scenarios are:

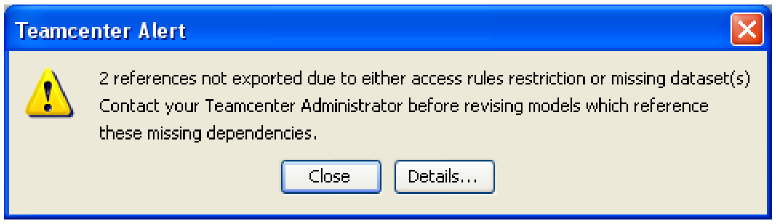
- The item, item revision, dataset, or some combination of these, has been assigned to a project to which the CAD user does not have read access.
- The item, item revision, dataset, or some combination thereof, has an ACL attached, or is owned by a security-enabled group, that explicitly or implicitly denies read access to the CAD user.
- Any of the above objects was imported into the local database via MultiSite import, and has since been deleted, leaving a stubbed reference in the local Teamcenter database. Teamcenter allows deletion of replica data even though it is referenced by higher-level objects in the local database. When that replica data is deleted, the reference remains.
- The user is saving an assembly which has suppressed components, and those suppressed components have never existed in Teamcenter. This can occur when the components

remain suppressed the first time the assembly is saved to Teamcenter. Because they are not available in session, the integration cannot save them to Teamcenter as items[6].

Because inaccessible models may result from deliberate customer security practices, such as assignment to secure projects, the integration does not consider them an error condition. Instead, it informs the user about the situation during both **Open** and **Save** operations.

Opening a model with access restrictions

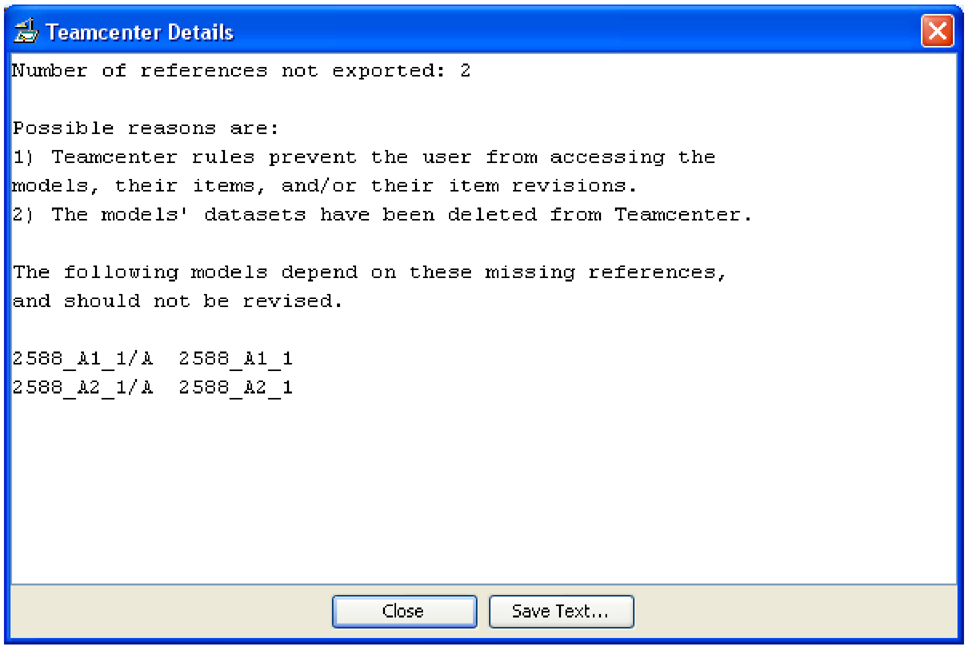
When a model with restricted components is opened from Teamcenter, the integration displays an informational dialog, stating how many inaccessible dependencies were encountered. Depending upon the user's access to the objects involved in the operation, additional details such as Item ID and model name may also appear in the dialog:


Teamcenter Alert

2 references not exported due to either access rules restriction or missing dataset(s)
Contact your Teamcenter Administrator before revising models which reference
these missing dependencies.

