

**Teamcenter Integration for SolidWorks®**  
**Users Guide**  
**Version 9.2.0**

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Milford OH, 45150

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## Introduction

The Teamcenter® Integration for SolidWorks® provides access to Teamcenter from within the SolidWorks environment. You can search Teamcenter for models and open them in SolidWorks, save models back to Teamcenter and update bills of material (BOMs), create new items and revisions, reserve models so others cannot modify them while you are working on them, and cancel your reservations.

This guide assumes you already know how to use SolidWorks. You should also have a basic familiarity with the Teamcenter concepts of an item, item revision, dataset, and BOM. The Teamcenter Integration for SolidWorks stores each SolidWorks model in its own dataset, and gives the dataset the same name as the model. Each dataset is stored in an item revision, and each item revision belongs to an item. When this guide mentions a SolidWorks model in Teamcenter, it is referring to the combination of the item, revision, and dataset that contain the SolidWorks model file.

## Technical Support

For technical support, please contact the UGS 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.industrysoftware.automation.siemens.com/services/global\\_number.pdf](http://support.industrysoftware.automation.siemens.com/services/global_number.pdf) for additional telephone numbers, or call (714) 952-5444.

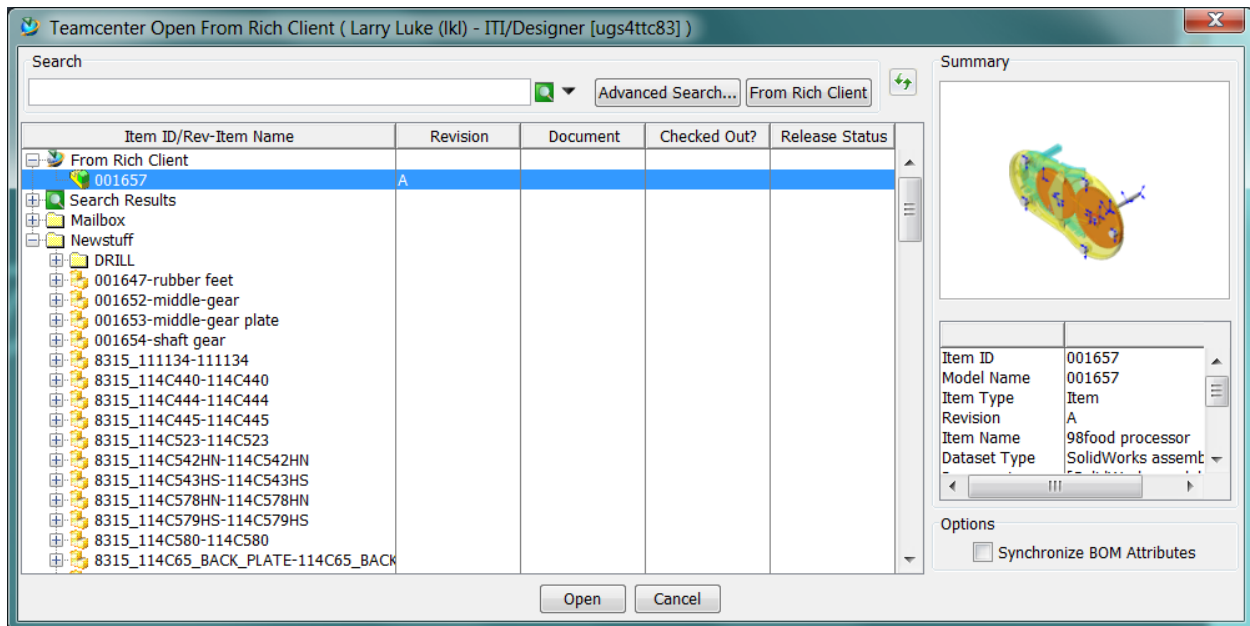
## Starting SolidWorks and the Teamcenter Integration for SolidWorks

### *From the Desktop*

Start the Teamcenter Integration for SolidWorks by selecting it from your Start menu, or double-click on the startsw.bat script found in the bin subdirectory of the Teamcenter Integration for SolidWorks installation.

### *From the Teamcenter Rich Client*

If you are working in the Teamcenter Rich Client, you can double-click on a SolidWorks dataset, or you can select the dataset and use the **Open** command in the Rich Client's **File** menu. In the Rich Client's Navigator application, you can also select one or more items, item revisions, and datasets, then click the  button, located in the toolbar near the top of the window. This starts a new SolidWorks session, displays the Integration's Open dialog, and shows the selected model in the dialog's **From Rich Client** folder. If a SolidWorks session already exists when the  button was selected, the select model(s) will be routed directly to the active integration session, and the user will see the integration's **Open** dialog, as shown below.



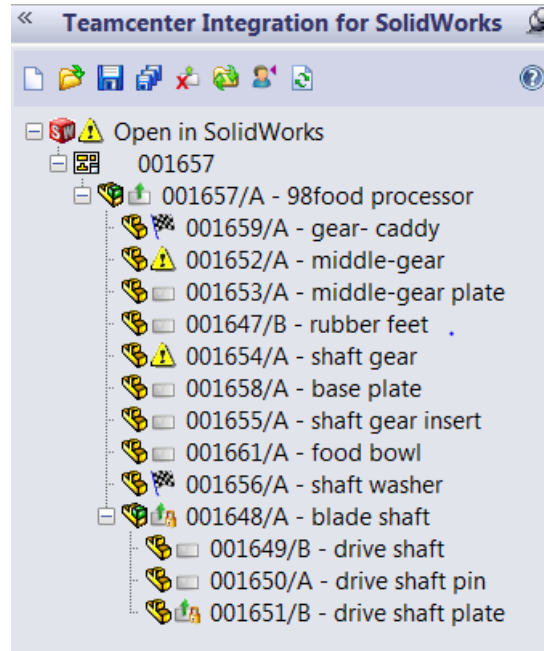
**Figure 1 The Teamcenter Open dialog**

See **Opening a SolidWorks Model Previously Saved to Teamcenter** for more information on the Open dialog.

## The Teamcenter Task Pane

When the Teamcenter Integration for SolidWorks is available in SolidWorks, the Teamcenter task pane appears on the right side of SolidWorks window<sup>1</sup>. Click on the Teamcenter tab to access the task pane.

<sup>1</sup> All Teamcenter operations are initiated from the Teamcenter task pane. The integration does not override any native SolidWorks functionality.



**Figure 2 the Teamcenter Task Pane**

The **Teamcenter** task pane is divided into three sections: Teamcenter Quick Launch Bar, Teamcenter Status Tree, and Teamcenter Operation Progress. The top section, Quick Launch Bar, contains a row of icons to launch Teamcenter operations. The operations are used to save data to Teamcenter, retrieve data from Teamcenter and manage checkouts. The middle section, Status Tree, shows the Teamcenter status of in-session models. The bottom section, Operation Progress, provides an indication of operations in progress.

### ***Teamcenter Quick Launch Bar***



**Figure 3 The Teamcenter Quick Launch bar**









Action icons are used to perform Teamcenter functions. Through these operations, users can retrieve (Open) data from Teamcenter and save data to Teamcenter. Additionally, the icons are used to check out, and cancel the checkouts of data. Below is a description of each of the icons. Each icon has an implied title. Sections of this document are dedicated to describing each operation in detail.

#### **Create**

Creates an Item in Teamcenter and creates the associated model in SolidWorks. You select the SolidWorks template file and specify the Teamcenter Item Type, the Item Id, and the Item Name. The

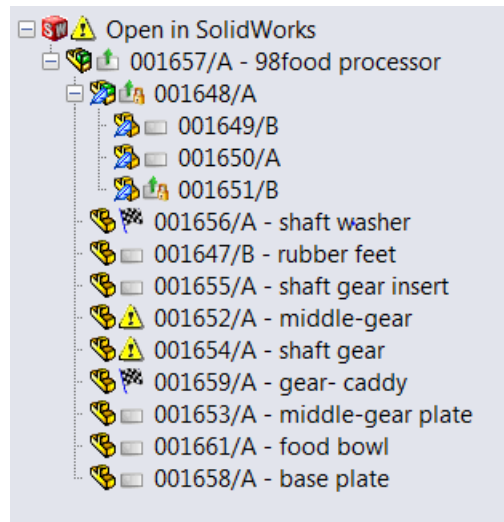


dataset is checked out of Teamcenter but the SolidWorks file is not saved to Teamcenter. The SolidWorks file is saved to disk.

-  **Open** Finds a model in Teamcenter and opens it in SolidWorks. You may choose to obtain a read-only copy of the model or to check out the model for modification. *Checkout* means Teamcenter reserves the model for you. Nobody else can check out the model and save a modified version back to Teamcenter while you have it reserved.
-  **Save** Saves the current model to Teamcenter. Optionally, you may choose to create a new revision of the model or save the model to a different Teamcenter item.
-  **Save All** Saves to Teamcenter all models in the SolidWorks session.
-  **Update Directory** Updates, deletes or purges SolidWorks files in a selected directory.
-  **Cancel Checkout** Cancels your reservation on SolidWorks models currently checked out to you. You do not have to open a model in SolidWorks to cancel your reservation on it.
-  **User Settings** Changes your Teamcenter group and role.
-  **Refresh** Refresh the Teamcenter status icons. For all models in session, the status icon will be updated based on the latest information from Teamcenter. The integration does not refresh automatically, not even when expanding the nodes of the status tree. The integration only refreshes after an integration operation or selecting the refresh button. Users should periodically refresh to ensure the icons and attribute information are current.
-  **About** Version information pertaining to the integration.

The first time you use one of these commands in a SolidWorks session, the Integration will prompt you for your Teamcenter user name and password. Once successfully logged in to Teamcenter, you will not have to log in again for the remainder of that SolidWorks session.





## Teamcenter Status Tree



**Figure 4 Status in the Teamcenter Task Pane**







From the status tree, the user can determine the status of all in-session models and use Right Mouse Button (RMB) pop-up menus to perform operations on selected models. By default, only the first level of structure is displayed, when an assembly is opened into session. The user must expand the tree nodes to see the full structure. Each node consists of two icons followed by a text string. The first icon indicates the SolidWorks type and the second icon indicates the Teamcenter status. By default, the text contains the Teamcenter Item Id and Revision followed by the SolidWorks Description. If a model has not been saved to Teamcenter, any Teamcenter attribute information will be blank.

### SolidWorks Type Icons

|   |  |
|---|--|
|  <b>Drawing</b>      | SolidWorks type is a drawing (.sldprt). Drawings will not contain configuration text.    |
|  <b>Assembly</b>     | SolidWorks type is an assembly (.sldasm) Assemblies will always have configuration text. |
|  <b>Part</b>         | SolidWorks type is a part (.sldprt). Parts will always have configuration text.          |
|  <b>Toolbox Part</b> | SolidWorks type is a toolbox part. Toolbox parts will always have configuration text.    |

A feather appears on the icon if the model is lightweight within the SolidWorks session. Because the models are not fully loaded into the SolidWorks session, not all integration actions can be invoked on the models.

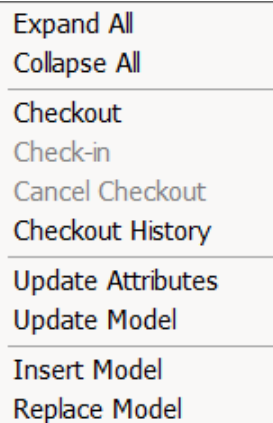
## Teamcenter Status Icons

|  |  |
|--|--|
| <b>New</b>   | If no icon is present, then the model has not been checked into Teamcenter or was moved from one directory to another. When the status is blank, the text will not contain Teamcenter information. |
|  <b>Ambiguity</b>           | The grey question mark (ambiguity) status is displayed if the model's status in Teamcenter is uncertain.   |
|  <b>In Teamcenter</b>       | The user has the latest revision of the model. The revision is neither released nor checked out by any user.   |
|  <b>Checked Out</b>         | The user has checked out the latest revision of the model.   |
|  <b>Checkout Restricted</b> | The user has the latest revision of the model. The revision is checked out by another user.  |
|  <b>Released</b>            | The user has the latest revision of the model. The revision is released.   |
|  <b>Yield</b>               | The user does not have the latest model. A newer revision or newer version exists in Teamcenter.   |

Operations can be performed directly on nodes in the tree via a shortcut menu. Nodes are selected by one of three ways:

- 1) Left mouse click on a single node
- 2) Shift left mouse click or control left mouse click to select multiple nodes
- 3) Selecting one or more models in the SolidWorks feature tree.

Once selected, the shortcut menu is exposed by right mouse button click on one of the selected nodes. Selecting an option from the menu initiates an operation on the selected nodes' models.



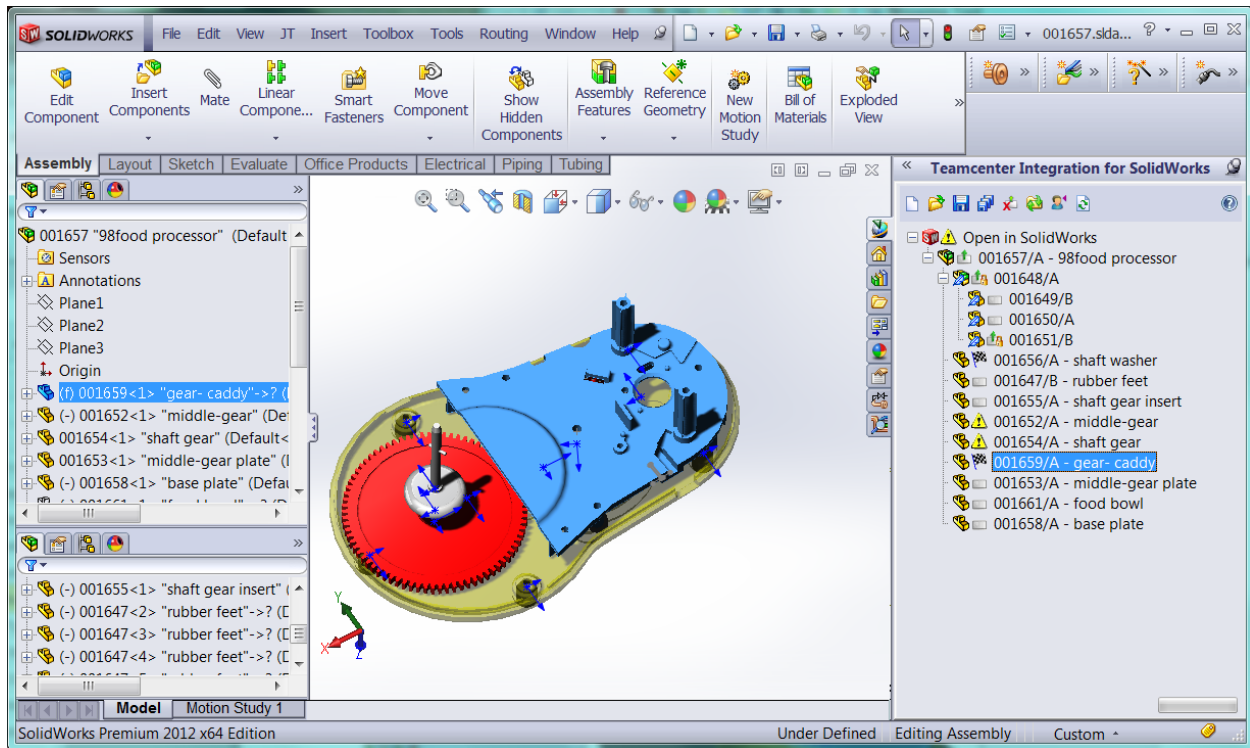
**Figure 5 The Right Mouse Button shortcut menu**

### **Collapse All**

All selected nodes and their children are collapsed as if the “-“ button had been selected on all of the nodes.

|                          |  |
|--------------------------|--|
| <b>Expand All</b>        | All selected nodes and their child nodes are expanded as if the “+” button had been selected on all of the nodes.  |
| <b>Checkout</b>          | Check out the model. If the checkout cannot be obtained the operation will not succeed and the status will remain as it was.   |
| <b>Check-in</b>          | Check in the model. The save dialog will appear, enabling the user to store the model into Teamcenter.   |
| <b>Cancel Checkout</b>   | Cancel the checkout of the model.  |
| <b>Checkout History</b>  | A dialog is brought up to view the checkout history of the model.  |
| <b>Update Attributes</b> | Synchronize the attributes from Teamcenter for the selected models. Attributes are synchronized during open, but a user may wish to synchronize again because another user has changed the attributes in Teamcenter or the model was not brought into session during the open operations (was lightweight or suppressed). The site administrator should use the attribute mapping section of Administration Guide to configure attributes for synchronization. |
| <b>Update Model</b>      | Update the model from Teamcenter by fetching the model again and reloading it in the current SolidWorks session.   |
| <b>Insert Model</b>      | Find and insert a model from Teamcenter into the active assembly.  |
| <b>Replace Model</b>     | Replace the selected component with a substitute from Teamcenter. The success or failure of the actual replacement, within the SolidWorks session, depends upon the degree of similarity between the two components, as well as any mating constraints to the parent assembly.   |

## Highlighting of Selected Models

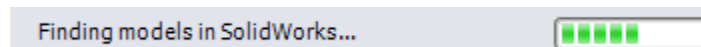


**Figure 6 Model highlighting**

Clicking on a model in the Teamcenter task pane highlights the model within the task pane, the SolidWorks feature manager and the SolidWorks graphics display. Similarly clicking on a model in the SolidWorks feature tree or graphics display will highlight the model in the task pane. Multiple models can be highlighted by using control key when selecting or by dragging within the graphics display.

*Ghost highlights can occur in the SolidWorks feature manager if dynamic highlighting is enabled. Disable dynamic highlighting from the SolidWorks FeatureManager System Options dialog to prevent the ghosts.*

## Teamcenter Operation Progress



**Figure 7 Teamcenter progress bar**

A progress bar is displayed indicating that an integration operation is in progress. While the bars scroll, the user should not use the SolidWorks session. At times, the integration must manipulate SolidWorks, so the user may observe changes on the screen. After the integration has completed interaction with SolidWorks, the integration attempts to preserve the session state when possible.

## Integration Dialog Windows

Each of the Teamcenter functions listed above leads to one or more dialog windows. The SolidWorks user interacts with Teamcenter using the buttons, check boxes, selection lists, and other elements of these dialogs. Except for the User Settings and About dialogs, each dialog is designed to display CAD models and allow the user to manage them. The CAD models are consistently displayed in a table format, allowing the user to select a single model, a contiguous range of models or a non-contiguous set of models. Object selection follows the usual paradigm: Use Ctrl-click to select non-contiguous sets, and use Shift-click to select a contiguous set. When at least one model is selected, a right mouse button pop-up menu is enabled. This is called the “Shortcut menu”. It provides a set of convenience functions appropriate to the context of the operation and the status of the selected object(s).

