ANARK CREO ADAPTER

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SUPPORTED FEATURES WITHIN THE ANARK CREO ADAPTER

Anark Advanced MBD CAD Adapter Feature Support for Creo Files			
Last Date Validated:	9/26/2024		
File Types Supported:	.prt .asm		

Multi-CAD Supported Features			
Category	Feature Name	Supported	Notes
Product Geometry	Part Geometry	YES	
Product Geometry	Part Bodies	YES	
CAD Entity Color	Part Body Color (color per body)	NO	
CAD Entity Color	Part color	YES	
CAD Entity Color	Part Face color	YES	
CAD Entity Color	Assembly Occurrence face color (set at assembly level)	NO	
CAD Entity Color	Assembly Occurrence part color (set at assembly level)	YES	
CAD Entity Color	Component Transparency/Opacity	NO	
CAD Entity Color	GDT Color	YES	
CAD Entity Color	Supplemental Geometry Color	PARTIAL	ANARK DOESN'T IMPORT DATUM PLANE COLOR - CREO Specific: AXES, QUILTS, NOT DATUM PLANES
Product Metadata	Product Attributes (Custom Parameters, File Properties, etc)	YES	
Product Metadata	Semantic Attributes for GD&T	YES	
Supplemental Geometry	Datum Planes	YES	
Supplemental Geometry	Fixed Size Datum Planes	YES	
Supplemental Geometry	Axis Systems	YES	
Supplemental Geometry	Fixed Size Axis Systems	YES	
Supplemental Geometry	Dashed lines: axes, witness lines, etc	PARTIAL	
Supplemental Geometry	Supp Geometry Surfaces	YES	
GD&T	Flat to screen notes	PARTIAL	- CREO Specific: IMPORTED AS DATA ONLY NOT GRAPHICS
GD&T	Graphical GD&T (3D dimensions, tolerances, notes, tables, weld symbols, etc	YES	
GD&T	GD&T->Part Associations	YES	
GD&T	GD&T->Occurrence Associations	YES	
GD&T	GD&T->Face Associations	YES	
GD&T	GD&T->Occurrence Face Associations	YES	
GD&T	GD&T->FEATURE CONTROL FRAME GTOL Datum Reference Associations	NO	Implicit associations; user not required to define in CAD
GD&T	GD&T->Datum Associations	YES	
GD&T	GD&T->Supp Geometry Associations	YES	
GD&T	GD&T->Edge Associations	NO	
Views	Views	YES	
Views	Entity visibility; ability to hide/show various entities in a view or capture	YES	
Views	View Placement (ie. Exploded)	YES	
Views	Views with Folded and Flat Patterns (Supporting Sheet Metal)	YES	
Views	Views with Complex Sections	NO	
Views	View-Specific Bodies	YES	
Cross Sections	Cross Sections: Single Plane	YES	
Cross Sections	Cross Sections: Curved/Sketched (CREO ONLY?)	NO	
Cross Sections	Cross Sections: Zone (Creo Only)	NO	
Cross Sections	Cross Sections: Selective Sections	NO	
Creo Specific Supported Features			
Creo	Hemispherical circle features - hole feature (two hemispherical surfaces) - Cr	NO	
Creo	Specify Rep/Instance	YES	
Creo	Import Scope, Quilts	YES	
Creo	Specific Default Display Preferences	YES	
Import / Output and Other Options			
Anark Collaborate	Importable to Anark Collaborate without Desktop publishing	NO	
Output Option	GD&T Show On Top	NO	
Import Options	Level Of Detail Settings	YES	
Import Options	Import Scope, GD&T	YES	
Import Options	Import Scope, Supplemental Geometry	YES	
Import Options	Import Scope. Attributes	YES	
Import Options	Ignore "Suppressed" Entities	NO	
Import Options	Combine Disjoint Bodies	NO	
Import Options	Ignore Surfaces (Might not be worth documenting)	NO	
Import Options	Repair Mesh (Un-Duplicate Vertices)	NO	