The content of the secondary **Details** dialog will vary, based upon whether or not the user has DBA privileges. For a non-DBA user, details about the restricted objects are omitted, and the following is displayed:

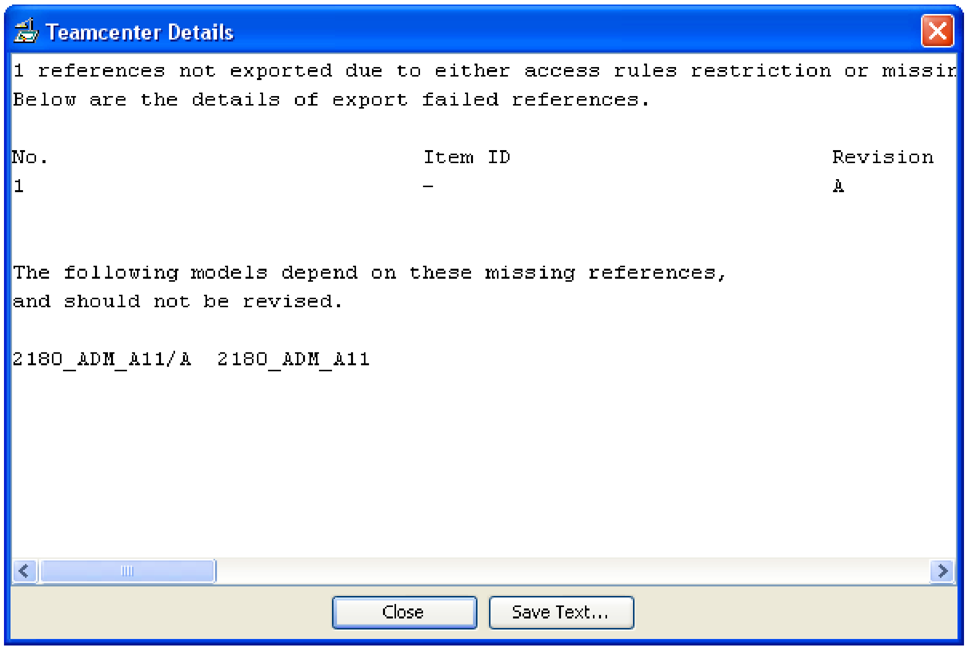
[6] This situation can be prevented by setting checkin.ignoremissing = never in swim.properties. This is not the default setting, because it is very common for customer data to contain "ghost references", which are pointers to non-existent, non-required dependencies. With the default setting of "prompt", the integration warns the user about these missing references, but does not prevent him from completing the save.



```
Number of references not exported: 2

Possible reasons are:
1) Teamcenter rules prevent the user from accessing the
models, their items, and/or their item revisions.
2) The models' datasets have been deleted from Teamcenter.

The following models depend on these missing references,
and should not be revised.

2588_A1_1/A  2588_A1_1
2588_A2_1/A  2588_A2_1
```

Figure 7 - Inaccessible models warning dialog, user level

A DBA user will see additional information about the identity of these missing models. For example[7]:



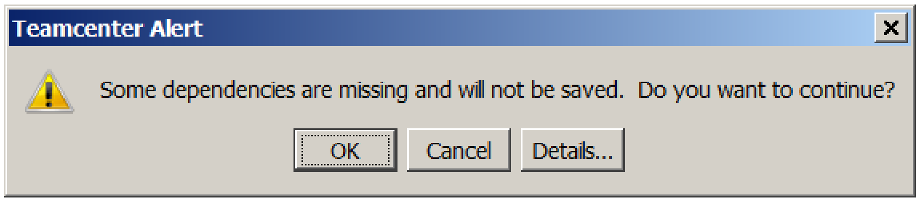
```
1 references not exported due to either access rules restriction or missir
Below are the details of export failed references.

No.                              Item ID                         Revision
1                                -                               A


The following models depend on these missing references,
and should not be revised.

2180_ADM_A11/A  2180_ADM_A11
```

Figure 8 - Inaccessible models warning dialog, admin level

[7] In this example, the item ID cannot be displayed because the access restriction is at the item level. If the item were accessible, but the item revision and dataset were not, then the item ID would be reported.

