SolidPractices: Working with Toolbox

3DEXPERIENCE SOLIDWORKS®

Last Update: May 2023 Revision 3.1



The **3DEXPERIENCE**[®] Company

Table of Contents

1)	PREFACE	4
2)	HOW DOES TOOLBOX INTEGRATION WORK?	5
A)	DESIGN WITH SOLIDWORKS	5
в)	SOLIDWORKS CONNECTED	11
3)	PLATFORM ADMINISTRATOR SETTINGS FOR TOOLBOX COMPONENTS	
4)	SAVING EXISTING ASSEMBLIES USING TOOLBOX COMPONENTS	16
5)	INSERTING NEW TOOLBOX COMPONENTS INTO AN ASSEMBLY	
6)	PHYSICAL PRODUCT CHARACTERISTICS FOR TOOLBOX COMPONENTS	
A)	MATURITY STATE	19
в)	ENTERPRISE ITEM NUMBER	20
C)	DELETE	21
7)	OPENING TOOLBOX COMPONENTS	
8)	ASSEMBLIES WITH TOOLBOX COMPONENTS LAST SAVED PRIOR TO 3DEXPERIENCE R2021X FD04	
9)	ADDITIONAL RECOMMENDATIONS	25



2



Revision History

Rev #	Date	Description
1.0	Jun 2021	Document created.
2.0	Jun 2022	Modified and updated using R2022x FD02.
3.0	Apr 2023	Modified and updated using R2023x FD02.
3.1	May 2023	Added additional information using R2023x FD02.

Note

All SolidPractices are written as guidelines. It is a strong recommendation to use these documents only after properly evaluating your requirements. Distribution of this document is limited to Dassault Systèmes SOLIDWORKS employees, VARs, and customers that are on active subscription. You may not post this document on blogs or any internal or external forums without prior written authorization from Dassault Systèmes SOLIDWORKS Corporation.

This document was updated using **3D**EXPERIENCE platform R2023x FD02 on Cloud and **3D**EXPERIENCE SOLIDWORKS Premium. The content also applies to Collaborative Designer for SOLIDWORKS and SOLIDWORKS Premium 2023 SP2.0. If you have questions or need assistance in understanding the content, then contact your designated reseller. You can download the most current version of this document from <u>https://my.solidworks.com/support/solidpractices</u>.





1) Preface

Seamless deployment of Toolbox components to the **3D**EXPERIENCE platform was implemented in **3D**EXPERIENCE R2021x FD04 on Cloud. Toolbox integration with **3D**EXPERIENCE provides easier access and management of Toolbox components to all the relevant SOLIDWORKS users within an organization.

When you open an assembly in SOLIDWORKS that contains references to Toolbox components, or you add new Toolbox components to it, the **3D**EXPERIENCE platform automatically recognizes those. The system enforces the generation of each Toolbox component as a part, to facilitate better collaboration through the platform.

This SolidPractices document provides information about Toolbox integration with **3D**EXPERIENCE in various scenarios.

References:

- **3D**EXPERIENCE Help Document: Working with the Toolbox Components
- SolidPractices Document: Setting up SOLIDWORKS Toolbox. Accessible from https://my.solidworks.com/support/solidpractices. This document provides more details on Toolbox functionality.

Your Feedback Requested

We would like to hear your feedback and suggestions for new topics. After reviewing this document, please take a few minutes to fill out a <u>brief survey</u>. Your feedback will help us create the content that directly addresses your challenges.





2) How Does Toolbox Integration Work?

Toolbox integration with **3D**EXPERIENCE supports the following scenarios:

- Save to **3D**EXPERIENCE existing assemblies referencing Toolbox components.
- Insert new Toolbox components in an assembly.

When you add Toolbox components to an assembly from the **Design Library** task pane tab, or open an assembly that includes Toolbox components, the integration automatically enforces the following:

- Parts for Toolbox components are imported to a designated **Collaborative Space**, or are retrieved from there if they already exist.
- Maturity state of the Physical Products created from the imported Toolbox parts are set to In Work or Released, based on the ToolBox settings deployed by the platform administrator.

To designate a **Collaborative Space** other than **Common Space**, you must set its visibility to **Public**.

,	Visibility		
	Public - Content visible to all users		•
		Create	Cancel

Toolbox components imported to **3D**EXPERIENCE by one user become available for re-use by other users without any duplication, and from any connected SOLIDWORKS client computer.

