PTC ®

Windchill® Part Creation
Utility Administrator's Guide

Windchill® 10.0 M050
Windchill PDMLink™ 10.0 M050
Pro/INTRALINK® 10.0 M050

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About This Guide

The Windchill Part Creation Tool Administrator's and User's Guide describes how to use the Part Creation Utility to bulk create WTParts related to previously existing CAD documents, and build product structures based on a set of rules configured by the user.

Related Documentation

The following documentation may be helpful to read and understand:

General Documentation

Guide Name	Description
Windchill PDMLink User's Guide	
Windchill ProjectLink User's Guide	

If books are not installed on your system, see your system administrator.

Technical Support

Contact PTC Technical Support through the PTC website, or by phone, email, or fax if you encounter problems using this product or the product documentation. The PTC eSupport portal provides the resources and tools to support your PTC Windchill implementation:

https://www.ptc.com/appserver/cs/portal/

For complete details, see the *PTC Customer Service Guide*:

http://www.ptc.com/appserver/support/csguide/csguide.jsp

You must have a Service Contract Number (SCN) before you can receive technical support. If you do not know your SCN, see "Preparing to contact TS" on the **Processes** tab of the *PTC Customer Support Guide* for information about how to locate it.

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  ○ Click any help icon in Windchill

  ○ Select **Help ▸ Windchill Help Center** from the **Quick Links** menu at the top right of any Windchill page

  ○ Use the following link to access all PTC help centers:

    https://www.ptc.com/appserver/cs/help/help.jsp

- **Reference Documents** website—The Reference Documents website is a library of all PTC guides:

  http://www.ptc.com/appserver/cs/doc/refdoc.jsp

Windchill® Part Creation Utility Administrator's Guide

A Service Contract Number (SCN) is required to access the PTC documentation from the Reference Documents website. If you do not know your SCN, see "Preparing to contact TS" on the **Processes** tab of the *PTC Customer Support Guide* for information about how to locate it:

http://www.ptc.com/appserver/support/csguide/csguide.jsp

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- Click the PTC help center feedback icon in the Windchill Help Center toolbar and complete the feedback form. The title of the help topic you were viewing when you clicked the icon is automatically included with your feedback.

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Getting Started

This section provides an overview of the Part Creation Utility.

What Does It Do?

The Windchill Part Creation Utility is used to bulk create WTParts related to previously existing CAD documents and build product structures based on a set of rules configured by the user.

This utility might be used as a post Pro/INTRALINK 3.x migration to PDMLink, a Pro/INTRALINK 8+ upgrade to PDMLink, or an existing PDMLink customer that has only been using the system for CAD data management to date and now wants to introduce WTParts and products structure into PDMLink.

If you have migrated Pro/INTRALINK data to a Windchill PDMLink target system, you can use the Windchill PDMLink Part Creation utility to selectively decide which of your migrated CAD Documents should be made into WTParts.

In its default configuration, the Part Creation utility does not search for CAD documents. The PartCreation.xml files MUST be created or changed for every system before the tool is run.

When using the Windchill Part Creation utility, you can select objects based on the following criteria:

- CAD document authoring application
- Document type
- CAD document name
- CAD document number
- Attribute value
- Release level
- Folder location

Bulk creating WTParts is a three-step process:

- Begin by configuring the properties files.
- Use elements to create XML scripts.
- Complete the process by using the Part Creation Utility to build parts and product structure for selected EPM documentation and EPM document iterations.

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Configuring the Properties Files

Review and setup the following information in the proimigration.properties file located in the <Windchill> directory. For example,

```
<Windchill>\codebase\com\ptc\windchill\pdmlink
\proimigration
```

These files MUST be configured before working with the Part Creation tool.

Configure the Partmigration.properties File

The partmigration.properties file can be modified to update the property values that the system assigns to parts created by the Part Creation tool.

The partmigration.properties file can be found in the following directory:

```
<Windchill>\codebase\com\ptc\windchill\pdmlink
\proimigration
```

The table below lists the rules and definitions of the partmigration.properties file.

Rule (Shown with Default Value)	Definition and Usage
`Persist-EPMDocuments =100`	This rule specifies what value to be assigned as the persist number of EPM documents. It can be set to the following value. `Persist-EPMDocuments =100`
`newPartLink=active`	This rule provides association between EPMDocuments and WTParts where it uses a link to be active or passive. The default value is "active." If the default value is changed to passive, it creates a passive association between EPMDocuments and WTParts. Values: • `newPartLink=active` • `newPartLink=passive`
`newPartView=Design`	This rule sets the default view of WTPart. The default value is "design." However, you can use any view that is configured for your system. For example, `newPartView=Design` Values: • `newPartView=Design` • Any view that is configured for your system
`partNumber=docNumber`	This rule sets the WTPart number to be identical to the document number.