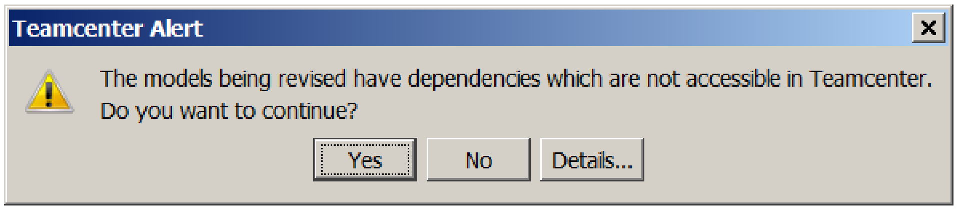
Saving a model with access restrictions

The integration does not display any special warnings when the assembly or drawing is checked out and saved as a new dataset version. There is no risk of lost references in this case; the integration simply ignores relationships to the inaccessible objects and does not alter them. The following dialog will appear, but this is a general informational dialog that appears in several other contexts. It is not specific to restricted object scenarios:


Teamcenter Alert

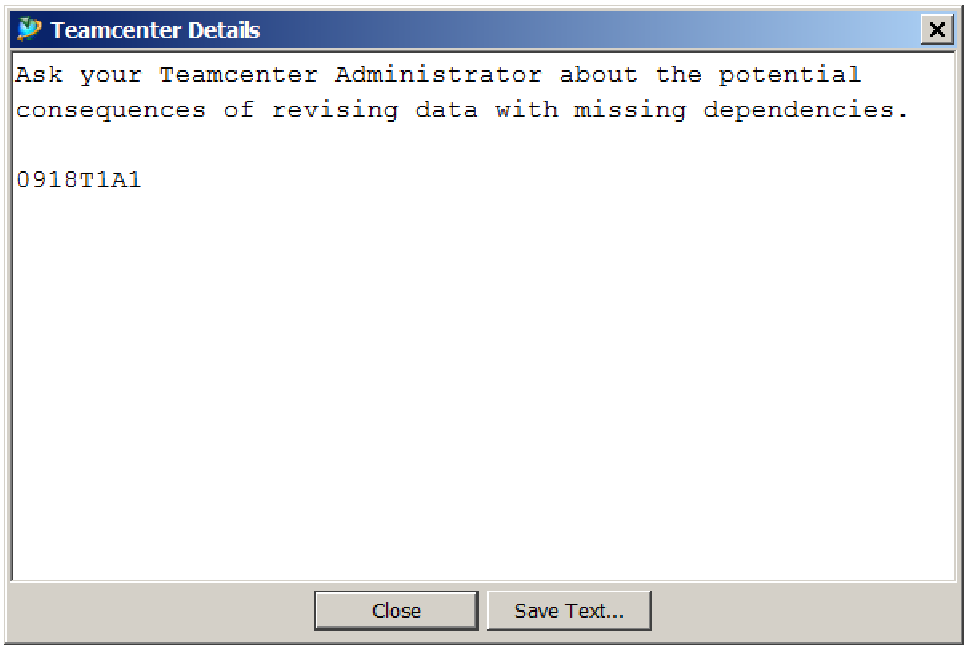
Some dependencies are missing and will not be saved. Do you want to continue?

When the model is being saved as a new revision, however, the integration displays a warning dialog:


Teamcenter Alert

The models being revised have dependencies which are not accessible in Teamcenter.
Do you want to continue?

The **Details** button will display a secondary dialog that identifies the model(s) which have dependencies on the inaccessible objects, and should therefore not be revised unless the user, or the Teamcenter Administrator, has judged it safe to do so:

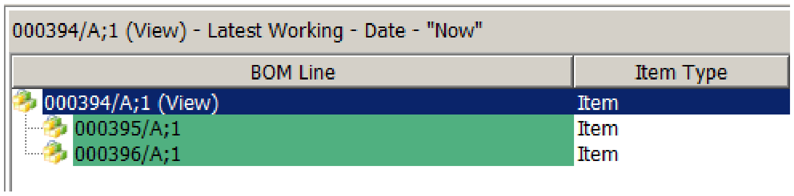

Teamcenter Details

```
Ask your Teamcenter Administrator about the potential
consequences of revising data with missing dependencies.

0918T1A1
```

