Hole Wizard and Toolbox Guide Packet

This document contains guides to many different common user actions and their best practices involving the SOLIDWORKS Hole Wizard and Toolbox Database. This acts as a guide during Administration, Customization, Access or Display of the Hole Wizard and Toolbox.

SECTIONS

ADMINISTRATION

Upgrade Version

This runs through the steps necessary to manually update the Hole Wizard and Toolbox Database to the current version under a non-PDM Pro environment. It contains two parts: one to update the database and the secondly to update the Hole Wizard and Toolbox content itself.

From the host computer or server:

Part 1: Update the Admin Side of Hole Wizard and Toolbox Database

STEP 1: Make a backup copy of current Hole Wizard/Toolbox Database in a .zip folder (Default location: Start menu > Computer > C: > **SOLIDWORKS Data**) by right-clicking and **paste** to Desktop (or preferred convenient location). Then right-click and **send** to zip:

Note: Toolbox location is verified by accessing System Settings>Hole Wizard/Toolbox
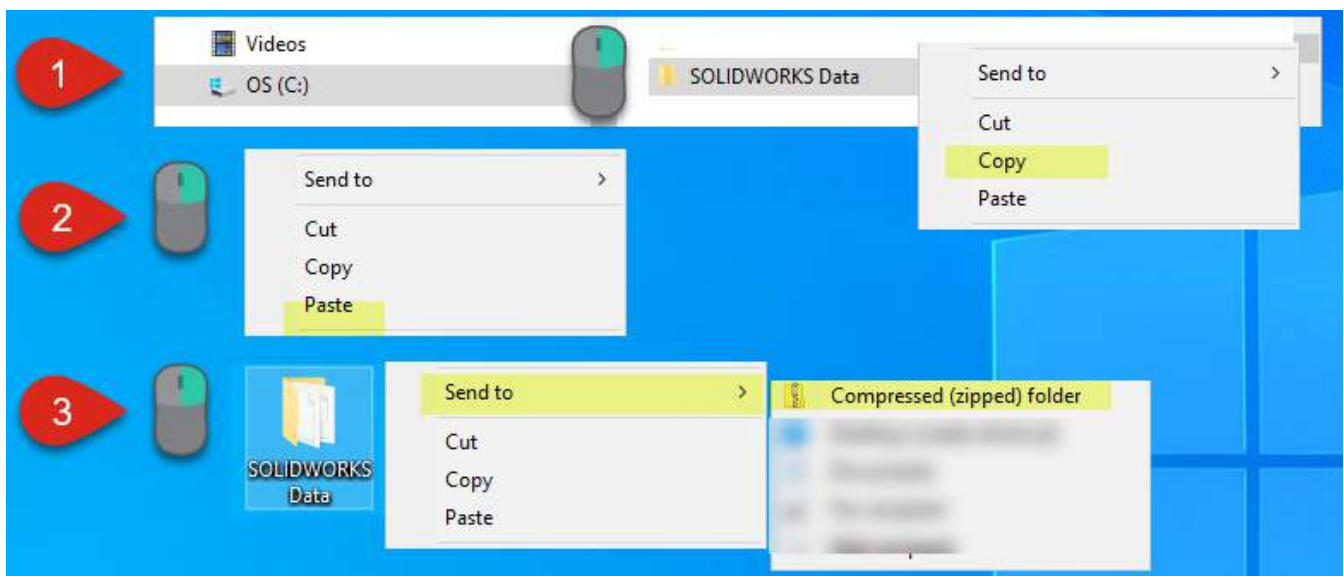
STEP 2: **Remove** Read-only status of swbrowser.sldedb, if it is Read-only. From SOLIDWORKS Corp folder of the new install version of SOLIDWORKS (Start > Programs > Computer > Local Disk > Program Files > **SOLIDWORKS Corp**), type **swbrowser.sldedb** into search bar on top-right. Then right-click and select **Properties** (Figures 1 and 2):
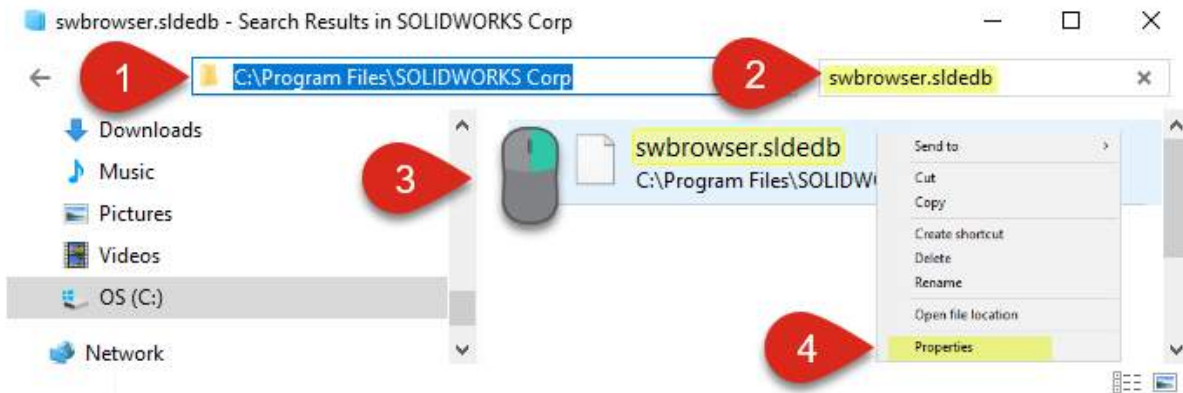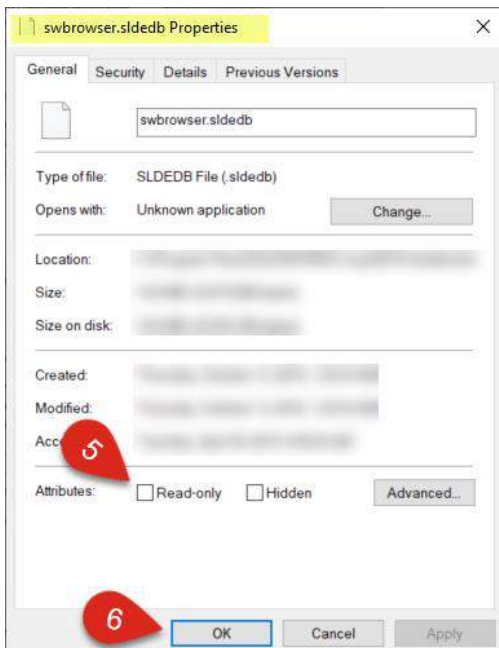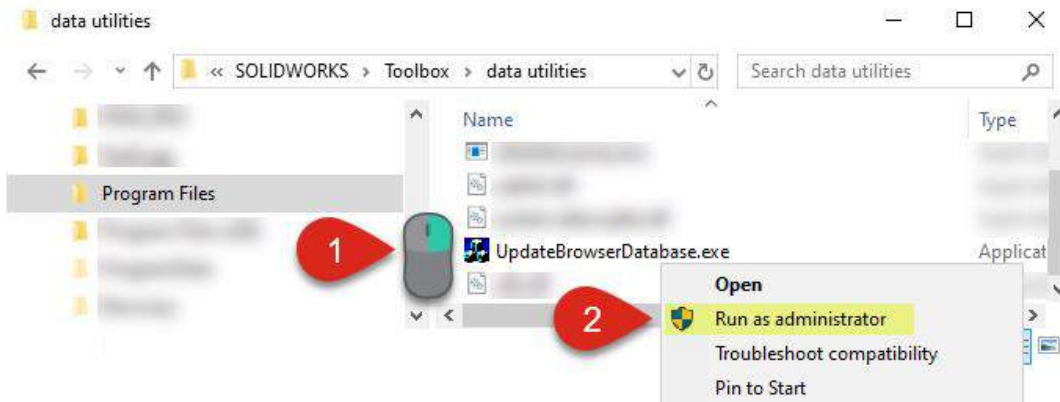

Figure 1

Figure 2

STEP 3: **Open** the UpdateBrowserDatabase.exe program to update the browser file (Start > Programs > Computer > Local Disk > Program Files > SOLIDWORKS Corp > SOLIDWORKS>Toolbox > **Data Utilities**) by right-clicking and selecting **Run as administrator**:

Make sure this is the New Install version

STEP 4: **Verify** that the path is correct for "Database To Update:" (C:\SOLIDWORKS Data\lang\English\swbrowser.sldedb). **Verify** the path is correct for "Updating Database:" click **Update:**

***Note:** Path for "Updating Database" needs to be the latest version installed if multiple versions are installed. Example path: C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS\Toolbox\data utilities\lang\English\swbrowser.sldedb
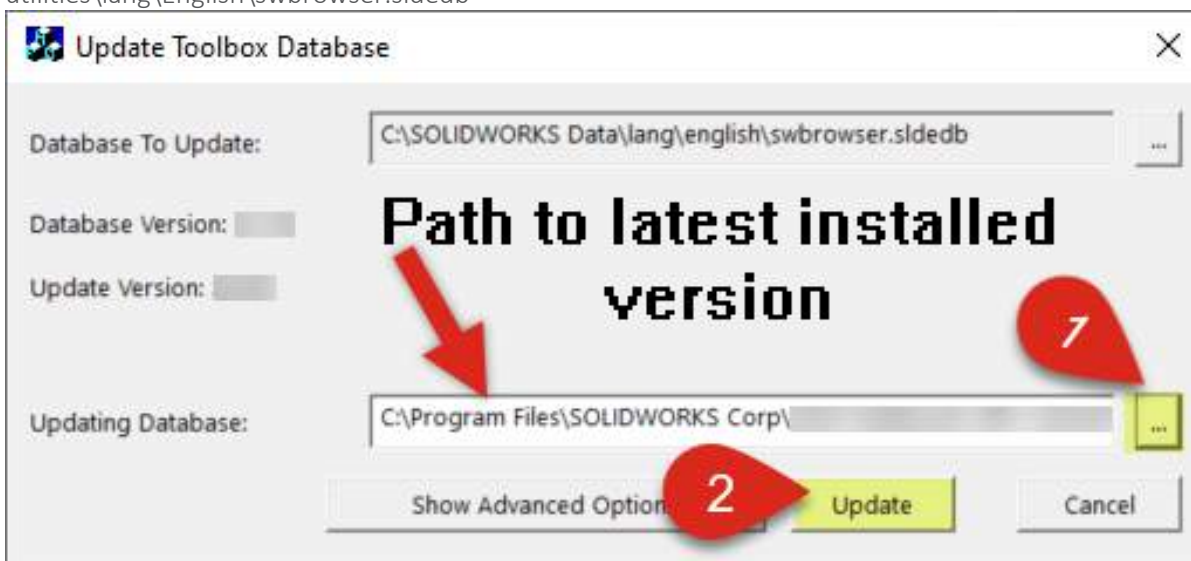
STEP 5: From within SOLIDWORKS, update the sourcing of the Toolbox. Click the **system options icon** (gear icon), then select **Hole Wizard/Toolbox** from the menu and choose the **Browse** button (right of text area):
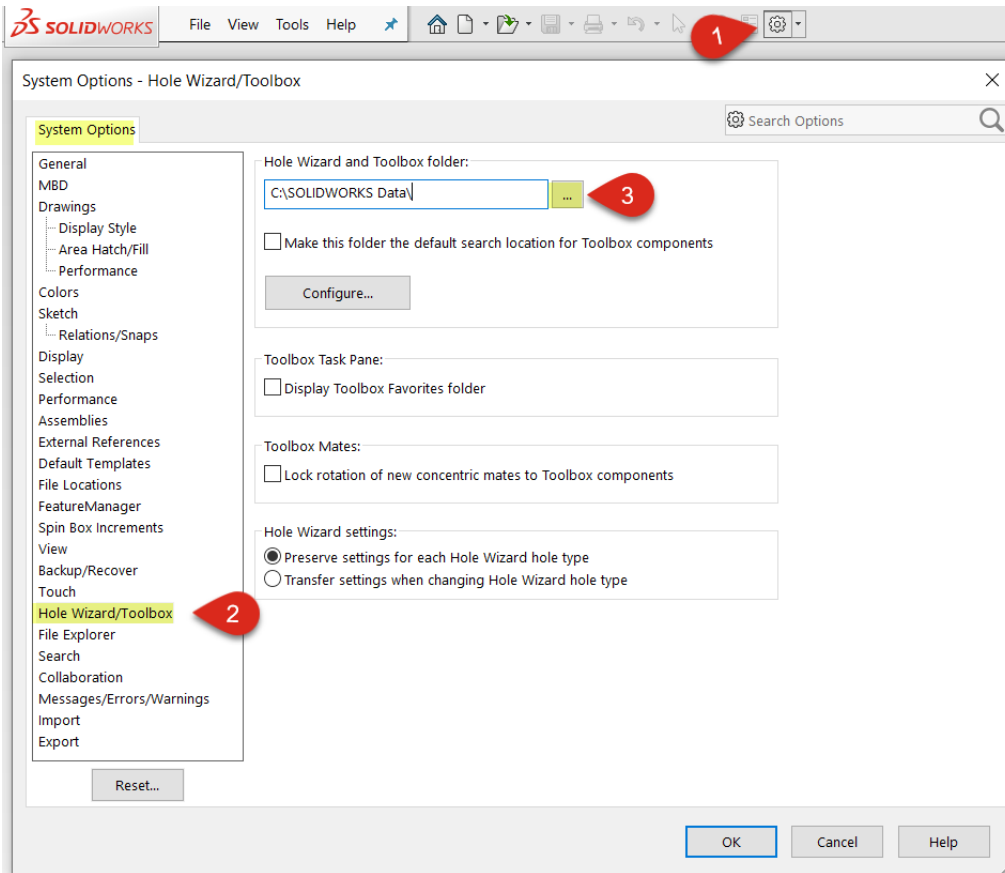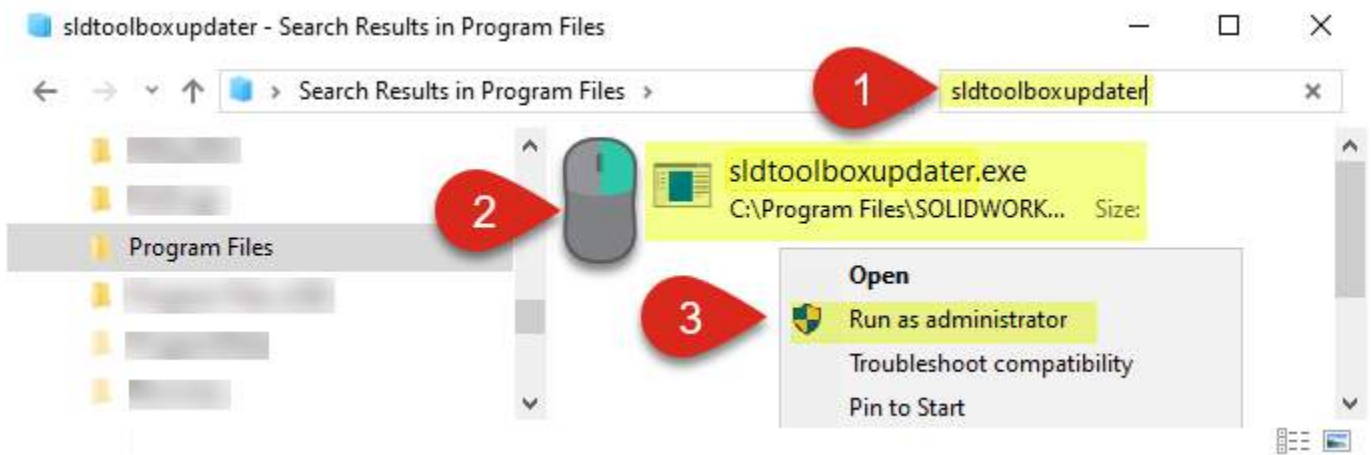

STEP 6: Browse for the **SOLIDWORKS Data** folder and select. Click **OK**:
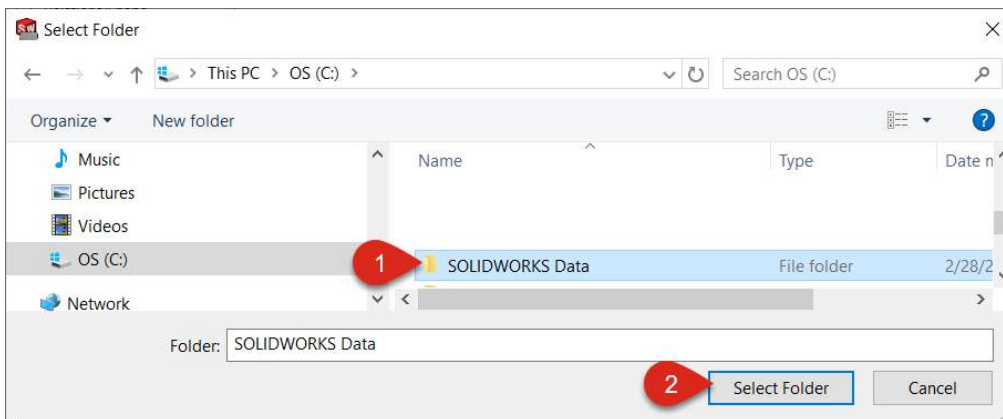
Part 2: Update Hole Wizard and Toolbox Database Content

STEP 1: **Browse** to the new install location of SOLIDWORKS (usually C: > Program Files > SOLIDWORKS Corp > SOLIDWORKS) and type **sldtoolboxupdater** in the search bar, then right-click and **Run as administrator:**