In the figure below, a set of models is selected in the Save dialog, and only those functions which can be performed on the selected models are enabled; the remainder are grayed out:

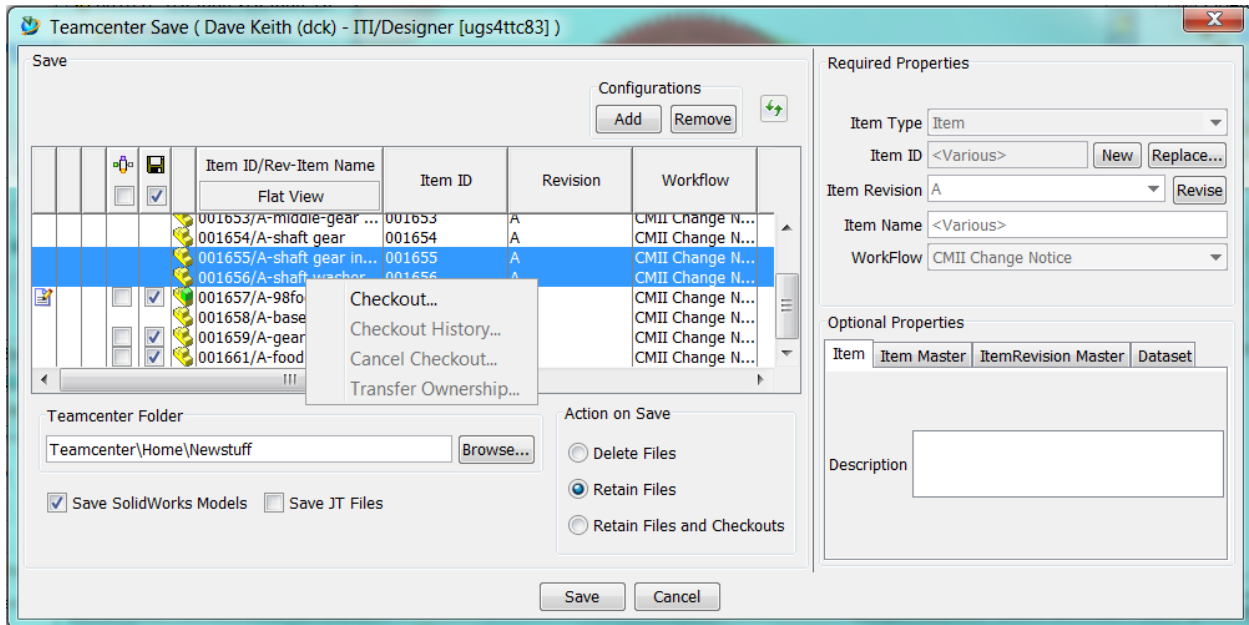


Figure 8 the Save Dialog

## Shortcut Menu Functions

Select one of the following commands from the shortcut menu.

- Checkout** Reserves the model. This does not export the model’s file from Teamcenter.
- Checkout History** Shows details about previous check out and check in operations performed on the selected model.
- Cancel Checkout** Cancels your reservation on selected models.
- Delete** Deletes searches in the **Open**, **Insert** and **Replace** dialogs and deletes models from disk in the Teamcenter **Update** dialog.

|                           |  |
|---------------------------|--|
| <b>Transfer Ownership</b> | Transfers ownership of the model from a remote site to your site, making it possible to modify and save the model at your site. This transfer affects the model's entire item, including all revisions of the model. |
| <b>Where Referenced</b>   | In the <b>Open</b> , <b>Insert</b> , and <b>Replace</b> Dialogs, results in a "where referenced" search being added to the Open Dialog's search folder.  |

These commands are described in more detail in the following sections.

## ***Checkout***

The **Checkout** command reserves the selected models. The command skips models that are already reserved and models that are not in Teamcenter.

Before any models are checked out, you will be prompted to enter a comment and change ID, similar to the information you are asked to provide when checking out a model with the Open dialog.

This command does not export the model's file from Teamcenter, so you can safely check out a model that is already open in your SolidWorks session or is in a local directory. Changes you have made to the local model will not be lost.

Once you have reserved the model, you can save it to Teamcenter, although you should always be careful when saving a model that was not reserved until long after its file was exported. For example, you may fetch a model from Teamcenter without checkout, and later use the shortcut menu to reserve it when you are ready to save your changes back to Teamcenter, but between the time you fetch the model and the time you reserve it, other users can change the model in Teamcenter. A recommended practice when using the **Checkout** command is to verify first that Teamcenter does not have a newer version of the model. See the section titled Detecting Newer Model Versions for more information on determining whether a model in Teamcenter has changed.

## ***Checkout History***

The Checkout History command can only be initiated for one row of a table. When selected a popup dialog appears with a table of the model's check-in and checkout history.

## ***Cancel Checkout***

The **Cancel Checkout** command cancels your reservation on selected models. The command skips models that you have not reserved.

Before any of your reservations are canceled, you will be prompted to confirm that you want to cancel your checkouts

## ***Delete***

The **Delete** command deletes the model from disk in the Manager Dialog and deletes searches on the Open Dialog.

## Transfer Ownership

The **Transfer Ownership** command transfers ownership of the selected model from a remote site to your site. This command actually transfers the model's entire item to your site, including all revisions of the model and the other datasets, forms, and Teamcenter objects attached to the item. If several rows are selected, all of the items that are owned by remote sites will be transferred to your site. The command skips models that are already owned by your site.

The Integration does not allow you to check out models owned by remote sites, so transferring ownership to your site is a first step if you need to modify and save such a model back to Teamcenter. After transferring ownership, you can check out the model in the usual way, using either the **Checkout** command in the shortcut menu or the **Open** command in the **Teamcenter** menu. Once you have made your changes and checked in the model, users back at the remote site may use the **Transfer Ownership** command to return ownership to their site. The Rich Client's Navigator application also makes it possible to transfer ownership with its **Tools | Import | Remote** command.

## Where Referenced

The **Where Referenced** command performs a Teamcenter "Where Referenced" search on the selected row of the **Open**, **Insert** and **Replace** dialogs. The result is a new "Where Referenced" folder in the list of searches.

## Detecting Newer Model Versions

Most integration dialogs can indicate whether a change has occurred since a model was exported from Teamcenter. As shown in preceding figures, three table columns display the icons described below when the local model has unsaved changes, or when a new dataset version or item revision is available in Teamcenter.



The first column displays this icon when the model in the SolidWorks session has never been saved to Teamcenter, or when an existing model is being saved to a new Item Revision.



The first column displays this icon when the model in the SolidWorks session has changes that have not been saved to Teamcenter.



The second column displays a dataset icon when a newer version of the model's dataset is available in Teamcenter. You may want to use one of the integration's **Update** commands to fetch the new version of the model, although when using **Update Directory**, you should first close your local copy in the SolidWorks session, otherwise the Integration will not be able to load the new version into the session.



The third column displays an item revision icon when a later item revision is available in Teamcenter. If you want this later revision in your SolidWorks session, first close the current version in the SolidWorks session, use one of the integration's **Update** commands to find the model's latest revision.

*This icon simply indicates a later item revision is available. It does not indicate whether that item revision also has a dataset for the SolidWorks model.*







A question mark appears in a column when the Integration does not have enough information to determine whether a change has occurred. For example, a question mark might appear if the model in Teamcenter was not saved with version 2.0 or later of the Teamcenter Integration for SolidWorks, or if cached information about exported models is not available.

The Integration gleans several pieces of information from a model to determine whether a change has occurred since the model was last exported from Teamcenter. It does not compare the contents of one SolidWorks file against another, so an icon should be treated as a hint that a change may have occurred, not as a guarantee that a model has or has not been modified. If the model is currently in the SolidWorks session, the information available to the Integration about that local version of the model is more up to date, but if the model is not in the current session, the Integration will attempt to use cached information collected the last time the model was saved to Teamcenter.

## Opening a SolidWorks Model Previously Saved to Teamcenter

The  **Open** command in the **Teamcenter** task pane displays a dialog that allows you to find a model in Teamcenter and open it in SolidWorks. You may choose to obtain a read-only copy, or you may choose to check out the model for modification.

You must check out the model if you wish to save it back to Teamcenter at a later time. While the model is checked out to you, Teamcenter will not allow others to modify the model, although they may obtain read-only copies.

The dialog displayed by the  **Open** command is shown in Figure 9. At the top of the dialog is the **Search** box, which you can use to query Teamcenter for SolidWorks models. The table below the **Search** box displays the contents of your Teamcenter folders or search results. Your folders may contain a variety of Teamcenter objects, but the table will show only items, item revisions, SolidWorks models and other folders. At the right side of the dialog is a summary pane that includes a preview of the model, if available, and a table of details.

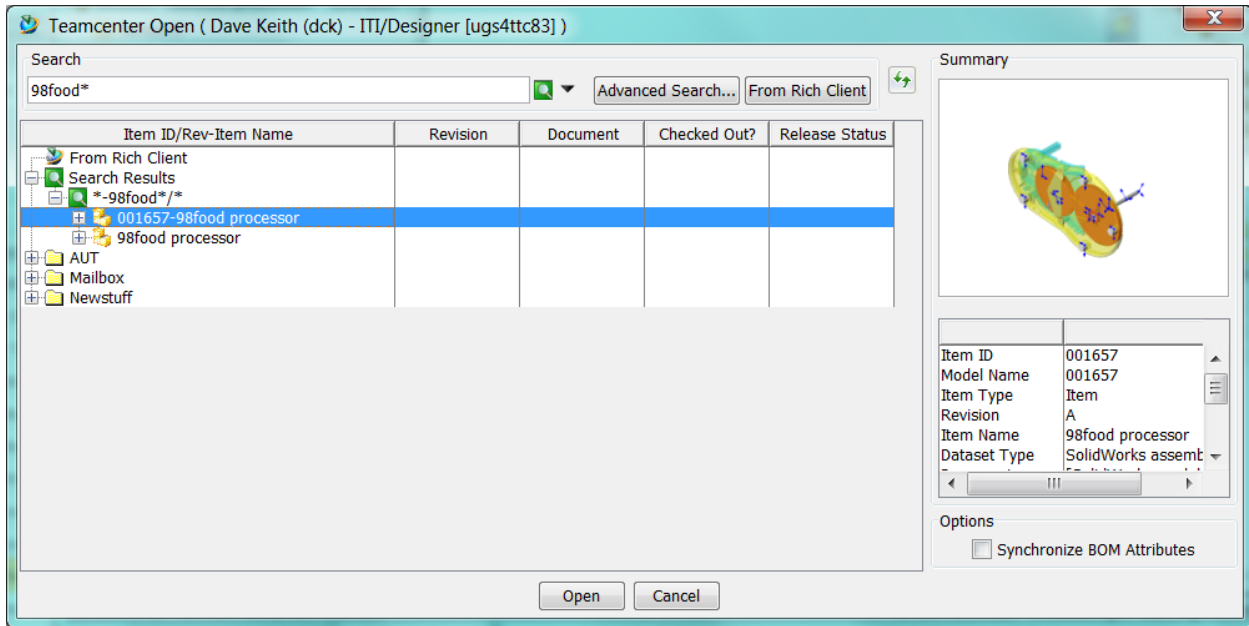


Figure 9 the Open Dialog

## The Open Dialog

The Open dialog initially shows your Teamcenter home folder, or if you started SolidWorks from the Teamcenter Rich Client, it shows the model you selected in the Rich Client in a folder named **From Rich Client**.

To find a model in Teamcenter and open it in SolidWorks, follow these steps.

### Find the top level model

#### Browsing Teamcenter Folders and items


Double-clicking on a folder, item, or item revision icon opens it and displays its contents in the table.

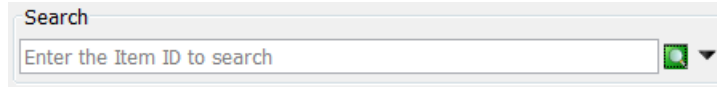
*If items are added to a folder or removed after you have opened it, click the **Refresh** button to display the folder's current contents.*

#### Browsing the Rich Client Selection

If you started SolidWorks from the Teamcenter Rich Client, clicking the **From Rich Client** button in the **Search** box at the top of the dialog will re-display the model you selected in the Rich Client. The model you selected in the Rich Client will appear in a folder named **From Rich Client**.

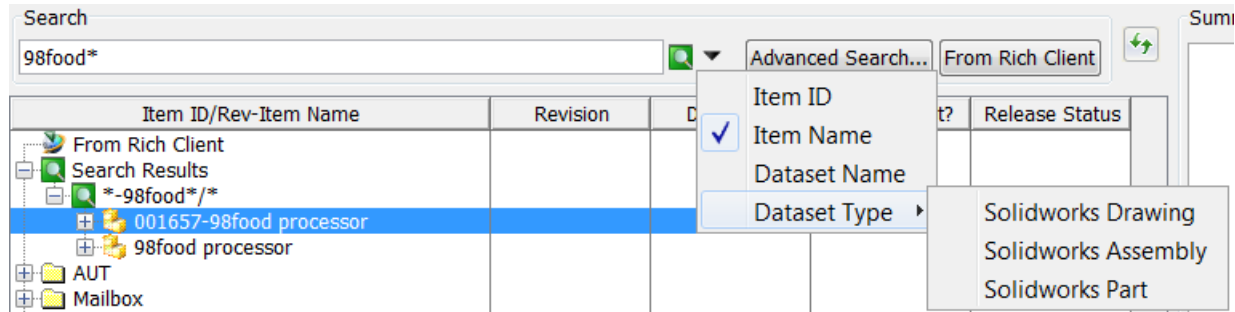
#### Searching Teamcenter

You can search for a model by entering its item ID into the text field. Click on the search button  or use the enter key to complete the search.



**Figure 10 Searching by Item ID**

Click on the down arrow button ▼ to search by Item Name, Dataset Name (model name) or Dataset Type. SolidWorks drawings, assemblies or parts are searchable by Dataset Type.

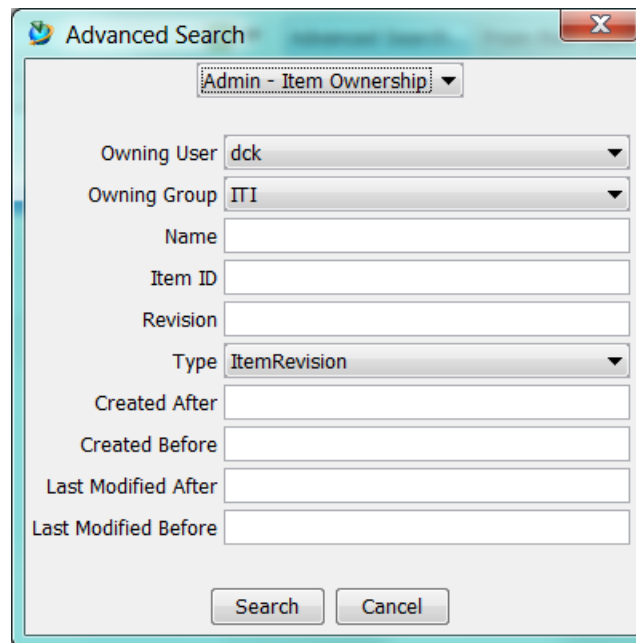


**Figure 11 Specifying search by Item ID, Item Name, Dataset Name, or Dataset Type**

More advanced searches are available by selecting the **Advanced Search** button.

The search results are put into a temporary folder within the **Search Results** folder, located near the top of the table. Each search adds another temporary folder to **Search Results**. All of these temporary folders will be discarded when your SolidWorks session ends (these folders are not available in the Rich Client).

The **Advanced Search** dialog uses the saved Teamcenter queries. At the top of the dialog is a drop-down list of Teamcenter queries. When a query is selected, the dialog will display the criteria configured for it. Enter as much as you know about the model(s) you want to find, then click the **Search** button to execute the query. The Advanced Search Dialog will close and the search results will be added to the Open dialog's **Search Results** folder.



**Figure 12 The Advanced Search dialog**

## Select the top level model

Select a specific item, item revision or model row in the table by clicking on the row. Once selected, a preview image and details for the selected CAD model will appear in the summary pane. The preview only appears if the preview had previously been saved to Teamcenter. When multiple rows are selected, no image is displayed, and the image frame contains the text “Multiple Selection”.

The preview pane may contain other text indicating specific conditions to the user:

- No Selection – The user has not yet selected an Item, Item Revision or Dataset in the table at left. A selection must be made before selecting the Open button.
- No Preview – A preview does not exist in Teamcenter for the selection. This most commonly occurs for data saved before Teamcenter Integration for SolidWorks 8.1.0 was installed.

If a specific dataset is not selected, the integration will infer the identity of the top-level model, based on the following logic:

- If an item is selected, the integration will select the highest-level CAD model under the latest item revision.
- If an item revision is selected, the integration will select the highest-level CAD model under the selected revision.

The integration allows you to save multiple CAD models under a single item revision. Often this is done to group a drawing and its 3D models together, to simplify release management. The integration’s interpretation of “highest-level” CAD model is based on this hierarchy:

- SWDrw (highest)
- SWAsm
- SWPrt (lowest)

## Choose top level options

If desired, select the **Synchronize BOM Attributes** check box on the Open dialog. This option will map a specified property or note<sup>2</sup> from each component’s BOM line in the Teamcenter Bill of Materials to the ComponentReference property of the corresponding occurrence of that component within the top-level assembly<sup>3</sup>. This synchronization does have a performance impact on the load operation, and so users should apply it selectively. It is most appropriate when opening a SolidWorks assembly drawing containing annotations which are driven from the assembly’s componentReference properties.

Click the **Open** button at the bottom of the dialog and proceed to the **Open Dependencies** dialog.

---

<sup>2</sup> The source property or note for this mapping is specified in the integration’s configuration file. See the Administration guide for more details.

<sup>3</sup> This capability is only available when SolidWorks 2010 or a later version is used.

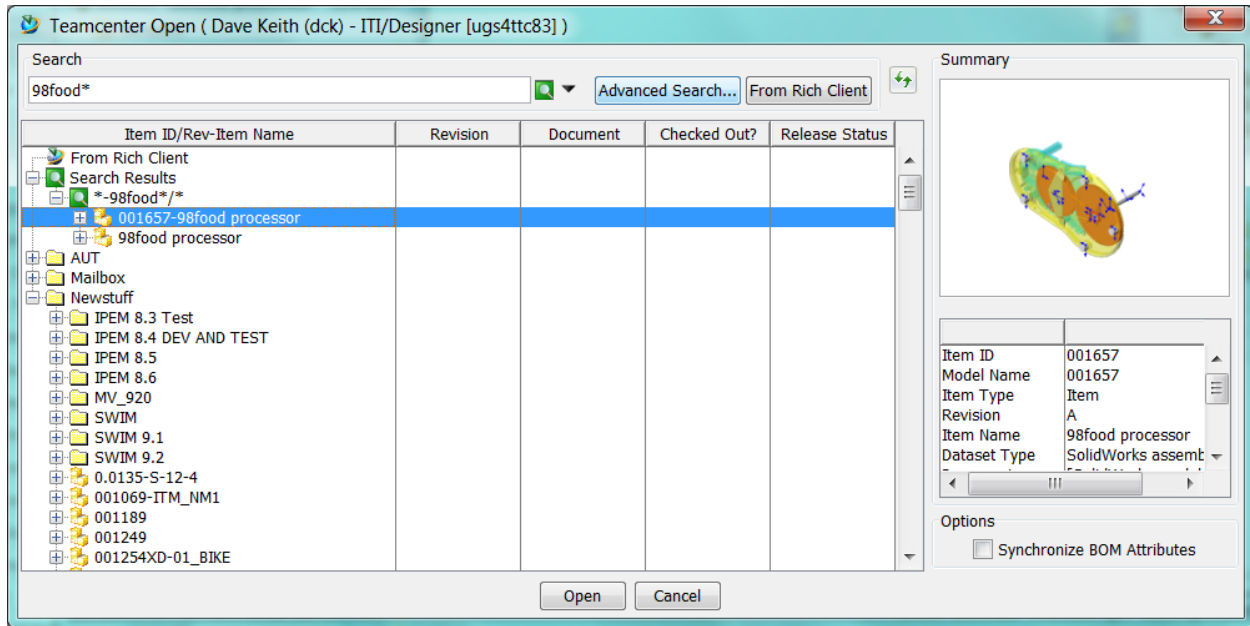


Figure 13 selecting the top-level model in the Open dialog

## Where Referenced: Searching for References to a Selected Model

If you want to know whether any SolidWorks models in Teamcenter reference a model found in the **Open**, **Insert** or **Replace** dialog, right-click on the model's row<sup>4</sup>, then select **Where Referenced** from the pop-up menu. The results of this "where referenced" search will be added to the Open dialog's **Search Results** folder.

The results include referencers of all models in the highlighted model's item revision. For example, if an item revision contains a part and a drawing, highlighting the drawing and clicking the **Where Referenced** drop down will find referencers for both the drawing and the part.

## Using seed parts

The Integration can create a new SolidWorks model when given an empty dataset to open. For example, you can start a new part by creating an item with a SWPrt dataset in the Teamcenter Rich Client, then sending the part to SolidWorks. The Integration will create a model for the part while opening it into SolidWorks.