Figure 9 - Revising a model with inaccessible dependencies

Proceeding with the Revise operation will usually result in an inconsistency between the Teamcenter Bill of Materials and the integration's CAD-specific dependency relationships, as shown in this example, where the component "000396" was inaccessible to the CAD user who performed the Revise:

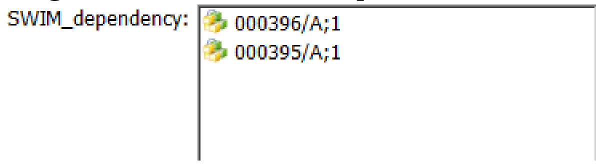
Teamcenter Bill of Materials before Revise:



000394/A;1 (View) - Latest Working - Date - "Now"

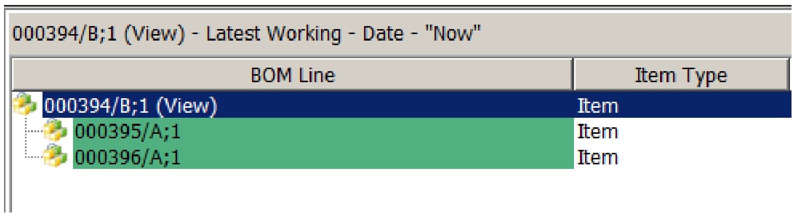
BOM Line	Item Type
000394/A;1 (View)	Item
000395/A;1	Item
000396/A;1	Item

Integration CAD relationships before Revise:



SWIM_dependency:
- 000396/A;1
- 000395/A;1

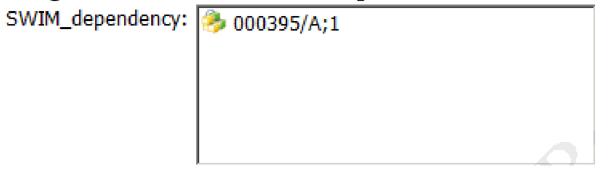
Teamcenter Bill of Materials after Revise:



000394/B;1 (View) - Latest Working - Date - "Now"

BOM Line	Item Type
000394/B;1 (View)	Item
000395/A;1	Item
000396/A;1	Item

Integration CAD relationships after Revise:



SWIM_dependency:
- 000395/A;1

Figure 10 - Result of revising a model with inaccessible dependencies

The integration cannot create or maintain a CAD relationship between the new assembly revision and the inaccessible component, because it cannot access it in the CAD session or in Teamcenter. Teamcenter itself propagates the relationship to the new revision's Bill of Materials, however.

Troubleshooting

General guidelines

Logging

- Sometimes problems must be sent to Siemens GTAC for analysis. We will need a txdlog.txt file to better understand the issue. To generate this file, add the following lines to the beginning of the swim.properties file, on the client:

  ```
  log.enable = true
  log.file = c:\\temp\\txdlog.txt
  log.suppress = 10000
  ```

  The SolidWorks session must be restarted after making this change, in order for the logging options to take effect.

- Each integration session will also produce a client_socket.log file. This file is written to the integration's installation directory on each client.

Reporting problems

- When submitting a case to GTAC, include enough detail to enable the assigned analyst to reproduce the problem. This always requires a full description of the test case and data involved, and all available log files. The minimum set required are the txdlog.txt file and client_socket.log file mentioned above, but when Teamcenter server errors are involved, GTAC will also need the tcserver syslog file.