STEP 2: Click **Process** to start the update (this may take a minute):

USER SETTINGS: SINGLE AND MULTIPLE USERS

Hole Wizard and Toolbox Database settings will vary depending on the amount of users. This guide will show the different system settings required for both the single user and multi-user setup.
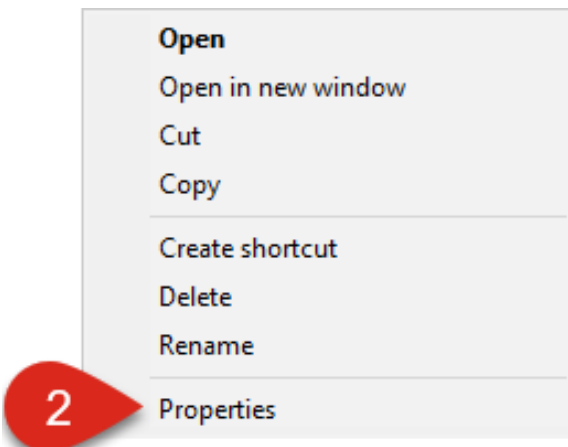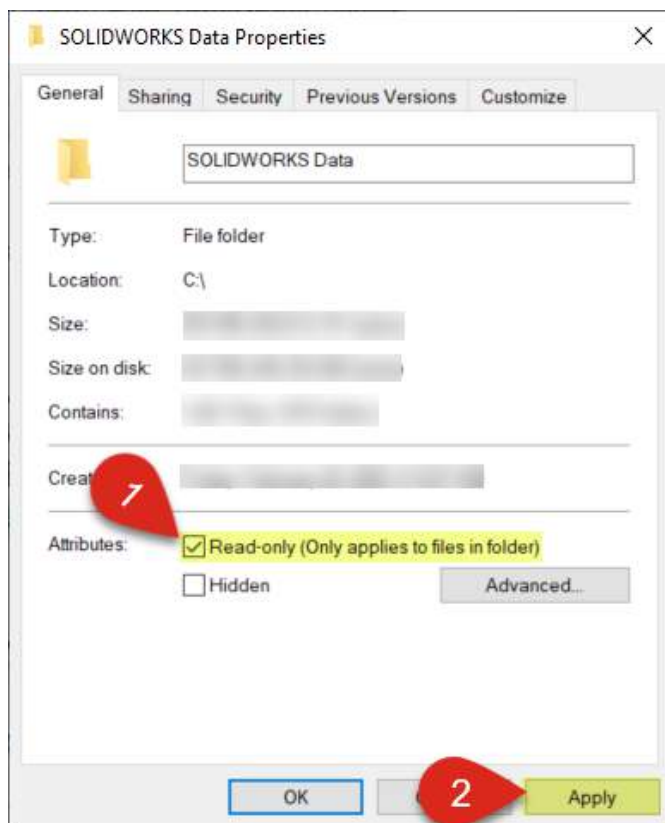
From User Computer

Part 1: Single User

STEP 1: To set files to read-only (with the exception of swbrowser.sldedb and Toolboxfiles.index), **browse** to the **SOLIDWORKS Data** folder and right-click (Start menu > Computer > C: > **SOLIDWORKS Data**):
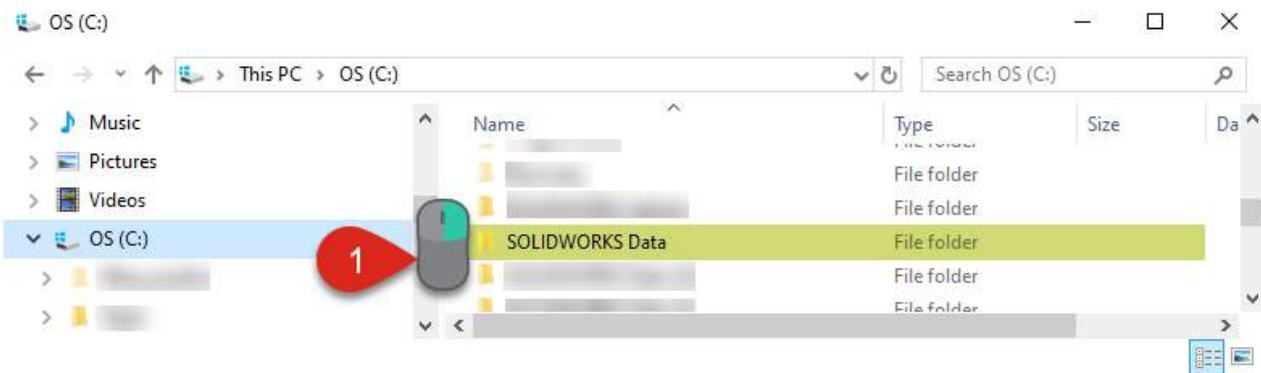
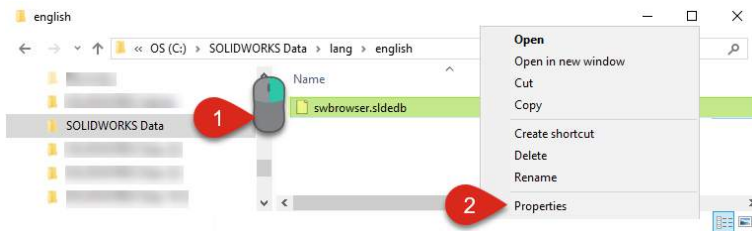**STEP 2:** Choose **Properties** from drop-down menu:

STEP 3: Click the **Read-only** check box twice so it is checked, then choose **Apply** (this will process changes):

SOLIDWORKS Data Properties

General Sharing Security Previous Versions Customize

SOLIDWORKS Data

Type:	File folder
Location:	C:\
Size:	
Size on disk:	
Contains:	

Creat **7**

Attributes: ☑ Read-only (Only applies to files in folder)
☐ Hidden   Advanced...

OK   **2** ⟶ Apply

STEP 4: Click **OK** to exit out of properties:

Attributes: ☑ Read-only (Only applies to files in folder)
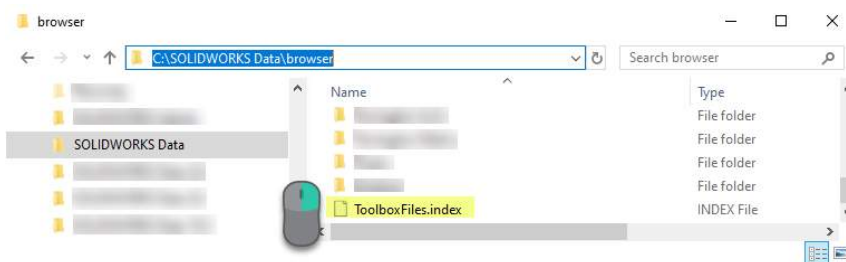☐ Hidden   Advanced...

1 ⟶ OK   Cancel   Apply

STEP 5: Now we need to correct for the Swbrowser.sldedb. **Browse** for the files (SOLIDWORKS Data > lang > English > **Swbrowser.sldedb**). Right-click, then choose **Properties**:

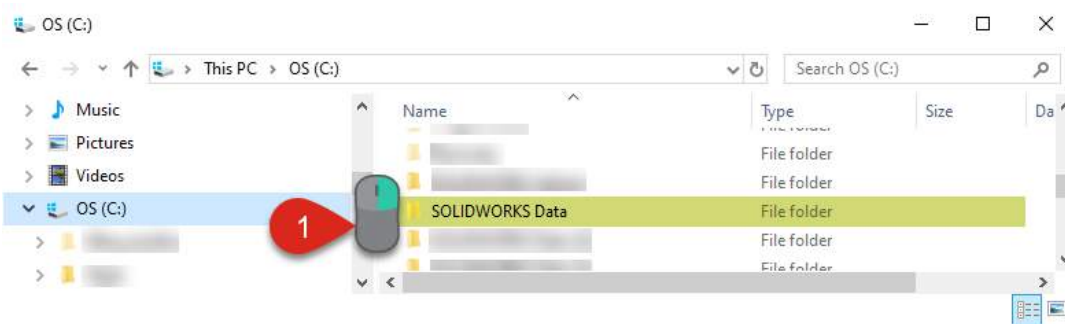**STEP 6:** Unselect the **Read-only** checkbox and click **Apply.** Click **OK** to close the window:
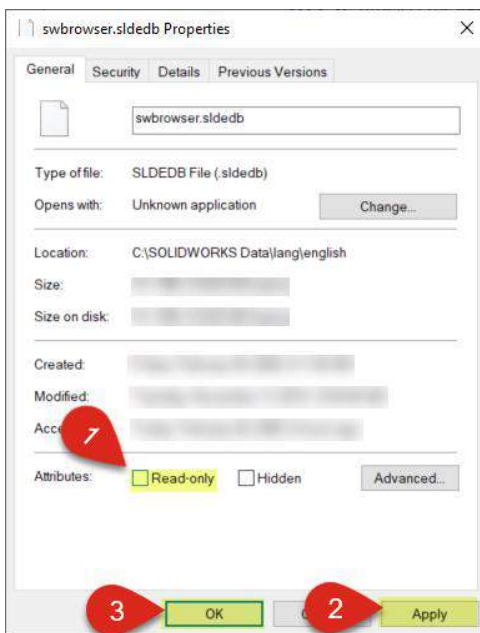
STEP 7: **Repeat** Step 6 to change Read-only status of ToolboxFiles.index (C: > SOLIDWORKS Data > **browser**):
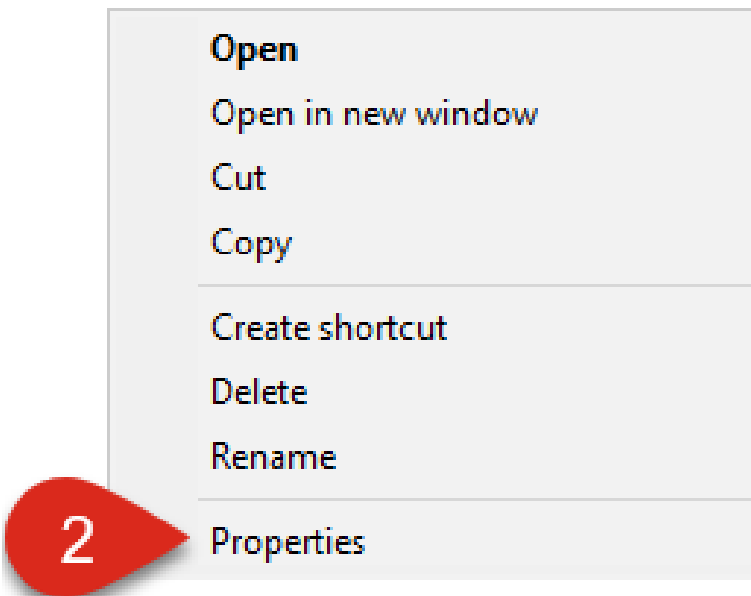
Part 2: Multi-User

Set Database Files to Read-Only

STEP 1: To set files to read-only in the Hole Wizard and Toolbox Database (with the exception of Swbrowser.sldedb and toolboxfiles.index), **browse** to the SOLIDWORKS Data folder (Start menu > Computer > C: > **SOLIDWORKS Data**) and **right-click**:

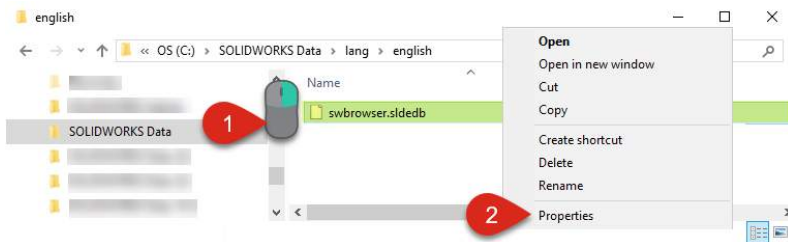**STEP 2:** Choose **Properties** from the drop-down menu:

STEP 3: Click the **Read-only** checkbox twice so it becomes checked, then hit **Apply** (this will process changes):
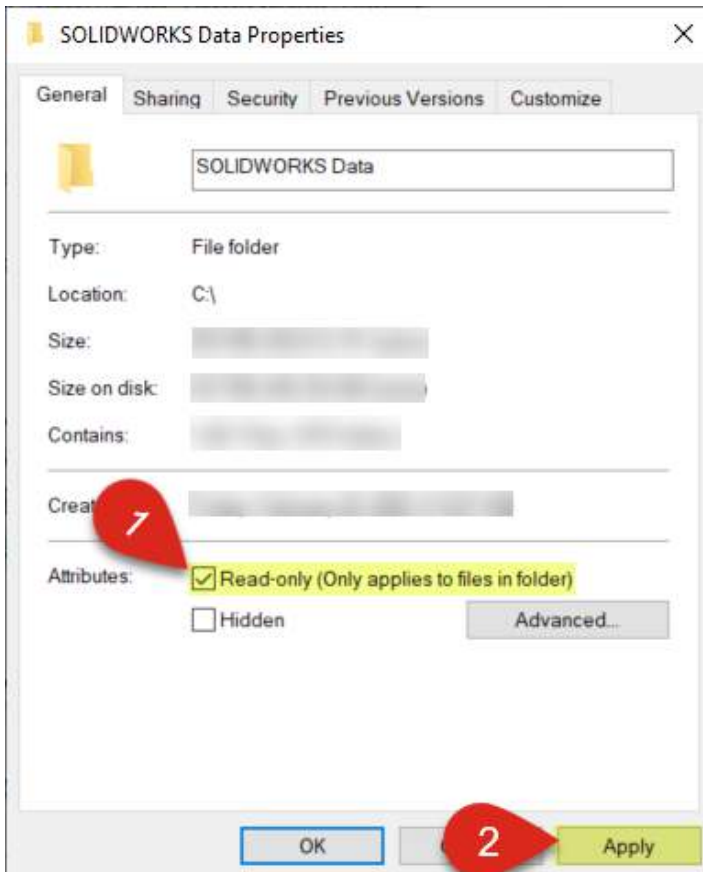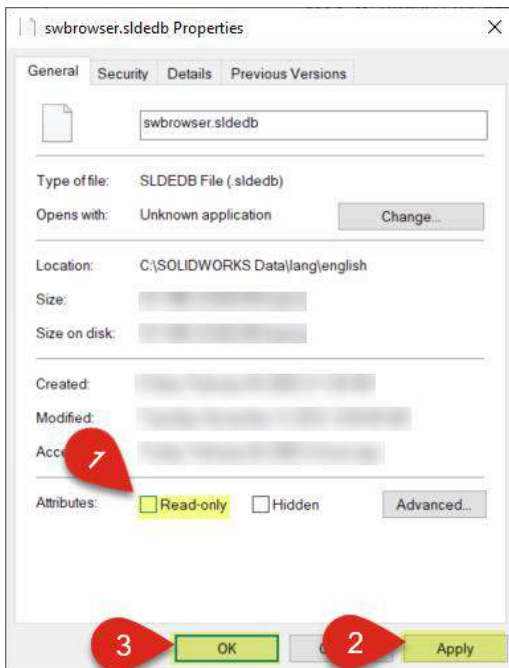

**STEP 4:** Click **OK** to exit out of properties:

STEP 5: Now we need to correct for the Swbrowser.sldedb. **Browse** for the file (SOLIDWORKS Data > lang > English > **Swbrowser.sldedb**) and right-click. Then choose **Properties**:

STEP 6: **Unselect** the Read-only checkbox and click **Apply**. Click **OK** to close the window:


STEP 7: **Repeat** Step 6 to change Read-only status of Toolboxfiles.index (C: > SOLIDWORKS Data > **browser**):
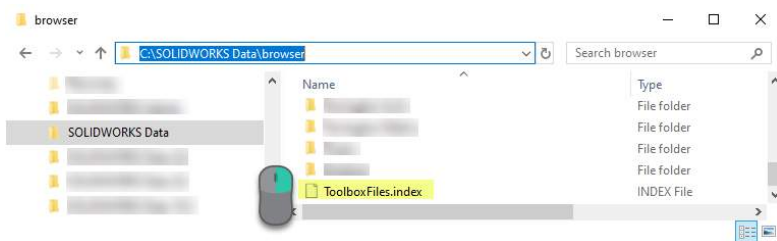
**Move Hole Wizard and Toolbox Database Location to Shared Folder**

STEP 8: **Copy** Toolbox to network location. **Find** and **cut** SOLIDWORKS Data folder (Start menu > Computer > C: > **SOLIDWORKS Data**):