When creating a dataset in the Teamcenter Rich Client for a SolidWorks model, keep these points in mind:

- The dataset name must be exactly the same as the name of the SolidWorks model it represents. If you are creating a dataset for a part to be named WIDGET, the dataset's name must be WIDGET. Do not include a revision or type in the dataset name. For example, WIDGET/A and WIDGET.PRT should not be used as dataset names for SolidWorks models.

<sup>4</sup> You must select the model's dataset in order to perform the "where referenced" search. The "where referenced" search is not available when the item or the item revision is the selected object.

- If you add a SolidWorks file to the dataset, the file's base name must match the dataset name. For example, if you create a dataset for a part file named `widget.prt.1`, the file's base name is `widget`, so the dataset name should be `WIDGET`.
- A dataset for a SolidWorks model should be attached to only one item revision. If you copy a dataset from one item revision and paste it into another, the dataset will appear in both item revisions and changes made to one will appear in the other. It is usually better to create a new, empty dataset in an item revision rather than using copy and paste to add the dataset to the item revision.
- SolidWorks datasets should be attached to their item revisions using a specification relation (`IMAN_specification`). This is the default when the Rich Client creates a new SolidWorks dataset.

To create a new item or revision for a SolidWorks model in the Teamcenter Rich Client:

1. Create the item or revision in the Rich Client as usual.

*When revising an item using the Teamcenter Rich Client, do not copy any existing SolidWorks datasets to the new item revision. Relationships to other SolidWorks models are not copied when datasets are copied.*

2. In the Rich Client's Navigator application, select the new item revision, then use the **File | New | Dataset** command to create a new SolidWorks dataset attached to that item revision. Give the new dataset exactly the name you want the new model to have, and use one of the following dataset types.

| Use This Dataset Type... | For This Type of Model... |
|--------------------------|---------------------------|
| SWDrw                    | Drawing (.slddrw)         |
| SWAsm                    | Assembly (.sldasm)        |
| SWPrt                    | Part (.sldprt)            |

3. If you leave the dataset empty, the Integration will create a model of the appropriate type when you attempt to open the model in SolidWorks using one of the commands in the Teamcenter menu. Alternatively, you may wish to import a SolidWorks model file into the new dataset in the Rich Client, if you do not want the Integration to create the model from a default template. Remember that the file's base name must match the dataset name.

## Multi-Site: Working with Models Owned by Remote Sites

You can open SolidWorks models owned by remote sites if Teamcenter Multi-Site is installed and your administrator has replicated those models at your site. Replicated models can be fetched and opened in the same manner as any other model. However, the Integration does not allow users to checkout models owned by remote sites. If you plan to modify and save changes to a model owned by a remote site, several additional steps are necessary.

## Transfer Ownership

The simplest way to checkout and modify a SolidWorks model owned by a remote site is to transfer ownership of that model's item to your site. Once your site has ownership of the item, you will have all of the normal access privileges allowed for locally owned SolidWorks models. You will be able to check models in and out of Teamcenter, and create new revisions.

If the Integration was configured during installation to show a Transfer Ownership command in the shortcut menu, you can right-click on one or more models in the Open dialog, use the Transfer Ownership command to transfer remotely-owned items to your site, then proceed as usual to open them in SolidWorks. The Transfer Ownership command is also available in the Integration's other dialogs. See the section on using the Shortcut Menu for more information about the shortcut menu and the **Transfer Ownership** command.

If the shortcut menu has not been configured to show the **Transfer Ownership** command, you or your site administrator may be able to transfer ownership using the Teamcenter Rich Client.

## Remote Checkout

It is recommended to use **Transfer Ownership** to gain modification privileges for any object owned by a remote site. This function is available from within the SolidWorks session, as well as the Teamcenter Rich Client.

Should you choose to use **Remote Checkout** instead, the checkout must be performed in the Rich Client. Click on an object to select it, then use the **Tools | Check-In/Out | Check-Out** command to reserve the object.

*If you use remote checkout to reserve a SolidWorks dataset, do not simultaneously export the file. After checking out the dataset, its CO property should be Y, not Y/E.*

If you choose to use remote checkout, you must use the Rich Client to check out every object in the item revision that may be modified by the Integration when later saving the model back to Teamcenter. In some cases, it may also be necessary to check out the item.

| <b>Check out this object...</b> | <b>If...</b>  |
|---------------------------------|---|
| SolidWorks Dataset              | You are modifying the SolidWorks model.   |
| Auxiliary File Dataset          | The auxiliary file map imports files into other datasets in the item revision.  |
| BOMViewRevision (BVR)           | You are modifying a SolidWorks assembly.  |
| Form                            | The attribute map copies SolidWorks properties to attributes in the form. This includes the Item Master form and the Item Revision Master form. |

| <b>Check out this object...</b> | <b>If...</b>   |
|---------------------------------|--|
| Item Revision                   | You will be creating a new SolidWorks or auxiliary file dataset when saving the model to Teamcenter, or when the attribute map copies SolidWorks properties to item revision attributes. |
| Item                            | The attribute map copies SolidWorks properties to item attributes.   |

After checking out the appropriate objects in the Rich Client, you can use the Integration to open the SolidWorks models and save them back to Teamcenter. The Integration will not release your reservation on any object that is owned by a remote site, even if you choose not to retain checkouts when saving models, or if you attempt to cancel checkouts using the **Cancel Checkout** command. When you are ready to release your reservations, use the Rich Client to check in the objects you previously checked out.

If you want to create a new revision of an item that is owned by a remote site, you must first create the new item revision in the Rich Client:

1. Check out the item in the Rich Client.
2. Use the Rich Client's **File | Revise** command to create the new item revision.
3. Check out the new item revision in the Rich Client.
4. Cut all SolidWorks datasets from the new item revision.
5. Use the Teamcenter Integration for SolidWorks to save the model to the new item revision.

### ***The Open Dependencies Dialog***

The Open Dependencies dialog will appear after the Open dialog closes. The Open Dependencies dialog shows the model you selected for opening, plus all of the models on which it depends. A check mark in the **Fetch** column means Teamcenter will put the model's file in the destination directory. A check mark in the **Checkout** column means the model will also be reserved for you. If you intend to modify a model and save it back to Teamcenter, you must put a check mark in the **Checkout** column.

*You will not be able to check out a model that is already checked out, even if you are the user who reserved it. However, you can always fetch a read-only copy no matter who holds the reservation.*



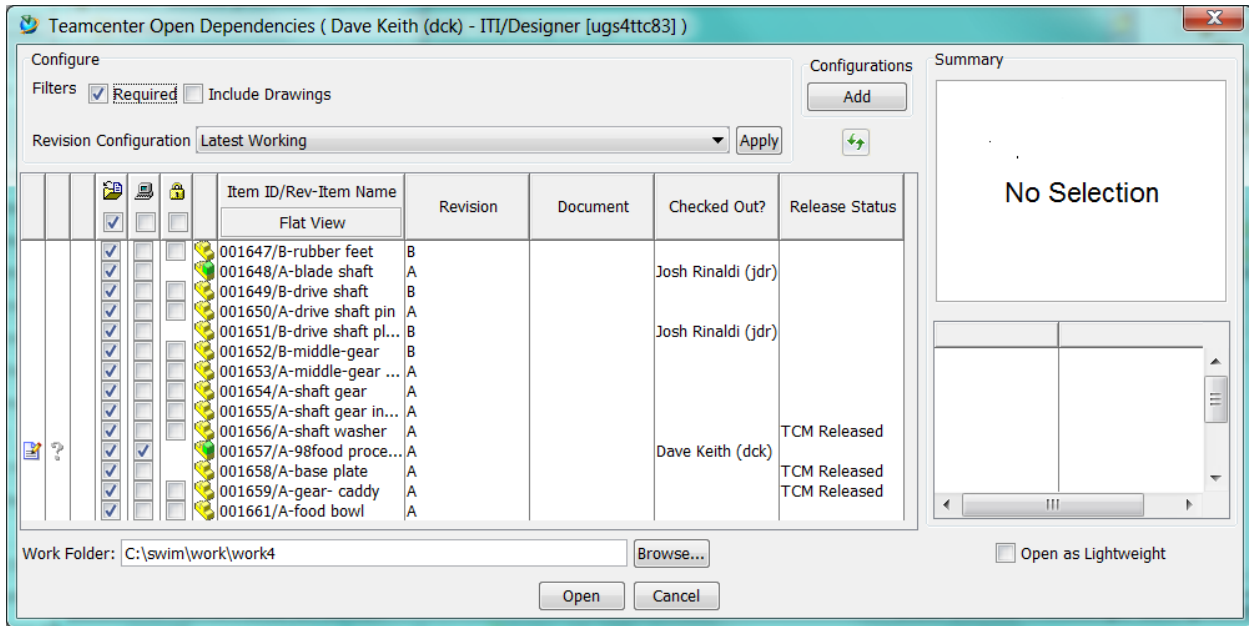


Figure 14 The Open Dependencies dialog

The first three columns, to the left of the **Fetch** column, indicate whether a change has occurred in a model. For more information, see the section on Detecting Newer Model Versions. The icons to the right of the **Checkout** column indicate the type of model in each row.

If you want to view the models' relationships to each other, click the **Tree** button above the table. This button toggles between the ordinary table view and a hierarchical view. The configuration rule displayed at the top of the table determines which revisions are shown.

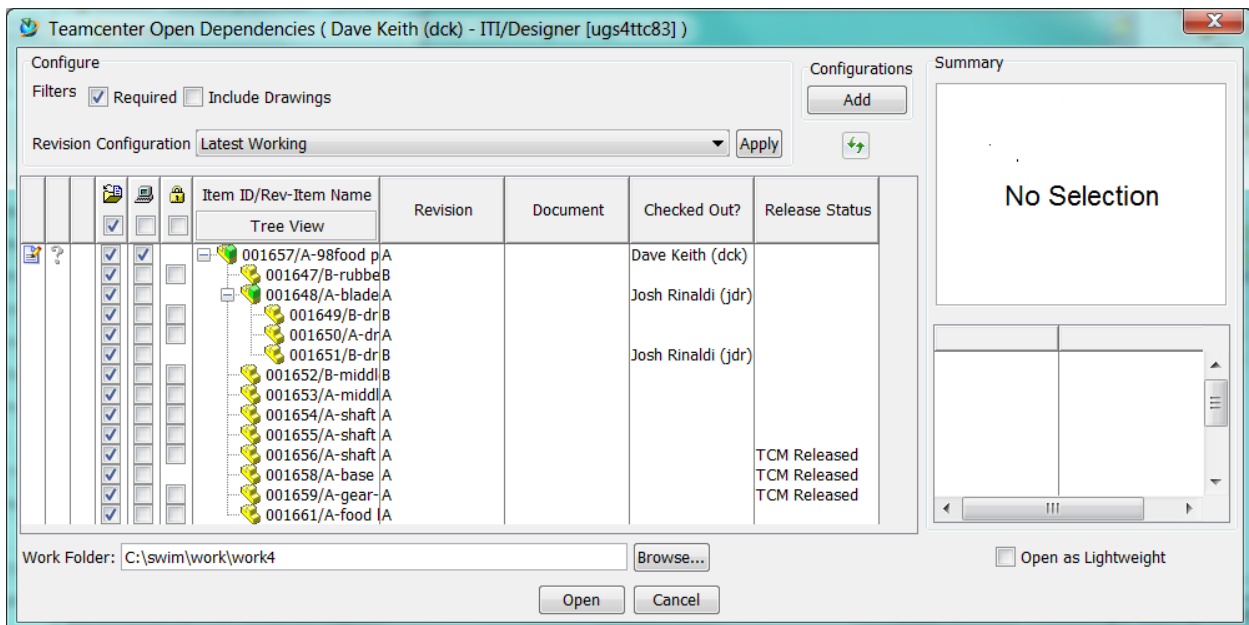


Figure 15 Tree View in the Open Dependencies dialog

To open a model and its dependencies in SolidWorks:

## Configure the structure

If you want to change the revision rule, use the **Revision Configuration** box, then click the **Apply** button to rebuild the table with your new configuration. See the Revision Configuration section for more information on how to use these settings. If you also want to apply a filter, use the **Filters** box before clicking the **Apply** button. See the section on **Filters** and the **Filter Dialog** for more information about filters.

If a model contains additional unreferenced configurations, you can include them by clicking on the model's row to select it, then clicking a **Configurations | Add** button. If no row is selected, these buttons affect every family table generic in the dialog. See Opening Configurations for more information.

## Select the actions to be applied



Put a check in the **Fetch** column to tell Teamcenter that the model's file should be placed in the destination directory.



Put a check in the **Open** column if you want the model displayed in its own SolidWorks window.



Put a check in the **Checkout** column if you also want to reserve the model for modification.

## Lightweight

If the **Open as Lightweight** checkbox is selected, the models will be opened lightweight in the same manner as SolidWorks opens with lightweight. **Open in Lightweight** will not synchronize attributes from Teamcenter to the lightweight SolidWorks models. The attributes can be synchronized later by resolving the models and using the **Update Attributes** button from the integration shortcut menu in the task pane. The **Open as Lightweight** button will be disabled if the user selected the **Synchronize BOM Attributes** checkbox on the Open dialog.

## Revision Configuration

The **Revision Configuration** box contains a list of revision rules. The specific rules included in the list are determined by Teamcenter configuration. When the Open Dependencies dialog appears, the current rule is the default rule which is specified in the swim.properties configuration file, or in the user's Teamcenter preferences. If you choose a different rule in the Open Dependencies dialog, click the **Apply** button to rebuild the table with the new rule.

## Opening the precise configuration of an assembly

The precise configuration of an assembly is the set of component item revisions that the assembly was using the last time it was saved to Teamcenter. To open the precise configuration of an assembly, select the **Precise Only** revision rule, then press **Apply**.

## Filters

When opening a model, the Integration by default also attempts to open all other models on which it depends. These models appear in the Open Dependencies dialog. However, by

applying a filter, you can limit the Open Dependencies dialog to only models that satisfy certain conditions. A filter can also set the **Fetch** and **Checkout** checkboxes for these models.


The **Filters** box in the Open Dependencies dialog contains several pre-defined filters. If a filter's checkbox is marked in the Open Dependencies dialog, the integration uses that filter when the **Apply** button is clicked in the **Configure** box. The pre-defined filters available by default are described below.

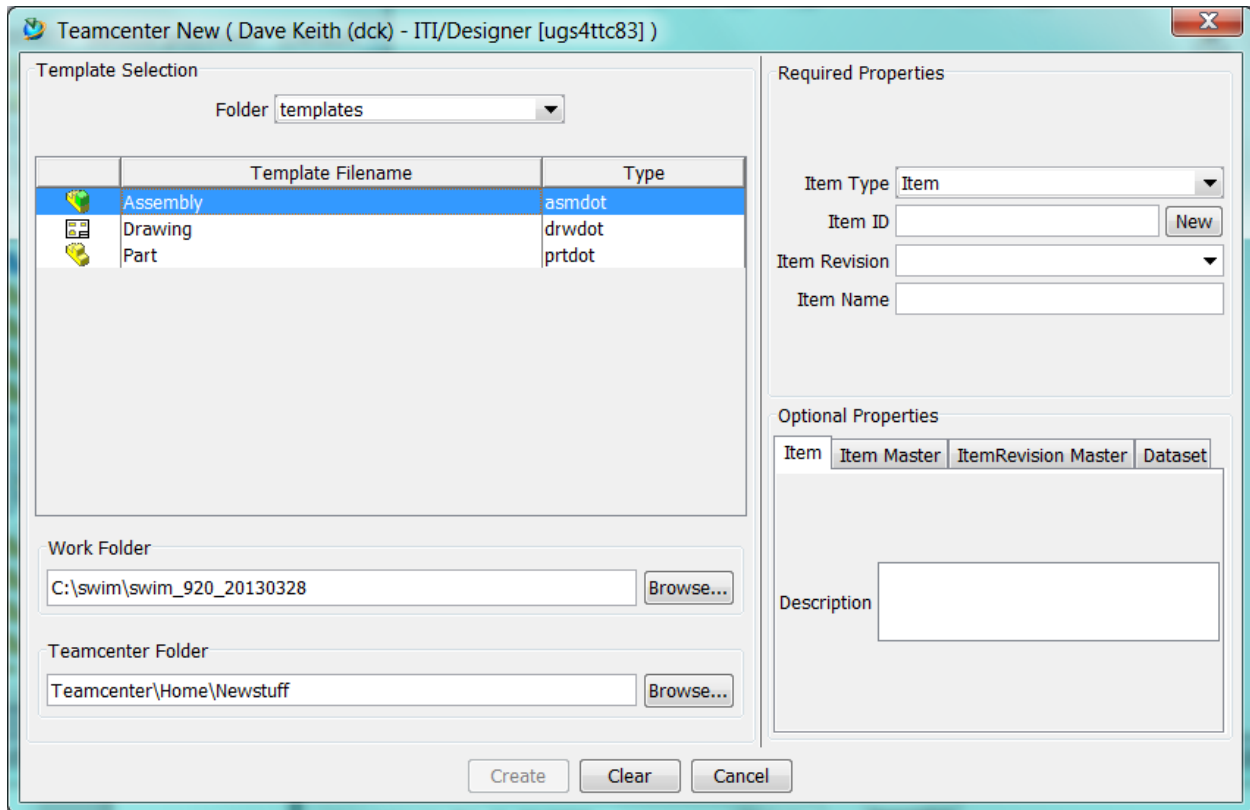
|                         |   |
|-------------------------|---|
| <b>Required</b>         | Models required by the models selected in the Open dialog are added to the Open Dependencies dialog. Other models are excluded from the Open Dependencies dialog. |
| <b>Include Drawings</b> | Drawings of all models are included in the Open Dependencies dialog.  |

## Opening SolidWorks Configurations

The configurations within a SolidWorks model may be saved as separate items in Teamcenter, as if they are distinct models. For example, if a model has a design table, each SolidWorks configuration might be a separate Teamcenter item. You can add such configurations to the dialog by clicking on the model's row to highlight it, then clicking the **Add Configurations** button in the **Configurations** box. Those configurations that exist as separate items in Teamcenter will be added to the dialog. If no row is highlighted, every model in the dialog is examined for configurations that were saved separately.

## Creating a new Teamcenter item and SolidWorks model

The  **Create** command in the **Teamcenter** task pane creates a new Teamcenter Item and a new SolidWorks model simultaneously. You should use this function whenever creating a new SolidWorks model that will be saved to Teamcenter, whether immediately or at some future time.



**Figure 16 The Create New Teamcenter Item dialog**


The **Create** dialog contains key attributes that must be provided whenever creating a new Teamcenter item. These include item type, item ID and item name. To create a new item, do the following:


1. Select the desired template part from the list. If your SolidWorks preferences define multiple template locations, you can switch between them using the drop-down list.
2. Accept the default **Work Folder** (which is based on your session's current working directory), or **Browse** to an alternative **Work Folder**.
3. Accept the default **Teamcenter Folder** (which is based on your session's current Teamcenter directory), or **Browse** to an alternative **Team Folder**.
4. Select an **Item Type** from the drop down list. Depending upon configuration, the choice of template part may determine the default item type that is selected. The list of available types is controlled by your Teamcenter administrator.
5. Set the **Item ID**. You may manually type an item ID, or ask Teamcenter to assign one, with the **New** button. The format of the assigned ID will depend upon the selected item type. The item revision, item ID is populated to the item Name and Model Description fields. If you manually typed in the item ID, you must click outside the box for the display to update.
6. Set the **Item Revision**. The integration filled in the default **Item Revision**. Click in the field to type in a different value.

7. Set the **Item Name**. The integration filled in the default **Name**. Click in the field to type in different value.
8. Depending on configuration, there may be additional **Required Properties** to fill in. You cannot complete the creation until all required properties have a legal value.
9. Use the tab to find and fill in the values of **Optional Properties**. The Create New operation can be completed without setting these values.

When you press the **Create** button, a new SolidWorks CAD model of the specified name and type will be created in the specified directory. A new Teamcenter item will be created with the specified type, ID, and name. When the CAD model is later saved to Teamcenter, the integration will find the matching item revision and save the CAD model as a dataset attached to the item revision.

