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# SolidWorks integration recommended settings

This section documents the recommended changes to the default settings for a successful SolidWorks to JT translation implementation; it’s strongly recommended to read and understand this section; additionally, you will find example of configuration files below, as well as a macro to rename JT configuration file names (don’t replace your configuration files with these file, use it as a reference example only)

This document was written for SolidWorks integration 4.5 and JT translator 7.0 (Dec. 2007)



Table of Contents

[1.1. JT files saved into Teamcenter 2](#_Toc187453279)

[1.2. JT translation for all parts within an assembly 2](#_Toc187453280)

[1.3. PDF and other auxiliary files save into Teamcenter 2](#_Toc187453281)

[1.4. Correct file naming for JT files (prevent characters substitution for JT files) 3](#_Toc187453282)

[1.5. Trouble shooting: “\_csi\_sw.dll cannot be loaded” 3](#_Toc187453283)

[1.6 Attribute mapping 3](#_Toc187453284)

[1.7 JT file save for SolidWorks configurations: RenameConfigJT macro 3](#_Toc187453285)

## JT files saved into Teamcenter

In order to have an automatic save of the JT file(s) into Teamcenter, you need to configure the following lines in the **swim.xml** file:

**<jt\_file cad\_type="sldprt" action="none">**

**<file\_name pattern="{cad\_name}.jt"/>**

**</jt\_file>**

**If the goal is to save the JT files to Teamcenter, then action should be set to “create”, not “none”. “none” will cause the integration to ignore JT files, even when they have already been translated using the embedded translator.**

**NB**: SolidWorks doesn’t create automatically the JT file; It requires a user action (Teamcenter/ Convert to JT); This must be done after the SolidWorks data got an item ID into Teamcenter; (so you have to save the SolidWorks data first and then generate JT, and then save again)

True, but it is not required to actually complete the save the first time. It is sufficient to (a) Select all rows and press New ID, then reply “yes” to the prompt about renaming models. (b) Cancel the save. Now you have all the SW models in your session with the names assigned by Teamcenter, and the JT translation will produce JT files with the correct names.

**NB**: By default, swim.xml is not configured for JT file saving into Teamcenter

This is true, the default choice in the SWIM installer is “no action”. However this can easily be changed during the installation, and furthermore when we release SWIM 7.0.0 “no action” will be the correct default, since we will have ETS support for SW data at that time.

## JT translation for all parts within an assembly

In order to have a successful JT translation of all the part within an assembly (loaded into SolidWorks) , you need to set **structureOption** to “**FULL\_SHATTER**” in the **tessSW.config** file (JT translator directory; for example: C:\swtojt\etc )

**version "EAITranslator" 1.0.0 "EAITranslator"**

**EAITranslator {**

**OutputDirectory = "./"**

**CommonPartsPath = ""**

**chordalOption = "ABSOLUTE"**

**structureOption = "FULL\_SHATTER"**

**NB**: with the default setting “PER\_PART, the JT files for each part are not saved into Teamcenter

Yes, that is a translator issue; I agree the translator default options should be changed

## PDF and other auxiliary files save into Teamcenter

SolidWorks provides capabilities to save auxiliary files for a part, drawing or assembly; This is a user action; In order to have these files automatically captured into Teamcenter, you have to configure the following settings:

**<auxiliary\_file cad\_type="slddrw" direction="cadtopdm">**

**<pdm\_location named\_ref="PDF" pdm\_type="PDF" relation\_type="IMAN\_specification"/>**

**<file\_name pattern="{cad\_name}\*.pdf"/>**

**</auxiliary\_file>**

For more auxiliary files support (JPEG, TIF,…) you can configure the following settings:

**<auxiliary\_file cad\_type="slddrw" direction="cadtopdm">**

**<pdm\_location named\_ref="Image" pdm\_type="DirectModelMarkup" relation\_type="IMAN\_specification"/>**

**<file\_name pattern="{cad\_name}\*.jpg"/>**

**</auxiliary\_file>**

**<auxiliary\_file cad\_type="slddrw" direction="cadtopdm">**

**<pdm\_location named\_ref="Image" pdm\_type="DirectModelMarkup" relation\_type="IMAN\_specification"/>**

**<file\_name pattern="{cad\_name}\*.tif"/>**

**</auxiliary\_file>**

**<auxiliary\_file cad\_type="slddrw" direction="cadtopdm">**

**<pdm\_location named\_ref="Image" pdm\_type="DirectModelMarkup" relation\_type="IMAN\_specification"/>**

**<file\_name pattern="{cad\_name}\*.DXF"/>**

**</auxiliary\_file>**

**<auxiliary\_file cad\_type="slddrw" direction="cadtopdm">**

**<pdm\_location named\_ref="Image" pdm\_type="DirectModelMarkup" relation\_type="IMAN\_specification"/>**

**<file\_name pattern="{cad\_name}\*.DWG"/>**

**</auxiliary\_file>**

**NB**: To save a PDF file for a SolidWorks drawing, follow these steps:

* Open the drawing into SolidWorks
* Save it to Teamcenter (if not already saved) (it will get the correct Teamcenter file name)
* Then in SolidWorks, select :
  + Save As (don’t use Print !)
  + PDF
  + Accept the name
  + Done
  + Save again the drawing into Teamcenter… the PDF file will be automatically saved under the same item rev.

**NB**: with the default setting, the JT files for each part within an assembly are not saved into Teamcenter

## Correct file naming for JT files (prevent characters substitution for JT files)

In some cases, the JT translator substitutes characters in the JT file name. Unfortunately, it prevents Teamcenter to save the JT files into Teamcenter (JT file name does not match any longer the SolidWorks file name); For example, a SolidWorks file with Teamcenter ID containing “-“ character will be change: “PART-12345.jt” will become “PART\_12345.jt” ;

To prevent characters substitution during the JT translation, set "**autoNameSanitize**" set to "**false**" in the tessSW.config, located in your translator/etc directory: (for example C:\swtojt\etc)

**version "EAITranslator" 1.0.0 "EAITranslator"**

**EAITranslator {**

**OutputDirectory = "./"**

**CommonPartsPath = ""**

**chordalOption = "ABSOLUTE"**

**structureOption = "FULL\_SHATTER"**

**writeWhichFiles = "ALL"**

**pmiOption = "NONE"**

**partMonitor = false**

**compression = true**

**triStripOpt = false**

**seamSewing = false**

**seamSewingTol = 0.001**

**includeBrep = true**

**brepPrecision = "DOUBLE"**

**autoNameSanitize = false**

**deleteUnusedParts = false**

**…..**

**NB**: the default setting leads to characters substitutions (when file name contains “-“ for example) , which leads to issues.

Another translator default issue

## Trouble shooting: “\_csi\_sw.dll cannot be loaded”

If you have the following error when launching swim, remember that Microsoft Visual C++ 2005 Redistributable Package SP1 is required on each client for the JT translator and Swim ; This information is valid for the JT translator v7.0 (Dec 2007); Check in the documentation for different releases.

## 1.6 Attribute mapping

You will find an example of aattribute mapping between Teamcenter and SolidWorks below:

**<attribute\_map cad\_type="sldprt">**

**<attribute>**

**<cad\_name value="DB\_PART\_NO"/>**

**<pdm\_name value="Item.item\_id"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="pdmtocad"/>**

**</attribute>**

**<attribute>**

**<cad\_name value="DB\_PART\_REV"/>**

**<pdm\_name value="Item.item\_revision\_id"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="pdmtocad"/>**

**</attribute>**

**<attribute>**

**<cad\_name value="DB\_PART\_DESCRIPTION"/>**

**<pdm\_name value="Item.object\_desc"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="cadtopdm"/>**

**</attribute>**

**<attribute>**

**<cad\_name value="DB\_PART\_NAME"/>**

**<pdm\_name value="Item.object\_name"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="pdmtocad"/>**

**</attribute>**

**<attribute>**

**<cad\_name value="Creator"/>**

**<pdm\_name value="Item.owning\_user"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="pdmtocad"/>**

**</attribute>**

**<attribute>**

**<cad\_name value="DrawnDate"/>**

**<pdm\_name value="Item.creation\_date"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="pdmtocad"/>**

**</attribute>**

**</attribute\_map>**

**<attribute\_map cad\_type="sldasm">**

**<attribute>**

**<cad\_name value="DB\_PART\_NO"/>**

**<pdm\_name value="Item.item\_id"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="pdmtocad"/>**

**</attribute>**

**<attribute>**

**<cad\_name value="DB\_PART\_REVISION"/>**

**<pdm\_name value="Item.item\_revision\_id"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="pdmtocad"/>**

**</attribute>**

**<attribute>**

**<cad\_name value="DB\_PART\_DESCRIPTION"/>**

**<pdm\_name value="Item.object\_desc"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="cadtopdm"/>**

**</attribute>**

**<attribute>**

**<cad\_name value="DB\_PART\_NAME"/>**

**<pdm\_name value="Item.object\_name"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="pdmtocad"/>**

**</attribute>**

**<attribute>**

**<cad\_name value="Creator"/>**

**<pdm\_name value="Item.owning\_user"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="pdmtocad"/>**

**</attribute>**

**<attribute>**

**<cad\_name value="DrawnDate"/>**

**<pdm\_name value="Item.creation\_date"/>**

**<missing\_attribute\_action value="create"/>**

**<prompt value="conditional"/>**

**<direction value="pdmtocad"/>**

**</attribute>**

**</attribute\_map>**

## 1.7 JT file save for SolidWorks configurations: RenameConfigJT macro

**(author: C Mazeau, Siemens PLM Software Spain)**

**NB: The macro is unsupported !**

**SWIM behaviour with configuration**

By default, SWIM only generate one item for the document (father) and all the configuration, but you can configure SWIM so that it generates different items for each configuration and for the document (father). This is probably the best configuration, because on the other way the BOM is never correct (for different assembly configurations) and you cannot have the JT file for each component / assemblies.

You have to change the following preferences in **swim.properties**

**sw.configurations.default = Multi**

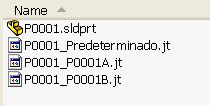
**sw.configurations.ignore = Défaut (for french)**

**or Predeterminado (for Spanish)**

**JT for configuration using the macro**

As you have seen, you have to generate the JT for each configuration manually. If the document (father) name is P0001.sldprt and the configurations names are ”P0001A” and ”P0001B” then the JT files will be ”P0001\_P0001A.jt” and ”P0001\_P0001B.jt”. I haven’t found a way to change that behaviour.

Note that ”P0001\_Predeterminado” is the JT file for the document, as ”Predeterminado” is the default configuration. This JT will be uploaded correctly.



For the purpose of renaming the JT files for the different configuration, a macro has been written; The user has to run the macro, and the JT files are renamed to ”P0001A.jt” and ”P0001B.jt” because The configuration name is by default the ItemId is TcEng. Then when saving the JT files are uploaded correctly.



**SW Macro Macro setup and use**

You have to rename the file .swp\_swp 🡪 .swp (just to evitate firewall issues)

To run the macro in SW: Tools\macro\run

You will have to edit the macro first to fit you system: Tools\macro\edit

        I have translated the comments to english for better understanding

        You have to change the working directory, which is where files are generated



        You have to change the default configuration name (Predeterminado is for ”Default” in Spanish)



This is a good trick but it should not be necessary to do all this work. You can configure the swim.xml file to recognize and store the configuration JT files when they carry their documents’ names, without actually renaming them. Just add the following to the auxiliary file map in swim.xml:

<auxiliary\_file\_map>

<jt\_file action="create" cad\_type="sldprt"/>

<auxiliary\_file cad\_type="sldprt">

<pdm\_location named\_ref="JTPART" pdm\_type="DirectModel" relation\_type="IMAN\_Rendering"/>

<file\_name pattern="{document\_cad\_name}\_{cad\_name}.jt"/>

<cadtopdm\_control label="Save JT Files" user\_preference\_default="true"/>

</auxiliary\_file>

This will cause the integration to look for JT files with two naming conventions: filename.jt for the documents, and filename\_configname.jt for the configurations. It will recognize and store both of them.