*Note:** If there is no toolbox currently (never installed), set the install location to the network folder.
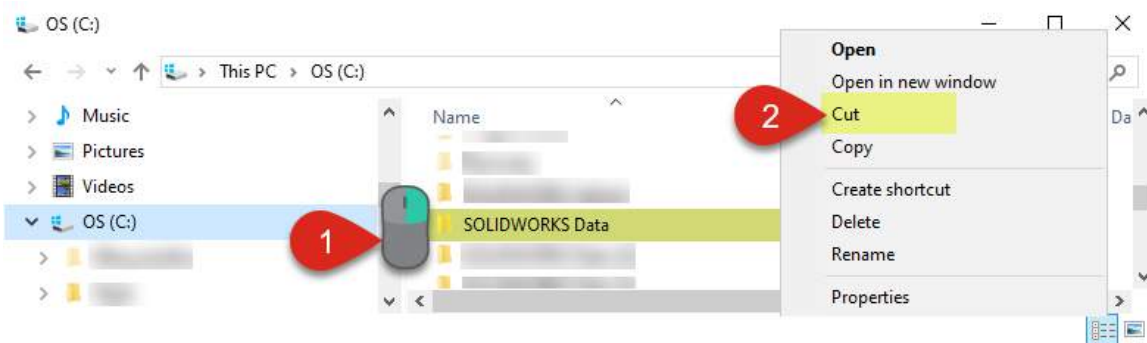
*Note:** Copying the existing Hole Wizard and Toolbox Database as a .zip for backup somewhere on host machine is advised. Then, find and delete any other SOLIDWORKS Data folders that might exist on the client machines that could get confused with the main one (Start > Computer > Local Disk > SOLIDWORKS Data).
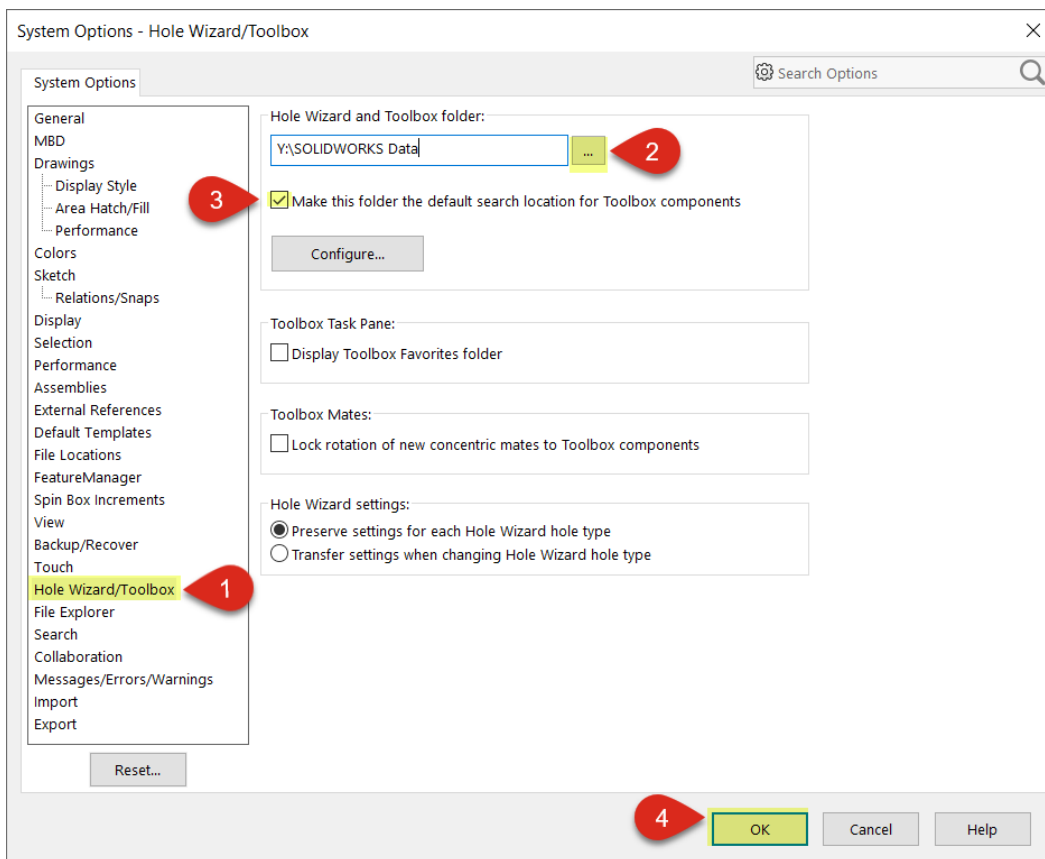
STEP 9: Paste folder into network/shared folder location (example: Train (//Network):
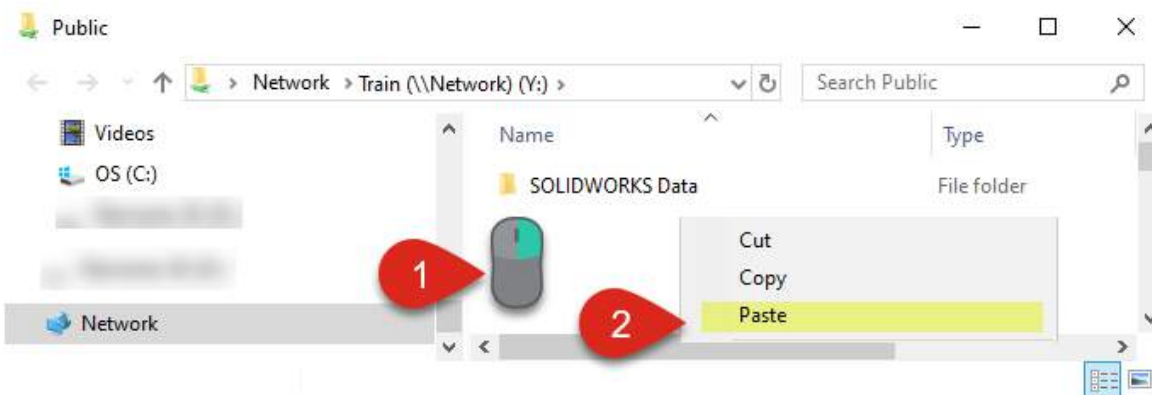
STEP 10: **Open** system options (within SOLIDWORKS) by choosing the icon or going to Tools > **Options**:
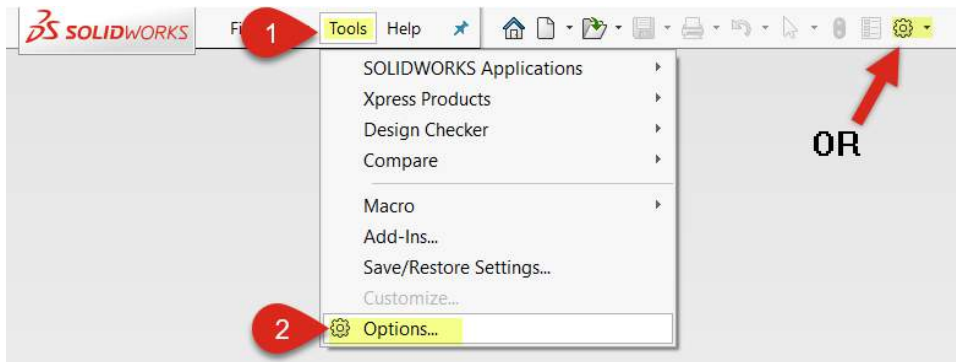
STEP 11: **Select** Hole Wizard/Toolbox and **set** the correct share location where the Toolbox exists. **Verify** "Make this folder the default…" is **checked**:

Standard Setup

Some users will prefer to use the default Hole Wizard and Toolbox Database settings when there is less need for custom part numbers and part descriptions. This guide will illustrate the setup needed for the non-custom environment.

*Note: Settings need to be adjusted on each user system if there are multiple users.

STEP 1: Find and select the **Toolbox Settings** program (Start > All Programs > SOLIDWORKS Tools > **Toolbox Settings**):
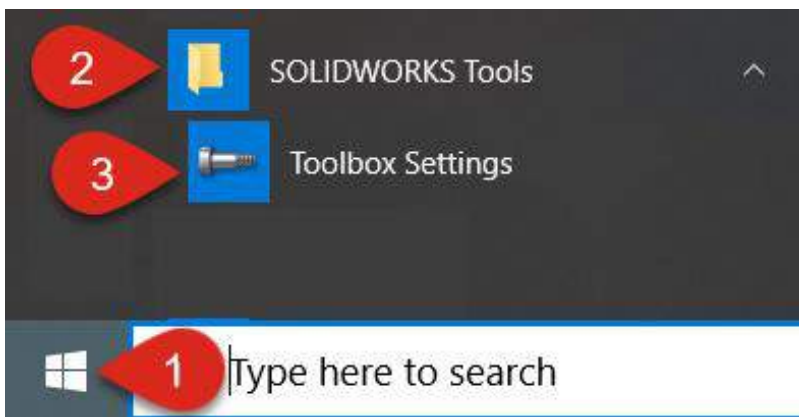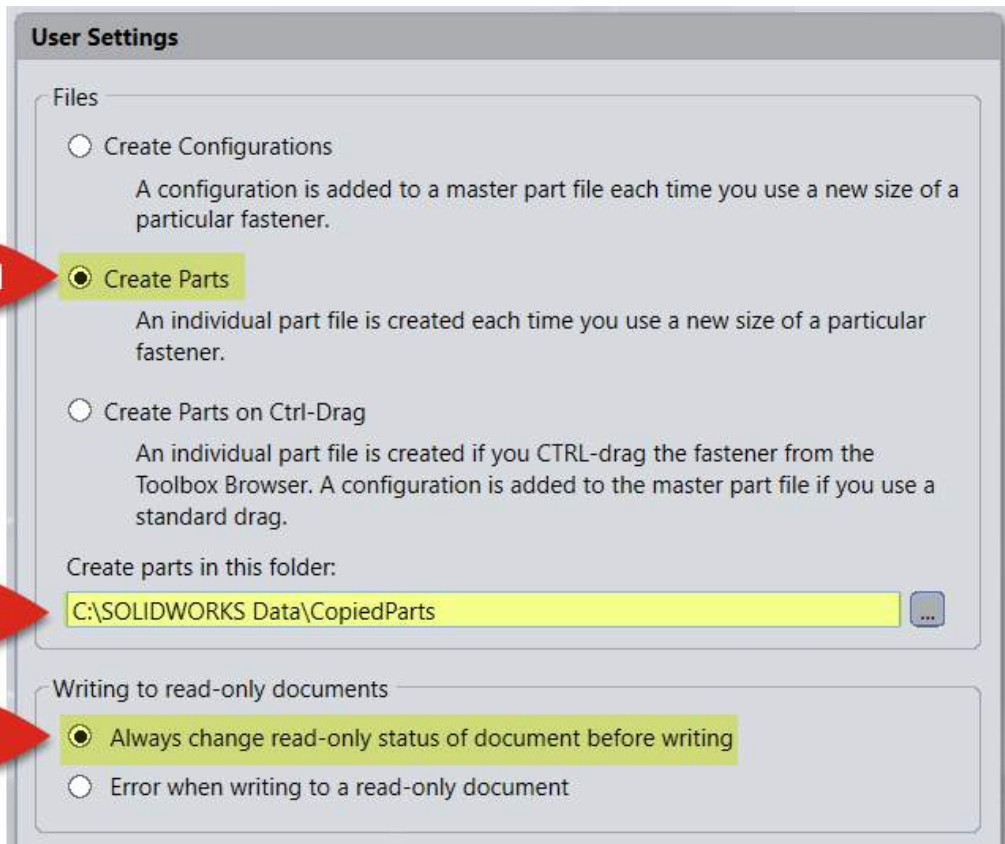

**STEP 2:** From the main menu, select **Define user settings**:
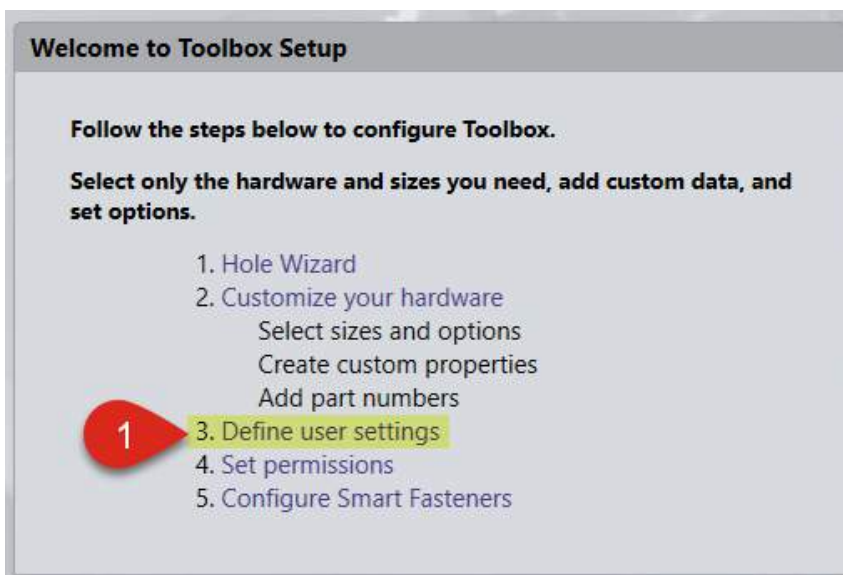
STEP 3: Make sure to select **Create Parts**, and "**Always change read-only…**" and set folder destination to **C:\SOLIDWORKS Data\Copied Parts**:

Note: Creating parts allows for higher performance within SOLIDWORKS by not adding size to the configurations.

STEP 4: **Open** SOLIDWORKS itself to check the folder search setting for Hole Wizard and Toolbox Database. Open **System Options** from the main window or Tools > **Options**:
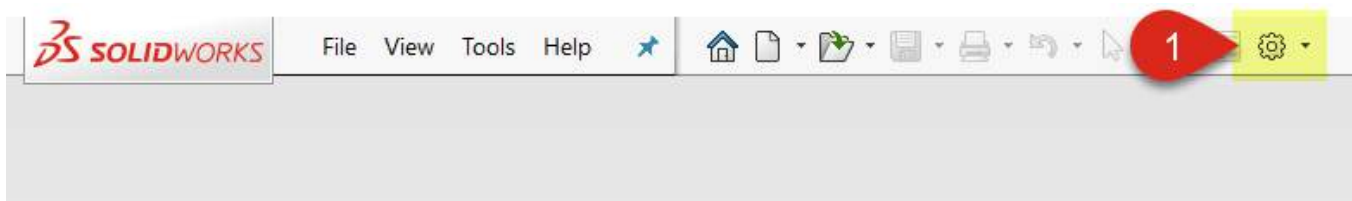
STEP 5: Click **Hole Wizard/Toolbox** link. Make sure the "Make this folder the default search location…" option is **checked** and **exit** out of Options:
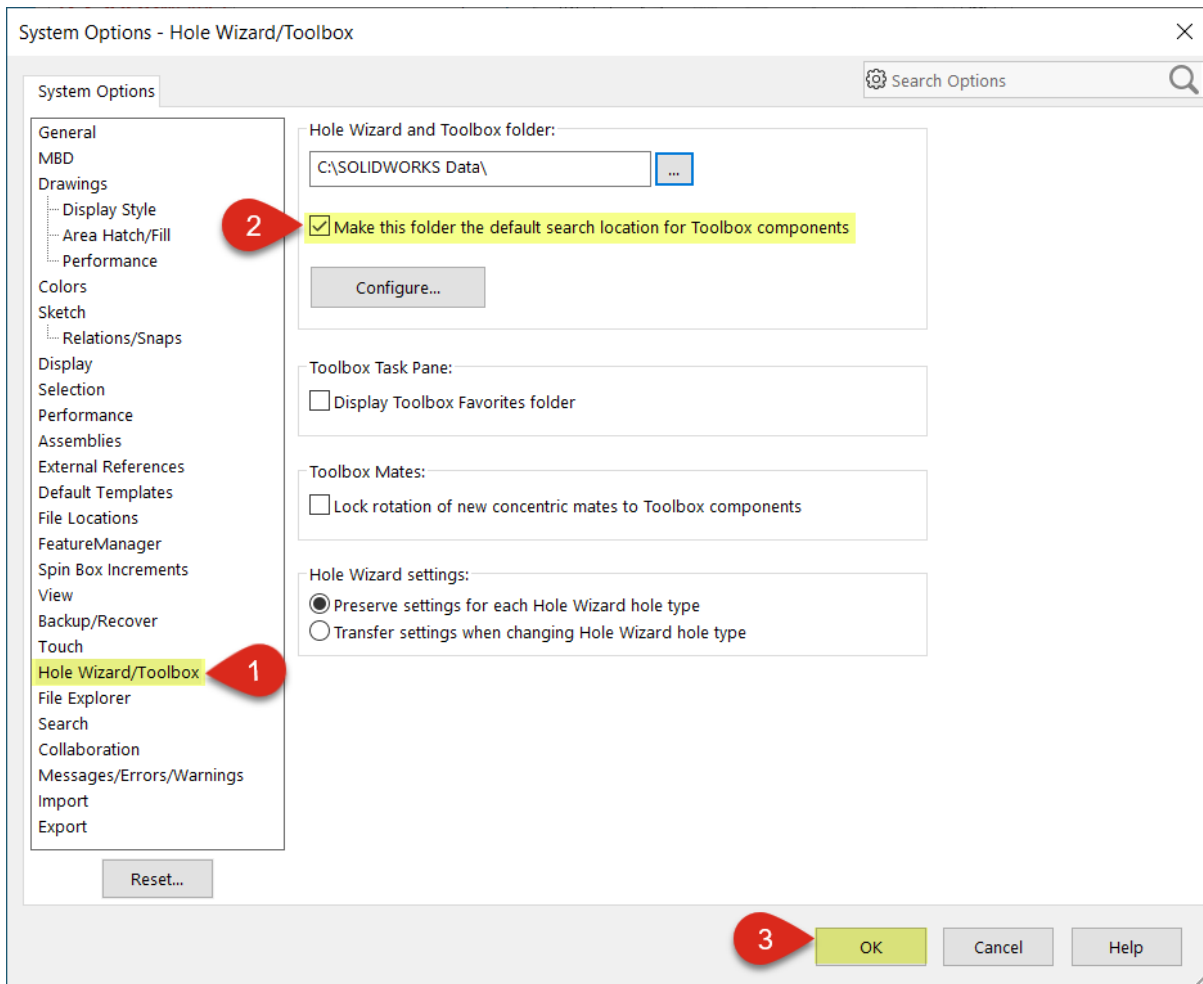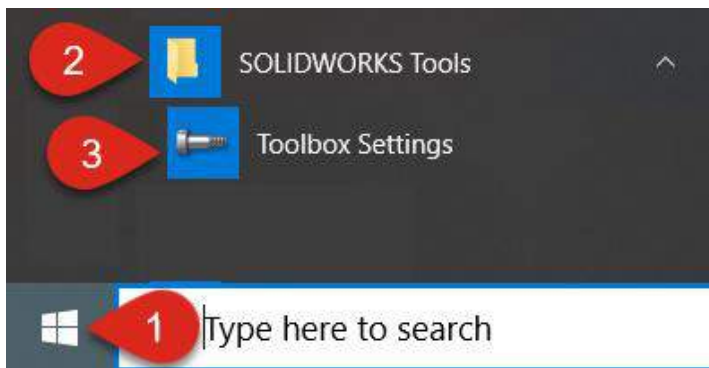
CUSTOMIZATION

ADDING A TOOLBOX PART

This guide shows the steps involved in adding a custom or downloaded part to your toolbox for quick access during modeling and assembly.