Rule (Shown with Default Value)	Definition and Usage
	`partNumber=docNumber`
`SearchDescendantSub TypesForExisitingPart=True`	This rule sets the search descendant sub types for an existing part. The search values are "True" or "False." The default value is "True."
	Values:
	• `SearchDescendant SubTy-pesForExistingPart= True`
	• `SearchDescendant SubTy-pesForExistingPart =False`
`SearchExistingPartBy Name=True`	This rule sets the search for existing parts by name. The search values are "True" or "False." The default value is "True."
	Values:
	• `SearchExistingPart ByName=True`
	• `SearchExistingPart ByName=False`
`SearchExistingPartBy Number=True`	This rule sets the search for existing parts by number. The search values are "True" or "False." The default value is "True."
	Values:
	• `SearchExistingPart ByNumber=True`
	• `SearchExistingPart ByNumber=False`

Configure the Lifecyclemap.properties File

The lifecycle.map properties is used to assign lifecycle to WTPart which gets created using this tool. All the relevant states of lifecycle gets attached to WTPart. It is necessary that the mappings be specified in this file.

The lifecyclemap.properties file can be found in the following directory:

```
<Windchill>\codebase\com\ptc\windchill\pdmlink
\proimigration
```

Note

- *If a lifecycle mapping is not found in this file, the lifecycle and state for newly created part is determined by the object initialization rule for the part.*

- *If lifecycle mapping does not exist in this file and if the CAD-document has lifecycle mapping, an exception may occur when using the Part Creation tool if the lifecyclemap.properties is NOT mapped properly.*

The table below lists the rules and definitions of the lifecyclemap.properties file.

Rule (Shown with Default Value)	Definition and Usage
`LifeCycleTemplate=Life CycleTemplate`	This rule specifies that the left side of the lifecycle is equal to the right side of the lifecycle. For example, `Basic=Review`
`LifeCycleTemplate. Statei- feCycleTemplate .State`	This rule specifies that the left side of the lifecycle state is equal to the right side of the lifecycle state. The left side of lifecycle can be different from the right side of lifecycle. For example, `Basic.INWORK=Review. UNDERREVIEW`

Check the Log Report Files for Errors

During WTPart migration, log reports are generated using the logging.properties and debug.properties files.

These files and the log report files can be found in the following directory:

```
<Windchill>\codebase\com\ptc\windchill\pdmlink
\proimigration
```

The Proimigration logging tool reads relevant log file configuration information at runtime from the logging.properties file.

The logging.properties file accesses the debug.properties file for the debug% SYSTEM=1 entry. If this entry is greater than -1, logging is enabled at all severity levels, and debug level and above are outputted to a debug log file.

The proibaseline.log file contains all log statements at the debug level and up and contains the severity level. This file can be appended. For example,

```
log%base.append=
truelog%baseline.file=$(wt.log.dir)$(dir.sep)proibaseline.log
log%baseline.severity=3
```

The proibasedebug.log file contains all log statements at the debug level and up. It doesn't contain a severity level. This file can be appended. For example,

```
log%baseline debug.append=true
log%baseline
debug.file=$(wt.logs.dir)$(dir.sep)proibaselinedebug.log
```

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Using Elements to Create XML Scripts

This section describes how to create and use elements to create XML scripts for the Windchill Part Creation utility.

Before You Begin

At its simplest part selection is determined by criteria defined in an XML file which contains one or many PartAction elements. This XML file is used by the part creation tool at runtime and parts are created according to the rules laid out within the PartAction elements.

It is best to think of the PartAction element as a filter that selects the CAD Documents you want and weeds out the ones that you don't want. The PartAction Element contains the Folder Specification (FolderSpec), the Product Item Specification (PISpec) and the Product Item Version Specification (PIVSpec) elements.

The FolderSpec, PISpec and PIVSpec elements allow you to filter CAD Documents by folder location, product item attributes and product item version attributes.