## Saving SolidWorks models to Teamcenter

Two commands are available for saving SolidWorks models to Teamcenter. The  **Save** command will save the SolidWorks session's current model, and optionally its dependencies, to Teamcenter. It also saves checked-out models on which the current model depends. **Save** displays the dialog shown in Figure 17, allowing you to assign an item ID and other properties associated with the model in Teamcenter.

The  **Save All** command also displays the Save dialog, but populates it with all models in the SolidWorks session. If a model to be saved depends on others in the SolidWorks session, those models will also appear in the dialog.

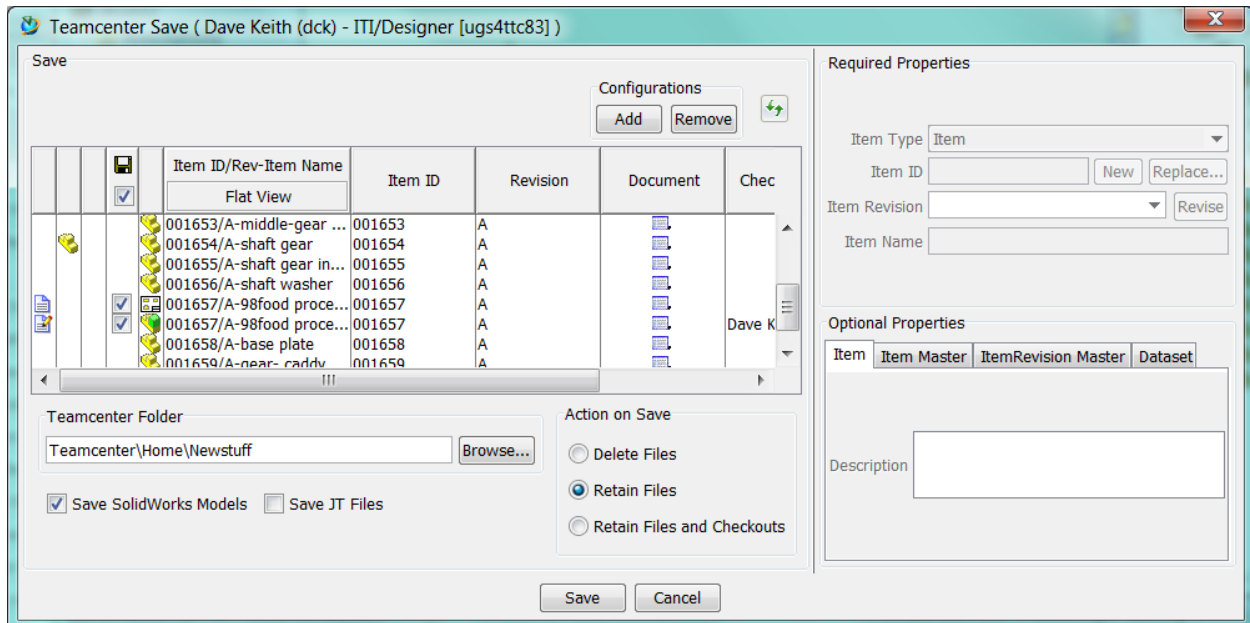


Figure 17 the Save Dialog

## The Save Dialog





Regardless of the command you use to save a SolidWorks model to Teamcenter, the Save dialog is the same. To save models to Teamcenter:

1. Put a check mark in the  **Save** column next to each model you want saved.

*You can save only models that are new to Teamcenter or models that you have checked out.*

*When saving a model that depends on another, and the latter has never before been saved to Teamcenter, be sure to put a check mark in the **Save** column for both models. The relationship between the models cannot be saved unless both models are saved to Teamcenter.*

*If you save a configuration, you must also save its document. If you do not save the document, the file that contains the configuration will not be saved to Teamcenter and any modifications you may have made to the configuration will be lost.*

The three columns to the left of the Save column indicate whether a change has occurred. A  new model icon will appear in the first column if the model does not exist in Teamcenter or the Item is not modified and is being saved to a new Item Revision. A  modified icon will appear in the first column if the model has been changed. A new  dataset icon will appear in the second column to indicate that a newer dataset version exists in Teamcenter. An  item revision icon will appear in the third column to indicate that a later item revision is available. Finally, if a checkbox does not appear in the fourth column, then the user has not checked out the dataset. See the section titled **Detecting Newer Model Versions** for more information on these columns.

The icons to the right of the **Save** column indicate the type of model in each row. The **Item ID/Rev-Item Name** column shows the objects Item ID as well as the Revision and Item Name, if the object exists in Teamcenter. You can click on the **Tree View** button in the **Item ID/Rev-Item Name** column to toggle the display of the tree view for hierarchical display. Click the button again to return to flat view.

On the right are the properties panels. The **Required Properties** panel contains a list of properties that must be given values the first time a model is saved. The **Optional Properties** panel contains tabs of properties that may be populated. Highlight one or more models to edit the properties. In the **Required Properties** panel, you can click in the **Item ID** field and type an item ID into the cell, or, you can click the **New** button to generate a new Item ID from Teamcenter, or you can click the **Replace** button to open a dialog for changing the item ID, using a pattern you define. Refer to the following section on **Creating a New Item** for more information about saving models to other item IDs, and a technique for starting new models from seed parts in Teamcenter.

*If you assign the same item ID to several models, such as a part and its drawing, they will be saved to the same item.*

If a model is new to Teamcenter, you can choose its item type. Click the dropdown to select from a list of eligible item types.


*If the item already exists in Teamcenter, the **Item Type** cannot be changed, unless you generate a new item ID for the model.*

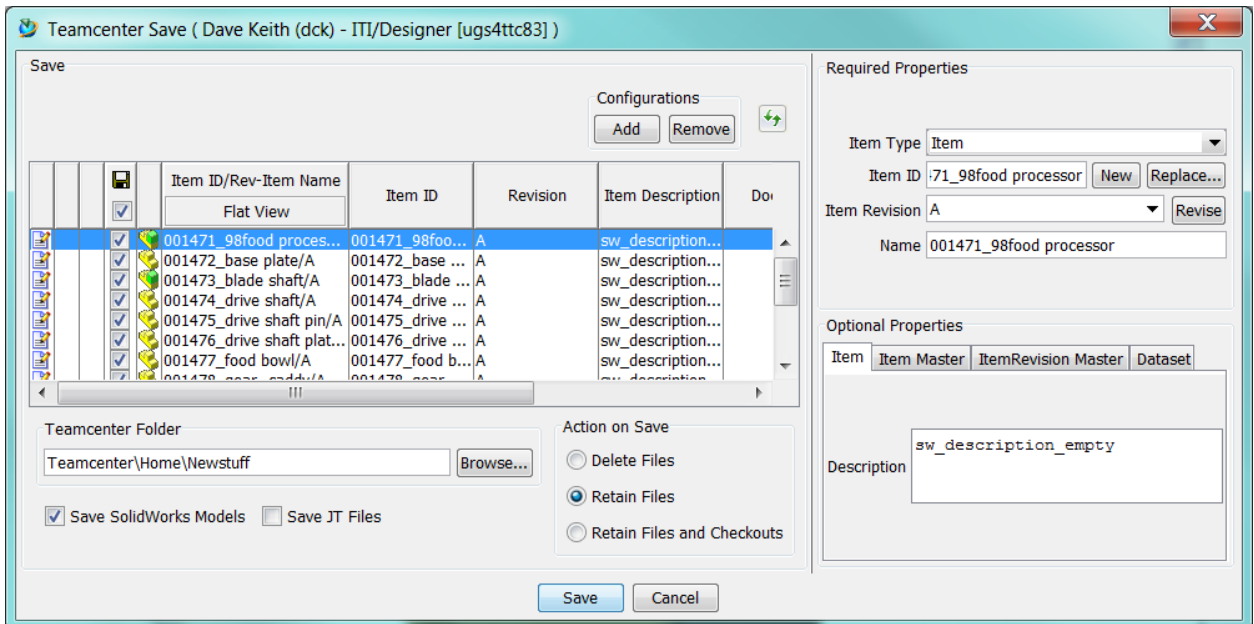
Other Teamcenter properties, such as Item Name and Description, may also be specified in the **Required Properties** panel. Depending on site configuration, more required properties may be shown. All required properties must be set before a model can be saved.

A property can be set on multiple models. Select several rows in the table, and then enter the property values that apply to all of the models.

*Some fields are not modifiable when multiple models are selected. In such cases, the field will not be editable. The Item ID and Revision fields will not be editable, but you can still use the **New**, **Replace** and **Revise** buttons.*

*When multiple models are selected, some value fields will show <Various>, indicating the value is not the same for all models.*

If a model's required properties are not fully populated in the Save dialog, the integration will alert the user by displaying a special  icon in the leftmost column, as shown in **Figure 18**.



**Figure 18 – One or more required property values is missing**

2. The save will not be allowed until all required property values are provided.

3. Select the Teamcenter folder to store all new data in Teamcenter. The **Browse** button enables the user to change the default folder. Once selected, the folder will be remembered throughout the session.

*In the Teamcenter Folder path, the top level “Teamcenter” is not an actual Teamcenter folder. Instead it is a pseudo folder representing Teamcenter as a whole. Models saved to “Teamcenter” will not be displayed in a user’s personal folder.*

4. Choose what you want done with the local model files after they have been saved to Teamcenter. From the “Action on Save”, select one of the following:

**Delete Files**

After saving the models, Teamcenter deletes their local files. The models are no longer checked out after they are saved.

**Retain Files**

After saving the models, the local files are left where they are. The models are no longer checked out after they are saved.

**Retain Files and Checkouts**

The models remain checked out to you after they are saved to Teamcenter.

5. Click the **Save** button at the bottom of the dialog. This saves the models that have a check in the **Save** column.

### ***Previewing models while Saving***

You can see a preview of any model by pausing the mouse over its row in the table. A flyover will appear listing the SolidWorks model type and model name along with a preview of the model.

*Previews may not be accurate. The preview comes directly from the file and is the active configuration at the time the file was saved to disk. As such, all configuration and document models for a file will show the same preview. If a file has never been saved to disk, its model(s) will not have a preview image.*



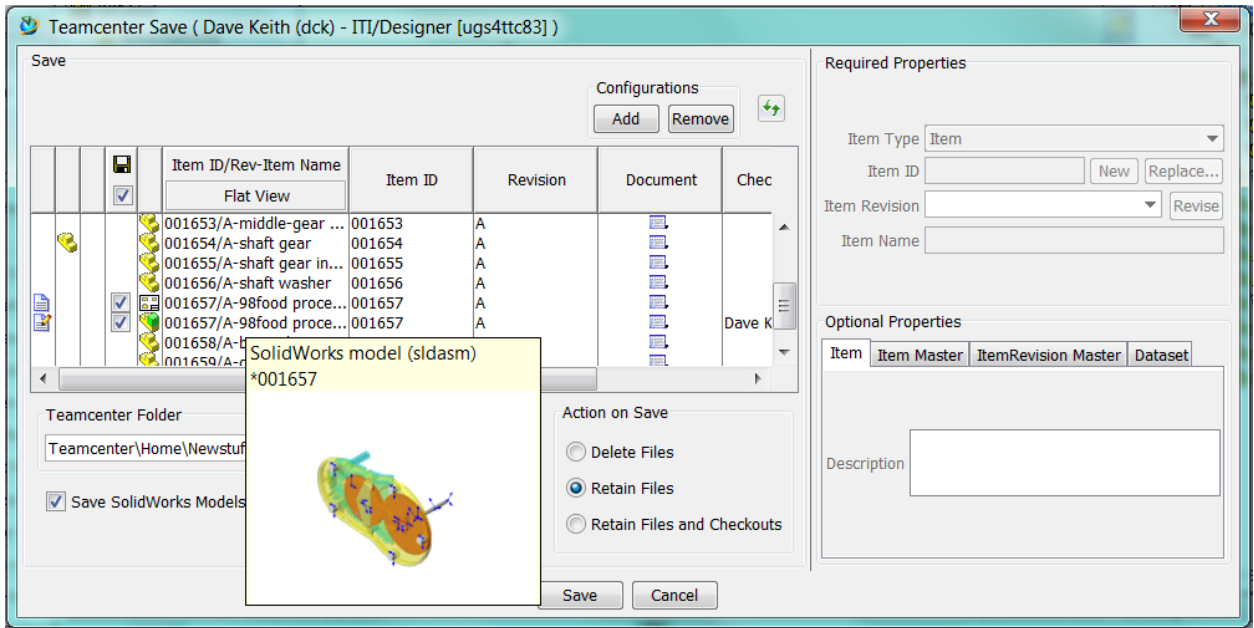


Figure 19 – Save dialog showing preview

## Updating Model Names while Saving

If a model is renamed in Teamcenter after it has been opened in SolidWorks, the local copy must be updated with the new name before it or any models that reference it can be saved back to Teamcenter. If the Integration finds models in the Save dialog whose local names no longer match their names in Teamcenter, a prompt similar to the one shown below will appear when the **Save** button is clicked. The **Details** button displays a list of the models whose names have changed in Teamcenter.

*A model must remain in the directory where it was placed when fetched from Teamcenter. If you move the model to another directory, the Integration will no longer know where the model came from in Teamcenter except by matching the model's name to a dataset, so if the Teamcenter dataset has been renamed, the Integration may assume the local model is new.*

The **Rename All** button renames models in SolidWorks to match their current names in Teamcenter, then proceeds with saving the models selected in the Save dialog. The **Rename None** button saves models to Teamcenter without renaming any in SolidWorks. The **Rename None** button will not be available if any model being saved requires renaming or references a model that requires renaming. Click **Cancel** to return to the Save dialog.

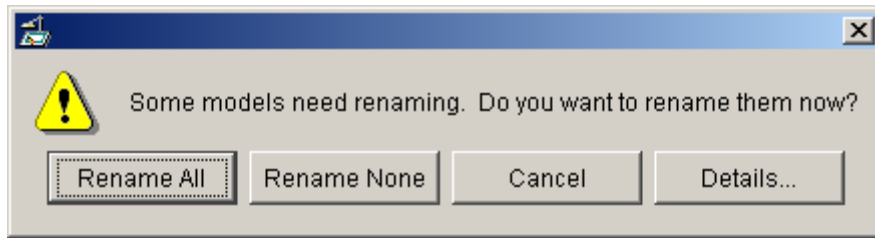


Figure 20 Renaming in-session models to synchronize with Teamcenter

## Creating a New Item

You can make the Integration save a model to another item by changing its **Item ID** column in the Save dialog. You also have the option of renaming the model in the SolidWorks session to match its new item ID.

*If the model is checked out to you, saving the model to a new item does not affect your reservation on the model in the “old” item. The dataset in the original item will remain checked out to you.*

Select the model's row in the **Save** dialog, and then use one of the following techniques to enter a new item ID.

1. Type the new item ID into the cell of the **Required Properties** panel.
2. Click the **New** button to have Teamcenter generate the next available item ID.
3. Click the **Replace** button in the Required Properties panel. Refer to the section on **The Replace Item ID Dialog** for more information.

If the original ID in the **Item ID** column referred to an item that already exists in Teamcenter, you will be asked to confirm that you want the model saved to a different item. A typical prompt is shown below. Click **Yes** to confirm that you want to assign a new item ID to the model. Click **No** if you do not want the model saved to a different item. **No** will also restore the previous item ID and discard any new value you might have entered in the **Item ID** column.

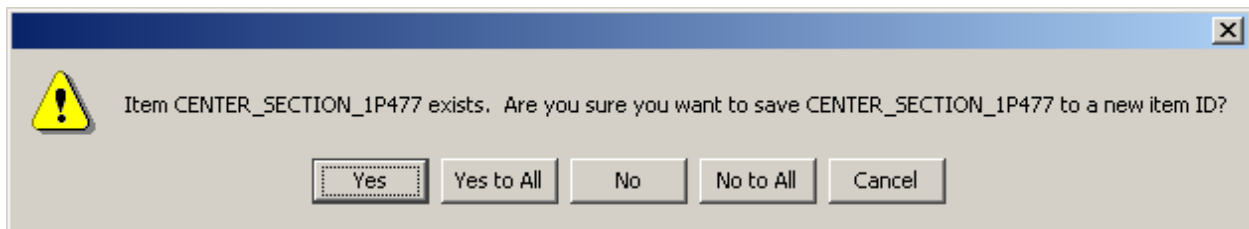
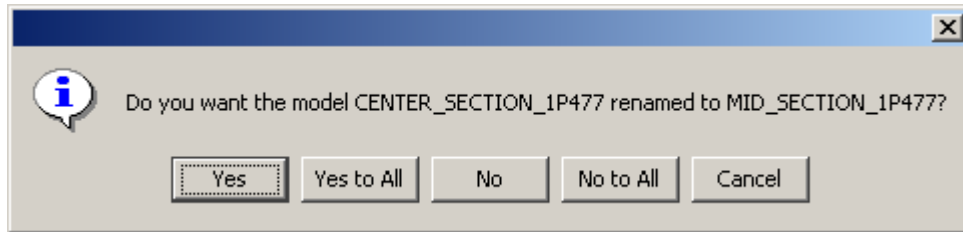


Figure 21 Confirmation dialog when assigning a new item ID

If more than one row is selected, you may receive this prompt several times. The **Yes to All** and **No to All** buttons have the same meaning as **Yes** and **No**, respectively, but they also apply your response to the remaining models in the selection. For example, if you want to assign item IDs only to models that do not yet exist in Teamcenter, click **No to All**. This prevents Teamcenter from generating new item IDs for rows that already refer to existing items and allows it to complete the assignment of item IDs to rows that refer to new items, without further prompting. The **Cancel** button stops Teamcenter from assigning any more IDs to the selected models, but it does not discard IDs assigned before the prompt.

If only one row is selected, **Yes** and **Yes to All** are equivalent, and **No** is the same as **No to All** or **Cancel**.

If you change an item ID, the Integration will also offer to rename the model to match the new item ID. For example, if you assign the item ID MID\_SECTION\_1P477 to a model named CENTER\_SECTION\_1P477, you would receive the prompt shown below. Click **Yes** if you want the model renamed in the SolidWorks session to match the new item ID. Click **No** if you do not want to rename the model.



**Figure 22 renaming the model to match the assigned item ID**

If more than one row is selected, you will receive this prompt several times. The **Yes to All** and **No to All** buttons have the same meaning as **Yes** and **No**, respectively, but they also apply your response to any remaining models in the selection, if their item IDs are also changed. The **Cancel** button stops Teamcenter from assigning any more IDs to the selected models, but it does not discard IDs or change model names assigned before this prompt appears.

If only one row is selected, **Yes** and **Yes to All** are equivalent, and **No** is the same as **No to All** or **Cancel**.

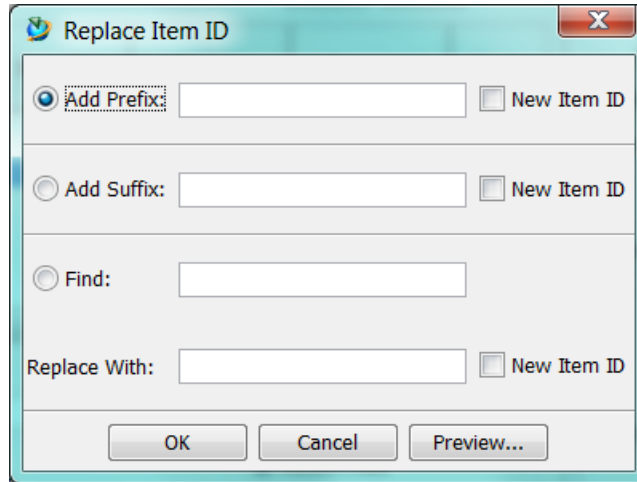
*Renaming a model in the Save dialog immediately renames it in the SolidWorks session. The SolidWorks session will continue to refer to the model by its new name even if you cancel the Save dialog.*

*You will not be prompted to rename a model if its item ID does not change.*

*If a model was previously saved to Teamcenter, renaming it in the Save dialog does not rename it in Teamcenter. The Integration will save the renamed model under its new item ID without affecting prior copies of the model stored in other items or under another name.*

## **The Replace Item ID Dialog**

The **Replace** button in the Save dialog's **Item ID** box displays the Replace Item ID dialog shown here. This dialog changes item IDs according to one of the following patterns.