From the Hole Wizard and Toolbox Administrator:

*Note: Before starting, make sure to have a fully configured part saved in a temporary location. Once the part is brought into the toolbox, it does not carry a link to that original part for changes, so we will delete that file. Because it is a custom part, it won't have editable properties like normal Toolbox parts.
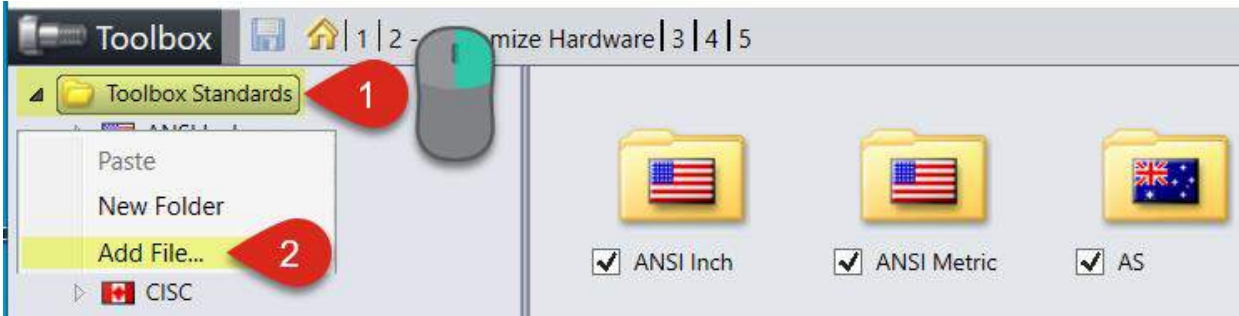
STEP 1: Browse to Start > Programs > SOLIDWORKS Tools > Toolbox Settings:

STEP 2: From main Toolbox Settings menu, select **Customize your hardware** link:
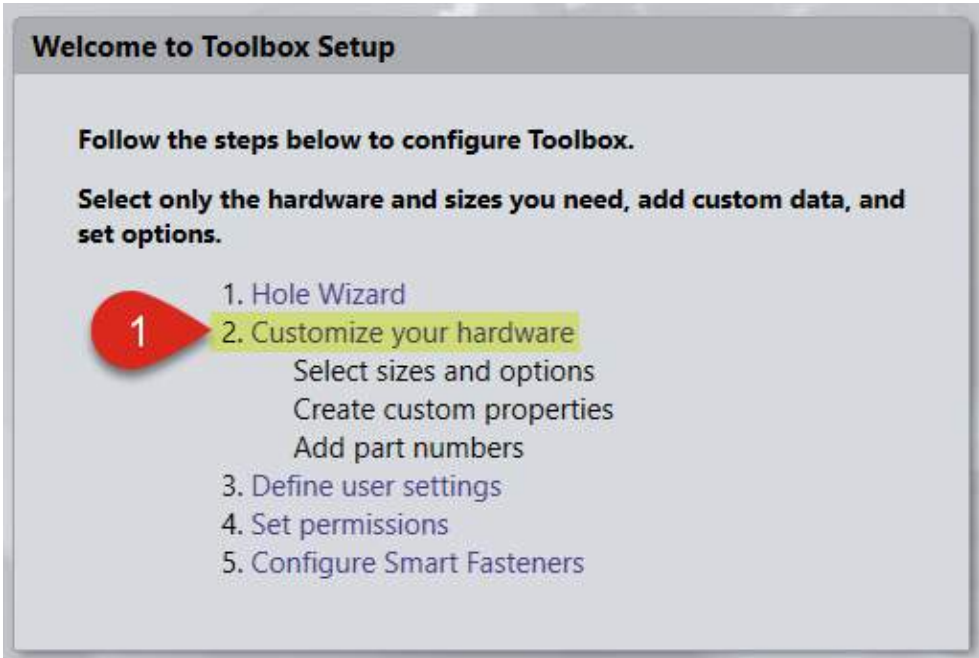
STEP 3: Right-click on **Toolbox Standards** (or preferred folder in tree) and select **Add File:**

- **3a)** **Select** desired part (ex. Toolbox 1 from Desktop location – Figure 1). There should now be an icon in the main thumbnail window of the new part (Figure 2):
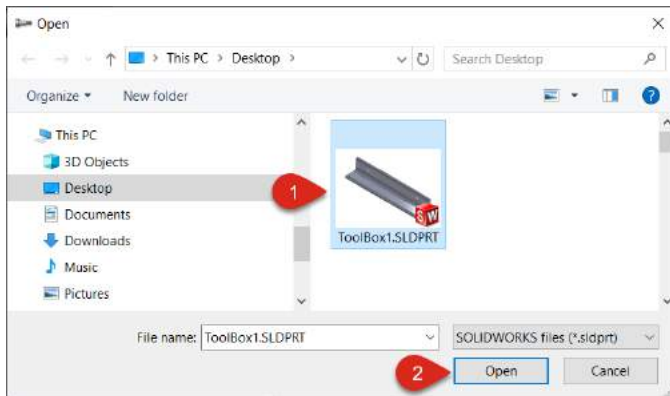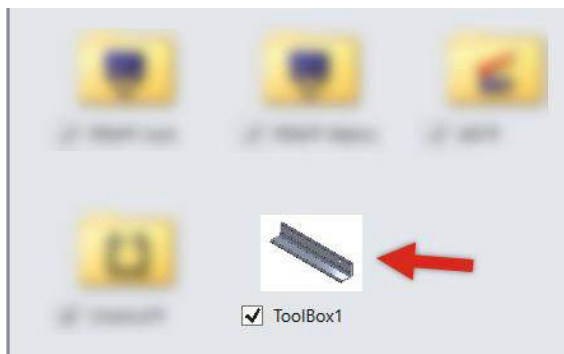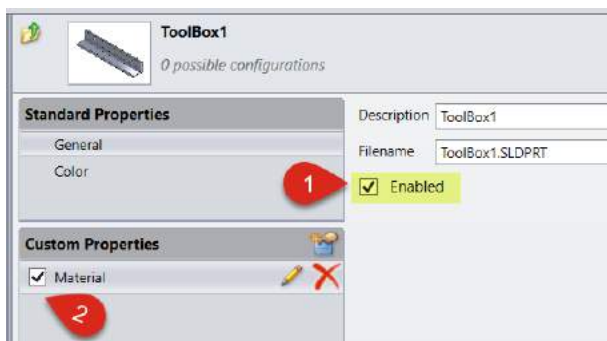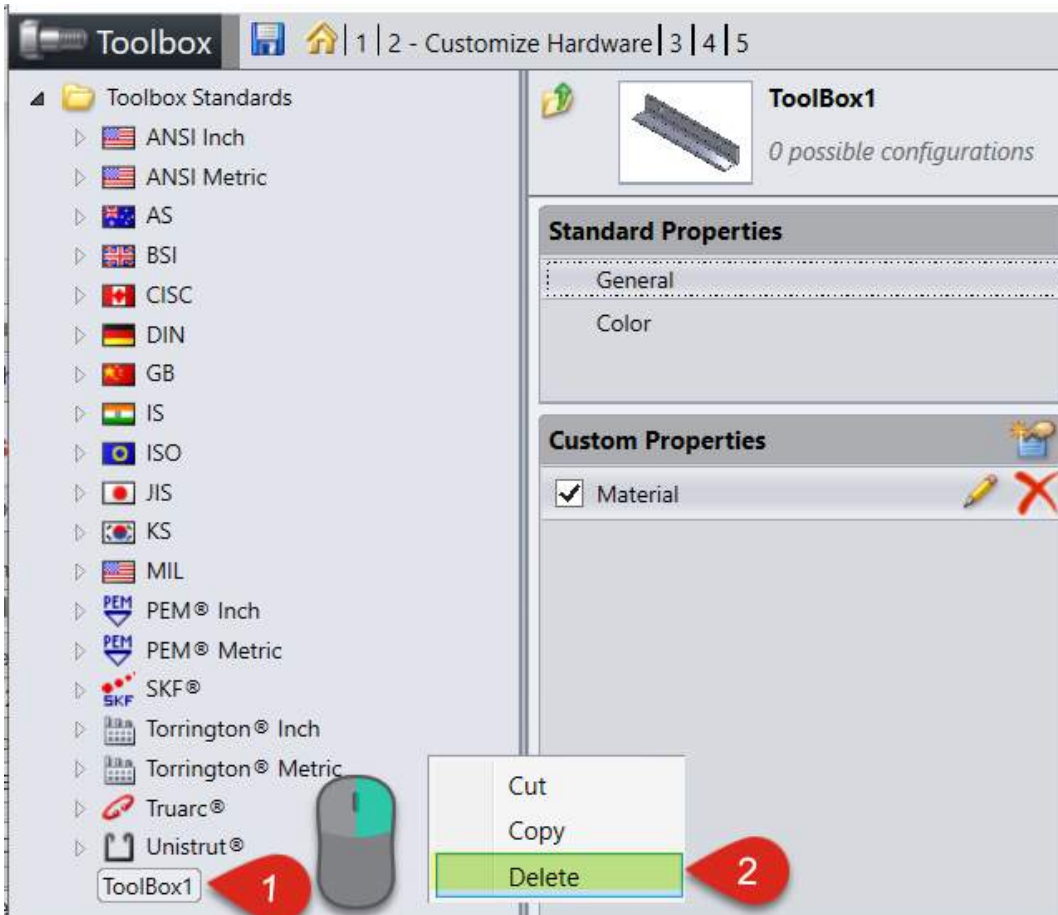

Figure 1

Figure 2

- **3b)** By clicking the thumbnail (Toolbox 1), it can be **enabled/disabled** (shown in SOLIDWORKS) or set with a **custom property** (ex. Material):
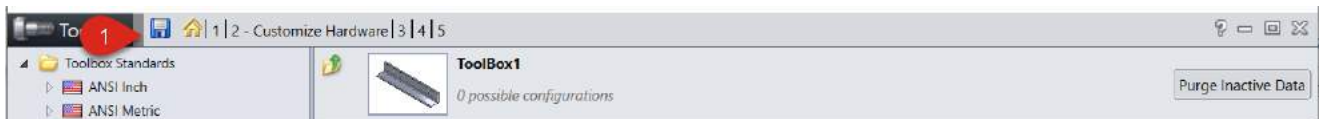 ***Note:** The custom property is user-specified and the user can select which type.
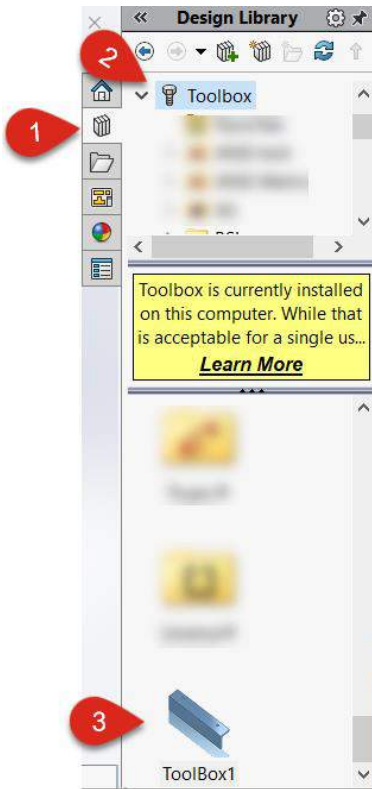
- **3c)** The part can later be deleted from the Toolbox by right-clicking on the part in the Standard tree and selecting **Delete:**
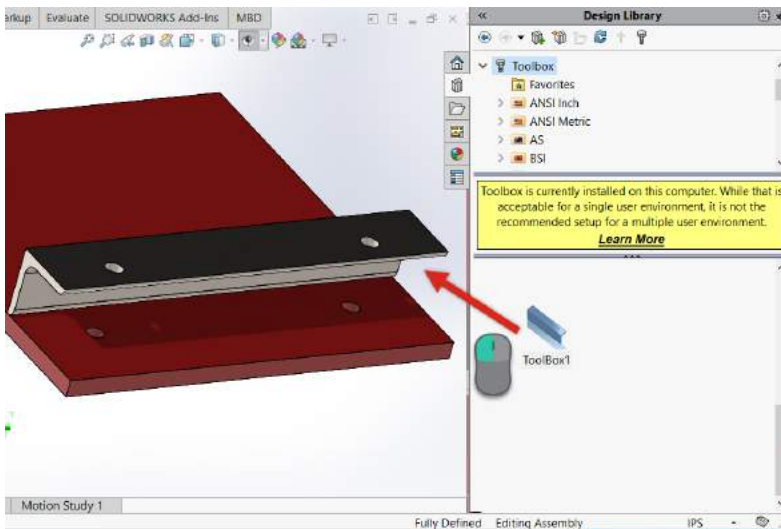

STEP 4: Before we bring the part into SOLIDWORKS, **save** it so it will be recognized at that time:
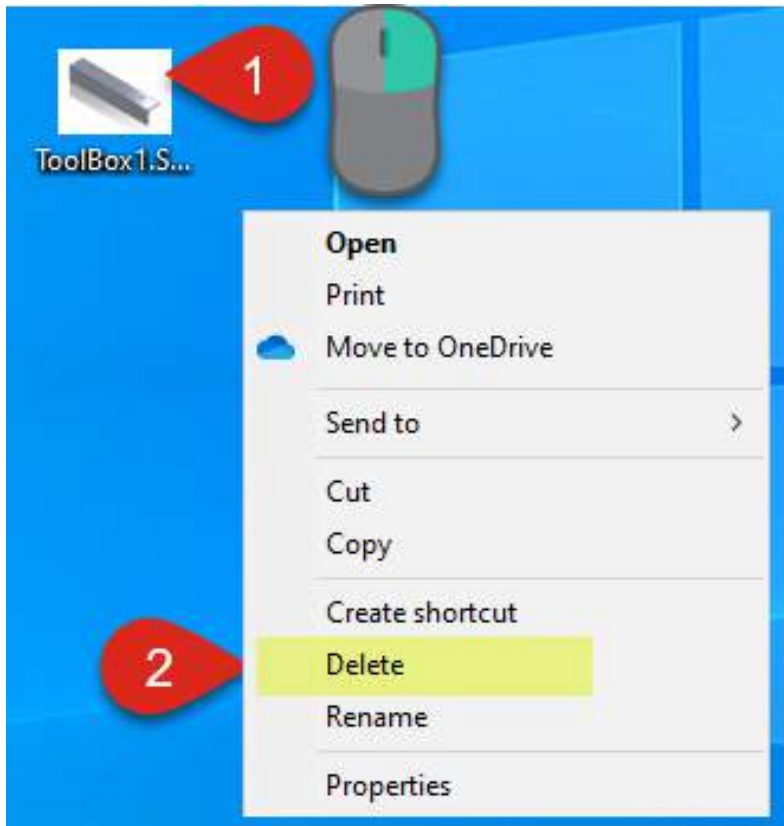
STEP 5: From the Task Pane, select **Design Library and Toolbox**, then scroll to find the part (ex. Toolbox 1):

STEP 6: **Drag and drop** file into an assembly to verify functionality:

STEP 7: **Delete** original toolbox file (or compress/archive, if necessary):

Adding or Removing a Toolbox Flag (Toolbox Part ID)

This guide shows the steps involved in changing the Document Property (Flag) of a Toolbox part so that it is recognized as preferred as either a Toolbox part or a Standard part. For example, this may be done after editing a standard Toolbox part and wanting it recognized as regular.

From Windows Desktop:

## Part 1: Remove Toolbox Flag (No Document Property)

*Note: This is done when you want the toolbox part to become a regular part.