SECURITY

After the file {**CoreProToolkitTranslator.exe**} is extracted from the zip file, right click on the file, select Properties, and check whether there is a security warning ("This file came from another computer and might be blocked to help protect this computer") as shown below. If there is a warning, click on the "Unblock" button.
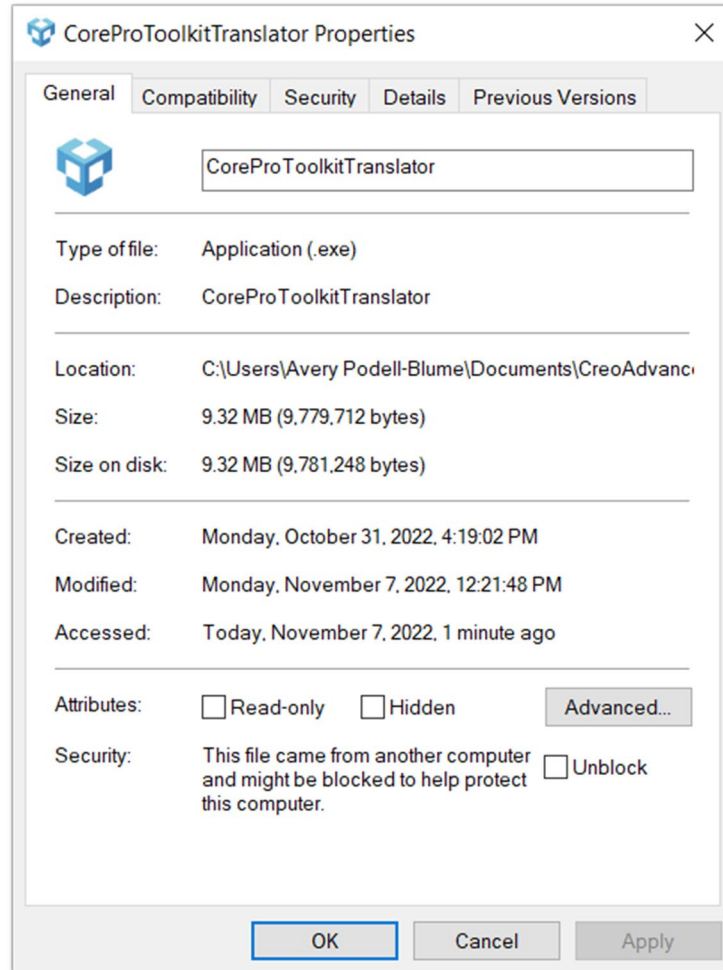
This must be done at most once for the file and then the .exe file can be copied to different machines.

If you do not perform this step, a translator which is built on .NET will not run as a full-trust application

and the import will fail. The warning is created by the built-in Windows zip extractor tool, so this might not be an issue if you use another tool to extract the zip file.

ANARK WORKSTATION INSTRUCTIONS

The **Anark Creo Adapter** optional integration software uses the Creo TOOLKIT API to translate Creo Parametric files. The following versions of Creo installations are supported: **Creo Parametric 8, 9, 10, and 11**. The Creo installation must be installed on the same machine Anark Workstation is installed on. The "Legacy Toolkit Application Runtime" option must be enabled for the Creo installation as well, which can be done in the Creo installer. If this option is not enabled, you may see an error regarding a missing file called "ucore46.dll" and the import will fail.

To use CREO TOOLKIT in Anark Workstation, follow these steps:

1. Copy the executable **CoreProToolkitTranslator.exe** from the ZIP to the **[Anark Workstation installation path]\Formats** folder.

2. This step is required the first time that **Anark Creo Adapter** is installed and it is not necessary to perform this step when upgrading Anark. If **ProEngineerAdvancedFormat.dll.config** file does not already exist in the **Formats** folder, copy the file **ProEngineerAdvancedFormat.dll.config** from the ZIP to the **[Anark Workstation installation path]\Formats** folder. Verify the Creo install path strings in ProEngineerAdvancedFormat.dll.config file (XML file), located in the **Formats** folder. By default, it is set to a **Creo 9.0.5.0** installation. If Creo Parametric is installed in a different folder or the version differs, modify the value elements in the XML file. Example configurations for Creo Creo 8.0 and Creo 9.0 are listed below.