**Figure 23 The Replace Item ID dialog**

- **Add Prefix** adds text to the beginning of every item ID highlighted in the Save dialog. The prefix text is whatever you type into the **Add Prefix** box. For example, if the **Add Prefix** box contains NEW\_, an item ID 1234567 will become NEW\_1234567. Alternatively, if **New Item ID** is checked, the **Add Prefix** box is ignored and the prefix is a new item ID generated by Teamcenter.
- **Add Suffix** is similar to **Add Prefix**, but adds text to the end of every item ID highlighted in the Save dialog. For example, if the **Add Suffix** box contains \_OLD, an item ID 1234567 will become 1234567\_OLD.
- **Find** attempts to match the contents of the **Find** box to text within each highlighted item ID in the Save dialog. If a match is found in an item ID, the matching text is replaced with the contents of the **Replace With** box. For example, if the **Find** box contains 345 and the **Replace With** box contains XYZ, an item ID 1234567 would be changed to 12XYZ67. Alternatively, if **New Item ID** is checked, the **Replace With** box is ignored and matching text is replaced with an item ID generated by Teamcenter. Only the last match is replaced in each item ID, so an item ID 345345 would be changed to 345XYZ. Wildcard characters have no special meaning in the **Find** box, so entering \* or . in the **Find** box will exactly match those characters.
- **Advanced** opens the Advanced Replace Item ID dialog, which may be used for more complex patterns, including wildcard characters. See The Advanced Replace Item ID Dialog section in the appendix for more information.

To change item IDs using a pattern:

Choose one of the three pattern types in this dialog and enter text in the appropriate box for your choice, or click the **Advanced** button to open the Advanced Replace Item ID dialog.

Click the **Preview** button to see how the item IDs in the Save dialog will change. If you have checked **New Item ID** to make Teamcenter generate new item IDs, a placeholder value will be substituted wherever a new item ID would appear. New item IDs will not be generated until the **OK** button is clicked.

Click the **OK** button to change the highlighted item IDs in the Save dialog.

## Creating a New Item Revision

You do not have to check out a model to revise it, but the model must be in the SolidWorks session. To create a new revision of a model, use the **Save** command to display the Save dialog, then:

- Click on the model's **Revision** column in the Save dialog to select it and use one of the following techniques to enter a new revision ID.
  - Type the new revision ID into the selected table cell.
  - Alternatively, click the **New** button in the **Revision** box above the table to generate the item's next available revision ID.

The new revision will be created when you click the **Save** button at the bottom of the Save dialog.

*When the Integration creates a new revision, it creates new datasets and new BOMs instead of copying the previous revision's contents to the new item revision. The new item revision's contents are derived solely from what you check in with the Save dialog, so models that are not saved during this operation, and non-SolidWorks datasets and BOM lines, will not appear in the new revision.*

## Saving Configurations and Design Tables

SolidWorks configurations can be saved as separate items in Teamcenter. This is possible for any configuration in a part or assembly document, including configurations that are created from a design table. For example, a part document might define a family of bolts in a design table. Each bolt should be treated as a distinct part in Teamcenter so it can be included in a bill of materials, with its own item ID and revisions. The Save dialog and several user preferences make this possible<sup>5</sup>.

If you want to create separate items for a model's configurations, you can add the configurations to the dialog by clicking on the model's row to highlight it, then clicking the **Add** button in the **Configurations** box. If no row is highlighted, configurations from every model are added to the dialog.

*Configurations that are not resolved, or are otherwise not in the SolidWorks session, will be brought into the session when the **Add** button is clicked.*

If you want to remove a configuration so a separate item is not created for it, click on the configuration's row to highlight it, then click the **Remove** button in the **Configurations** box.

## Exclude from bill of materials Flag

The SolidWorks provides a means to flag models for exclusion from the Bill of Materials. The SolidWorks integration to Teamcenter honors the flag, excluding the component from the Teamcenter BOM for the assembly during save. Instead of creating a SWIM\_dependency

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<sup>5</sup> See the Administration Guide for more details on user preferences.

relation for the component, a SW2BOM\_exclusion relation is created. During open, SW2BOM\_exclusion relations are considered required, so they appear in the open dependencies dialog even when the required filter is selected. If a model which has “exclude from bill of materials” selected is also suppressed at the time of save, then the model will not be in the BVR and will be stored as a SWIM\_suppressed\_dependency. As a suppressed component, the model will not be considered as required during the assembly load operation.

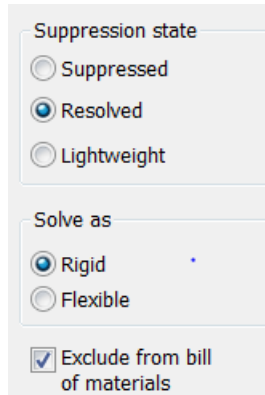


Figure 24 SolidWorks Exclude from Bill of Materials setting

## Saving Auxiliary Files

If the Integration has been configured to save *auxiliary files*, a set of checkboxes may appear across the bottom of the save dialog. Auxiliary files might be derived from your SolidWorks models, such as JT files for visualization in the Teamcenter Rich Client or PDF files of drawings, or they might be other files that you want to associate with your models. Checkboxes in the **Auxiliary Files** box allow you to choose the kinds of auxiliary files to save.

- The **Save SolidWorks Models** checkbox is always on by default. If you turn off this checkbox, the Integration will not save your models to Teamcenter, but it will still save auxiliary files for the models that have a check in the table’s **Save** column. Turn off the **Save SolidWorks Models** checkbox if you are using the Save dialog to save only auxiliary files to Teamcenter without their models.

*If Save SolidWorks Models is off, models will still be saved from the SolidWorks session to the local file system, in case an external program must be executed to create auxiliary files from your models.*

- If a **Save JT Files** checkbox is shown, a check tells the Integration to save JT files to Teamcenter. Turning off the check prevents the Integration from saving JT files.
- Additional checkboxes may be shown in the **Auxiliary Files** box, depending on how the Integration has been configured at your site.

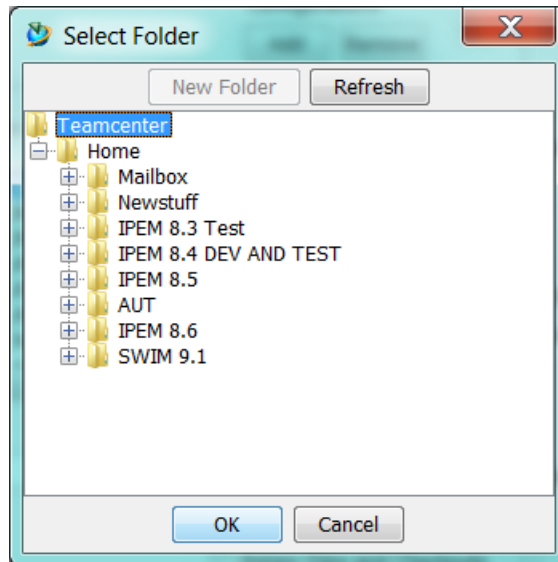
## Adding Items to a Teamcenter Folder

All models saved to Teamcenter are put into a folder. The first time you save models to Teamcenter, the destination folder is set to “Home\Newstuff” by default. You can change the destination by clicking the Browse button, then selecting a different folder, or creating a new

folder. For the remainder of the SolidWorks session, your last folder selection is retained so that subsequent saves will reuse that folder, unless you once again change the destination.

*The destination folder can only be controlled for models being saved to new items in Teamcenter, that is, being saved for the first time. Once a model and its item have already been saved, they are in a folder, and the Integration will not attempt to move or copy them to a different folder, regardless of what the user selects. However, the integration may create links to additional selected folders, if the Teamcenter Administrator has configured the system to do so.*

If you do not wish to explicitly select a folder destination for the new items, then you may select the top-level Teamcenter folder in the folder selection dialog:

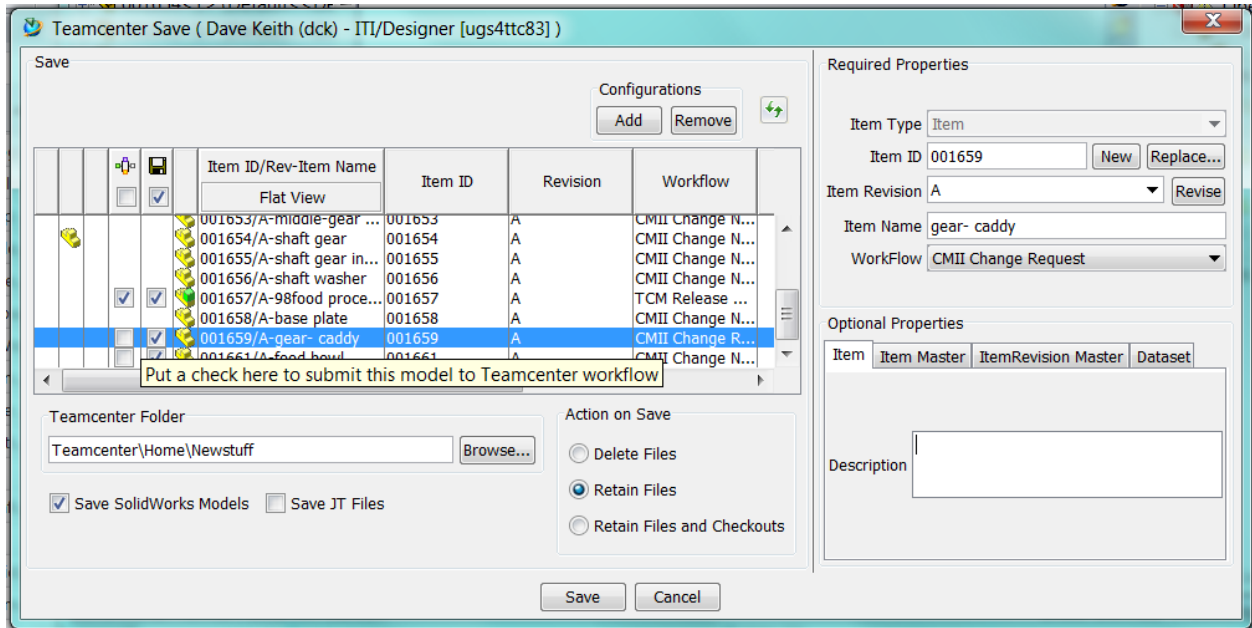


**Figure 25** Selecting the Teamcenter pseudo-folder

This is an abstract pseudo-folder that does not exist in Teamcenter, but serves as a placeholder for the Teamcenter database itself. When you select this folder as the destination, then the actual destination will be determined by a Teamcenter preference, which is documented in the Integration's Admin guide.

### ***Submitting Items to a Teamcenter Workflow***

The integration can submit selected item revisions to a designated Teamcenter workflow, during save to Teamcenter. The save dialog includes a column with a **Submit to Workflow** icon heading:



**Figure 26 Submitting an item revision to a workflow**


The default workflow is determined by your Teamcenter Administrator, but can be changed by selecting a different workflow from the drop-down list in the **Workflow** column. You can submit all saved models to a workflow, or a subset of them, by selecting the checkboxes under the **Submit to Workflow** column on the left.

The integration will submit the item revisions for the selected models to the designated workflow(s), but will not track their workflow status, and will not report errors that occur within the workflow, after the initial submission.

## Managing models in session

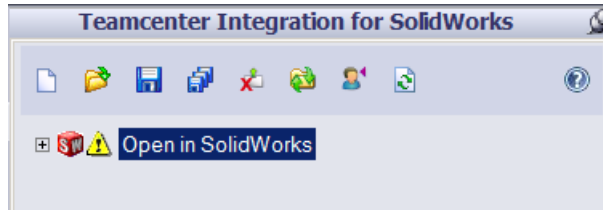
The integration provides several functions for monitoring the status of in-session models, and for updating models with newer versions or revisions from Teamcenter. All of these functions are available in the Teamcenter task.

### ***Detecting updates in Teamcenter***

While a user is working on an assembly in SolidWorks, other users may check in new versions or new revisions of components which are used by that assembly. This creates an “out of date” or “superseded” condition which must be recognized and managed. Each user should periodically press the **Refresh**  button in his Teamcenter task to check for updates in Teamcenter<sup>6</sup>. When one or more of his models has become superseded, the integration will display a warning icon at the top level of the model tree:

<sup>6</sup> The integration will automatically refresh the Teamcenter task after every Teamcenter operation is performed. But the user can work for long intervals of time in SolidWorks, without interacting with Teamcenter. Therefore it is important to explicitly refresh the session regularly.

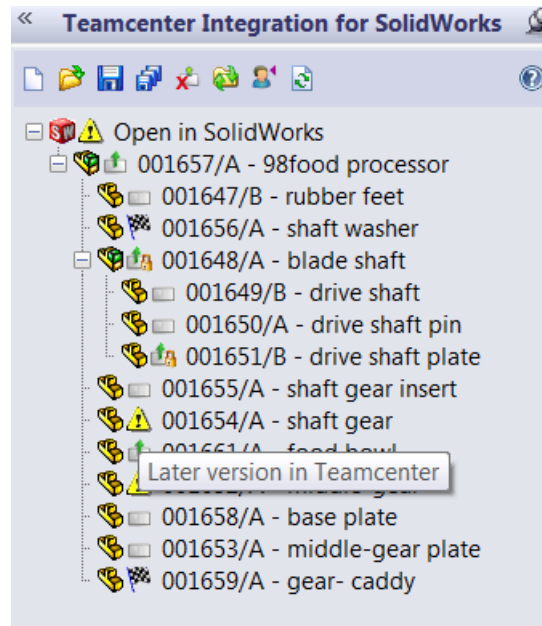




**Figure 27 Top level out of date warning**

This is simply an indication that something has been updated in Teamcenter, and that he should expand his model tree to see more details.

When he expands the tree, he will see the same warning icon next to each superseded model, as shown below. By hovering his mouse over the icon, he can learn whether his in-session model has been superseded by a new version, or by a new revision:



**Figure 28 Model is superseded by a new version**

In many cases, the user will only want to perform an update when new versions are available. A new revision implies that a more formalized change has taken place, and he will want to close, then reopen, his top-level assembly with the appropriate revision rule.

### ***Updating in-session models from Teamcenter***

When the user has decided to perform an update, he can select an assembly at the desired level in the structure (note that in the above example, the new version is a component of the top-level assembly, so he must update at the top level), and then select **Update Model** from the right mouse button pop-up menu:

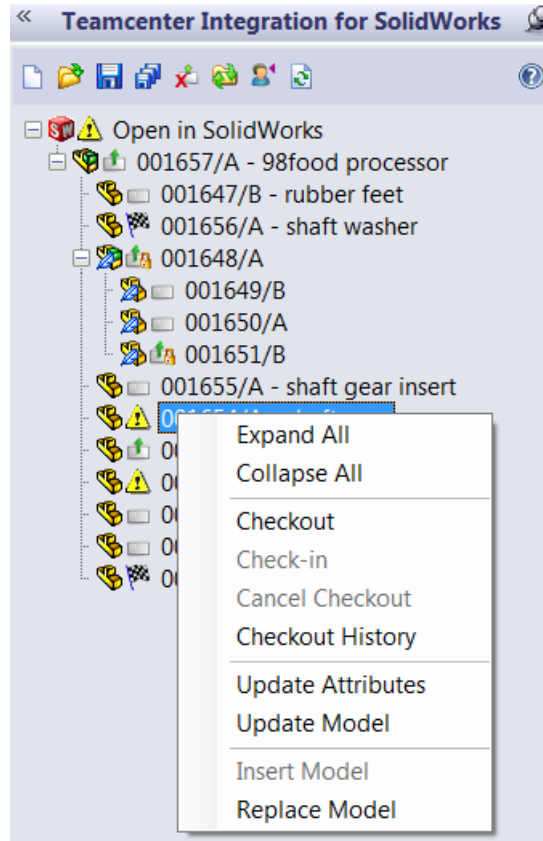
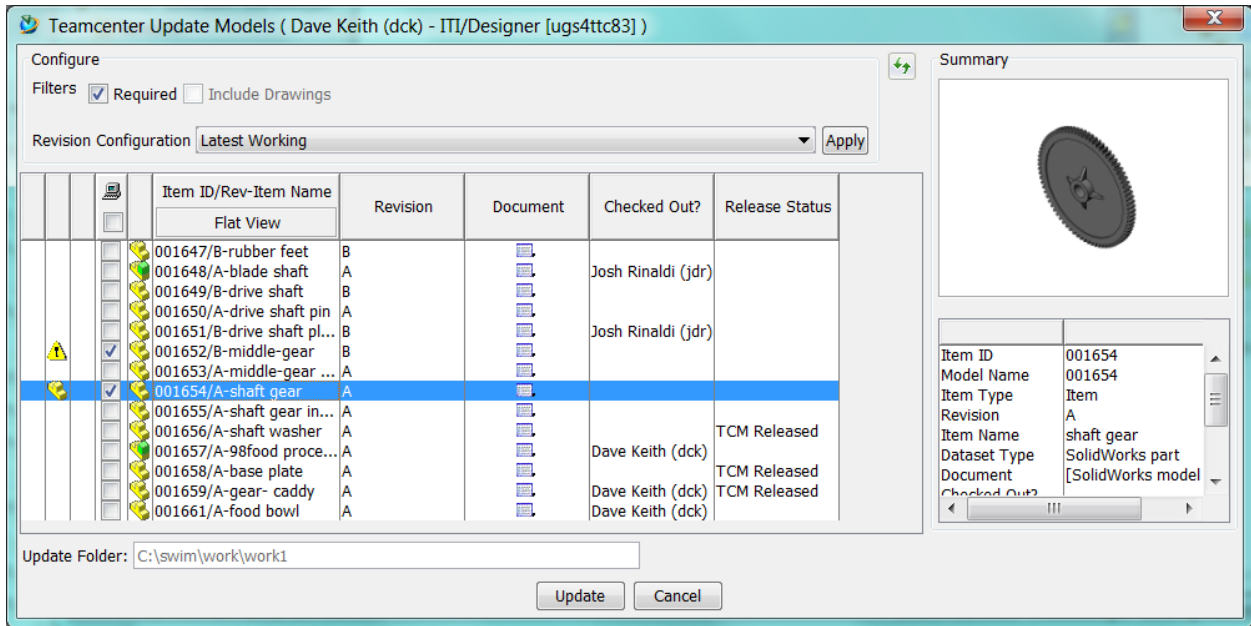


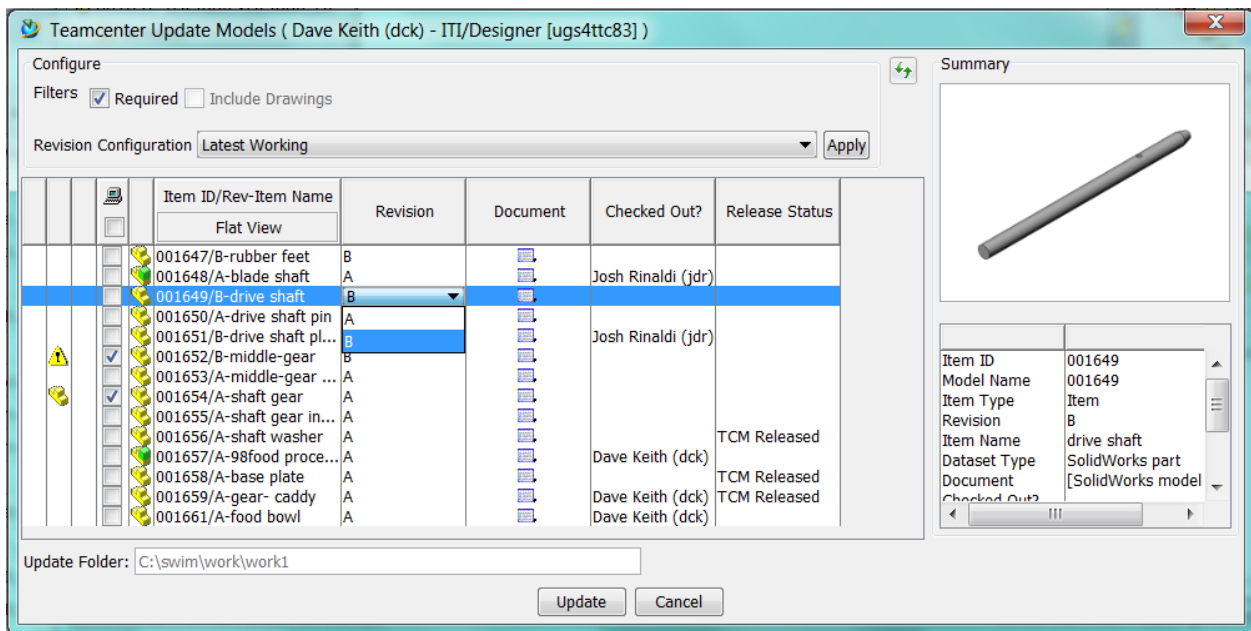
Figure 29 Updating the top-level assembly

This function displays the **Update Models** dialog, which contains the selected model and all of its dependent models. All models out of date according to the revision rule are automatically selected for update. Simply press the **Update** button, to get the desired models in session.