STEP 1: Browse to the **sldsetdocprop** app (Start > Computer > Local Disk > Program Files > SOLIDWORKS Corp > SOLIDWORKS > Toolbox > Data Utilities > **sldsetdocprop**):

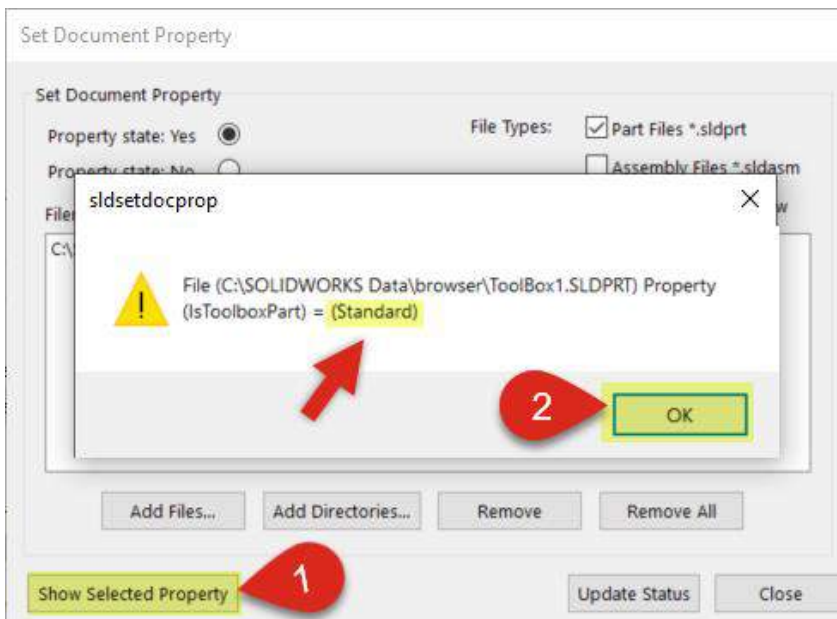***Note:** It is also possible to add folders or directories of multiple parts to change at once.
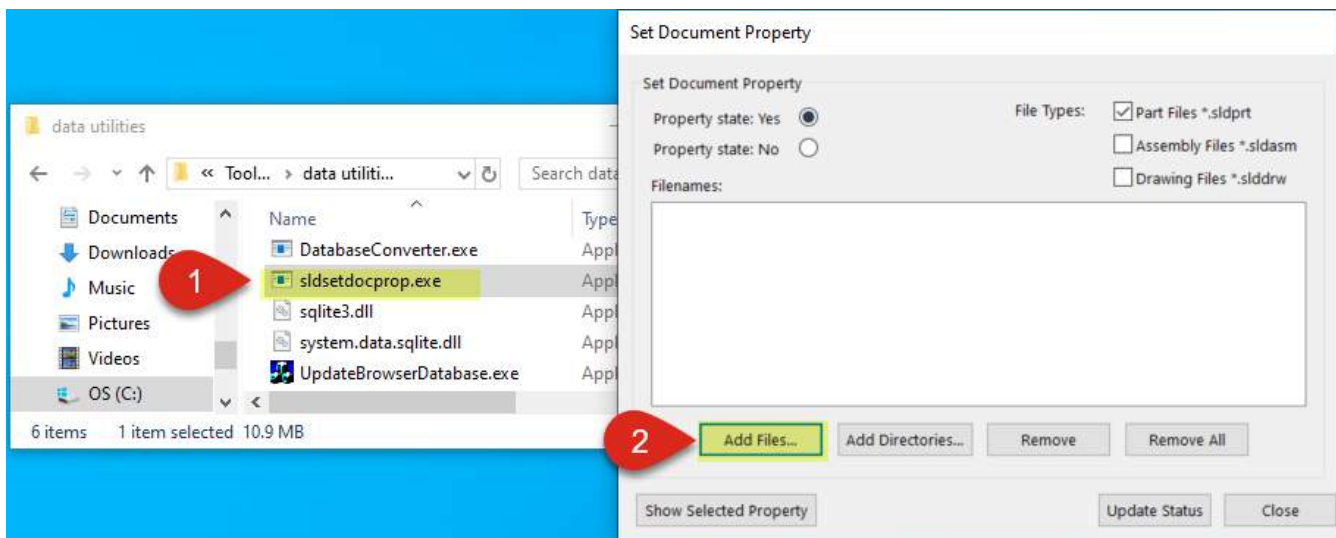
STEP 2: Select and **open** preferred part to be changed (ex. Toolbox 1):
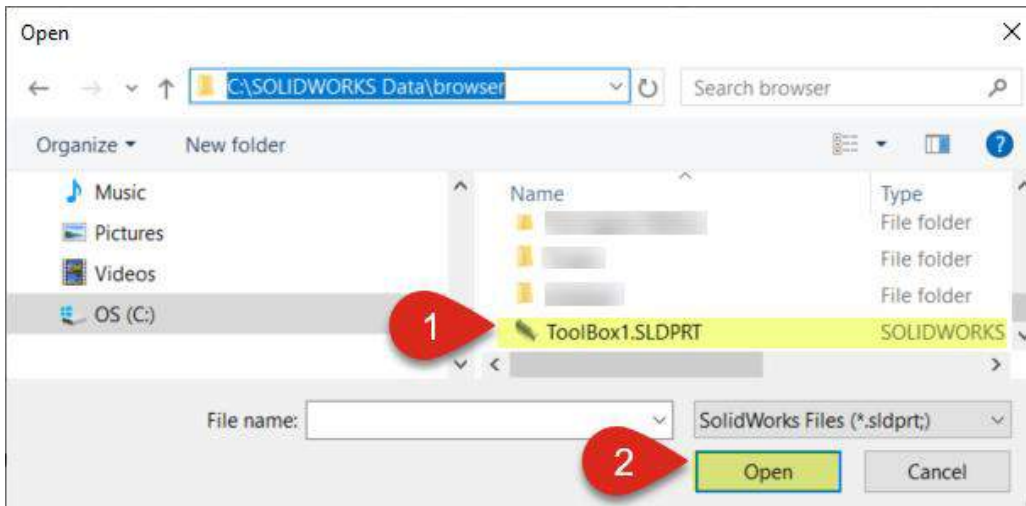
**STEP 3:** To check current status of part, select **Show Selected Property.** This should show **Standard** by default. In order to have the part recognized as a regular part (not flagged) by PDM, we will change this. Select **OK**:

STEP 4: Select **No** in the Property State, then **Update** to remove the flag:
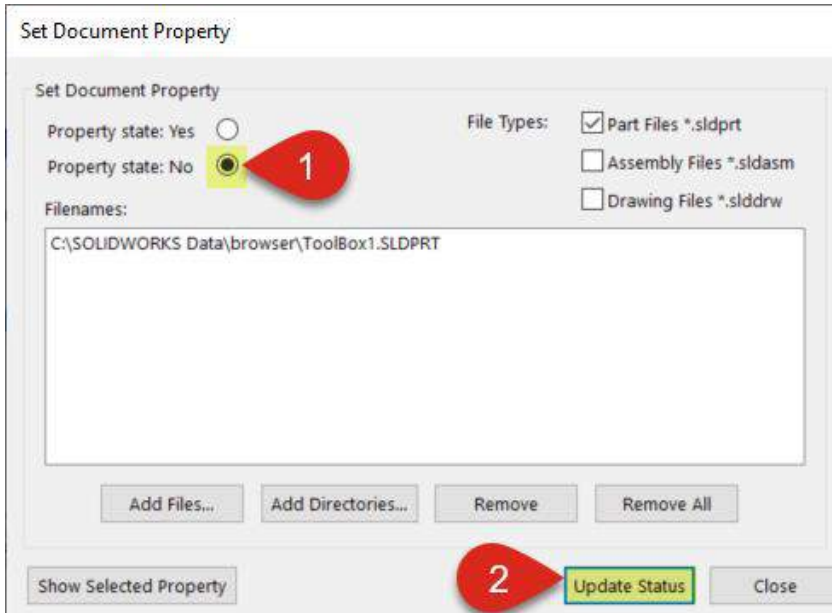
Now it shows **"No"** for the property value when selecting "Show Selected Property." Choose **OK** to close:
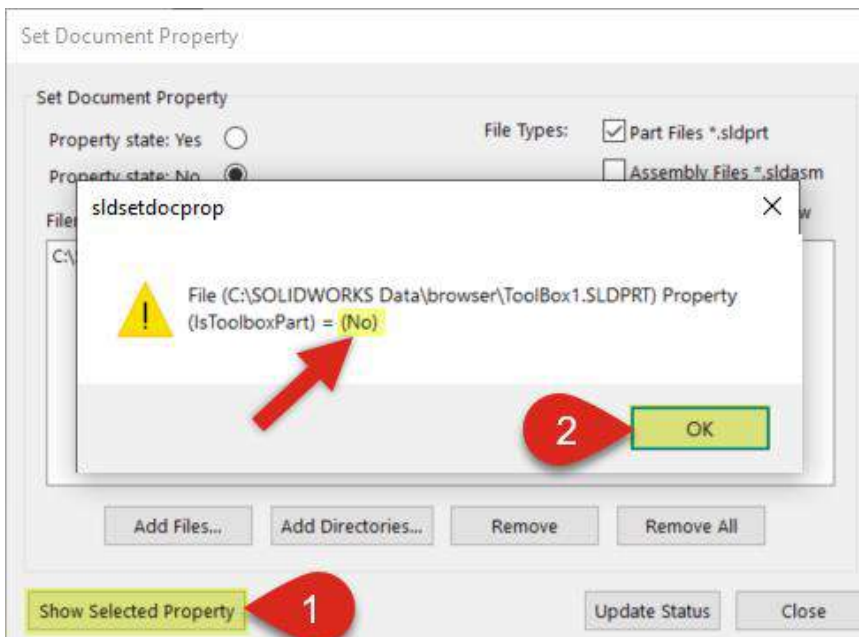
Part 2: Add Toolbox Flag (Yes Document Property)

*Note: This may be done if you want the part to become a toolbox part or treated as such by the PDM.

STEP 1: **Repeat** Part 1: Steps 1 & 2 (above).

STEP 2: First, **verify** the part has a No Document property and is not recognized as a Toolbox party already. Select **Show Selected Property** – it should list "No." Click **OK** to close:
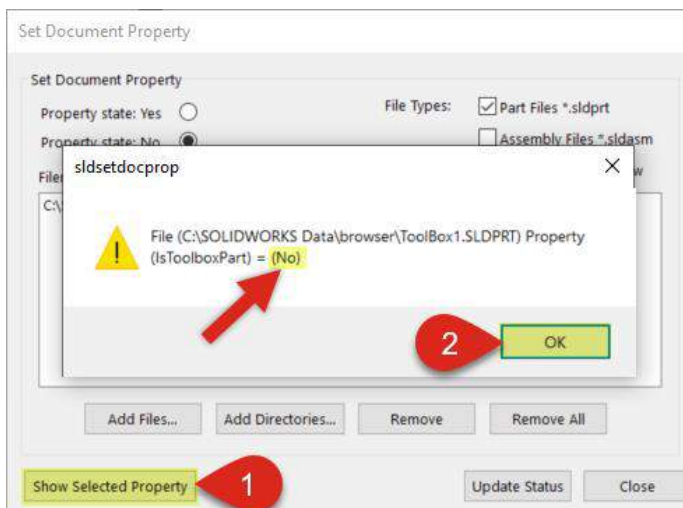
STEP 3: Choose **Yes** in Property State at top, then click **Update Status**. When the file attribute window appears, select **Yes** to confirm:
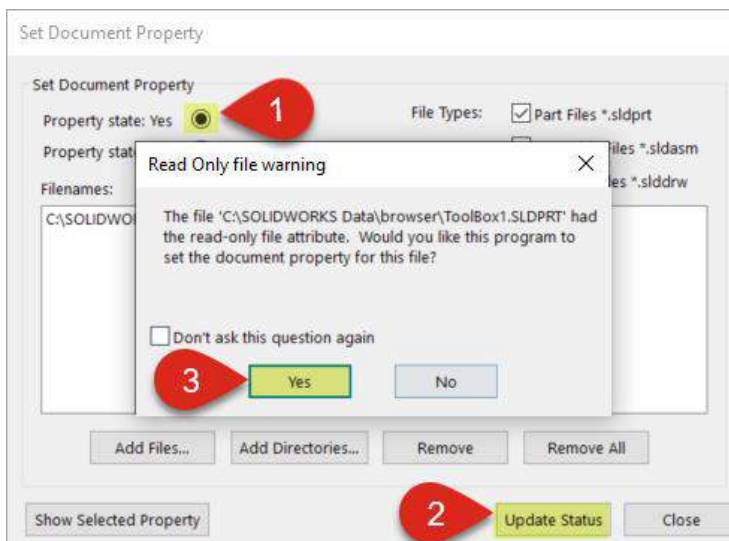
CUSTOM PROPERTY

This guide shows the steps involved in creating a custom hardware property in the Toolbox (ex. Material) with multiple fasteners of the same size, but different material. The same process is used to add other properties (ex. Density or Weight).

From User Desktop:

*Note: Make sure to verify access to SOLIDWORKS Data folder in Windows Permissions before continuation to allow for saving. See File permission section.

**STEP 1:** **Browse** to Start > Programs >SOLIDWORKS Tools > **Toolbox Settings**:
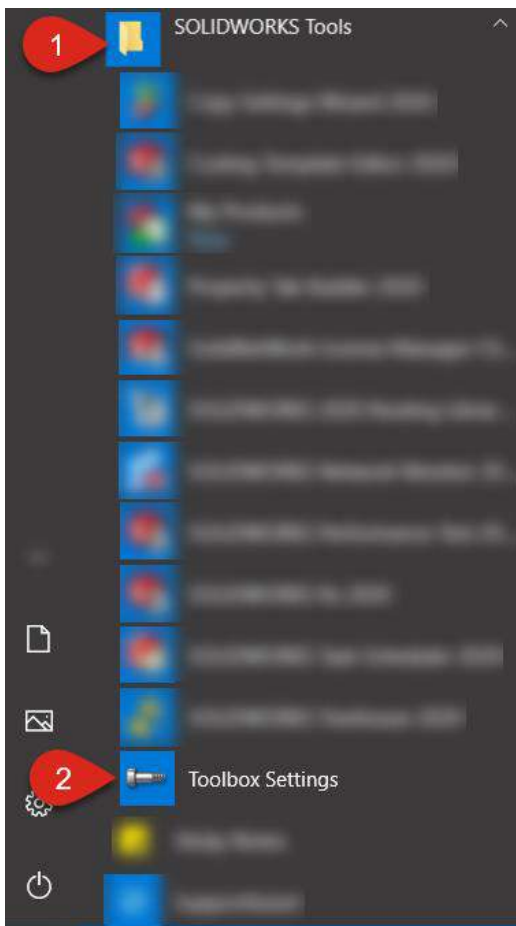
Note: For an Alternate Method of accessing the Toolbox Settings, see Section XX: Creating a Custom Hole Wizard Threaded Hole.

**STEP 2:** From main Toolbox Settings menu, select **Customize your hardware** link:

STEP 3: To open the custom property dialog, click the **new-page icon**. As an example, we type "Material" as our new property:
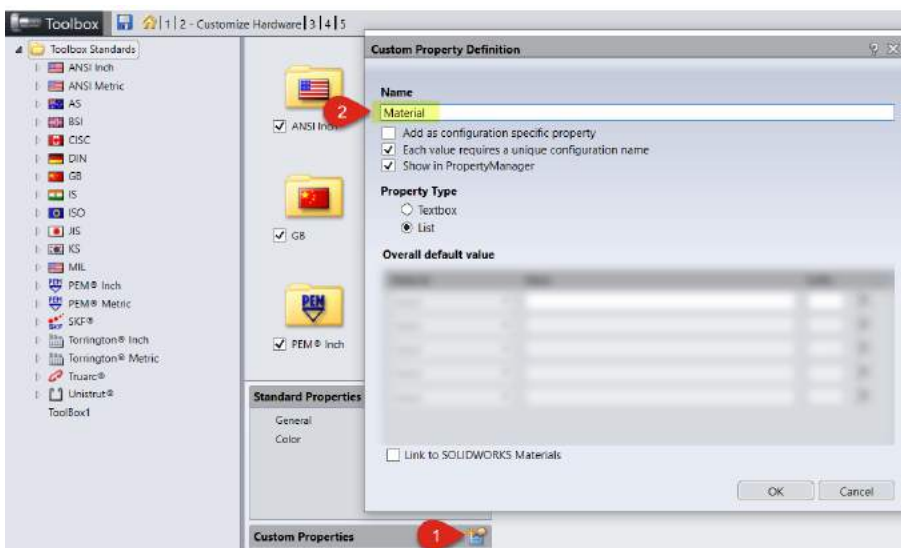
STEP 4: Verify that these **three options** are selected and "Add as configuration…" is **unchecked**:

- "Each value for this property…" –Ensures that a new part file is created for each new property.

- "Show in PropertyManager…" –This enables a label to appear in SOLIDWORKS.

- "List." –This will allow us to add our material and create a link for it.