Note: Toolbox integration with **3D**EXPERIENCE only supports the **Create Parts** option that automatically generates individual parts. This does not support managing Toolbox components as configurations.

a) Design with SOLIDWORKS

The **Create Parts** option is available in the Toolbox **User Settings** for **Files** in the SOLIDWORKS desktop software that is used with the **Design with SOLIDWORKS** app (*Collaborative Designer for SOLIDWORKS* role).

When you install this app, it automatically enforces the following in User Settings for Files:





- Activates the **Create Parts** option.
- Specifies the following file path in the Create parts in this folder field -

%LOCALAPPDATA%\DassaultSystemes\CATTemp\ENOUSWC\Resources\[tenant ID]\Toolbox

User Settings				
C Files				
O Create Configurations				
A configuration is added to a master part file each time you use a new size of a particular fastener.				
Create Parts				
An individual part file is created each time you use a new size of a particular fastener.				
 Create Parts on Ctrl-Drag 				
An individual part file is created if you CTRL-drag the fastener from the Toolbox Browser. A configuration is added to the master part file if you use a standard drag.				
Create parts in this folder:				
C:\Users\ \AppData\Local\DassaultSystemes\CATTemp\ENOUSWC \Resources\R \Toolbox				

The app generates *.sldprt files in this folder for each of the Toolbox components integrated with **3D**EXPERIENCE. For each of the *.sldprt files integrated with **3D**EXPERIENCE, the app also generates the required cache files, i.e. *.v6 and *.srv.v6, in the same folder.

> WINDOWS (C:) > Users >	> AppData > Lo	cal → DassaultSystemes → CATTe	emp > ENOUSWC > Resources > R1 > Toolbox
Name	Date modified	Туре	Size
CR-THMS 0.216-28x0.25x0.25-N.SLDPRT	18/04/2023 17:33	SOLIDWORKS Part Document	145 KB
CR-THMS 0.216-28x0.25x0.25-N.sldprt.srv.v6	18/04/2023 17:34	V6 File	1 KB
CR-THMS 0.216-28x0.25x0.25-N.sldprt.v6	18/04/2023 17:35	V6 File	1 KB

However, these cache files are hidden by default, so only visible with the relevant folder option enabled.





Folder Options	×			
General View Search				
Folder views You can apply this view (such as Details or Icons) to all folders of this type. Apply to Folders Reset Folders				
Advanced settings:				
Hiles and Folders Always show icons, never thumbnails Always show menus Display file icon on thumbnails Display file size information in folder tips				
 Display the full path in the title bar Hidden files and folders Don't show hidden files, folders, or drives Show hidden files, folders, and drives 				

These cache files are essential for the Toolbox *.sldprt files to successfully integrate with **3D**EXPERIENCE. More details in the **Note:** below.

In User Settings for Files, the two other available options are not supported for Toolbox integration with 3DEXPERIENCE:

Create Configurations Create Parts on Ctrl-Drag

You should not change **User Settings** to either of these two options, as it will prevent the integration and will show the following message when you use a Toolbox component:



Note: After you install the Design with SOLIDWORKS app on a computer, if you disable the **3DEXPERIENCE** add-in,

[Add-Ins			×
	Active Add-ins	Start Up	Last Load Time	^
	SOLIDWORKS Toolbox Library	\checkmark	< 1s	
	SOLIDWORKS Toolbox Utilities	\checkmark	1s	
	SOLIDWORKS Utilities			
	TolAnalyst			
	SOLIDWORKS Add-ins			_
	D 😳 3DEXPERIENCE		8 s	





any new Toolbox component you insert into an assembly will follow the rules below:

1. If you change the User Settings for Files to Create Configurations –

I	User Settings	
	Files	1
	Create Configurations	
	A configuration is added to a master part file each time you use a new size of a particular fastener.	

Toolbox components will be referenced from the SOLIDWORKS Data folder (default location 'C:\SOLIDWORKS Data').

f) needle roller bearing_nrbb_b	si<1> (BS 5773-2 - 490511 - 6,Sl,NC,6) <display state-4=""></display>		
Find References			
		nclude broken references	
Title	In Folder		
🖃 🎯 Assem-No3DEXP_add-in_creat	E:\SolidPractice-TB-23xFD02		
needle roller bearing_nrbb_bsi.	e roller bearing_nrbb_bsi. C:\SOLIDWORKS Data\browser\BSI\bearings\roller bearings		

2. If you retain the User Settings for Files to Create Parts -

User Settings				
Files				
 Create Configurations 				
A configuration is added to a master part file each time you use a new size of a particular fastener.				
Create Parts				
An individual part file is created each time you use a new size of a particular fastener.				