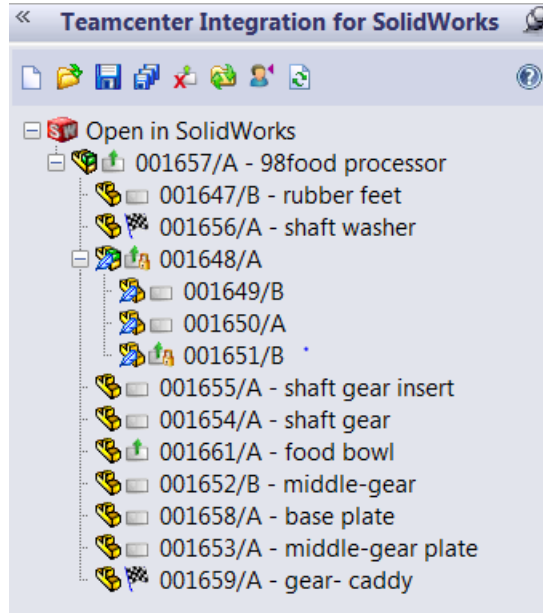
**Figure 30 The Update Models dialog, with versions pre-selected**

If the user wished to change the revision rule or filtering for the update, he can make changes in the configuration panel and press “Apply”. The table of models will be refreshed according to the new configuration. Users can choose a different revision of any model by clicking in the revision column and selecting a new drop down. When the revision of a model is changed, all of its dependents are reconfigured in accordance with the setting in the configure panel. Whenever the table is updated, the Update check boxes are updated for the models. If the user wishes not to update a model, he can uncheck the box.



**Figure 31 Updating to a newer revision**

After the update has completed, his Teamcenter task will be automatically refreshed to remove the out of date warning icons:

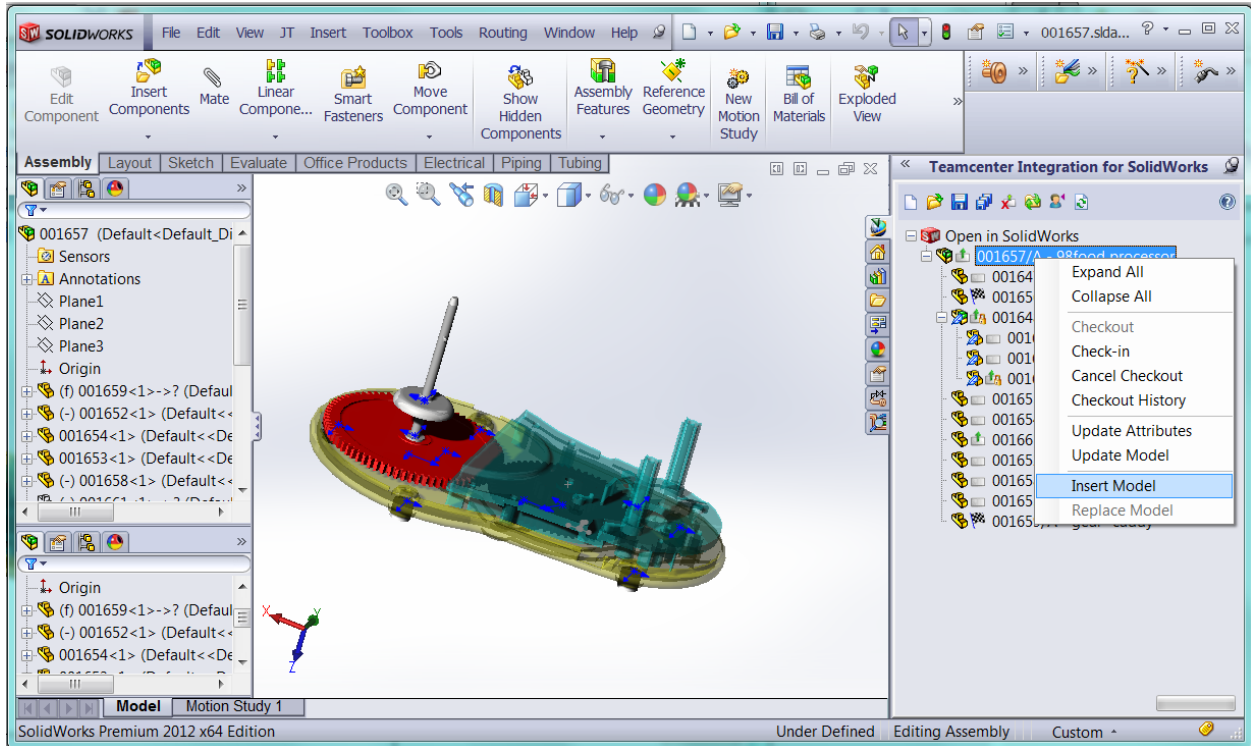


**Figure 32 Up to date session**

*Lightweight models can be updated even though they are not open in the SolidWorks session. However, the attributes cannot be updated on the model. If desired, the attributes can be updated later by resolving the model, selecting the model in the Teamcenter tree, and then performing right mouse button **Update Attributes**.*

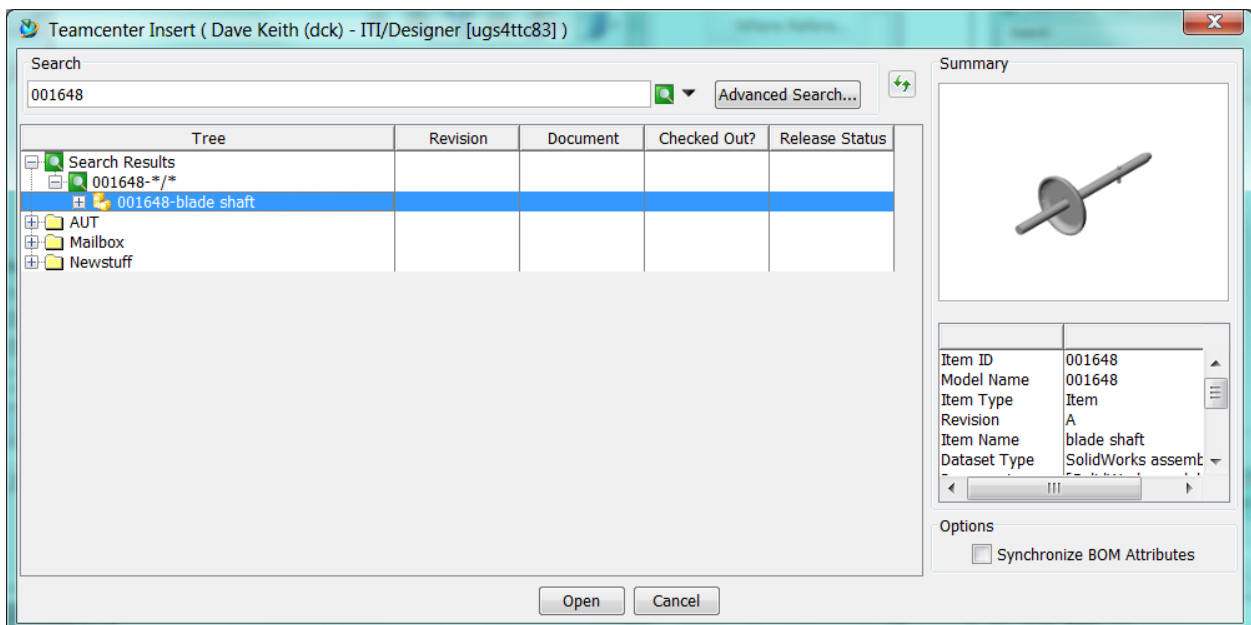
### ***Inserting models from Teamcenter***

While working with an assembly, the user can conveniently locate a part in Teamcenter and add it to his assembly, without opening it into a separate window. This is done by selecting the assembly to which the new model should be added, then selecting **Insert Model** from the right mouse button pop-up menu:



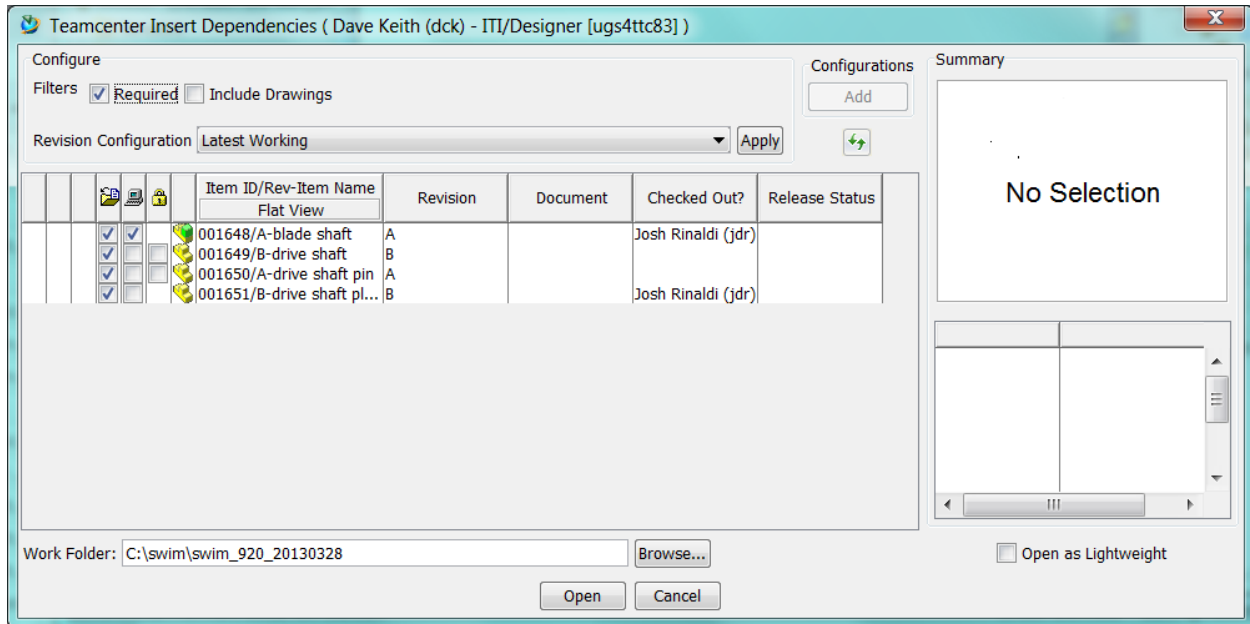
**Figure 33 Adding a new model to a selected assembly**

The **Teamcenter Insert** dialog is displayed, allowing the user to navigate to, or search for, the desired model.



**Figure 34 The Teamcenter Insert dialog**

Once he selects the desired model, pressing **Open** will display the **Teamcenter Insert Dependencies** dialog:



**Figure 35** the Teamcenter Insert Dependencies dialog

Here he can change the revision rule, change filter options, and perform all the other configuration tasks which are available in the **Teamcenter Open** dialog.

Finally, when he presses the **Open** button, the selected model, and its dependencies, will be fetched from Teamcenter and displayed in the SolidWorks **Insert Model** dialog, from which he can use standard SolidWorks functionality to drag and drop the model into his assembly:

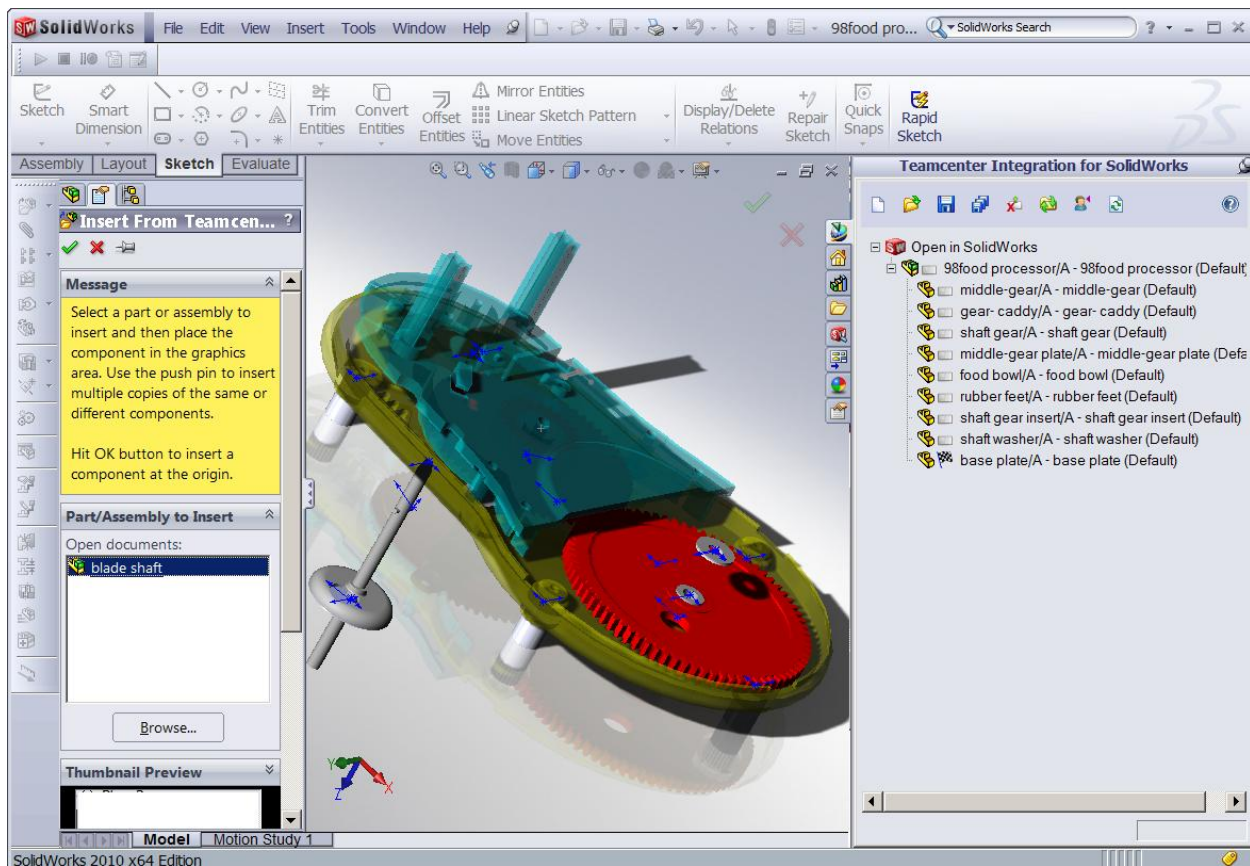
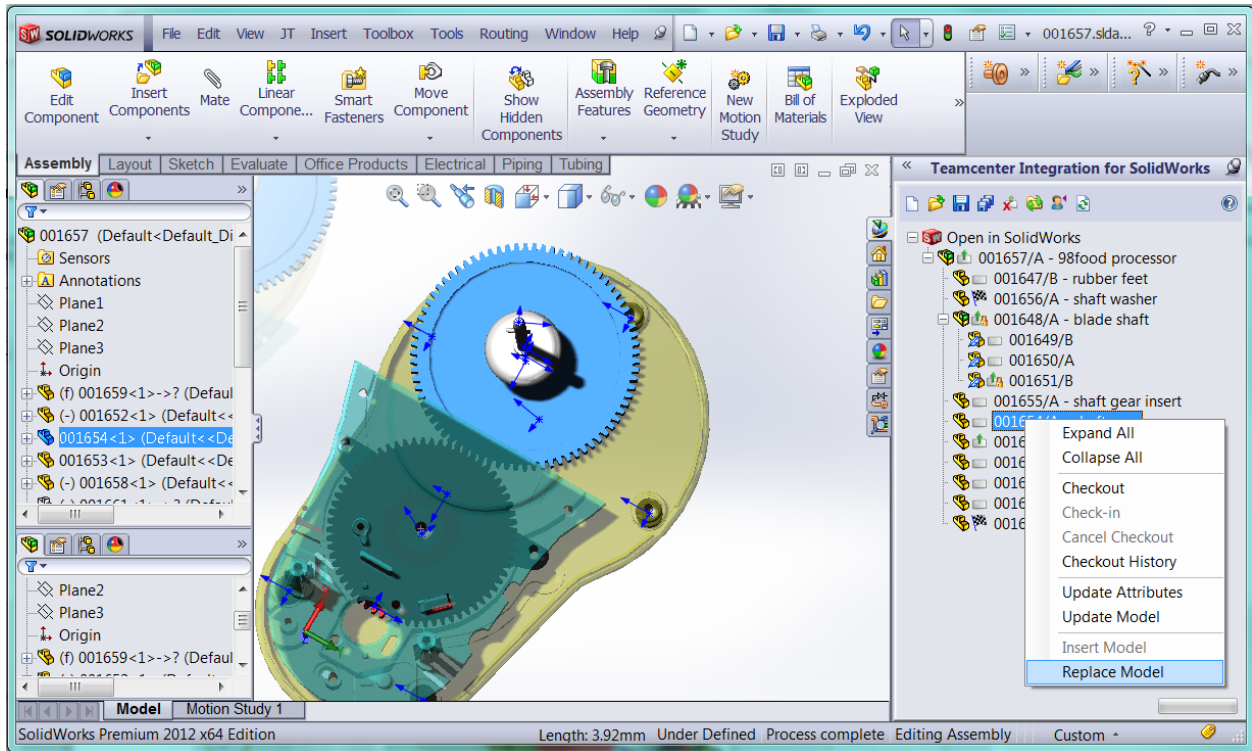


Figure 36 Inserting the model from Teamcenter

### ***Replacing models from Teamcenter***

The user can replace a selected component within his assembly with a substitute component from Teamcenter. He selects the component to be replaced, in the Teamcenter ask, then selects **Replace Model** from the right mouse button pop-up menu:



**Figure 37** Selecting a model for replacement

The **Teamcenter Replace** dialog is displayed, which allows him to search for the substitute model in Teamcenter:



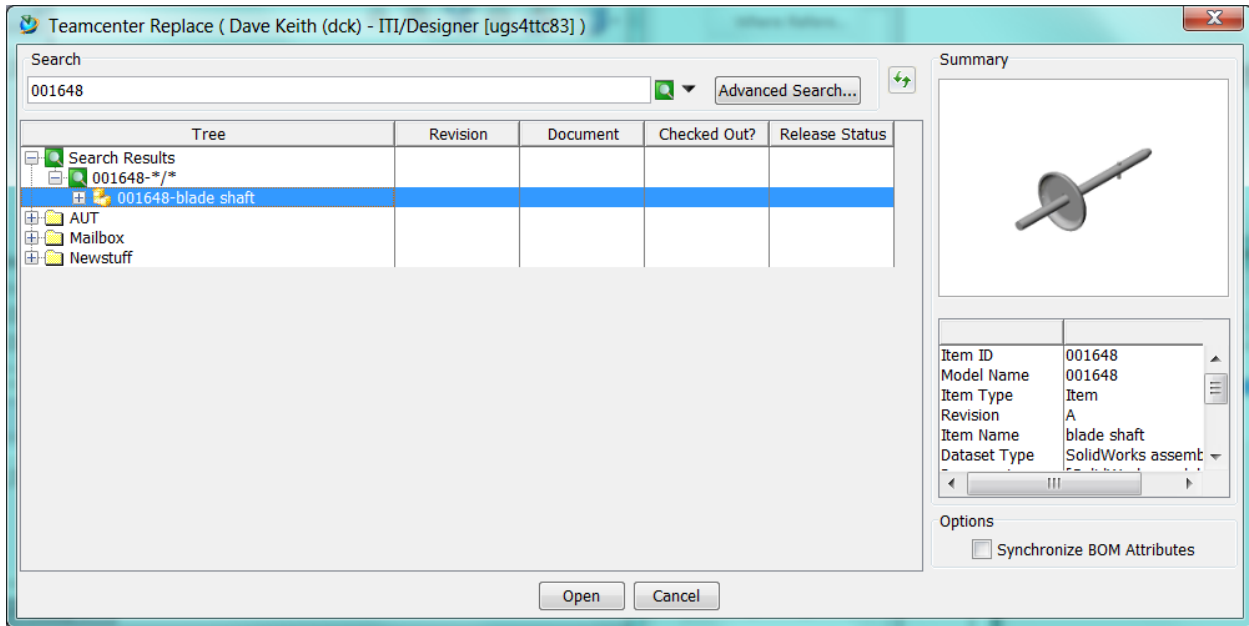


Figure 38 Searching for the substitute component

When he presses **Open**, the **Teamcenter Replace Dependencies** dialog will appear. If the substitution model is an assembly, he can change the revision rule, modify filter options, and perform all other configuration tasks which are available in the **Teamcenter Open** dialog:

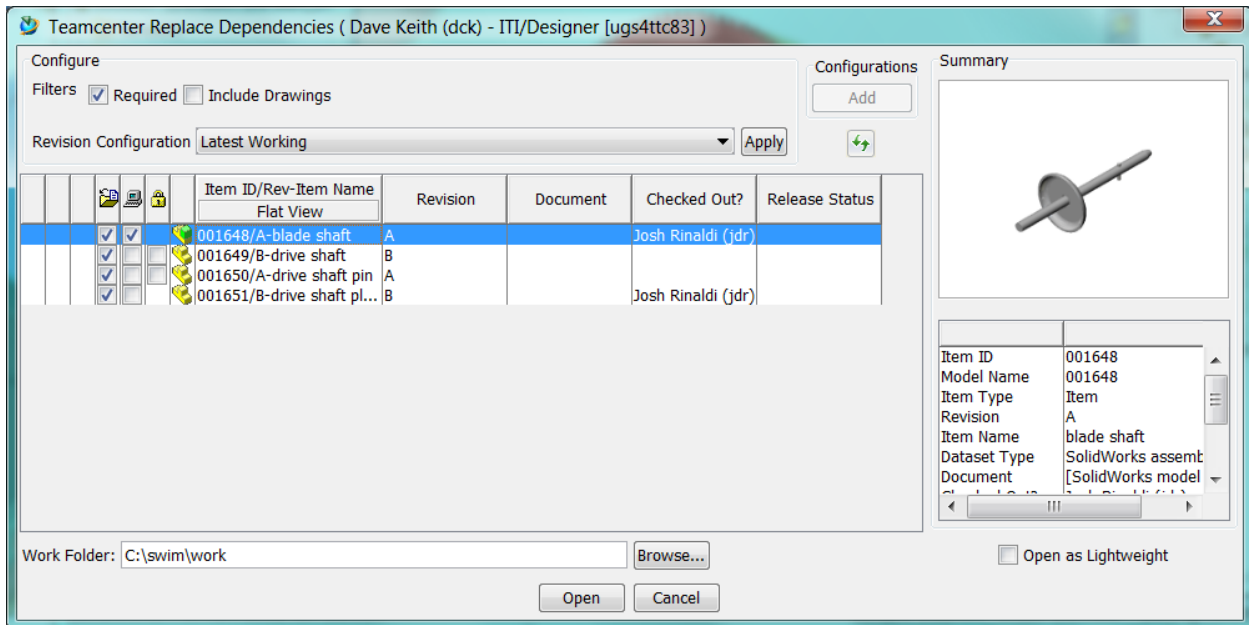
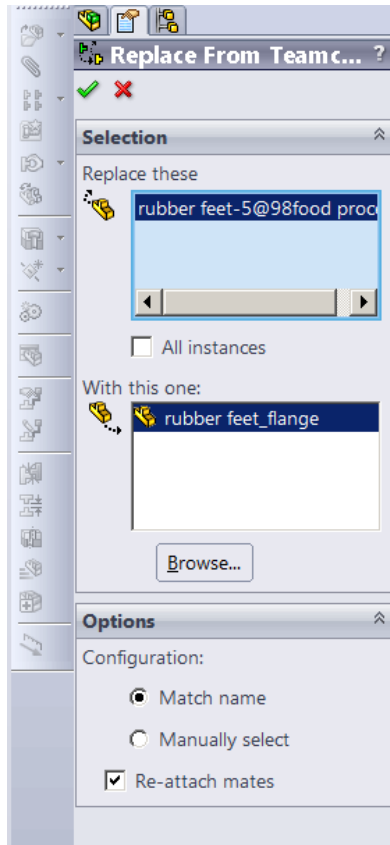


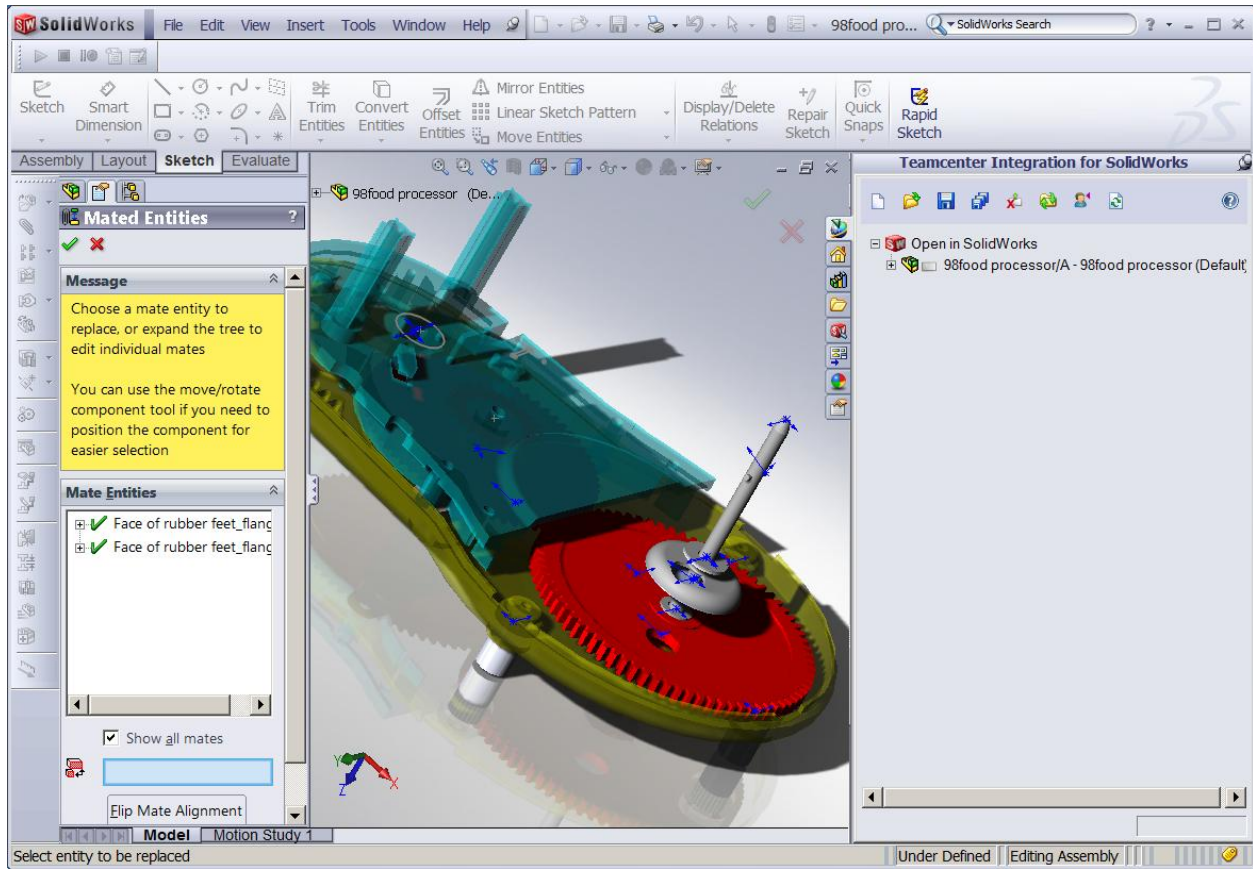
Figure 39 Replacing one part with another from Teamcenter

When **Open** is pressed in the **Teamcenter Replace Dependencies** dialog, the selected model will be fetched from Teamcenter and displayed in the SolidWorks **Replace Component** dialog, from which the user can select options for the replace operation:



**Figure 40 The SolidWorks Replace Component dialog**

When he proceeds with the replacement, standard SolidWorks functionality allows him to manage assembly constraints such as mates:



**Figure 41 The SolidWorks Mated Entities dialog**

And the component is replaced in the assembly.

*The success or failure of the replace operation, within the CAD session, is not under the control of the Teamcenter integration. It is dependent upon the similarity of the CAD models, and the compatibility of the substitution model with the parent assembly.*

As illustrated in this example, only the first occurrence of a selected component will be replaced. This is because the Teamcenter task shows “packed” occurrences, and it is not possible to select an individual occurrence for a replacement. If the user wishes to replace all occurrences of the component, he must do so using standard SolidWorks functionality, by selecting each remaining occurrence and performing the replacement operation.

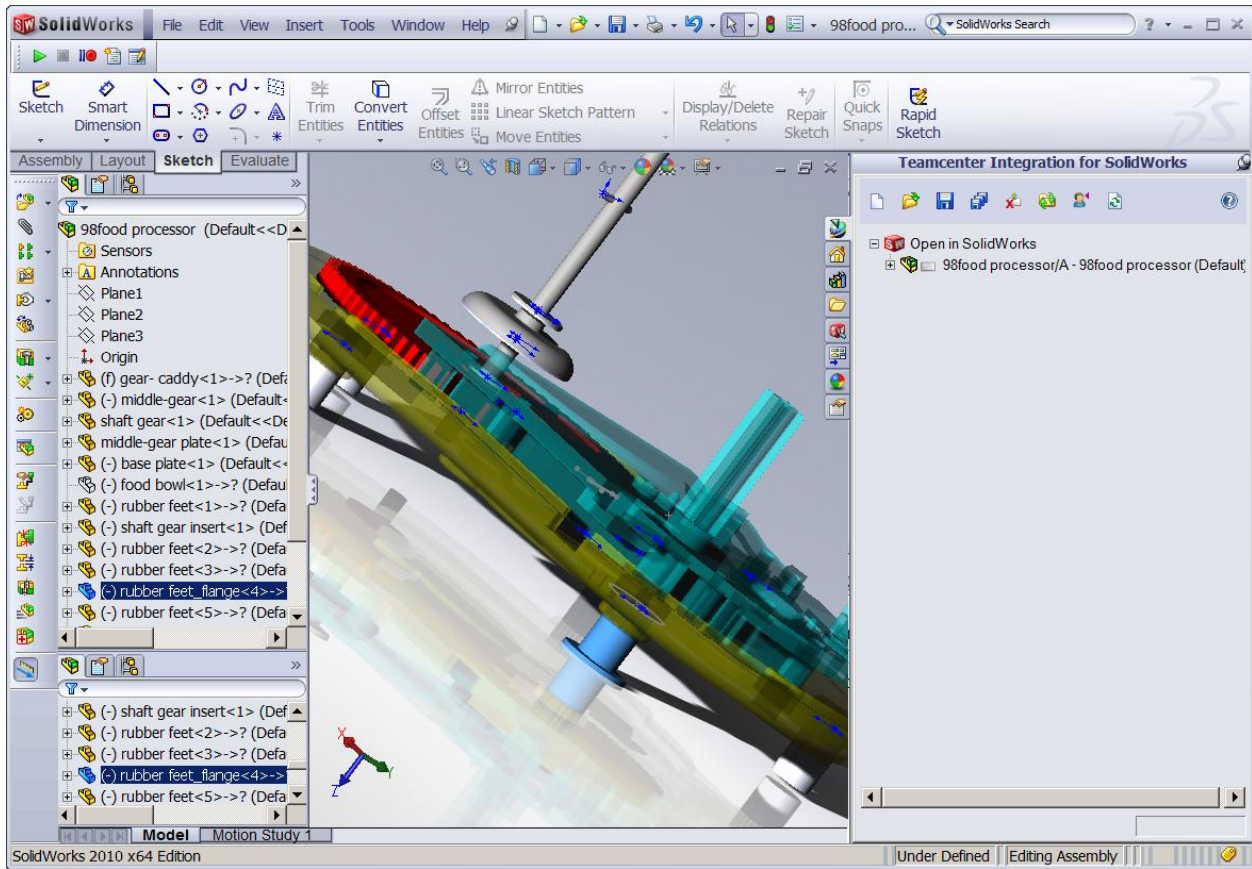



Figure 42 Only the first of 4 occurrences is replaced

## Managing Work Directories

Two commands are available for updating local copies of SolidWorks models when newer versions become available in Teamcenter. The **Update Directory** command displays the Update dialog shown below, which initially shows models in the current directory. You can browse to other directories on your file system, update models to export their latest versions from Teamcenter into the directory, open models in your SolidWorks session, or delete files from the directory.

### *The Update Dialog*

The  **Update Directory** command displays the contents of a directory. The Update dialog shows the models in the directory, but it does not show relationships between the models, so the **Tree** button gives only a flat view of the directory's contents. Also, only SolidWorks documents will be shown. Configurations will not be visible in a directory.

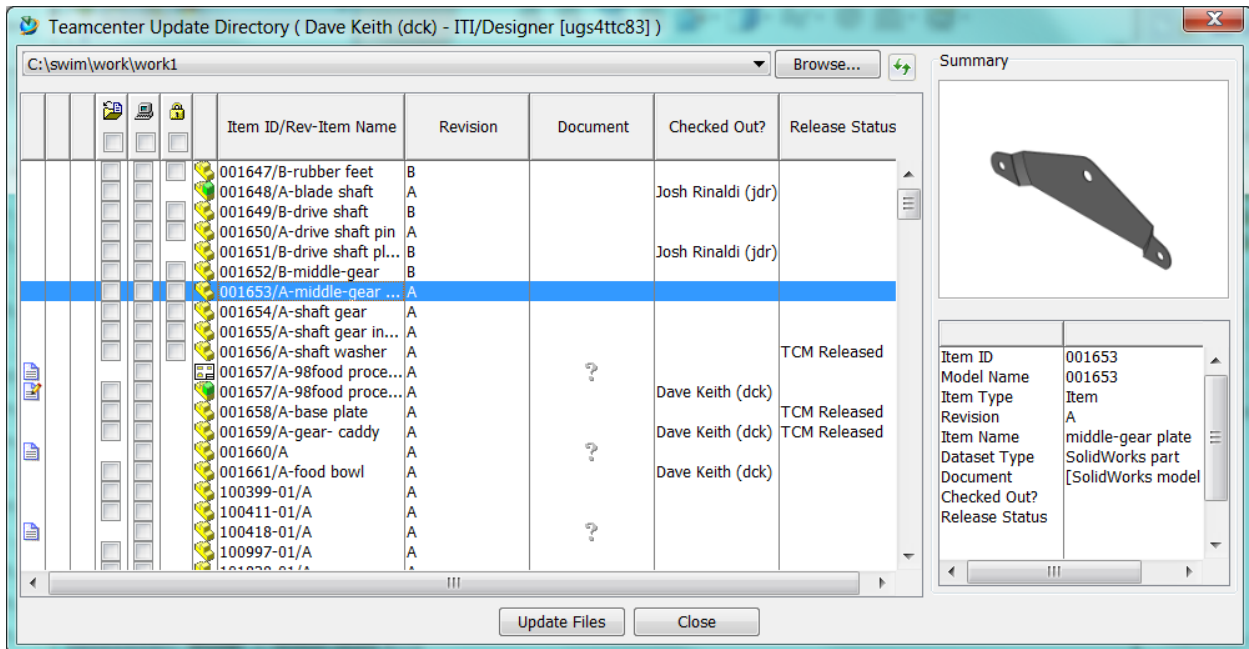




Figure 43 The Update Directory dialog


## Browsing and Updating Files

To update model files with their latest versions from Teamcenter:

1. Check the columns that apply to what you want to do:

 Put a check in the **Fetch** column to indicate that the latest version of the model's file should be exported from Teamcenter to the directory. Models that do not exist in Teamcenter will not have a checkbox in this column.

 Put a check in the **Open** column if you want to open the model in SolidWorks after its file is updated. Models which are already open in the SolidWorks session cannot be fetched or opened.

 Put a check in the **Checkout** column if you also want to reserve the model when the file is exported.

*You can put a check in the **Open** column without checking the **Fetch** column if you want to open the model without updating its file. The model will be opened when you click the **Update Files** button.*

The first three columns, to the left of the **Fetch** column, indicate whether a change has occurred in a model. By default, a check will appear in the **Fetch** column if a newer version of the model is in Teamcenter and no unsaved changes have been detected in your local copy of the model. For more information on these columns, see the section on Detecting Newer Model Versions.

The icons to the right of the **Checkout** column indicate the type of model in each row.

2. If you want to update your model with a file from a different item revision, click on the model's cell in the **Revision** column to choose from a menu of the available revisions in Teamcenter.

If a model you have selected to update has new dependencies in Teamcenter, the Dependencies dialog will appear later with all of these models in it. If you want to change the rule that selects revisions before the Dependencies dialog appears, select the rule from the menu in the **Revision Configuration** box. A filter may also be configured for use by the Dependencies dialog by clicking on any checkbox in the **Filters** box. The **Revision Configuration** box is described in the section on The Open Dependencies Dialog, and filters are described in the section on Filters.

*If you are updating a directory, the Teamcenter Integration for SolidWorks will look for dependencies in that directory before attempting to fetch them from Teamcenter. If you are updating models in the SolidWorks session, the Integration will look for dependencies in the SolidWorks session. Models that are locally available elsewhere could be overlooked and fetched again from Teamcenter.*

3. Click the **Update Files** button at the bottom of the dialog to update the model files, reserve models in Teamcenter, and open models in SolidWorks.

Unlike other Integration dialogs, the Update dialog does not close after you click the **Update Files** button, unless **Close This Dialog After Update** is checked. When the files have been updated, you can browse to another directory by clicking the **Browse** button in the dialog's top row of buttons. The directory's contents will appear in place of the contents previously shown in the Update dialog.


4. When you are finished with the Update dialog, click the **Close** button at the bottom of the dialog.

## **Deleting Files**

When the Update dialog displays the contents of a directory, the Right Mouse Button **Delete** can delete models from that directory.

1. Click on a row in the dialog's table to highlight it. More than one row may be highlighted if you want to delete several models at one time.
2. Select **Delete** from the RMB popup menu to delete the highlighted models from the directory.
3. When you are finished with the Update dialog, click the **Close** button at the bottom of the dialog.

## **Canceling a Checkout**

The  **Cancel Checkout** command displays the Cancel Checkout dialog shown below. The table in this dialog lists all of the SolidWorks models that you have checked out for modification.

A model need not be open in the SolidWorks session to appear in this table. If you put a check in the **Cancel** column, your checkout of that model will be canceled when you click the **OK** button.

Canceling a checkout means you can no longer save the model back to Teamcenter. Other users may checkout the model for modification after you cancel your checkout.

If a model shown in the table is in the SolidWorks session, the first three columns indicate changes that may have occurred to it. For models that are not in the SolidWorks session, the icons shown in these columns depend on information gathered when the models were previously saved to Teamcenter. For more information, see the section on Detecting Newer Model Versions.