Creo 8.0 Example

```
<ProEngineerAdvancedFormat.Settings1>
    <setting name="ProEInstallPath" serializeAs="String">
        <value>C:\Program Files\PTC\Creo 8.0.7.0\Common Files</value>
    </setting>
    <setting name="ParametricExePath" serializeAs="String">
        <value>C:\Program Files\PTC\Creo 8.0.7.0\Parametric\bin\parametric.exe</value>
    </setting>
</ProEngineerAdvancedFormat.Settings1>
```

Creo 9.0 Example

```
<ProEngineerAdvancedFormat.Settings1>
    <setting name="ProEInstallPath" serializeAs="String">
        <value>C:\Program Files\PTC\Creo 9.0.3.0\Common Files</value>
    </setting>
    <setting name="ParametricExePath" serializeAs="String">
        <value>C:\Program Files\PTC\Creo 9.0.3.0\Parametric\bin\parametric.exe</value>
    </setting>
</ProEngineerAdvancedFormat.Settings1>
```

ANARK PUBLISH INSTRUCTIONS

The **Anark Creo Adapter** optional integration software uses the Creo TOOLKIT API to translate Creo Parametric files. The following versions of Creo installations are supported: **Creo Parametric 8, 9, 10, and 11**. The Creo installation needs to be installed on the same machine as Anark Publish is installed. The "Legacy Toolkit Application Runtime" option must be enabled for the Creo installation as well, which can be done in the Creo installer. If this option is not enabled, you may see an error regarding a missing file called "ucore46.dll" and the import will fail.

To use Anark Creo Adapter with Anark Publish, follow these steps:

1. Copy the executable file **CoreProToolkitTranslator.exe** from the ZIP to the **[Anark Publish installation path]\CadTransformerWebInterface\bin\Formats** folder.

2. This step is required the first time that **Anark Creo Adapter** is installed and it is not necessary to perform this step when upgrading. If **ProEngineerAdvancedFormat.dll.config** file does not already exist in the **Formats** folder, copy the file **ProEngineerAdvancedFormat.dll.config** from the ZIP to the **[Anark Publish installation path]\ CadTransformerWebInterface\bin\Formats** folder. Verify the Creo install path in ProEngineerAdvancedFormat.dll.config file (XML file), located in the **Formats** folder. By default, it is set to a **Creo 9.0.5.0** installation. If Creo Parametric is installed in a different folder or the version differs, modify the value elements in the XML file. Example configurations for Creo 8.0, Creo 9.0 and Creo 10.0 are listed below.

Creo 8.0 Example

```
<ProEngineerAdvancedFormat.Settings1>
    <setting name="ProEInstallPath" serializeAs="String">
        <value>C:\Program Files\PTC\Creo 8.0.7.0\Common Files</value>
    </setting>
    <setting name="ParametricExePath" serializeAs="String">
        <value>C:\Program Files\PTC\Creo 8.0.7.0\Parametric\bin\parametric.exe</value>
    </setting>
</ProEngineerAdvancedFormat.Settings1>
```

Creo 9.0 Example

```
<ProEngineerAdvancedFormat.Settings1>
    <setting name="ProEInstallPath" serializeAs="String">
        <value>C:\Program Files\PTC\Creo 9.0.3.0\Common Files</value>
    </setting>
    <setting name="ParametricExePath" serializeAs="String">
        <value>C:\Program Files\PTC\Creo 9.0.3.0\Parametric\bin\parametric.exe</value>
    </setting>
</ProEngineerAdvancedFormat.Settings1>
```

Creo 10.0 Example

```
<ProEngineerAdvancedFormat.Settings1>
    <setting name="ProEInstallPath" serializeAs="String">
        <value>C:\Program Files\PTC\Creo 10.0.1.0\Common Files</value>
    </setting>
    <setting name="ParametricExePath" serializeAs="String">
        <value>C:\Program Files\PTC\Creo 10.0.1.0\Parametric\bin\parametric.exe</value>
    </setting>
</ProEngineerAdvancedFormat.Settings1>
```

TROUBLESHOOTING

If you experience missing tolerances after importing or publishing, please insure that the Creo config.pro file has the setting **tol_display yes** set on the machine that imports the part or assembly with Anark Workstation and/or Anark Publish.