Toolbox components will continue to be referenced from the designated directory for **Create parts in this folder** mentioned in section a) above.

• 💡 (f) BS 6267 ABB - 170.6 - 8,SI,NC	C,8_68<1> (BS 6267 ABB - 170.6 - 8,SI,NC,8_68) <display state-4=""></display>			
Find References				×
	☑ Include br	roken references	◯ Flat view	
Title	In Folder			
🖃 🎯 Assem-No3DEXP_add-in_creat	E:\SolidPractice-TB-23xFD02			
BS 6267 ABB - 170.6 - 8,SI,NC,8	C:\Users\\AppData\Local\DassaultSystemes\CATTemp\ENOUSWC\Resources\R`\Toolbox			





3. If you change the User Settings for Files to Create Parts on Ctrl-Drag -

User Settings				
Files				
 Create Configurations 				
A configuration is added to a master part file each time you use a new size of a particular fastener.				
O Create Parts				
An individual part file is created each time you use a new size of a particular fastener.				
Create Parts on Ctrl-Drag				
An individual part file is created if you CTRL-drag the fastener from the Toolbox Browser. A configuration is added to the master part file if you use a standard drag.				

- i) With Ctrl-Drag, Toolbox components will be referenced from the designated directory for **Create parts in this folder** mentioned in section a) above.
- ii) With drag and drop, Toolbox components will be referenced from the SOLIDWORKS Data folder (default location 'C:\SOLIDWORKS Data').

	(f) double pitch conveyor chain (-) ISO - RH Helical gear 0.25M	wheel_iso<1> (Chain wheel ISO - 5Z C 208A5SA4.0NC1) <display state-4=""> 10T 45HA 20PA 12FW10A75H50L0.8N<1> (ISO - RH Helical gear 0.25M 10T</display>	
Find R	leferences		×
		✓ Include broken references	◯ Flat view
Title		In Folder	
- 🔍	Assem-No3DEXP_add-in_creat	E:\SolidPractice-TB-23xFD02	
Ē	double pitch conveyor chain w	C:\SOLIDWORKS Data \browser\ISO\power transmission\chain wheels	Drag and drop
Ĩ	ISO - RH Helical gear 0.25M 10	C:\Users\\\AppData\Local\DassaultSystemes\CATTemp\ENOUSWC\Resources\R	\Toolbox
		Ctrl-Dray	g

With the **3DEXPERIENCE** add-in disabled, the Toolbox *.sldprt files created in the designated directory for **Create parts in this folder** will not have the cache files necessary for successful integration with **3DEXPERIENCE**, e.g. in scenarios 2 and i) above.

> WINDOWS (C:) > Users > AppData > Local :	 DassaultSystemes > CA 	TTemp > ENOUSWC > Resource	s > R > Toolbox
Name	Date modified	Туре	Size
BS 6267 ABB - 170.6 - 8,SI,NC,8_68.SLDPRT	19/04/2023 16:09	SOLIDWORKS Part Document	422 KB
ISO - RH Helical gear 0.25M 10T 45HA 20PA 12FW10A75H50L0.8N.SLDPRT	19/04/2023 13:43	SOLIDWORKS Part Document	501 KB





As a result, assemblies previously locally saved in these two scenarios, if later opened in SOLIDWORKS with the **3DEXPERIENCE** add-in enabled (and the **Create Parts** option activated for the add-in to work), will show the Toolbox components as not yet saved to **3DEXPERIENCE**, i.e. not yet successfully integrated with **3DEXPERIENCE**.



This can be easily resolved by simply performing the save to **3D**EXPERIENCE command on the assembly. This will generate the required cache in the designated folder, and will successfully integrate those Toolbox components with **3D**EXPERIENCE.