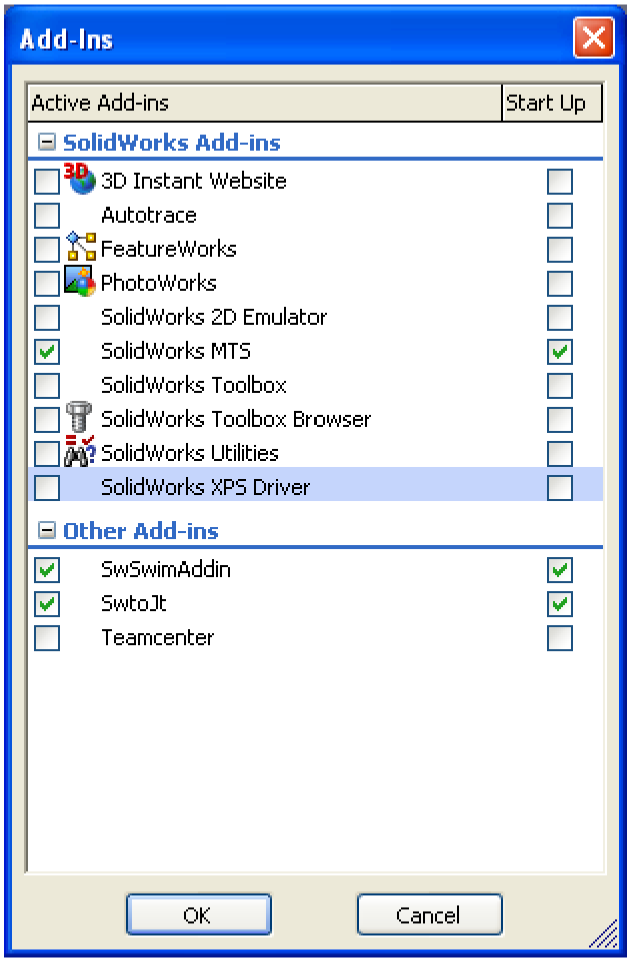
  It is very important to provide all necessary information with the initial submission to GTAC. Incomplete or ambiguous information will lead to unnecessary delay in resolving your problem.

Integration launch

If you have trouble starting SolidWorks, or if SolidWorks starts, but no Teamcenter Integration for SolidWorks sidebar tab appears:

- Make sure the minimum required build of SolidWorks is installed. See the Installation Guide section on "Prerequisites for the Teamcenter Integration for SolidWorks" for more information.
- If the Teamcenter Integration for SolidWorks sidebar tab does not appear in SolidWorks, make sure the SwSwimAddin is enabled. In SolidWorks, select **Tools | Add-Ins** to open the Add-Ins dialog. Make sure the dialog shows the SwSwimAddin in this list, and put a check next to its name. If the dialog does not show the SwSwimAddin, it may be necessary to repeat the client installation or you can double click on %SWIM_DIR%/bin/ RegisterSwimAddin.bat, to re-register the integration with SolidWorks.

- Customers who had previously installed an older version of the integration (prior to version 8.1.0), may need to remove the older version's add-in. This older add-in will be labeled as "Teamcenter" in the SolidWorks dialog, as shown below:

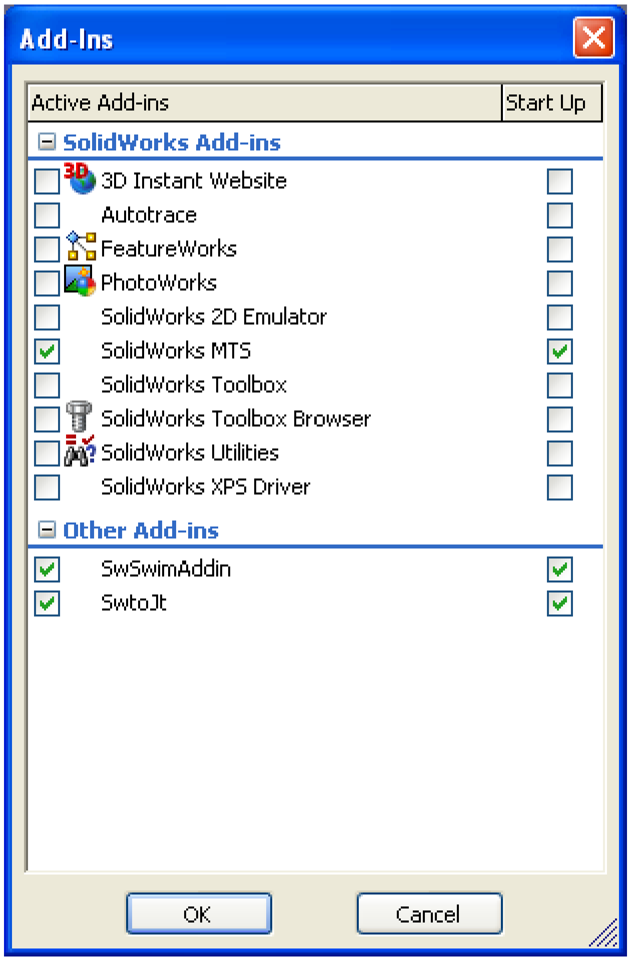


Add-Ins

Active Add-ins	Start Up
SolidWorks Add-ins	
☐ 3D Instant Website	☐
☐ Autotrace	☐
☐ FeatureWorks	☐
☐ PhotoWorks	☐
☐ SolidWorks 2D Emulator	☐
✓ SolidWorks MTS	✓
☐ SolidWorks Toolbox	☐
☐ SolidWorks Toolbox Browser	☐
☐ SolidWorks Utilities	☐
☐ SolidWorks XPS Driver	☐
Other Add-ins	
✓ SwSwimAddin	✓
✓ SwtoJt	✓
☐ Teamcenter	☐

Figure 11 Teamcenter add-in from older integration version

It should be removed by double clicking on %SWIM_DIR%/bin/Remove_cadscript.reg, after which the add-in dialog should look like this:



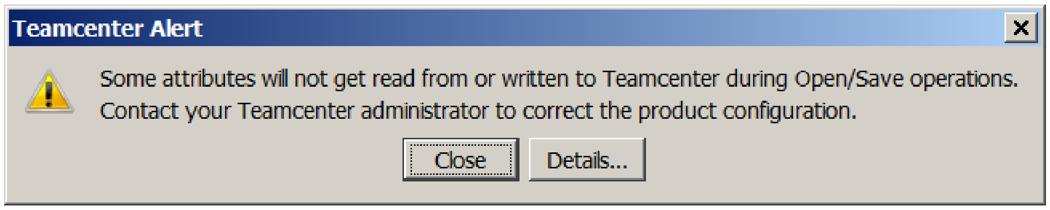
Active Add-ins	Start Up
SolidWorks Add-ins	
☐ 3D Instant Website	☐
☐ Autotrace	☐
☐ FeatureWorks	☐
☐ PhotoWorks	☐
☐ SolidWorks 2D Emulator	☐
✓ SolidWorks MTS	✓
☐ SolidWorks Toolbox	☐
☐ SolidWorks Toolbox Browser	☐
☐ SolidWorks Utilities	☐
☐ SolidWorks XPS Driver	☐
Other Add-ins	
✓ SwSwimAddin	✓
✓ SwtoJt	✓