The part selection criteria are comprised of a number of named elements organized into a nested hierarchy. The figure below shows the major parts of that hierarchy and how they fit together.
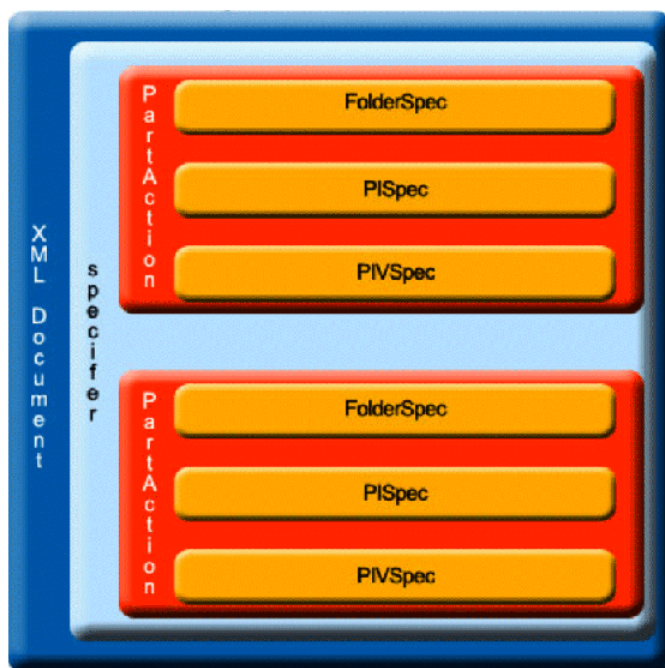
Syntax

As an XML document, the part specification is comprised of one or more named elements organized into a nested hierarchy. An element is an opening tag, some data and a closing tag. A tag is an element name preceded by a less-than symbol (<) and followed by a greater-than (>) symbol. For any given element, the name of the opening tag must match that of the closing tag. A closing tag is identical to an

opening tag except the less-than symbol is immediately followed by a forward-slash (</). Tag names are case-sensitive and should use the same case as defined in the DTD file.
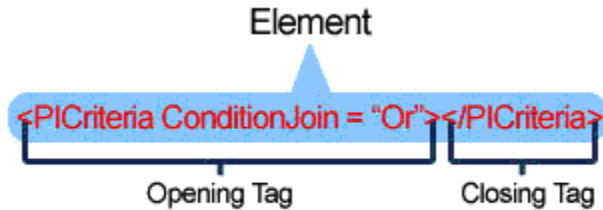
Note

Some elements (such as <FolderCondition>, <PICondition> and <PIVCondition>) contain no data between the opening and closing tags. As such, you can end these opening tags with "/>" and leave out the closing tag all together. For example,

```
<PICondition Lookup= "name" Operator= "equal"
Value="bishop_b.asm"> </PICondition>
```

Is the same as, or becomes:

```
<PICondition Lookup= "name" Operator= "equal" Value= "bishop_b.asm"/>
```

Part Selection

Part Selection allows you to select CAD Documents by specific folder locations using the <specifier>, <FolderCriteria> and <PartAction> elements.

The <specifier> Element

The <specifier> tag is a container for all of the part action elements within the part specification. The <specifier> tag is the very first tag in the part specification definition and the </specifier> is the very last. All other elements in the part selection criteria reside between the two <specifier> tags.

Usage Example:

```
<specifier>

<PartAction>

.

.

</PartAction>

</specifier>
```

The <PartAction> Element

The <PartAction> element can contain three elements that are used to filter and select your CAD Documents. These elements are the FolderSpec, the PISpec and the PIVSpec. The <PartAction> element MUST contain at least one of these elements or a combination of several. It is important to note that you cannot use any single element more than once in the <PartAction> element.

The <PartAction> tag defines whether or not the system creates or does not create a part for the CAD documents. PartAction also specifies which class of WTPart to create from the selected CAD Documents.

Attribute	Definition and Usage
Create	This attribute specifies whether or not the part creation tool creates a WTPart for the CAD Documents selected by the <PartAction> element. Values: • True (default) • False **_Note_** _The default value of this attribute is True. The only time you need to specify the Create attribute in the <PartAction> element is when you want to set it to False._
Class	This attribute determines the type of WTPart that the Part Creation tool creates for the CAD Documents selected by the <PartAction> element. This is a required attribute. This attribute is used with the End Item attribute. Value: • wt.part.WTPart
End Item	This attribute specifies whether or not the Part Creation WTPart for the CAD documents has an EndItem selected by the <PartAction> element. Values: • True • False (default)

Attribute	Definition and Usage
	Note *The default value of this attribute is False. The only time you need to specify the EndItem attribute in the \<PartAction\> element is when you want to set it to True.*

Usage Example:

The following \<PartAction\> statement specifies that the system should create WTParts of the class "wt.part.WTPart" for any CAD Document selected by the criteria contained within the \<PartAction\> element.

```
<specifier>

<PartAction Class="wt.part.WTPart">

.

.

</PartAction>

</specifier>
```

The following \<PartAction\> statement specifies that the system should not create WTParts for any CAD Document selected by the criteria contained within the \<PartAction\> element.

```
<specifier>

<PartAction Create="False">

.

.

</PartAction>

</specifier>
```

Folder Specification

The Folder Specification allows you to select CAD Documents by specific folder locations using the \<FolderSpec\>, \<FolderCriteria\> and \<FolderCondition\> elements.