Assembly Layout Sketch Markup Ev	×	»		3DEXPER	IENCE					
° ♥ ■ ℝ ♦ �;> } <			MySession -)~				Search	R' 'Q
₹-	0	Component Name	e	Status		Rev	ls	Maturi	ty State	Collaborative Space
Assem-No3DEXP_add-in_create_parts2	D	- 🞯 Assem-	-No3DEXP_add-in_create_parts2		ß	A.1	× .	In Wor	k	R14_test
History	27	🛛 🙆 BS	6267 ABB - 170.6 - 8,SI,NC,8 68	Ē,	-	A.1	~	Releas	ed	Common Space
Sensors Carlot Annotations Front Plane Right Plane Corigin Grigin Grigi										
Assembly Layout Sketch Markup Evaluate Lifecycle	and Colla	boration ×	»		3DEXP	ERIENCE				
· (1) (1) (1) (1) (1) (1) (1) (1) (1) (1)		>	ENOVIA MySession -)~			Se	arch R 🛛 🖗
7-			Component Name		Statu	6	Rev	ls	Maturity Sta	ate Collaborative Space
Assem-No3DEXP_add-in_create_parts_C-Drag (Default) <display History</display 	lay State-1		Assem-No3DEXP_add-in_create	parts_C-Drag			A.1	× .	In Work	The second







> WINDOWS (C:) > Users > AppData > Local > I	DassaultSystemes → CATTe	emp > ENOUSWC > Resources >	R > T
Name	Date modified	Туре	Size
BS 6267 ABB - 170.6 - 8,SI,NC,8_68.SLDPRT	19/04/2023 16:09	SOLIDWORKS Part Document	268 KB
BS 6267 ABB - 170.6 - 8,SI,NC,8_68.sldprt.srv.v6	19/04/2023 16:27	V6 File	1 KB
BS 6267 ABB - 170.6 - 8,SI,NC,8_68.sldprt.v6	19/04/2023 16:27	V6 File	1 KB
Chain wheel ISO - 5Z C 208A 5SA4.0NC1.SLDPRT	19/04/2023 15:22	SOLIDWORKS Part Document	209 KB
Chain wheel ISO - 5Z C 208A5SA4.0NC1.sldprt.srv.v6	19/04/2023 16:19	V6 File	1 KB
Chain wheel ISO - 5Z C 208A5SA4.0NC1.sldprt.v6	19/04/2023 16:24	V6 File	1 KB
ISO - RH Helical gear 0.25M 10T 45HA 20PA 12FW10A75H50L0.8N.SLDPRT	19/04/2023 16:24	SOLIDWORKS Part Document	501 KB
ISO - RH Helical gear 0.25M 10T 45HA 20PA 12FW10A75H50L0.8N.SLDPRT.srv.	.v6 19/04/2023 16:24	V6 File	1 KB
SO - RH Helical gear 0.25M 10T 45HA 20PA 12FW10A75H50L0.8N.SLDPRT.v6	19/04/2023 16:24	V6 File	2 KB

To avoid any unexpected complications, general recommendation is not to disable the **3DEXPERIENCE** add-in or to change the Toolbox **User Settings** from **Create Parts** to any of the other two options.

b) SOLIDWORKS Connected

Unlike the SOLIDWORKS desktop software, the **SOLIDWORKS Connected** app (*3DEXPERIENCE SOLIDWORKS* role) does not include the options for **Files** in Toolbox **User Settings**. This is because only the **Create Parts** option is hardcoded in this app.

User Settings	
Writing to read-only documents]
 Always change read-only status of docume 	ent before writing
 Error when writing to a read-only document 	nt
Part numbers	
Allow duplicate part numbers for geometri	cally equal components
Display options	
Show as Component Name in FeatureManager	Filename 💌
Show as Part Number in Bill of Materials	Configuration Name 💌
Show as Description in Bill of Materials	Description 💌
* Designation applies only to the AS, DIN, GB, IS	SO, IS and KS standards.

In the background, the Toolbox *.sldprt and **3D**EXPERIENCE cache files are generated in the same designated directory as mentioned in section a) above.



11



Also, unlike the SOLIDWORKS desktop software, the Add-Ins list in the SOLIDWORKS Connected app does not include the **3DEXPERIENCE** add-in, as this is hardcoded in online mode.

Add-Ins		×
Active Add-ins	Start Up	Last Load Time
SOLIDWORKS Premium Add-ins		
CircuitWorks		
E FeatureWorks		
ScanTo3D		
SOLIDWORKS Design Checker		< 1s
SOLIDWORKS Routing		
SOLIDWORKS Toolbox Library	\checkmark	< 1s
SOLIDWORKS Toolbox Utilities	\checkmark	2s
SOLIDWORKS Utilities		
TolAnalyst		
SOLIDWORKS Add-ins		
Autotrace		
Other Add-ins		
3DCloudByMe Plug-in		
Batch Save to 3DEXPERIENCE	\checkmark	< 1s
SOLIDWORKS XPS Driver 2023		
OK Cancel		





3) Platform Administrator Settings for Toolbox Components

The platform administrator can choose a designated **Collaborative Space** for Toolbox components. They can also set the initial **Maturity** state to either **In Work** or **Released**, for when those are uploaded to the designated **Collaborative Space** for the first time.