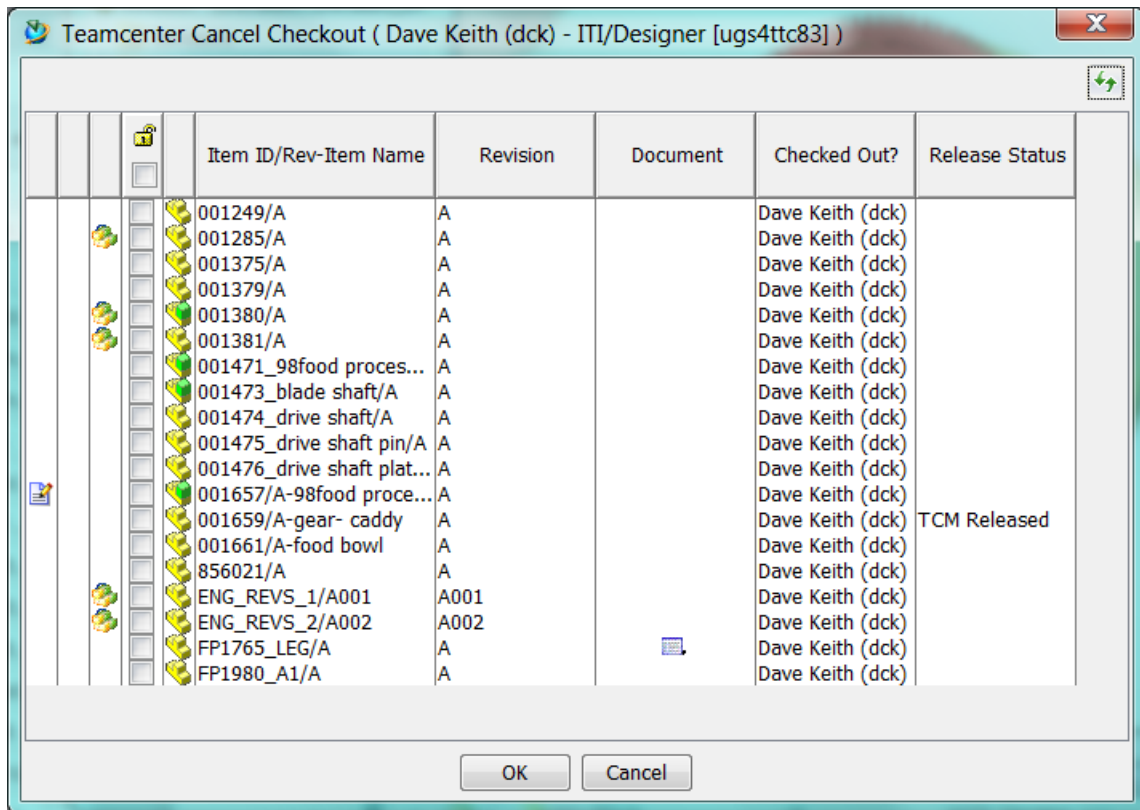


Figure 44 The Cancel Checkout dialog

To cancel the checkout of one or more models:


1. Put a check mark in the **Cancel** column for each checkout you want to cancel.

*When canceling the checkout of a document that has separately checked out configurations, put a mark in the **Cancel** column for all of the configurations that are checked out, too.*

2. Click the **OK** button at the bottom of the dialog.

*Local copies of the models are not affected by canceling checkouts. If you want to replace them with current versions from Teamcenter, use the Open command to fetch the models of interest after canceling checkouts.*

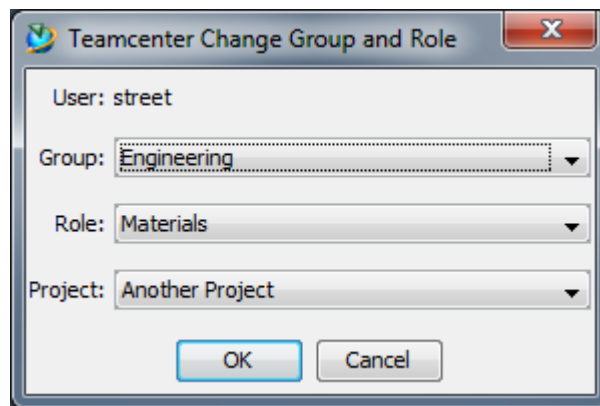
## Changing User Settings

The  **User Settings** command displays your current user name, group, role and project in Teamcenter and allows you to select a different group or role or project for subsequent operations.

### **Changing Group and Role and/or Project**

To change your group and role and/or project:

1. Select the group to which you want to change, using the **Group** pull-down. This may change the roles that are available in the **Role** pull-down.
2. Select a role from the **Role** pull-down
3. Select a project from the **Project** pull-down. This is optional and independent from the group and role selections.
4. Click the **OK** button to close the User Settings dialog.





**Figure 45 The User Settings Dialog**

## Integration to Teamcenter Rich Client

Customers may use the **Teamcenter Rich Client** to perform a limited set of functions on SolidWorks data. Several of these functions are also available from within the CAD session.



## Launch to SolidWorks Button

The integration installation process adds the  button into the toolbar of the Teamcenter Navigator application. In the Navigator, if a user selects an Item, Item Revision, or Dataset, and then selects the  button, the object will be sent to Solidworks. If no session is running, a new Solidworks session will automatically be started, and the integration's **Login** dialog will appear, followed by the **Open** dialog. If SolidWorks is already active, the **Open** dialog appears immediately, without requiring a new login. In both cases, the Open dialog will display the model sent from Teamcenter, and will automatically select it. In most cases, the user only has to press the **Open** button to proceed to the **Open Dependencies** dialog.

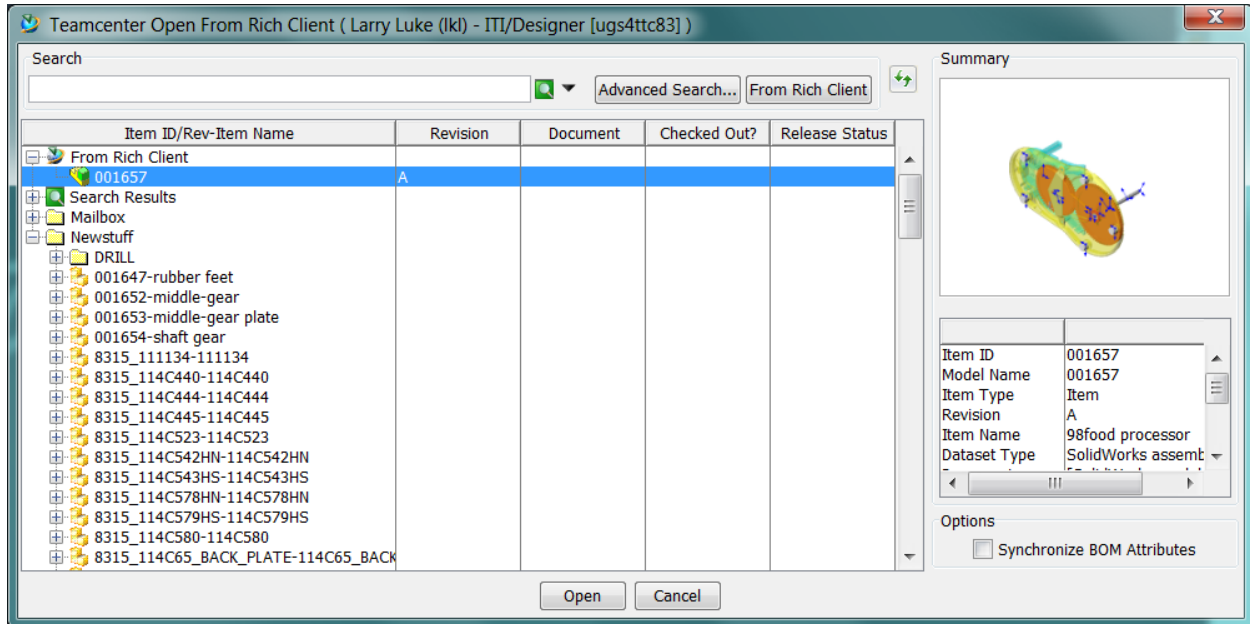


Figure 46 The Open dialog displaying a drawing sent from Teamcenter

## Revise

The Teamcenter Rich Client may be used to revise SolidWorks data. References are maintained such that the new revision of an assembly or drawing may be opened in SolidWorks. Without special configuration, SolidWorks datasets are always copied as objects, not references, when a new revision is created.

## Save As

The Teamcenter Rich Client may be used to save an existing item to a new item with a new item ID. SolidWorks references are maintained such that the new assembly or drawing may be opened in SolidWorks, however the dataset belonging to the new item retains its name; it is not automatically renamed to match the new item ID.

## Baseline

The Teamcenter Baseline function may be used on assembly structures that contain SolidWorks data. After the baselined item revision is created, it can be opened into SolidWorks for

reference. It cannot be checked out because by definition a Teamcenter baseline is frozen. It is recommended to always create precise baselines for structures containing SolidWorks data<sup>7</sup>.

## Appendix

### Configurations

Every SolidWorks part or assembly has one or more configurations. Not all configurations are used the same way. The design intent of the configuration affects how the configuration and its associated document should be managed. Configuration intended usage:

1. Product – A configuration which is intended to be a product, component of product or tool.
  - a. Family Member – The document as a whole is a family of parts or assemblies (products) which have a common definition and usage. The intent is that each member of the family is to be manufactured. Often family members are defined in a design table. A Family Member is a product, so it aligns with the definition of “Product”, and the term “Family Member” will not be used again in the Configuration text that follows.
2. Representation
  - a. Simplified Representation – Usually an assembly configuration, which is not intended for manufacture. The representation is used to simplify to work on a specific aspect of the structure during design
  - b. Visualization – Usually an assembly configuration, which is intended to show what a structure looks like in specific orientations or component positions

### Managed and Unmanaged

The integration enables configurations to be managed or unmanaged. Managed equates to deserving of an Item ID in Teamcenter. In general, product configurations should be managed and representation configurations should remain unmanaged. The integration provides preferences and buttons on the save dialog to establish whether a configuration is managed.

### Save Dialog Buttons

Also see the documentation of these buttons in the save dialog section of this document.

- **Add** – The Configuration **Add** button is used to add the non-active and non-referenced configurations of one or more documents to the save dialog.
- **Remove** – The configuration **Remove** button is used to remove a configuration from the save dialog. When removing an assembly configuration, the assembly’s relationships are merged back to the assembly document.

### Recommendations for working with configurations

- Limit use of the Configuration Add button on the save dialog. Add should be used when multiple configurations in a document must be managed, and the document is being saved to Teamcenter for the first time.


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<sup>7</sup> See the Teamcenter documentation for more information on baselines.

- Users should never remove or rename managed configurations from a document saved in Teamcenter. Doing so will result in a configuration which cannot be opened. If a rename is required, the recommendation is to create a new configuration, or to use swimrename.
- Users should never remove “Master Configurations” from an assembly document which has been saved to Teamcenter. Doing so can corrupt BOMs.
- The assembly configurations of a document often have different component relationships (a component may be suppressed in one configuration and not in another, or a component referenced within a configuration can be different) thus resulting in different BOM structures to be managed.
  - Assembly documents are best managed without “Master Configurations”. In this way all managed configurations will have their own Items and therefore BOMs are not subject to changes in other configurations.
  - If the assembly does contain a “Master Configuration” it is preferred that the “Master Configuration” does not have any suppressed components. If the “Master Configuration” does not have suppressions, then the document can be opened from Teamcenter and user can switch to any configuration as all data will be fetched.
  - If all the configurations of an assembly document have the same relationships, it is acceptable for the document to contain a Master Configuration.
  - If an assembly document has been managed in Teamcenter with a Master Configuration, do not remove the Master Configuration, or BOMs may become corrupted within Teamcenter.
  - Configurations with the same name as the document are not recommended for assembly documents.
  - While saving an assembly, if a configuration appears (either during generation of the save dialog or from selecting **Add** button) it is best not to use the configuration **Remove** button. Use of the remove button is best for part configurations.

## ***Toolbox Parts***

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.

Toolbox parts are managed similar to assemblies and parts, with the following exceptions:

1. Configuration of the toolbox directory is required to recognize toolbox parts. Configuration is accomplished by specifying the sw.toolbox.dirs in the swim.cfg file.
2. The open command only opens toolbox parts within an assembly. A toolbox part cannot be selected for open or fetch in the open dialog.
3. JT files cannot be generated via Teamcenter Dispatcher. The only means to generate JT files is during the save operation.
4. Toolbox parts cannot be duplicated or renamed by the integration. The model name must remain unchanged.
  - a. If a toolbox part has not yet been saved to Teamcenter, then the **New Item Id** and **Replace** buttons on the save dialog will allow the toolbox item id to be set, but the model name cannot be changed.
  - b. If a toolbox part has been saved to Teamcenter, then the **New Item Id** and **Replace** buttons on the save dialog will ignore toolbox files, to avoid duplication of item ids and renaming of the models.
  - c. The swimrename utility will not rename sw2Tbx datasets
  - d. Sites should avoid having multiple items containing the same toolbox part.
  - e. Once saved, the rename of the toolbox part item id can be performed within the Teamcenter Rich Client.
5. It is recommended that toolbox parts have world read access in Teamcenter to ensure that they can be accessed by all projects.
6. In a multi-site environment, it is recommended that sw2Tbx datasets be pushed to all sites as soon as they are created. Doing so will avoid the risk of duplicates in the database.
7. Revise of toolbox parts is allowed within Teamcenter, but it is not expected that a site would want to revise a toolbox part. The recommendation is to release toolbox parts once they are saved to Teamcenter and correctly contain all associated documents such as JT files.
8. Attribute mapping of toolbox parts is only supported in cadtopdm direction. When a sldtbx model is included in a “both” or “pdmtoCAD” mapping, the mapping will be silently ignored during the Open or Update operation.
9. Auxiliary files must be placed into the toolbox for the integration to load them into Teamcenter.
10. 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.
11. The swimimport utility must be executed on a system with the Toolbox enabled and configured, in order for the sw2Tbx datasets to be created for the Toolbox parts.

SolidWorks toolbox has three modes, “Create Configurations”, “Create Parts”, and “Create Parts on Ctrl-Drag”. Each of these methods has subtle differences which sites should consider carefully and establish design practices.

## Usage

### Create Configurations

- Create Configurations is the preferred mode for the integration to utilize. If possible, set design practices to only use “Create Configurations” and to only save assemblies that are using “Create Configurations” to Teamcenter.

### Create Parts

- To manage “Create Parts” as SW2Tbx datasets in Teamcenter, the “Create parts in this folder” path must be included in the sw.toolbox.dirs path in swim.cfg. The “Create Parts” path must be the same for all users because SolidWorks stores the path in the referencing models.
- The “Create Parts” path must be shared on the network for all users
  - a. Failure to make the path shared will result in assemblies that cannot be opened by all users.
- If sharing data across multiple sites, the “Create Parts” path must be synchronized outside of Teamcenter. Failure to synchronize the “Create Parts” directories will result in failures to open data.

### Create Parts on Ctrl-Drag

- As stated earlier, it is recommended that designers always use “Create Configurations”
- “Create Parts on Ctrl-Drag” enables users to use both the “Create Configurations” and “Create Parts” methods of Toolbox. Sites should manage both types of toolbox data as outlined above.
- If the “Create Parts” directory is not available and synchronized to all users, then some toolbox parts could be changed from “Create Parts” to “Create Configurations”. These changes could be problematic for users.

### Create Parts as SWPrt

- It is possible to manage “Create Parts” as regular part files. To do so, do not include the “Create Parts” path in the sw.toolbox.dirs in swim.cfg
- The integration will interpret the “Create Parts” as parts, and the mappings for sldprt will apply, instead of the mappings for sltdtbx.
- Sites would want to set sw.configurations.hide.document.name = true
- The “Create Parts” path must not be a network path, and should not be synchronized between sites. The path should be local to the users’ system.
- If managed as parts, the data files are saved to Teamcenter, and will therefore be fetched from Teamcenter during open.
- When saving “Create Parts” as SWPrt, the integration does not protect against designers changing the model name or creating duplicate items of the same toolbox part.
- The part icon, not the toolbox icon will be displayed in the integration dialogs and task pane for “Create Parts”.
- If modifications are made to a “Create Part” while inserting, it is possible that the resulting toolbox document will contain a PreviewCfg configuration a configuration which most sites will not want to manage. The recommendation is to add PreviewCfg to

sw.configurations.hide preference to avoid multiple PreviewCfg datasets in Teamcenter. PreviewCfg may not be the name for all locales.


## **Duplicating Items and Structures**

Using the integration client a user can duplicate items which are already in Teamcenter. The process is referred to as “In-Session Duplicate”

The term duplication refers to making a copy of an item, and selected child items, for example a complex assembly. The process of duplication is commonly used when a new design is to be created, based on an existing design. In such a case the existing design is duplicated to create the initial new design. The new design is then modified to finalize the design.


### **In-Session Duplicate**

The in-session Duplicate function uses the integration **Open** and **Save All** dialogs to load all models into SolidWorks session from Teamcenter, duplicate the models in session, then save the models as new Item IDs to Teamcenter. Here are the steps:

1. Start a new session of SWIM and SolidWorks. Since large assemblies and their information may take up a large amount of memory. There should only be one session running and minimal other activity on the client computer.
2. Use **Teamcenter | Open**  to search for a structure to be duplicated. This is usually a drawing or assembly with many children. Once the search is complete, the user should select both fetch and open into session checkboxes of the object to be duplicated.
3. For the duplicate to be successful, the user must ensure that all desired models will be loaded into the session. Thus the following should be checked or not checked.
  - a. In the main panel, select both the fetch and open in session checkboxes for the selected top level item
  - b. In the **Filter** panel, the **Required** filter must **not** be selected, to ensure all relationships are traversed (such as part to part or part to assembly)
  - c. Also in the **Filter** panel, the **Include Drawings** filter checkbox must be selected to include all drawing for all models of the structure<sup>8</sup>.
  - d. Do not select the “Lightweight” checkbox.
4. Set the destination directory to a new or empty directory to avoid conflicts
5. Select the open button
6. On the open dependencies dialog verify that all models (items) to be included in the duplication are present.
7. Once the models are loaded into session, verify that all models to be duplicated are resolved in session. Lightweight models will not be duplicated.

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<sup>8</sup> Drawings that reference models outside of the structure being opened are excluded, to prevent uncontrolled traversal of external references.

8. Once the models to be duplicated are resolved in the SolidWorks session, select **Teamcenter | Save All** .
9. Click the **Configurations | Add** button. Failure to do so will result in two documents within Teamcenter that contain configurations with the same names. The name clashes will degrade the performance of the integration.
10. Select all of the models in the save dialog (<ctl><A>) then use the **Item ID | New** or **Item ID | Replace** buttons to change the names of the models. When prompted to rename the items, click “yes to all” and for the prompt to rename the models, click “yes to all”. The files are now renamed within the session.
11. All models should now be selected for save. Click on the save button to complete the operation.

The above procedure is the recommended process to duplicate all objects in a structure inside the SolidWorks session. Some users may want to modify the procedure for specific situations by choosing different filter options, setting models to lightweight or unselecting objects in the open or save dialogs. A common procedure would be to unselect toolbox parts (nuts, bolts) when assigning new Item IDs in the save dialog.

Partial duplicates or bottom up duplicates are supported. If a user knows that five parts and their parent models are to be duplicated, then the user can use an iterative process, as described below, for step 11:

1. Select the five parts and assign new IDs and model names via Item ID new or Item ID replace buttons.
2. After the model names are changed, the modified hint will appear in the left column for the five models and the parent assemblies that reference them. The five models will have checkboxes in the save column and the modified parents will not.
3. Select the modified parents, and use Item ID new or Item ID replace buttons to assign new IDs and model names.
4. Repeat steps 2 and 3 until no more models have a modified hint without the save checkbox selected.

If a parent model is included in a duplicate, but a child model is not, then the duplicated parent will reference the original child. This presents a problem where circular relationships are involved, because the parent will know the original child, but the original child will not know the duplicated parent. For this reason, it is recommended that all data in a structure be included in duplication.