The <FolderSpec> Element

The <FolderSpec> element is a container in which one or more <FolderCriteria> elements reside. The <FolderSpec> element allows you to enter criteria for selecting CAD Documents based on one or many folder locations. This element contains both the <FolderCriteria> and the <FolderCondition> tags.

The <FolderSpec> tag uses the following attribute.

Attribute	Definition and Usage
CriteriaJoin	In the event that your <FolderSpec> element should contain multiple <FolderCriteria> statements, this attribute specifies if the system selects CAD Documents that meet all of the criteria or any of the criteria. Values: • "And"–the system only selects CAD Documents that meet ALL of the criteria specified in the <FolderCriteria> elements. • "Or"–the system selects CAD Documents that meet ANY of the conditions specified in the <FolderCriteria> elements.

Usage Example:

```
<FolderSpec CriteriaJoin="Or">

.

.

</FolderSpec>
```

The <FolderCriteria> Element

The <FolderCriteria> element is a container for all of the individual <FolderCondition> elements in the <FolderSpec>. You can create multiple <FolderCriteria> elements with in the <FolderSpec> element.

Attribute	Definition and Usage
Join	In the event that your <FolderCriteria> element should contain multiple <FolderCondition> statements, this attribute specifies if the system selects CAD

Attribute	Definition and Usage
	Documents that meet all of the criteria or any of the criteria. Values: • "And"–the system only selects CAD Documents that meet ALL of the criteria specified in the <FolderCondition> elements. • "Or"–the system selects CAD Documents that meet ANY of the conditions specified in the <FolderCondition> elements.

Usage Example:

The following <FolderCriteria> statement specifies that the system should select CAD Documents that meet any of the conditions contained with in multiple <FolderCondition> elements.

```
<FolderCriteria ConditionJoin="Or">

.

.

</FolderCriteria>
```

The <FolderCondition> Element

The <FolderCondition> element specifies an individual folder location in which you want to select (or not select) CAD Documents.

Attribute	Definition and Usage
Context	This attribute specifies the product context (including organization) for which the parts have to be created.
Operator	This attribute specifies an assignment operator that connects the Lookup attribute to the Value attribute.
Value	This attribute specifies the full path to the folder location. Value: "Root Folder/Some_location/"

Values:

- "equal"-indicates that selected CAD Documents must have a location that is equal to the specified value.
- "notEqual"-indicates that selected CAD Documents must have a location that is not equal to the specified value.

Usage Example:

The following <FolderCondition> statement specifies that the system should select any CAD Document that does not have folder location of "/Default/Design". Additionally the <FolderCondition> statement specifies that corresponding parts should be created in someProduct context in the SomeOrg organization.

```
<FolderCondition Operator="notEqual" Value="/Default/Design"
Context="/wt.inf.container.OrgContainer=SomeOrg/
wt.pdmlink.PDMLinkProduct=someProduct"/>
```

Product Item Specification

The Product Item Specification allows you to select CAD Documents based on Product Item criteria using the <PISpec>, <PICriteria> and <PIVCondition> elements.

```
<PISpec>
<PICriteria>
AttrCondition>
.
</AttrCondition>
</PICriteria>
</PISpec>
```

The <PISpec> Element

The <PISpec> element is a container in which one or more <PICriteria> elements reside. The <PISpec> element uses the following attribute.

Attribute	Definition and Usage
CriteriaJoin	In the event that your <PISpec> element should contain multiple <PICriteria> statements, this attribute specifies if the system selects CAD Documents that meet all of the criteria or any of the criteria. Values:

Attribute	Definition and Usage
	• "And" — the system only selects CAD Documents that meet ALL of the criteria specified in the <PICriteria> elements. • "Or" — the system selects CAD Documents that meet ANY of the conditions specified in the <PICriteria> elements.

Usage Example:

The following <PISpec> statement specifies that the system should select CAD Documents that meet any of the conditions contained with in multiple <PICriteria> elements.

```
<PISpec CriteriaJoin= "Or">
.
.
</PISpec>
```

The <PICriteria> Element

The <PICriteria> element is a container for all of the individual <PICondition> elements in the <PISpec>. You can create multiple <PICriteria> elements with in the <PISpec> element. The <PICriteria> element uses the following attribute.

Attribute	Definition and Usage
ConditionJoin	In the event that your <PICriteria> element should contain multiple <AttrCondition> statements, this attribute specifies if the system selects CAD Documents that meet all of the condition statements or any of the condition statements. Values: • "And" — the system only selects CAD Documents that meet ALL of the criteria specified in the <AttrCondition> elements. • "Or" — the system selects CAD Documents that meet ANY of the conditions specified in the <AttrCondition> elements.