These settings are available in the **ToolBox settings** section of the **Standard Libraries Management** widget of **Collaborative Spaces Configuration Center**.

<i>3</i> 5 ₀ №	3DEXPERIENCE Collaborative Space	tes Configuration Center - 🗸 🗸 🗸	and the second second) >> @ ::
V.R		E Standard Libraries Management	ा । द	
 Image: A set of the set of the	Project Management Configure Project Management Experience	Configuration of Standard Libraries settings	1	
	Quality Document Control	▼ Part Supply settings		
	Configure Quality Document Control experience	Collaborative Space	0	Common Space 🔻
•	RAMS and APQP Core Tools	State	0	RELEASED V
4	Dennes parameterization for RAMS and APUP Co	▼ ToolBox settings	Allows to set Collaborative Space for this context	-
<u>B</u>	Configure Requirements Management Experience	Collaborative Space	•	Common Space 🔻
	Retention policy for iterations	State	0	RELEASED V
	Allows to define the retention rules for the iteration	▼ Content Center settings		
<u>5</u> 2	Simulation Automation Define simulation automation deployment options	Collaborative Space	0	Common Space 🔻
	Specification Management Configure Specification Management Experience	State	Û	RELEASED V
	Standard Libraries Management Allows you to configure Standard Libraries behavi		* *	

Out-Of-The-Box (OOTB) values for these settings are -

Collaborative Space = Common Space State = RELEASED







▼ ToolBox settings	-		-	
Collaborative Space	3	0		Common Space 🔻
State	-	0		RELEASED V
	1		- 📌	RELEASED
▼ Content Center settings			4	IN_WORK

As mentioned in section 2), to designate a **Collaborative Space** other than **Common Space**, you must set its visibility to **Public**.

The values set in **ToolBox settings** only affect the Toolbox components that you upload after you define those values. This does not affect any Toolbox components that you previously uploaded.

As an example, consider an assembly that you previously saved to **3D**EXPERIENCE with Toolbox components uploaded to Common Space as **Released**.



Then in **ToolBox settings**, you defined a different **Collaborative Space** and set **Maturity** state to **IN_WORK**. If you then open the assembly in SOLIDWORKS from **3D**EXPERIENCE -



1. The previously saved Toolbox components will still be in Common Space as **Released**. This will still be the case even if you clear local cache for both the assembly and the Toolbox components. Any subsequent save command on the assembly will not change this.







2. You insert new Toolbox components in the same assembly – only the new Toolbox components will be uploaded to the new **Collaborative Space** as **In Work**, as per your values defined in **ToolBox settings**.

♥ ■ K ♥ ♥	6	> 🚱	<i>در</i> س					~	· ·
7.		1	Component Name	Status		Rev	ls	Maturity State	Collaborative
Family00019773 (Default) < Display State-1>			- 🞯 Family00019773	-	1	А	~	In Work	Common Space
Isotroy			BS 6267 ABB - 170.6 - 8,SI,NC,8_68<2>		L.	А	~	Released	Common Space
Sensors	1		BSI 3469 - 170602 - R,8,SI,NC,8<1>		d l	А	~	In Work	test_cs
[] Front Plane	1								
[] Top Plane	1	- <u></u>							
[]] Right Plane									
T (1) BS 6267 ABB - 170.6 - 8,51,NC,8_68<22 (BS 6267 ABB - 170.6 - 8,51,NC,8_68) < Display State-4> Q (1) BS 6267 ABB - 170602 - P. 9 SI MC 9/15 (PSI 2469 - 170602 - P. 9 SI MC 9) < Display State-4>	2								
II (-) bol 3409 - 170002 - n,o,ol,140,ok 12 (bol 3409 - 170002 - K,8,51,140,8) < Display State-42	4								

3. You have the flexibility to move the previously uploaded Toolbox components from Common Space to another **Collaborative Space**, e.g. to the newly designated **Collaborative Space** in **ToolBox settings**.







4) Saving Existing Assemblies Using Toolbox Components

When you open an existing assembly referencing Toolbox components, regardless of the Toolbox **User Settings** originally used (applicable to SOLIDWORKS desktop software) to insert the Toolbox components, the **Create Parts** option takes effect. All Toolbox components are retrieved from and created inside the designated directory for **Create parts in this folder**, as described in section 2).