*Note: By unchecking "Add as configuration…," we enable a new property to be created separately from any particular configuration.
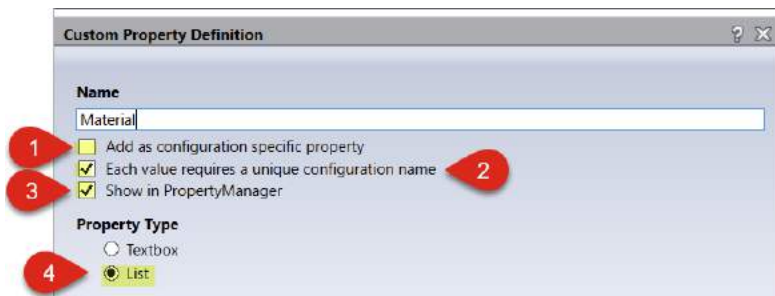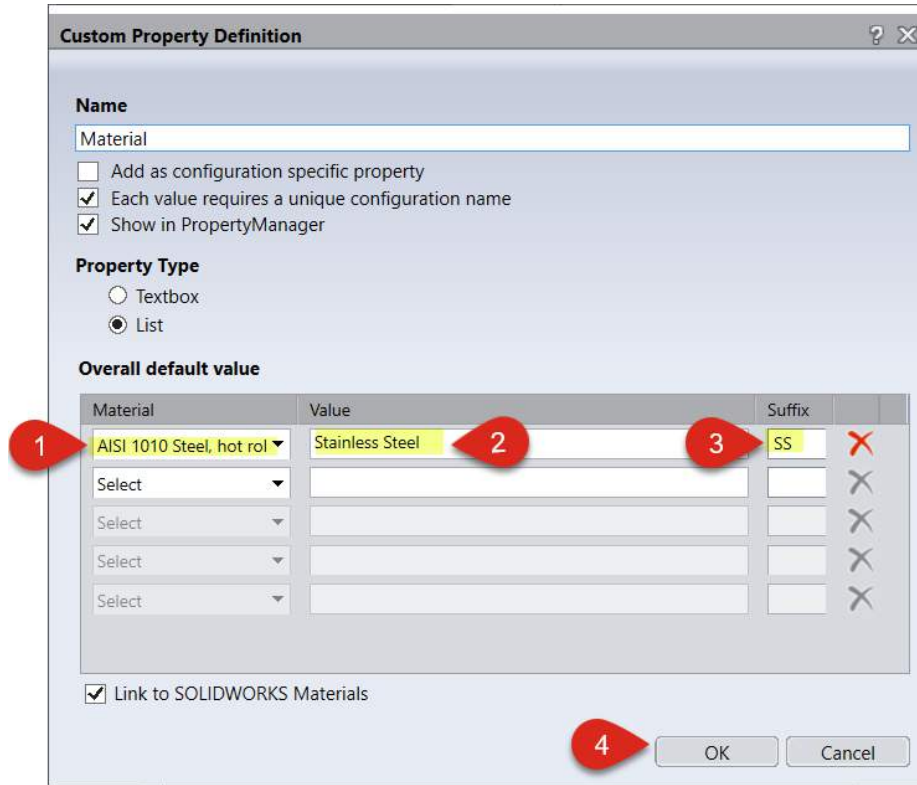

STEP 5: Check **Link to SW material** and choose the **Select** button that appears:
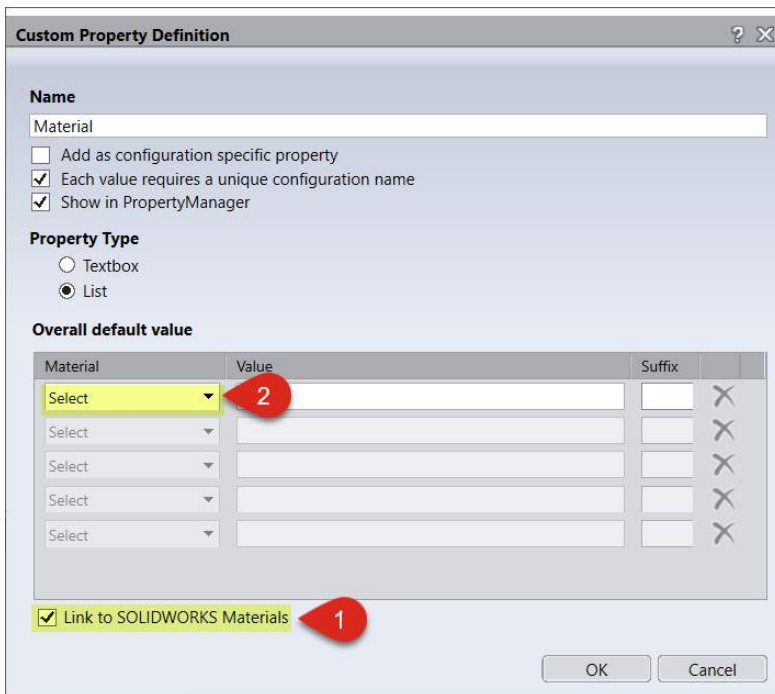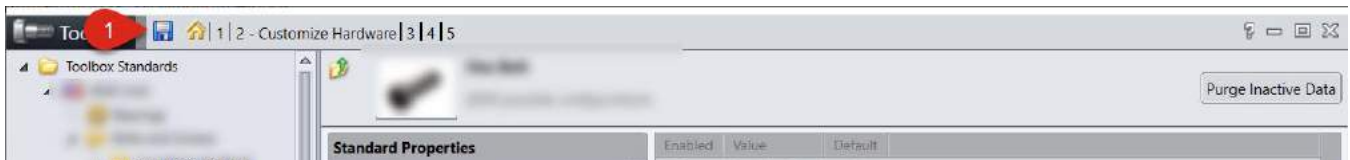
STEP 6: Browse through menu to select **AISI 1010 Steel** (SW Materials > Steel > **AISI 1010**), then enter in the desired name and suffix:

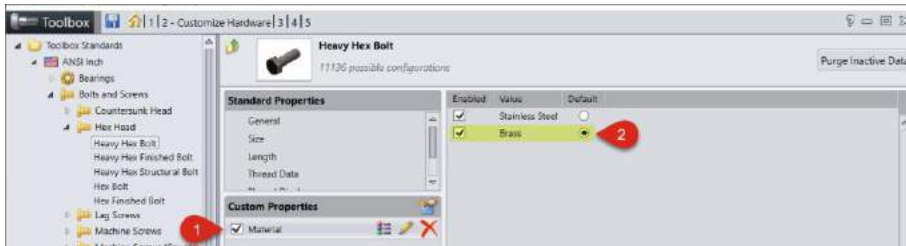*Note: Use **Tab** to move from Value to Suffix boxes. If materials are shifting around, widen your window.

STEP 7: At this point, it is suggested to **save** current progress in settings in case of power loss, etc.:
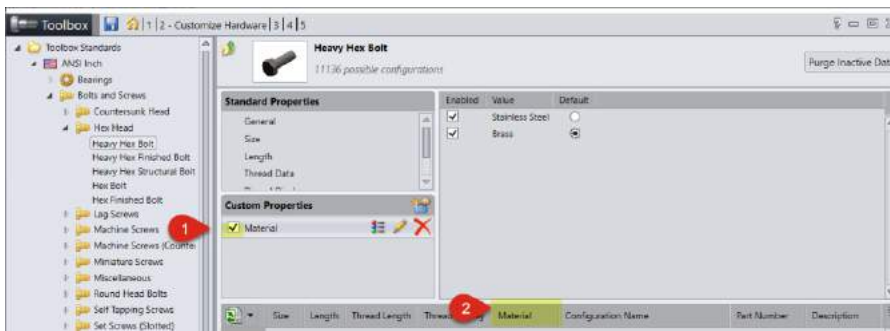
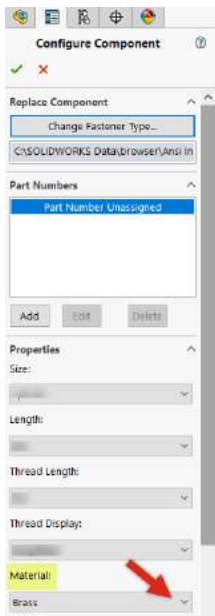**STEP 8:** **Select** the desired default material for usual use (ex. Brass):

*Note the changes to the windows below:

Material Column Added in Toolbox Settings

Material Option in SOLIDWORKS

STEP 9: Select the **Save** icon to secure your changes upon exit:

Adding Part Descriptions

This guide shows how to create custom part numbers and add descriptions to parts that are used by the Toolbox. The user can choose to do this individually or many at once, when a more descriptive nature is needed.

STEP 1: In the following steps, create a folder to save to (if it doesn't exist). From SOLIDWORKS Data (Start > Programs > Local Disk > **SOLIDWORKS Data**), right-click to **create a new folder** and name it "Copied Parts," or similar:
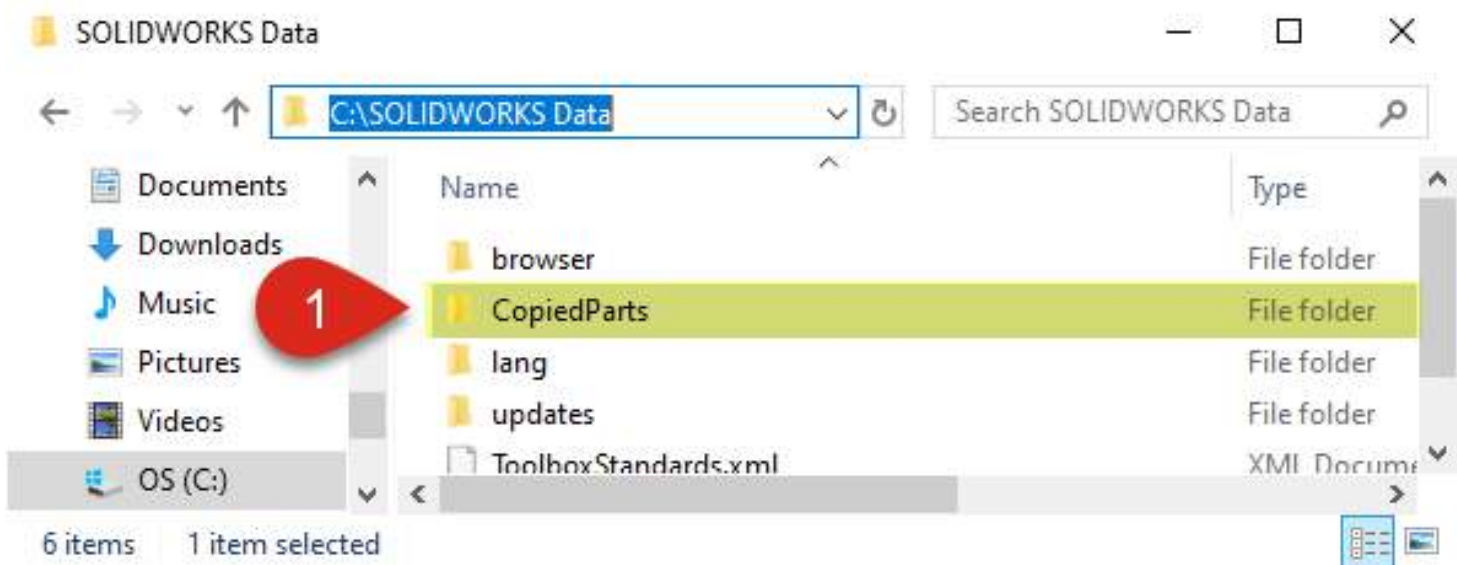
STEP 2: Click on **Define user settings**:

STEP 3: Click **Create Parts**:
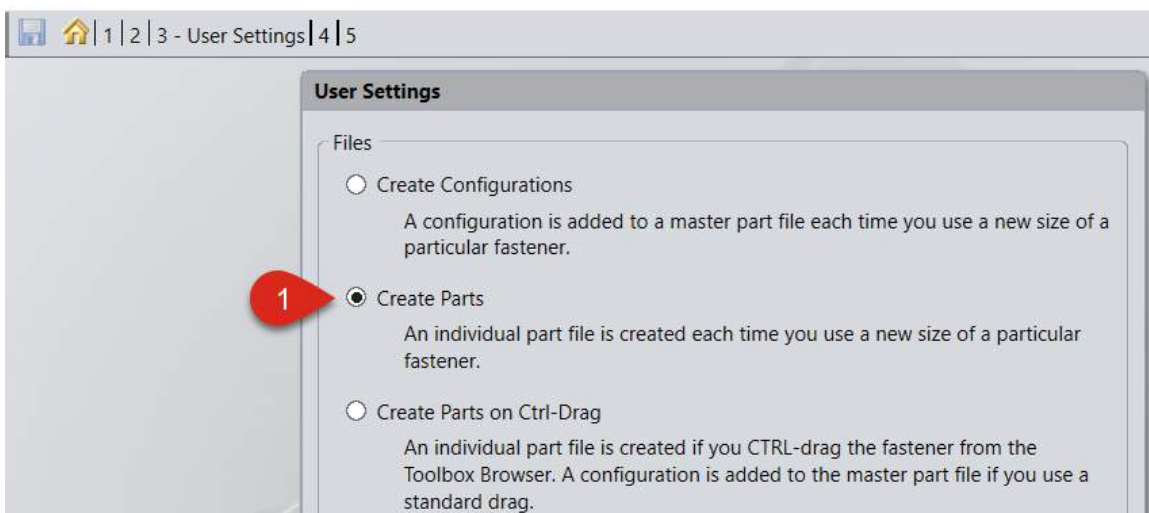
*Note:** In order to keep Toolbox part files from getting too large over time and slowing performance, make sure the **Create New Parts** option is checked.

STEP 4: Browse for a folder by clicking button to the right of text field:

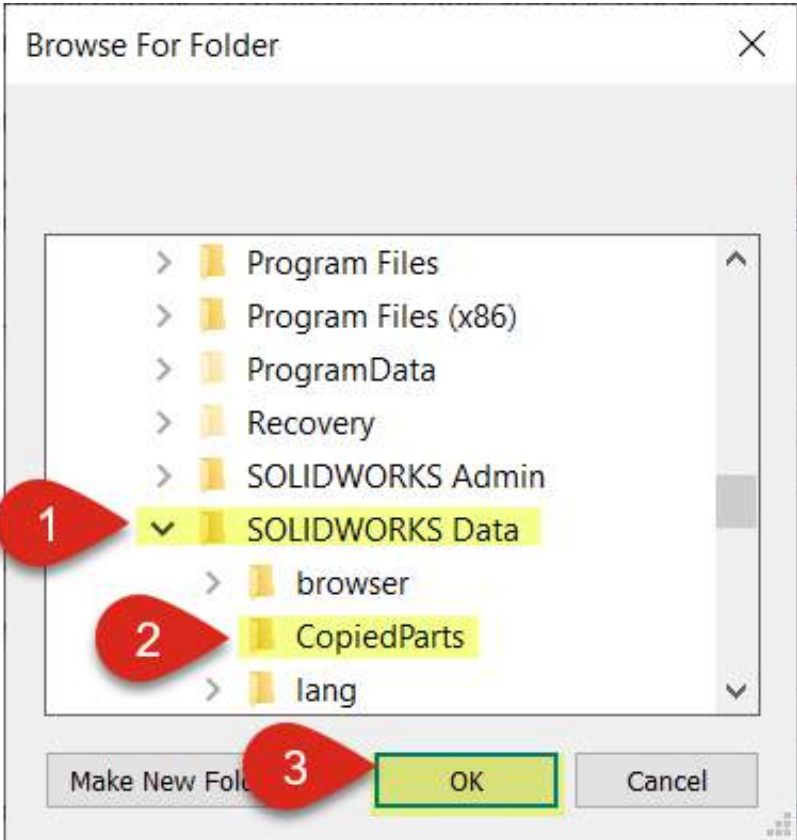**STEP 5:** Select **CopiedParts** and click **OK** (Computer > Local Disk > SOLIDWORKS Data > **CopiedParts**):

Part 1: Individual Change

STEP 1: From Main Graphics area with an assembly open in SOLIDWORKS, **browse** to the right-side task pane and click **Design Library** tab. Choose **Toolbox and folder** (ex. ANSI Inch > Bolts and Screws > Hex Head >Heavy **Hex Bolt**). Then **drag and drop** into graphics area.
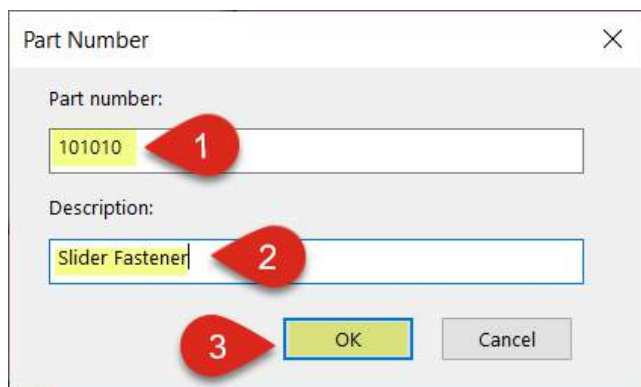
Note: If Toolbox hasn't been used in this manner before, click Add-in now in task pane, if prompted.
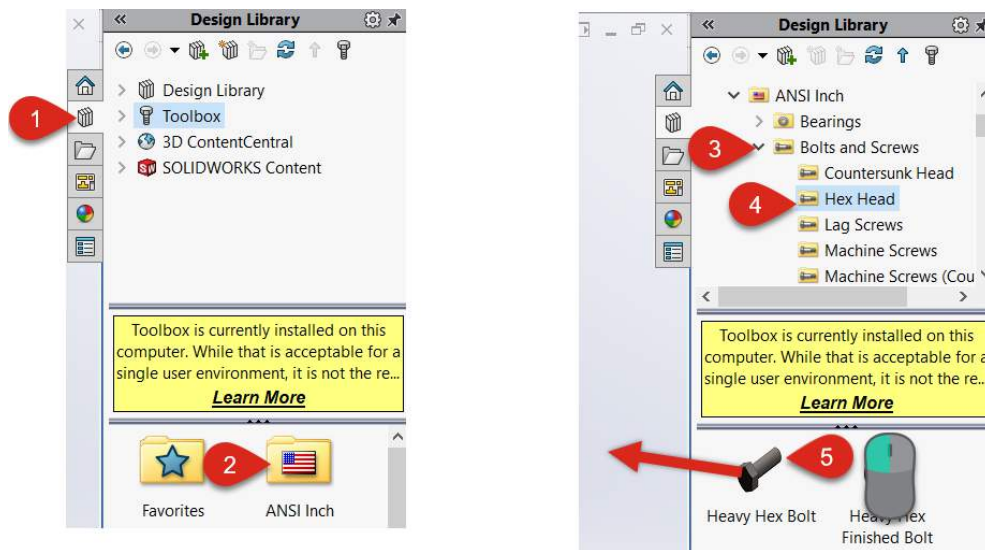
**STEP 2:** The Configure Component Dialogue box opens. From here, select **Add** to **edit** the part number and description of this specific bolt:

STEP 3: Enter in corresponding **part number and description**, then click **OK** to save:
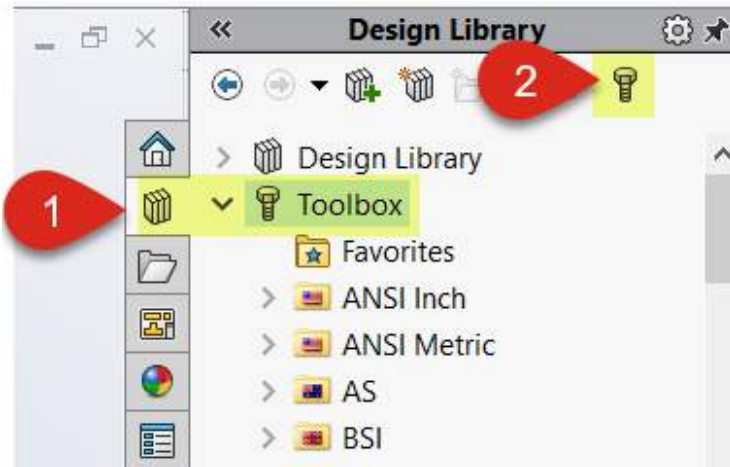

STEP 4: Click on the green check mark to **save** this info. Now the bolt will have this specific information available in drawings and it is saved with these properties:
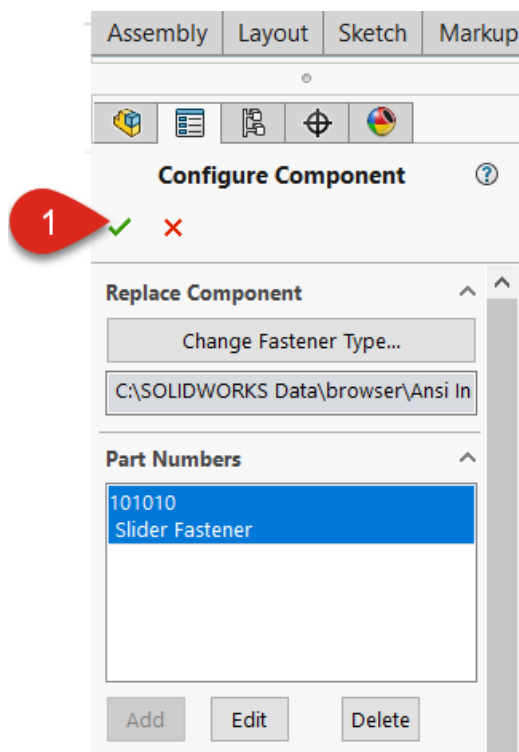
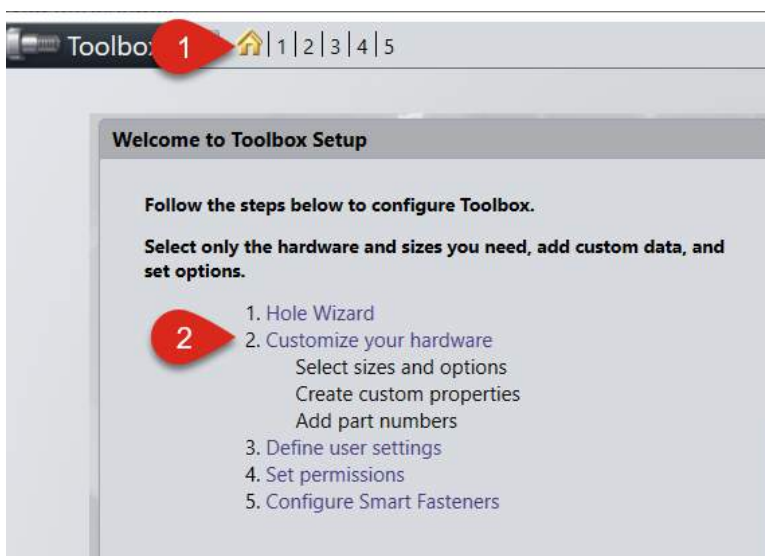## Part 2: Multiple Changes

STEP 1: With SOLIDWORKS open, select the **Design Library tab** from the right hand task pane and click **Configure…:**
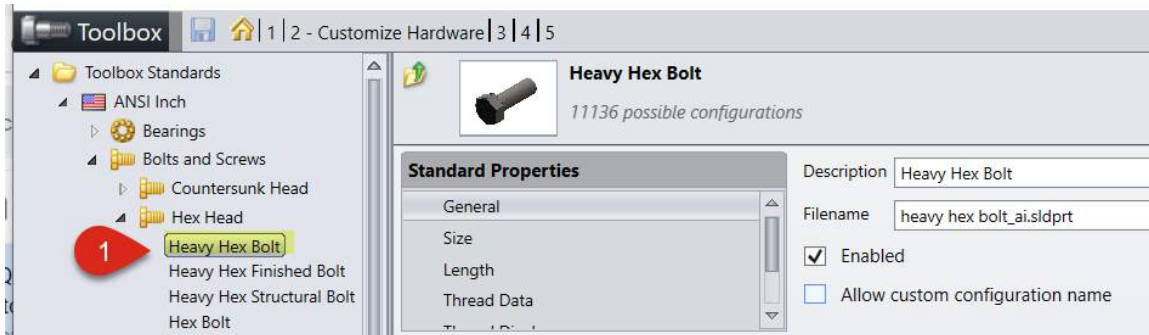

STEP 2: When the Toolbox Settings main page opens, select **Customize your hardware**:

*Note: When clicking **Configure**, it may jump straight to a certain part's settings. Click the **home icon** at the top of the page to get to this page.

STEP 3: From the left-side folder tree, **navigate** to desired part for editing (ex. Heavy Hex Bolt):

STEP 4: Enter in any **corresponding Part Numbers** and **Descriptions** below. These will then appear in SOLIDWORKS drawings and part properties:
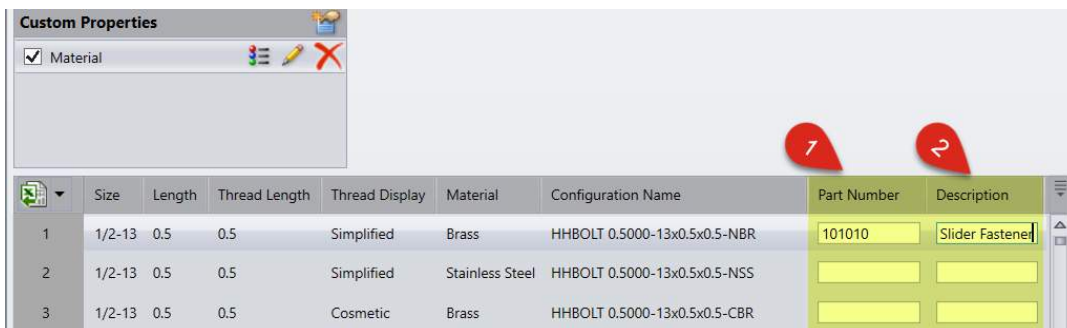
*Note: An alternative to entering these within Toolbox Settings is to use Excel.

- **4a)** Select **drop-down menu** on far left of title bar and click **Export data**:

- **4b) Save** Excel sheet to a desired location:
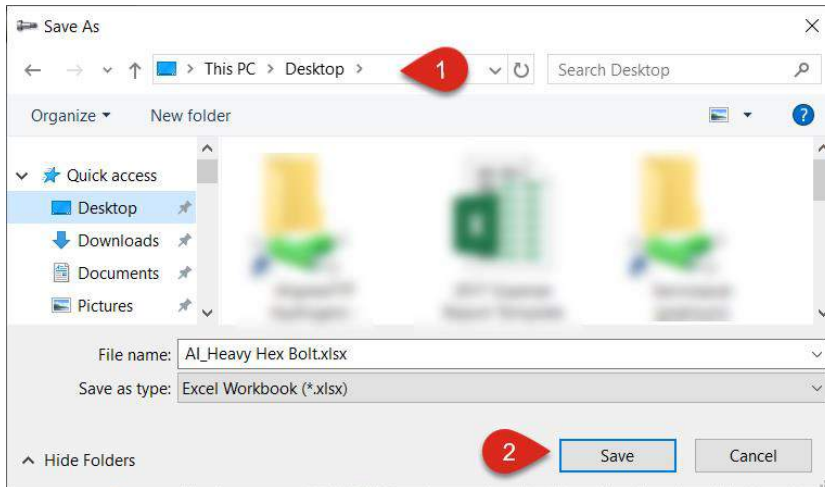
- **4c) Open** the sheet and do any needed **editing**: Note: Values are able to be entered under existing columns in the Excel interface but a new column added will not import.

- **4d) Save** the sheet and **import** back into the Toolbox:
  *Note: Administrator must be logged in to access the Import data button.
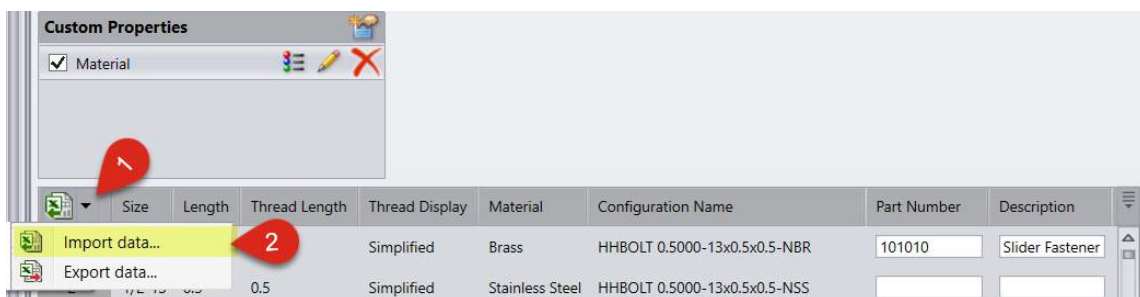
- **4e)** Select the **Save** icon to secure your changes upon exit:
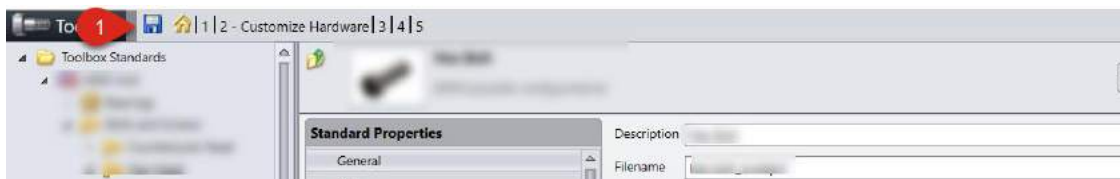
STANDARDS EDITING

Creating a Custom Hole Wizard Standard

STEP 1: From main Toolbox Settings menu, select the **Hole Wizard** link:

STEP 2: **Select** ANSI Inch (for example) and click on the **folder "copy" icon**, then type the **name** and select the **checkmark**. The new copied standard will appear at the bottom of the tree:
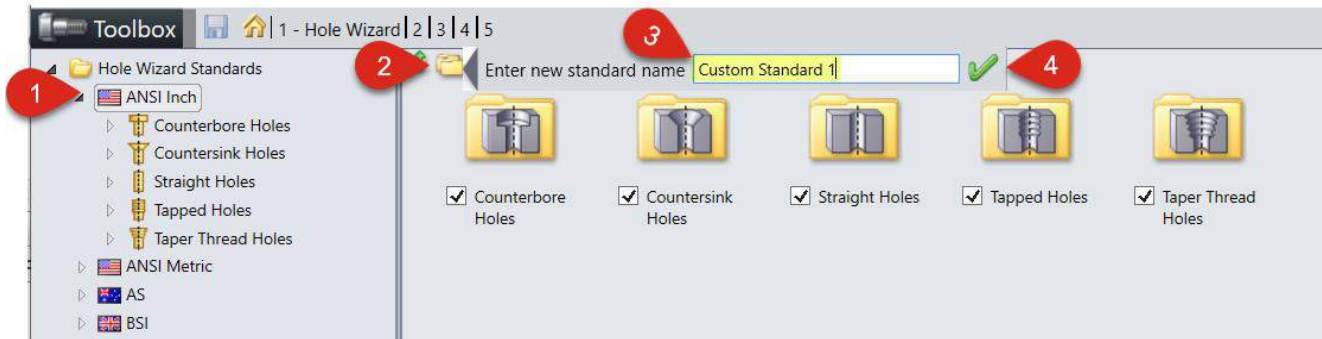
*Note: From here, customization of the new copied standard is now possible.

STEP 3: Select the **Save** icon to secure your changes upon exit:

Deleting a Custom Standard

STEP 1: Browse down the Standard tree and select your custom Standard (ex. CustomStandard1):

STEP 2: Click on the **X-marked folder icon** and (unless otherwise preferred) **check** the "Delete associated part files…"Hit **OK:**
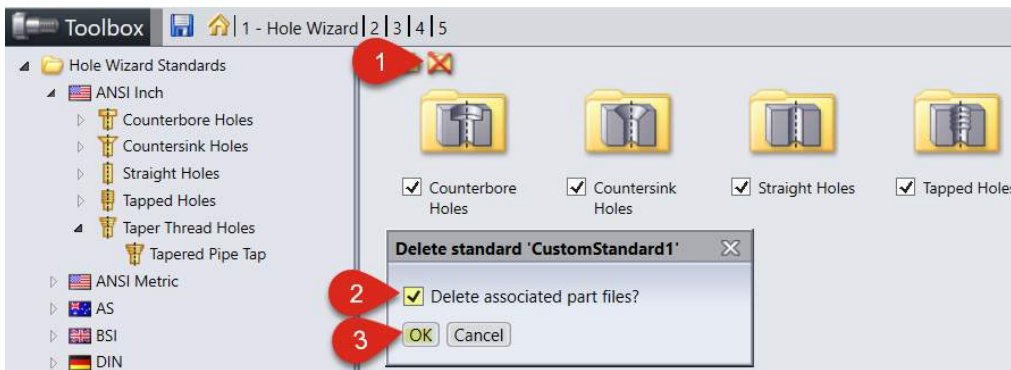
STEP 3: Select the **Save** icon to secure your changes upon exit:
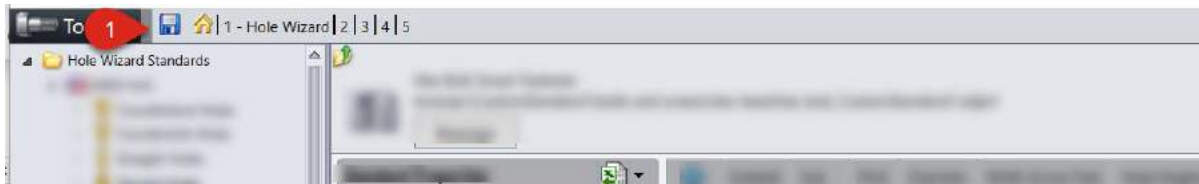
Creating a Custom Hole Wizard Threaded Hole

This instruction shows how to add a Custom Hole Wizard Threaded Hole to the Database. When complete, we will have a Custom Hole Size which works identical to the pre-made entries in the Database.