Usage Example:

The following <PICriteria> statement specifies that the system should select CAD Documents that meet any of the conditions contained within multiple <AttrCondition> elements.

```
<PISpec>
<PICriteria ConditionJoin = "Or">
.
</PICriteria>
</PISpec>
```

The <PICondition> Element

The <PICondition> tag contains the individual conditions under which a particular product item should be selected. The <PICondition> element has the following attributes.

Attribute	Definition and Usage
Lookup	This attribute indicates whether the <PICondition> element "looks up" an attribute name or a type definition. Values: • "name"–indicates that the focus of the <PICondition> element is an attribute name. • "typeahead"–indicates that the focus of the <PICondition> element is a type definition.
Operator	This attribute specifies an assignment operator that connects the Lookup attribute to the Value attribute. Values: • "equal"–indicates that selected CAD Documents must have a name or type definition that is equal to the specified value. • "notEqual"–indicates that selected CAD Documents must have a name or type definition that is not equal to the specified value.
Value	This attribute specifics the value to a "name" or "type_def" type.

Attribute	Definition and Usage
	Values:
	• If the Lookup attribute is set to "name", then the expected value is a PI attribute name in quotes.
	• If the Lookup attribute is set to "type_def", then the expected value is a PI type definition in quotes.

Usage Example:

The following <PICondition> statement specifies that the system should select PIs with the name of bishop_b.asm.

```
<PI'Condition Lookup= "name Operator= "equal">
Value= "bishop_b.asm">
```

Product Item Version Specification

The Product Item Version (PIV) Specification allows you to select CAD Documents based on PIV criteria using the <PIVSpec>, <PIVCriteria> and <PIVCondition> elements.

The <PIVSpec> Element

The <PIVSpec> element is a container in which one or more <PIVCriteria> elements reside. The <PIVSpec> element contains both the <PIVCriteria> and the <PIVCondition> tags. The <PIVSpec> tag uses the following attribute.

Attribute	Definition and Usage
CriteriaJoin	In the event that your <PIVSpec> element should contain multiple <PIVCriteria> statements, this attribute specifies if the system selects CAD Documents that meet all of the criteria or any of the criteria.
	Values:
	• "And" — the system only selects CAD Documents that meet ALL of the criteria specified in the <PIVCriteria> elements.
	• "Or" — the system selects CAD

Attribute	Definition and Usage
	Documents that meet ANY of the conditions specified in the <PIVCriteria> elements.

Usage Example:

The following <PIVSpec> statement specifies that the system should select CAD Documents that meet any of the conditions contained with in multiple <PIVCriteria> elements.

```
<PIVSpec CriteriaJoin = "Or">
.
</PIVSpec>
```

The <PIVCriteria> Element

The <PIVCriteria> element is a container for all of the individual <PIVCondition> and <AttrCondition> elements in the <PIVSpec>. You can create multiple <PIVCriteria> elements with in the <PIVSpec> element.

Attribute	Definition and Usage
ConditionJoin	In the event that your <PIVCriteria> element should contain multiple <PIVCondition> or <AttrCondition> statements, this attribute specifies if the system selects CAD Documents that meet all of the condition statements or any of the condition statements.
	Values:
	• "And" — the system only selects CAD Documents that meet ALL of the criteria specified in the <PIVCondition> elements.
	• "Or" — the system selects CAD Documents that meet ANY of the conditions specified in the <PIVCondition> elements.

Usage Example:

The following <PIVCriteria> statement specifies that the system should select CAD Documents that meet any of the conditions contained with in multiple <PIVCondition> elements.

```
<PIVSpec>
<PIVCriteriaConditionJoin = "Or">
```

```
.
.
</PIVCriteria>
</PIVSpec>
```

The \<PIVCondition> Element

The \<PIVCondition> tag contains the individual conditions under which a particular product item version should be selected.

Attribute	Definition and Usage
Version	This attribute specifies whether the \<PIVCondition> element selects all versions, the latest version or a specific version of a CAD Document. Values: • "latest"—indicates that the \<PIVCondition> element selects the latest version. • "all"— indicates that the \<PIVCondition> element selects all versions of a CAD Document. • "specific"— indicates that the \<PIVCondition> element select CAD Documents with a specific version value. ***Note*** *If the Version attribute is set to "specific" then you must also provide both the Operator and Value attributes.*
Operator	This attribute specifies an assignment operator. Values: • "equal"—indicates that selected CAD Documents must have criteria that is equal to the specified value. • "notEqual"— indicates that selected CAD Documents must have criteria that is not equal to the specified value. • "greaterThan"— indicates that

Attribute	Definition and Usage
	selected CAD Documents must have criteria that is greater than the specified value.
	• "greaterThanOrEqual"— indicates that selected CAD Documents must have criteria that is greater than or equal to the specified value.
	• "lessThan"— indicates that selected CAD Documents must have criteria that is less than the specified value.
	• "lessThanOrEqual"— indicates that selected CAD Documents must have criteria that is less than or equal to the specified value.
Value	Indicates the version value you want to search for.
	Value: Any valid version is an acceptable value.