When you perform the save to **3D**EXPERIENCE command on the assembly, all Toolbox components are automatically imported to the **3D**EXPERIENCE platform before the save transaction itself.

If you open the assembly without first establishing a connection to **3D**EXPERIENCE, the Toolbox components will appear in **MySession** with the status **File has not yet been saved into 3D**EXPERIENCE.



After establishing a connection, when saving to **3D**EXPERIENCE using the **Save with Options** command, the Toolbox components will appear in the **Save to 3DEXPERIENCE** dialog box with **Not modified** status, indicating that those have already been imported to the designated **Collaborative Space**. Toolbox integration with **3D**EXPERIENCE establishes this status by either importing the new Toolbox components to or retrieving the existing ones from the designated **Collaborative Space**.

Save to Sel	BDEXPERIENCE lect Bookmark, ▼	odified (0)) Warning or Error (0)				View: ① Flat	Export
Туре	SW Title	Save	Status	Revision	Collaborative Space	Maturity State	Bookmark	New F
9	Assem-TB		V New					
4	ISO 4027 - M1.6 x 2-N	\checkmark	✓ Not modified	A.1	Common Space	Released		
4	ISO 7434 - M1.2 x 2-N	\checkmark	✓ Not modified	A.1	Common Space	Released		
4	ISO 14586 - ST2.9 x 6.5-C-N	\checkmark	✓ Not modified	A.1	Common Space	Released		
<								>
Unloc	ck files after saving					Save	Cancel	Help





If you open the same assembly with an established connection to **3D**EXPERIENCE, the status bar at the bottom of the SOLIDWORKS window will show the progress of part creation for Toolbox components that have not yet been uploaded to **3D**EXPERIENCE.



Once the assembly is fully open, the Toolbox components will appear in **MySession** as already saved to **3D**EXPERIENCE, with the values defined in **ToolBox settings** for **Collaborative Space** and **Maturity** state.

		- 30	(11132			, 0.		Courter	
0	Component Name	▼	Status		Rev	ls	Maturity State	Collaborative Space	File Name
D	- 🞯 Assem-TB		•						Assem-TB.SLDASM
	– 🚳 ISO 7434 - M1.2 x 2-N			ď	A.1	~	Released	Common Space	ISO 7434 - M1.2 x 2-N.SLDPRT
	– 🔞 ISO 4027 - M1.6 x 2-N			ď	A.1	~	Released	Common Space	ISO 4027 - M1.6 x 2-N.SLDPRT
	🛯 🚳 ISO 14586 - ST2.9 x 6.5-C-I	N		ď	A.1	~	Released	Common Space	ISO 14586 - ST2.9 x 6.5-C-N.SL





5) Inserting New Toolbox Components Into An Assembly

To insert new Toolbox components into an assembly, use the Toolbox add-in and the **Design Library** task pane tab.

Add-Ins		×		«	Design Library
Active Add-ins	Start Up Last Lo	ad 🔨		۲	9 • 🗰 🐿 🗁 😂 🕆 📍
SOLIDWORKS Premium Add-ins			101	~ 9	Toolbox
	· · ·				Favorites
FeatureWorks	□ -		D	,	ANSI Inch
PhotoView 360	<u> </u>		F		ANSI Metric
🗌 🧠 ScanTo3D	<u> </u>		•		AS
SOLIDWORKS Design Checker	 < ' 	s	=		BSI BSI
SOLIDWORKS Motion	· · ·			1	
SOLIDWORKS Routing	· ·		<u><u><u>u</u></u></u>	1	Cise
SOLIDWORKS Simulation	· ·)	DIN
SOLIDWORKS Toolbox Library	× ·	s)	GB
SOLIDWORKS Toolbox Utilities		s)	is Is
SOLIDWORKS Utilities	- 1			>	ISO ISO
TolAnalyst	- I			>	ZIL 💽
)	KS
SOLIDWORKS Add-ins		_		;	MIL
3DEXPERIENCE	\checkmark	s		3	PEM ® Inch
Autotrace	-				PEM ® Metric
SOLIDWORKS CAM 2023	IZ 10	1e Y		ĺ,	• SKE®
OK	ancel				SKF SKI -
UK (ancer)	inch and a second second
		11)	Iorrington® Metric

As you insert each Toolbox component into an assembly, the software completes one of the following two actions:

- 1. If the Toolbox component already exists in the designated **Collaborative Space**, the software retrieves the component information, and adds the part to the assembly.
- 2. If the Toolbox component does not exist in the designated **Collaborative Space**, the software generates a new part file automatically, and saves it to **3D**EXPERIENCE as per the values defined in **ToolBox settings**.