Figure 12 Add-in dialog, after removal of older integration version

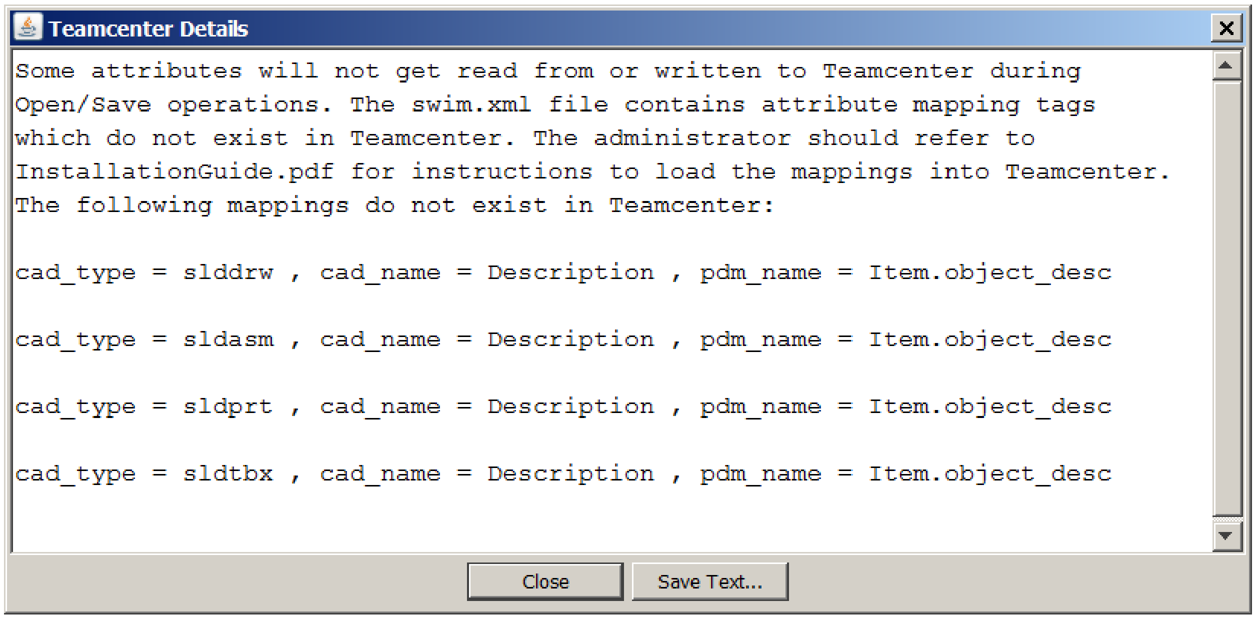
- Make sure any Windows paths specified for the JRE_DIR, IPR_DIR, or SWIM_DIR variables in scripts, or in the user's environment, do not have spaces. If necessary, use a command window and the MS-DOS command dir /x to determine the short MS-DOS name for a file or director, and update all scripts and environment variables to eliminate the spaces.
- Make sure the JRE_DIR variable in the %SWIM_DIR%/swimenv.bat script specifies the correct installation directory for the Java runtime environment. This script is located in the installation directory for the Teamcenter Integration for SolidWorks. It is recommended to use the JRE that is installed with the Teamcenter Rich Client.
- If the JRE_DIR variable is already defined in the user's environment, it will take precedence over the variable's definition in the %SWIM_DIR%/swimenv.bat script. If the user prefers setting this variable in his environment, make sure it is defined correctly, otherwise remove it from the user's environment so that the swimenv.bat definition takes effect.

Runtime problems

- If SolidWorks models cannot be opened successfully at remote sites after export via Teamcenter Multi-Site, make sure you are including the TXD_long_name_relation in the relation types that are exported.
- You may need to switch between different versions of the SolidWorks integration, on a single client workstation. This may be done by running the registration batch scripts (RegisterSwimAddin.bat and UnregisterSwimAddin.bat) in the SWIM bin directory. If you are using Windows Vista or Windows 7 you must run the batch script as administrator (right click it and select “Run as Administrator”)
  1. First unregister the currently active SWIM version by running UnregisterSwimAddin.bat in the SWIM bin dir.
  2. Next, go to the bin dir of the SWIM version you want to switch to and run RegisterSwimAddin.bat.
  3. Start SWIM using the version’s corresponding startsw.bat.
  4. You can verify the change by selecting **Tools | Addins** in the SolidWorks menu and hovering your mouse over the SwSwimAddin option. The tooltip that appears should display the integration install directory of the version you switched to.
- When logging into Teamcenter from SolidWorks, for the first time, users may see these dialogs:



Some attributes will not get read from or written to Teamcenter during Open/Save operations.
Contact your Teamcenter administrator to correct the product configuration.



```
Some attributes will not get read from or written to Teamcenter during
Open/Save operations. The swim.xml file contains attribute mapping tags
which do not exist in Teamcenter. The administrator should refer to
InstallationGuide.pdf for instructions to load the mappings into Teamcenter.
The following mappings do not exist in Teamcenter:

cad_type = slddrw , cad_name = Description , pdm_name = Item.object_desc

cad_type = sldasm , cad_name = Description , pdm_name = Item.object_desc

cad_type = sldprt , cad_name = Description , pdm_name = Item.object_desc

cad_type = sldtbx , cad_name = Description , pdm_name = Item.object_desc
```

Figure 13 - Attribute mapping warning message

The problem is explained in the dialog text itself, but sometimes causes confusion. This simply means that the attribute map in the swim.xml file contains definitions that have not been loaded into Teamcenter, or have been loaded incorrectly. Users will continue to see this dialog at each login until (a) the Administrator loads the attribute mappings correctly or (b) the Adminstrator removes (or comments out) the attribute mappings from the swim.xml file.