Usage Example:

The following <PIVCondition> statement specifies that the system should select the latest version, specific and all.

```
<PIVCondition Version= "Latest"/>
<PIVCondition Version= "specific" Value="A"/>
<PIVCondition Version= "all "/>
```

Putting it All Together – Applying Structure and Syntax

This section explains how the part selection criteria is structured. You can now start to put the structure and the syntax together into your own part selection criteria.

By default, the Part Creation select specification file is set up to create WTParts for each and every Creo Elements Pro Part and Assembly that the system migrates. The default part selection criteria for creating WTParts is located at:

```
codebase\src\com\ptc\windchill\pdmlink\proimigration\server\xml
```

In order to create customized part selection criteria, you must modify this file. This is an XML file and can be edited with a standard text editor. When opened, the part selection criteria should look as follows:

```
<specifier>
   <PartAction Class="wt.part.WTPart">
      <PISpec CriteriaJoin="Or">
         <PICriteria ConditionJoin="Or">
            <PICondition Lookup="type_def"
             Operator="equal" Value="Part"/>
            <PICondition Lookup="type_def"
            Operator="equal" Value="Assembly"/>
         </PICriteria>
      </PISpec>
   </PartAction>
</specifier>
<?xml version="1.0 encoding="UTF-8"?>
<!--
```

Note

When editing this file, keep in mind that the <PartAction> element can contain the following combinations of elements:

- *FolderSpec, PISpec, PIVSpec*
- *FolderSpec, PISpec*
- *FolderSpec, PIVSpecPISpec, PIVSpec*
- *FolderSpec*
- *PISpec*
- *PIVSpec*

Examples of Common PartAction Criteria

The following section contains examples of common PartAction criteria. Each example contains a table to explain important condition statements in the coding sample.

Example 1: Creating a WTPart for CAD Documents with the Following Attributes

For example, enter the following XML script to create a WTPart for all CAD documents whose PIs are a Creo Elements Pro part or assembly, and those CAD documents have the attribute (UDA) "ERP State" whose value is "Released."

```
<<PartAction Create="True", Class="wt.part.WTPart">
   <PISpec>
      <PISpec CriteriaJoin="And">
         <PICriteria ConditionJoin="And">
         <AttrCondition AttrName="number" Operator="notEqual"
         Value="0-0-0" AttrTypeIsDate = "false"/>
      </PICriteria>
      <PISpec>
   </PartAction>
</PartAction>
```

```
<?xml version="1.0 encoding="UTF-8"?>
<!--
```

Where	Indicates
`<PIVCriteria ConditionJoin = "And">`	The condition for joining PIV criteria is "And".
`<PartActionCreate="True",Class= "wt.part.WTPart"Create=False">`	Creates a WTPart for all CAD documents whose PIs are a Creo Elements Pro part or assembly.
`<ATTRCondition AttrName= "number"Operator="notEqual" Value="0-0-0"AttrTypeIsDate ="false"/>`	Creates a WTPart for all CAD documents whose number is not equal to 0-0-0.

Examples of Common Part Selection Criteria

The following section contains examples of specific part selection criteria. Each example contains a table to explain important condition statements in the coding sample.

Example 1: Selecting Singular CAD Documents with the Following Attributes

Here is an example XML script where the part selection criteria *only selects CAD documents with the following attributes*.

- Latest CAD document with a release level of "Released"

- In the folder "Root Folder/Ready_to_be_Released"

```
<Specifier>
    <PartAction Class="wt.part.WTPart">
        <FolderSpec CriteriaJoin="And">
            <FolderCriteria ConditionJoin="And">
                <FolderCondition Lookup= "Fullpath"
                 Operator="equal" Value="Root
                 Folder/Ready_to_be_released"/>
            </FolderCriteria>
        </FolderSpec>
        <PIVSpec CriteriaJoin="Or">
            <PIVCriteria ConditionJoin="And">
                <PIVCondition Version= "Latest"/>
        </PIVCriteria>
        </PIVSpec>
    </PartAction>
</Specifier>
<?xml version="1.0 encoding="UTF-8"?>
<!--
```

Note

The WTParts to be created in this example will be Standard WTParts. They are not end items and will not be traced.

Where	Indicates
`<FolderCondition Lookup ="Full path" Operator= "equal"Value="RootFolder /Ready_to_be_released"/>`	The folder name.
`<PIVCondition Version="Latest"/>`	Only the latest CAD documents with a release level of Released.