18

6) Physical Product Characteristics for Toolbox Components

a) Maturity State

The platform administrator can change the **Maturity** state of existing Toolbox component Physical Products from **Released** to **In Work**.

	ENOVIA - C	Collaborative Lifecycle (R)		Power	Frozen RELEASEDIOIN_	Released	Obsolote Make obsolete	-				
ENOVIA - Collaborative Lifecycle ISO 4027 - M1.6 x	Graph 🔻	Title ISO 4027 - M1.6 x 2-N	Revi	Maturity State Released			Creation Date April 20, 2023 at	Modificatio 5:42 April 20, 2	×	?	¢	а ;	* ×
		Pit	Vate In Make private	Vork Frazen	Released Cosolele				 	Maturity (Disabling following	index acc	ccessful. celeration ing opera	n X A
													1 -

For this, the platform administrator must ensure that the **Maturity** transition from **Released** to **In Work** is correctly applied to **Engineering Definition** of the **Maturity Graph** widget.

N	laturity G	Graph	€ Engineering Definition	+	×	*
		Q 📑	🛠 Edit 🞯 About			
	\$	Bookmark Workspace			~	
	\$	Change Action Change Action	Private Freese Consider			
	L	Change Order Fast track Change				
	\$	Change Request Request For Change	Current Controls for Maturity Graph			×,
*	<u>.</u>	Chapter Chapter	Object Type From To Rule Additional info Remove			<u>_</u>
	\$	Document Release				
	Ľ	Engineering Definition VPLM_SMB_Definition				

The platform administrator will need to have the required credentials for these. More details on this are documented in the following **User Assistance** topics:

Changing the Maturity of Objects

Configuring Maturity Graphs

The platform administrator can also use the **Default** • Administrator credentials for this.





Edit preferences	
3DEXPERIENCE Platform	
R	v
Credentials	
Default • Administrator	•
	Save

b) Enterprise Item Number

You can assign **Enterprise Item Numbers** (EIN) to Toolbox component Physical Products that are in **In Work Maturity** state.

							Properties	×
							a she she	and and a product
							é.	J 4521 - MIN AZ
							 Enterprise Information 	on
							Enterprise Item Number	DS000085
 							 Attributes 	
Component Name	Status	Re	ev Is	Maturity State	Collaborative S	pace File	Type	Physical Product
- 🧐 Assem-TB		A	.1 🗸	In Work	RII4_test	Asse	e Titlo *	ISO 4027 - M1 6 x 2-N
– 🚯 iso 4027 - m1.6 x 2	?-n<1>		.1 🗸	In Work	Common Space	ISO 4	Revision	A.1
Enterprise Item Nun	nber - 1 Object					×	Revision Comment	ومعاديه والمستحق المسوى المحال المحا
					×) +	www.stange	wurn.cange
Title	1	Enterprise Ite	m Numbe	r		Menu	Collaborative Policy	Engineering Definition
iso 4027 - m1.6 x 2-n<	:1>	DSXXXXXX				\sim	Modification Date	4/21/2023, 2:13:41 PM
							Maturity State	In Work
					Set	Cancel	uun	
							Collaborative Space	Common Space

You can also customize your assembly **Tree Display** to show EIN as the **Primary** name, which will also appear in the **Component Name** column of **MySession**.



20



c) Delete

You can delete existing Toolbox components from a Collaborative Space.

- 1. If a Toolbox component is in **In Work Maturity** state, you can delete it as the **Leader** of the **Collaborative Space**.
- 2. If it is in **Released Maturity** state, the platform administrator can delete it with the **Default Administrator** credentials. Alternatively, they can demote the **Maturity** state to **In Work** first, and then you can delete it as the **Leader** of the **Collaborative Space**.

More details on delete permissions with different access roles are documented in the User Assistance topic <u>Baseline Responsibilities</u>.





For Toolbox components, changing the **Maturity** state, moving to a different **Collaborative Space** or deleting those will affect all the users within a platform tenant. It is therefore recommended that only a designated Toolbox administrator performs these activities, and with caution.





7) Opening Toolbox Components

- As mentioned in section 2), when you open an assembly referencing Toolbox components, the Toolbox components are downloaded to the designated local folder, and not to your local work folder.
- You can search for Toolbox components in **3D**EXPERIENCE using **3DSearch**. However, the **Open** or drag operations into an assembly, or as a part, will return the following message:



You should always use the **Design Library** task pane tab to insert Toolbox components into an assembly.