Example 2: Selecting Any CAD Documents with One or More of the Following Attributes

Here is an example XML script where the part selection criteria selects *any CAD documents with one or more of the following attributes*:

- Parts and Assemblies with a name like "Turbine".

- Latest CAD documents only.

- In the folder "Root Folder/Thermal_Power_Plant"

```
<Specifier>
   <PartAction Class="wt.part.WTPart">
      <FolderSpec CriteriaJoin="And">
         <FolderCriteria ConditionJoin="And">
            <FolderCondition Context= "Fullpath"
               Operator="equal" Value="Default
               /Thermal_Power_Plant"/>
         </FolderCriteria>
      </FolderSpec>
      <PISpec CriteriaJoin="Or">
         <PICriteria ConditionJoin="Or">
            <PICondition Lookup="name" Operator="equal"
             Value="Turbine*"/>
         </PICriteria>
      </PISpec>
      <PIVSpec CriteriaJoin="Or">
         <PIVCriteria ConditionJoin="And">
            <PIVCondition Version= "Latest"/>
         </PIVCriteria>
      </PIVSpec>
   </PartAction>
</Specifier>
<?xml version="1.0" encoding="UTF-8"?>
<!--
```

Where	Indicates
`<FolderCondition Context ="Full"Operator="equal"`	The folder name.

Where	Indicates
`Value="Default Folder /Thermal_Power_Plant"/>`	
`<PICondition Lookup= "name"Operator="equal" Value="Turbine*"/>`	Parts and assemblies with a name that includes the word "turbine."
`<PIVCondition Version="Latest"/>`	Only the latest CAD documents.

Example 3: Selecting and Deselecting CAD Documents with the Following Attributes

Here is an example XML script where the part selection criteria selects CAD documents with specific criteria and eliminates CAD documents with specific criteria.

For this example, *All CAD documents with the following attributes are selected*:

• All CAD documents with a version of "A" and a version of "C"

For this example, *All CAD documents with the following attributes are not selected (eliminated)*:

• CAD documents in the folder "Root Folder/Back_Up"

```
<Specifier>
   <PartAction Class="wt.part.WTPart">
      <PIVSpec CriteriaJoin="Or">
         <PIVCriteria ConditionJoin="And">
            <PIVCondition Version= "specific" Value="A"/>
            <PIVCondition Version= "specific" Value= "C"/>
          </PIVCriteria>
      </PIVSpec>
   </PartAction>
   <PartAction Class="wt.part.WTPart" Create="False">
      <FolderSpec CriteriaJoin="And">
         <FolderCriteria ConditionJoin="And">
            <FolderCondition Lookup= "Fullpath"
             Operator="equal" Value="Root
             Folder/Back_Up"/>
         </FolderCriteria>
      </FolderSpec>
   </PartAction>
</Specifier>
<?xml version="1.0 encoding="UTF-8"?>
<!--
```

Where	Indicates
`<PIVCondition Version= "specific" Value="A"/>`	All CAD documents with a version of A.
`<PIVCondition Version= "specific" Value= "C"/>`	All CAD documents with a version of C.

Where	Indicates
`<PIVCriteria ConditionJoin="And">`	The condition for joining PIV criteria is And. In this example, this means that the criteria are all CAD documents with a version of A and a version of C.
`<PartAction Class="wt.part.WTPart" Create="False">`	The part selection criteria does not select CAD documents with the following attributes.
`<FolderCondition Lookup="Fullpath" Operator="equal"Value="RootFolder/Back_Up"/>`	The folder name.

5

Using the Part Creation Utility

This section explains how to use the Part Creation Utility to build parts and product structure for selected EPM documents and EPM document iterations.

How to Create Parts and Product Structure

Users can create parts and product structure for all CAD documents, including all versions and iterations representing a Creo Elements Pro part and assembly file types.

The Part Creation Utility can perform the following functions:

Function	Definition and Usage
Search for CAD documents to process	Search and identify which existing EPM documents require a corresponding WTPart.
Validate objects and view warnings	Validate and assign case to EPM Documents from the list of EPM documents requiring a corresponding WTPart.
Create new parts	Create new WTParts.
Create links to CAD documents	Link EPM Documents to new parts or existing parts.
Build part structures	Build the part structure.
Add parts to baselines	Optionally, add parts to baselines.
Delete a temporary status table	Optionally, delete the temporary status table.

The Windchill PDMLink Part Creation utility is a command-line driven tool that can be accessed from a Windchill shell.

This section contains the following information:

- Access and Log On
- Create XML scripts
- Review Reports
- Exit and Log Off

Access and Log On

This procedure explains how to access, log on, and run the Windchill PDMLink Part Creation utility.

1. Enter the following command in your Windchill shell:

```
java com.ptc.windchill.pdmlink.proimigration.server.WTPartCreator
```

After entering the command, you will be prompted to enter a username and password.