• Toolbox integration with **3D**EXPERIENCE, as described in section 2), only works within an assembly environment. If you open a Toolbox component on its own as a part, using the drag operation from the **Design Library** task pane tab, it will open from the folder location specified for **Hole Wizard and Toolbox folder** in SOLIDWORKS **System Options**. In this case, the automated integration with **3D**EXPERIENCE will not work. You should avoid opening Toolbox components on their own as individual parts to save those to **3D**EXPERIENCE.



8) Assemblies with Toolbox Components Last Saved Prior to 3DEXPERIENCE R2021x FD04

You may have existing assemblies referencing older version Toolbox components that were last saved before the release of the **3D**EXPERIENCE R2021x FD04 update. When you open these assemblies and save in session the next time, the legacy Toolbox components automatically convert to imported Toolbox parts, and are saved to the designated **Collaborative Space**. The software maintains the 'where used' references of the Toolbox components.

If you do not save those assemblies in session, those continue to reference the older version Toolbox components.





9) Additional Recommendations

• Toolbox integration with **3D**EXPERIENCE is designed to work when the following Toolbox option is active in SOLIDWORKS desktop software:

 $\label{eq:system} System \ Options > Hole \ Wizard/Toolbox > Make \ this \ folder \ the \ default \ search \ location \ for \ Toolbox \ components.$

System Options - Hole Wizard/Too	lbox		×
System Options		🚱 Search Options	Q
General 3DEXPERIENCE Integration MBD Drawings Display Style Area Hatch/Fill Performance	Hole Wizard and Toolbox folder: C:\SOLIDWORKS Data\ Make this folder the default search location for Toolbox components Configure		
Colors Sketch 	Toolbox Task Pane: Display Toolbox Favorites folder Toolbox Mates: Lock rotation of new concentric mates to Toolbox components		
Assemblies External References Default Templates File Locations FeatureManager	Hole Wizard settings: O Preserve settings for each Hole Wizard hole type Transfer settings when changing Hole Wizard hole type		
Spin Box Increments View Backup/Recover Touch Hole Wizard/Toolbox File Explorer	Include data for DELMIA applications		
Reset	[OK Cancel	Help

This setting is automatically enforced with the **Design with SOLIDWORKS** app installation, although users can later disable it. You should avoid disabling this option, as this can lead to unexpected behavior.

For **SOLIDWORKS Connected Premium** and **Professional** (that include the Toolbox addin), this option is greyed out.

- You should not attempt to revise Toolbox components, as it is currently not supported.
- It is possible to generate duplicates of Toolbox components using the **Duplicate** command from supported ENOVIA apps (e.g. **Collaborative Lifecycle**). It is also possible to open those duplicates in SOLIDWORKS from **3D**EXPERIENCE, using the drag operation from **3DSearch** into an assembly. Duplicates should only be generated when strictly necessary and with caution, to avoid unwanted complications managing Toolbox components.





• To ensure that model graphics for each Toolbox component size is generated, and is viewable on the **3D**EXPERIENCE platform, it is recommended that the following SOLIDWORKS option is enabled:

System Options - Assemblies			×	
System Options		😥 Search Options	Q	
General 3DEXPERIENCE Integration Drawings — Display Style — Area Hatch/Fill — Performance Colors Sketch — Relations/Snaps Display Selection Performance Assemblies External References Default Templates File Locations FeatureManager Spin Box Increments View	 Move components by dragging Optimize component placement when adding mates Allow creation of misaligned mates Save new components to external files Update model graphics when saving files Automatically check and update all components in Large Desig Change mate alignments on edit: Update out-of-date Speedpak configurations when saving files: Opening a large assembly Use Lightweight mode and Large Assembly Settings when the number of components exceeds: Use Large Design Review mode when the number of components exceeds: 	yn Review mode Always ✓ None ✓ 500 € 5000 €		
Backup/Recover Touch Hole Wizard/Toolbox Messages/Errors/Warnings Dismissed Messages Synchronize Settings Import Export Reset	Large Assembly Settings	OK Cancel Help	• •	

System Options > Assemblies > Update model graphics when saving files.

We hope that you find this document informational and useful and request that you leave a <u>brief feedback</u> about the topics that you want us to cover in the next revision of this document. Click <u>here</u> for a complete list of SolidPractices documents available from DS SOLIDWORKS Corp.