2. To start the utility, enter a username and password that belongs to an Administrator group.

3. The main window for the Windchill PDMLink Part Creation utility opens:

```
Windchill PDMLink Part Creation Tool

--------------------------------------------------

1. Search for CAD Documents to process.
2. Validate your objects, and see existing warnings.
3. Create new parts.
4. Create links WTPart->EPMDocument.
5. Build Part Structures.
6. Add Parts to Baselines.
7. Delete the temporary status table.
8. Exit

>>> Choose an option:
```

Create XML Scripts

The Windchill PDMLink Part Creation utility uses XML scripts to selectively create WTParts for CAD and EPM Documents.

1. From the Windchill PDMLink Part Creation Tool window, perform Options 1 - 7 in sequential order.

Note

• *In order to properly use the Windchill PDMLink Part Creation utility, you MUST perform Options 1–7 in sequential order.*

• *See for details on how to access and review an XML script report.*

```
Windchill PDMLink Part Creation Tool

--------------------------------------------------

1. Search for CAD Documents to process.
2. Validate your objects, and see existing warnings.
3. Create new parts.
4. Create links WTPart->EPMDocument.
5. Build Part Structures.
6. Add Parts to Baselines.
7. Delete the temporary status table.
8. Exit
```

```
>>> Choose an option:
```

2. Begin by choosing Option 1, "Search for CAD Documents to process," at the command line and press Enter. In this case, the window pertaining to Option 1 opens.

    a. At the command line, create an XML script using the appropriate elements and tags for the option you selected.

    b. Upon completion of the XML script, view the report for the XML script you just completed.

    c. If there ARE errors in the report, resolve those errors before proceeding to the next Option in the list.

    d. If there are NO errors in the report, proceed to the next step.

3. Choose Option 2, "Validate your objects and see existing warnings," by entering 2 at the command line and pressing Enter.

    a. At the command line, create an XML script using the appropriate elements and tags for the option you selected.

    b. Upon completion of the XML script, view the report for the XML script you just completed.

    c. If there ARE errors in the report, resolve those errors before proceeding to the next Option in the list.

    d. If there are NO errors in the report, proceed to the next step.

4. Choose Option 3, "Create new parts," by entering 3 at the command line and pressing Enter.

    a. At the command line, create an XML script using the appropriate elements and tags for the option you selected.

    b. Upon completion of the XML script, view the report for the XML script you just completed.

    c. If there ARE errors in the report, resolve those errors before proceeding to the next Option in the list.

    d. If there are NO errors in the report, proceed to the next step.

5. Choose Option 4, "Create links WTPart->EPMDocument," by entering 4 at the command line and pressing Enter.

    a. At the command line, create an XML script using the appropriate elements and tags for the option you selected.

    b. Upon completion of the XML script, view the report for the XML script you just completed.

c. If there ARE errors in the report, resolve those errors before proceeding to the next Option in the list.

d. If there are NO errors in the report, proceed to the next step.

6. Choose Option 5, "Build part structures," by entering 5 at the command line and pressing Enter.

a. At the command line, create an XML script using the appropriate elements and tags for the option you selected.

b. Upon completion of the XML script, view the report for the XML script you just completed.

c. If there ARE errors in the report, resolve those errors before proceeding to the next Option in the list.

d. If there are NO errors in the report, proceed to the next step.

7. Choose Option 6, "Add parts to baselines," by entering 6 at the command line and pressing Enter.

a. At the command line, create an XML script using the appropriate elements and tags for the option you selected.

b. Upon completion of the XML script, view the report for the XML script you just completed.

c. If there ARE errors in the report, resolve those errors before proceeding to the next Option in the list.

d. If there are NO errors in the report, proceed to the next step.

8. Choose Option 7, "Delete the temporary status table," by entering 7 at the command line and pressing Enter.

View Reports

After the XML script runs, visit the logs folder <Windchill>/logs to view log files produced based on the Windchill PDMLink Part Creation Tool option and criteria executed.

For example,

• proisearch.log

• proivalidate.log

• proicreate.log

Exit and Log Off

This procedure explains how to Exit the main window and Log Off the Windchill PDMLink Part Creation utility.

1. From the Windchill PDMLink Part Creation utility main window, enter 8 on the command line and press Enter.

```
Windchill PDMLink Part Creation Tool

-------------------------------------------------

1. Search for CAD Documents to process.
2. Validate your objects, and see existing warnings.
3. Create new parts.
4. Create links WTPart->EPMDocument.
5. Build Part Structures.
6. Add Parts to Baselines.
7. Delete the temporary status table.
8. Exit

>>> Choose an option:
```

2. When the Windchill shell opens, you have successfully exited and logged off from the Windchill PDMLink Part Creation utility.