Step 1: Open SolidWorks and Go to Tools->Options-(or gear icon)>System Options->Hole Wizard/Toolbox. In the Hole Wizard/Toolbox Window select "Configure"
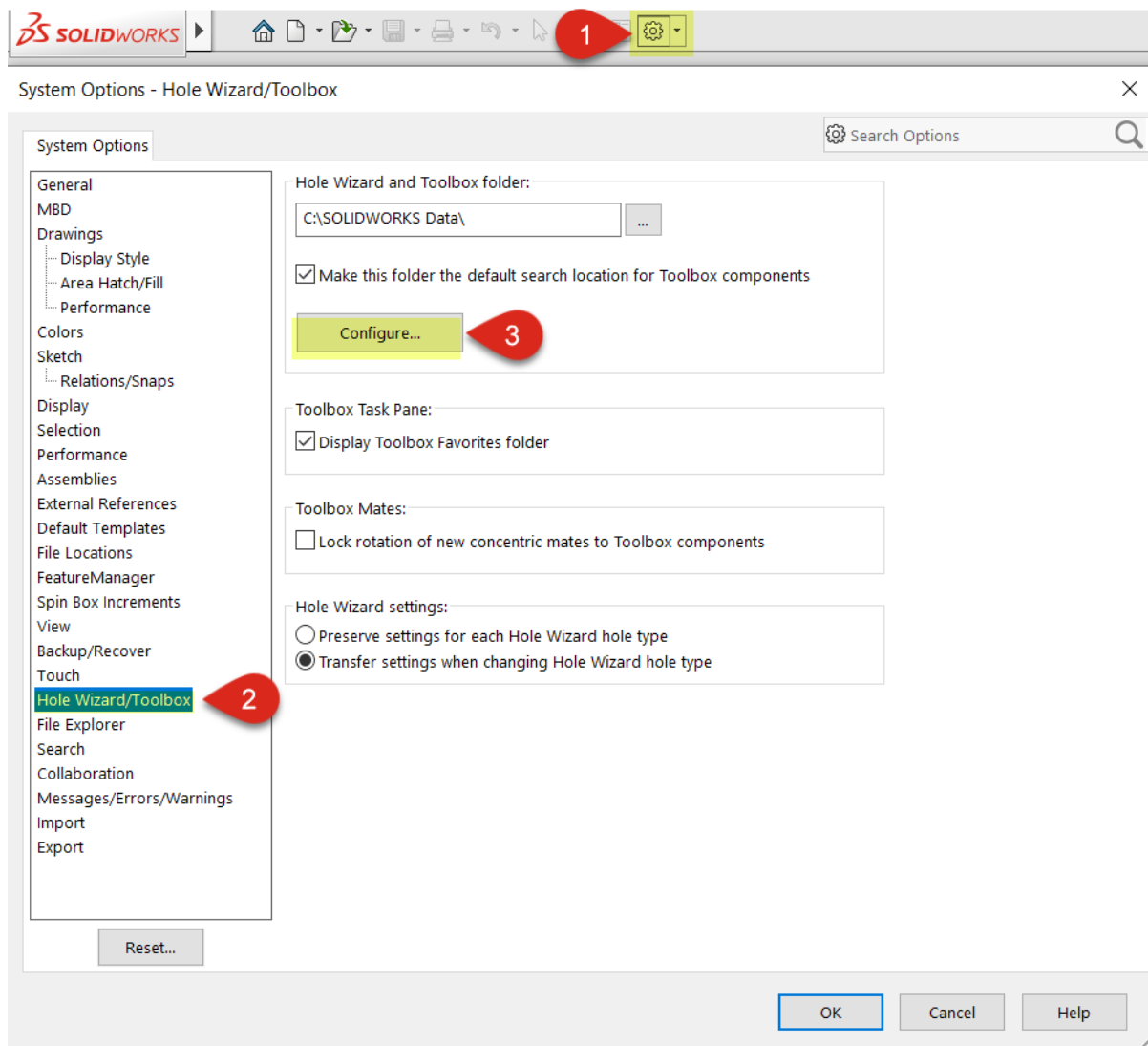
Step 2: In the Toolbox Settings Tool select the Hole Wizard link.
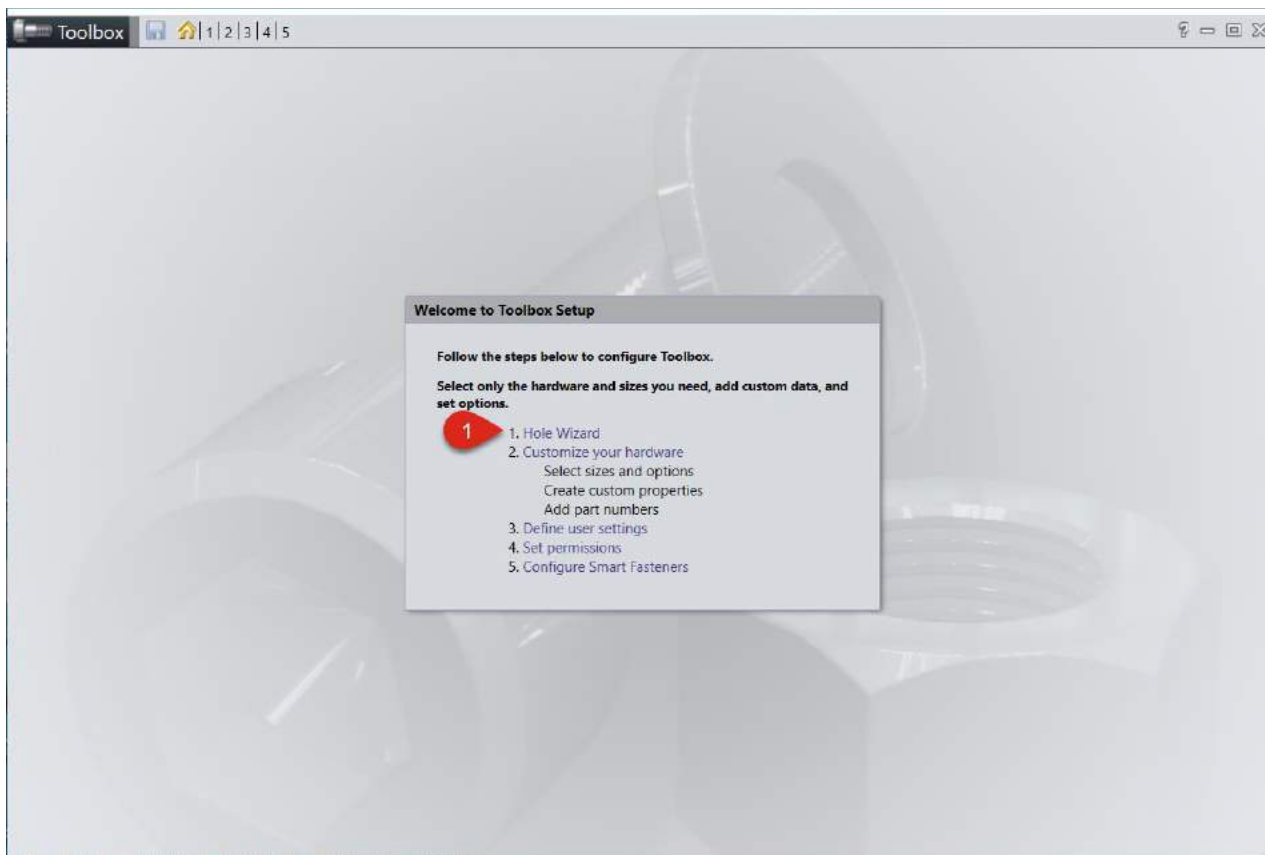
Step 3: Browse to the Standard and Hole Type to be added by using the arrows to expand each heading. Example: ANSI Inch, Tapped Holes, Tapped Hole.
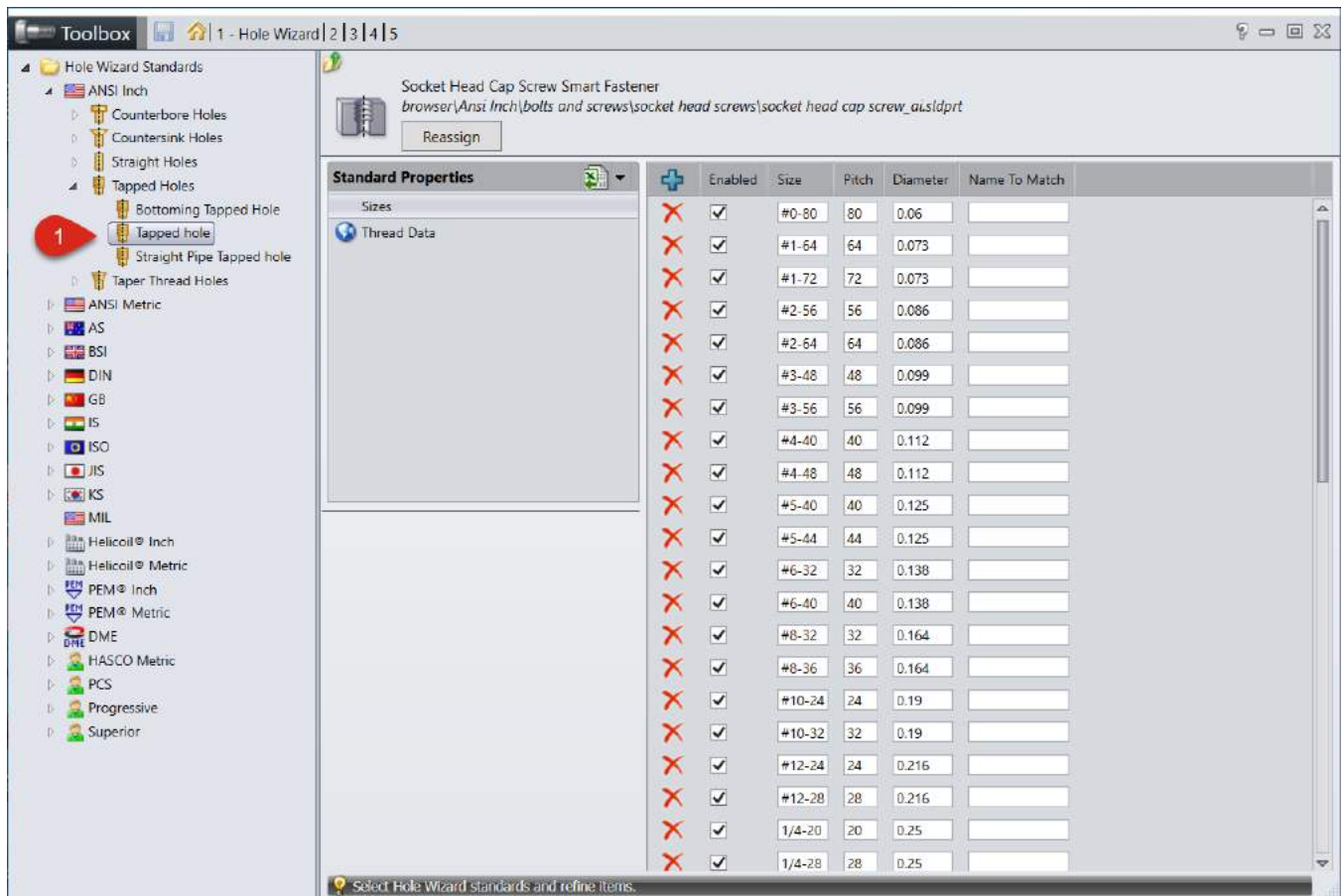
Step 4: Hit the "+" Icon to Add a Size and enter the Size Information. Make note of the name used in the **Size** Property here as it is needed for the Threaded Data in the next Step.

> Note: The **Name to Match** Property is used when we change the Hole Wizard Type or Standard in the Hole Wizard Property Manager. (Threaded Hole to Counter-Bored Hole) For example, if we are making a Custom ¼" Hole, we can add ¼" or M6 in the **Name to Match**. This will set the Size to use when switching the Hole Type or Standard. (If we want to bypass this function, add a "-" for the Property and delete it in the list after adding the Size.)
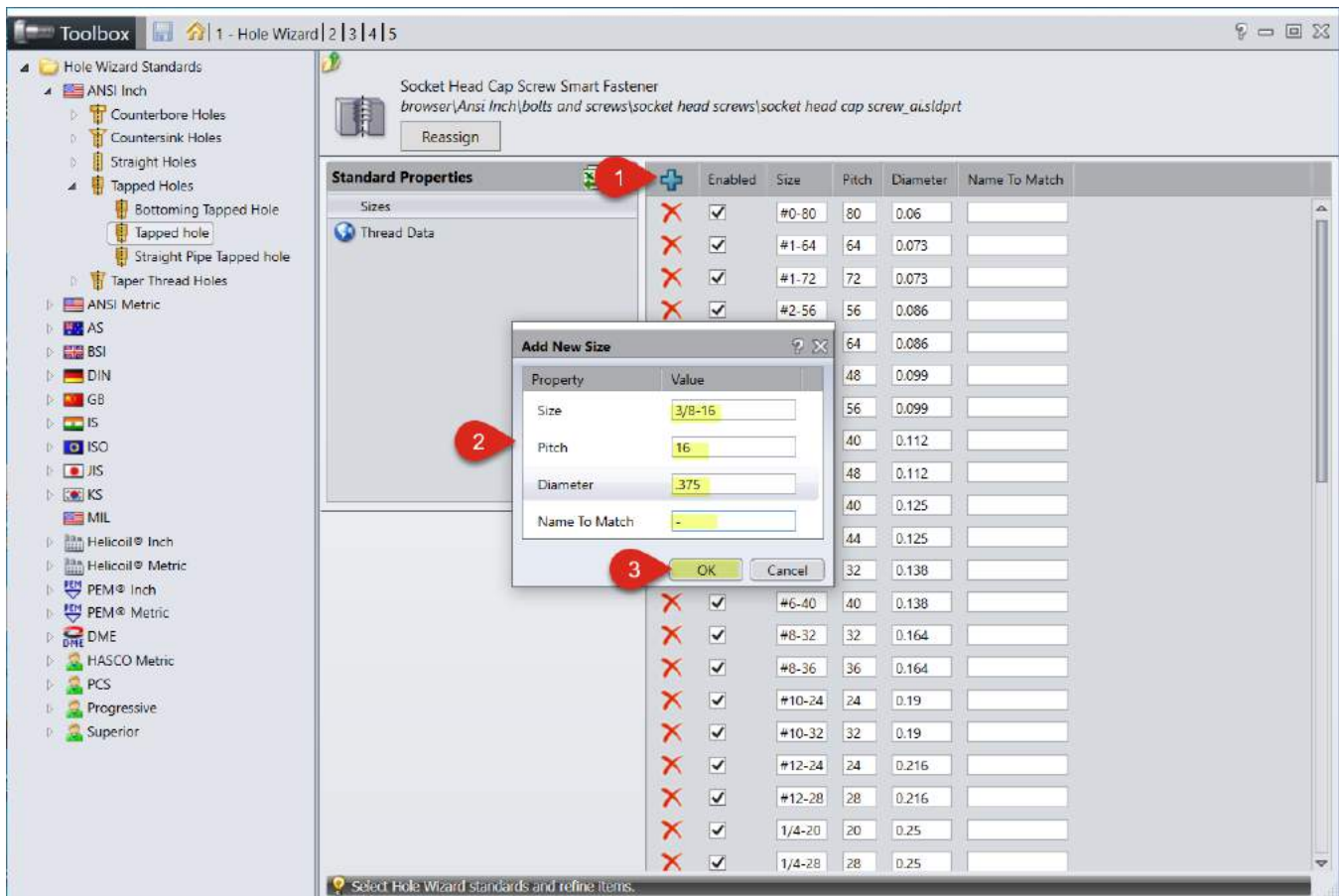
Step 4a: Select the Thread Data heading under Standard Properties and hit the "+" on that page to add the Thread Data. (Use the existing entries and reference Material like the Machinery's Handbook to help with what data is needed in the entries.)

Important: The **Full Size** Property in the Thread Data must match the **Size** Property in the previous step.
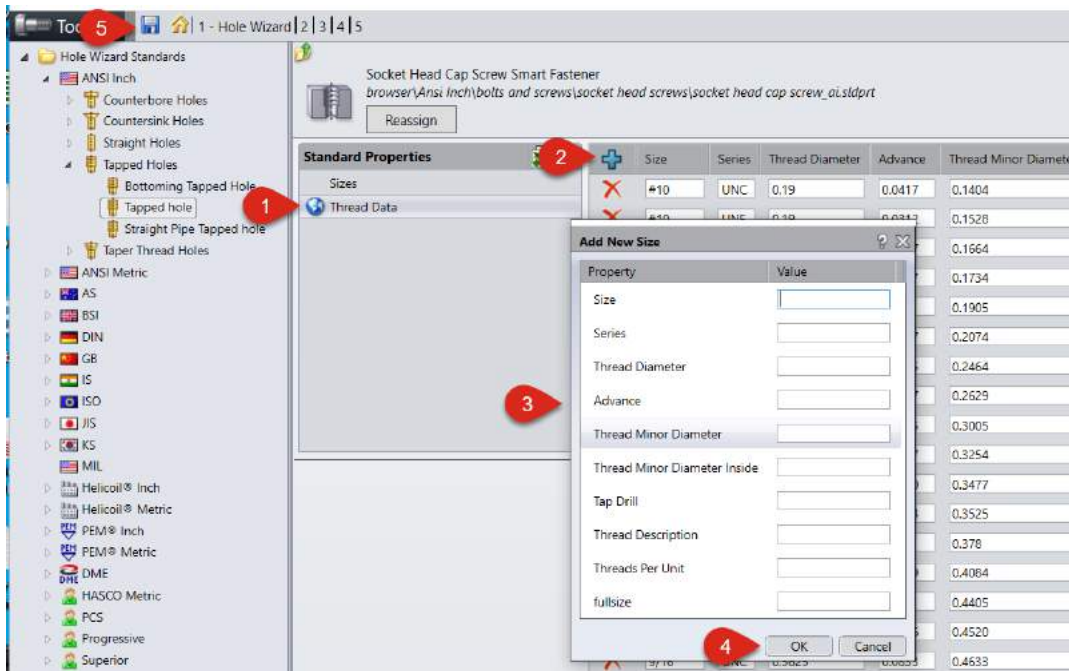
Step 5: Once all the information is entered, hit the Save Icon in the top toolbar to save the changes.

## Creating a Custom Hole Wizard Hole

This instruction shows how to add a Custom Hole Wizard Hole to the Database. When complete, we will have a Custom Hole Size which works identical to the pre-made entries in the Database.

The steps for the other types of Hole Wizard Holes, Counterbore or Countersink, are the same. However, we will be entering different Size information relevant to those types of Holes.

Step 1: Open SolidWorks and Go to Tools->Options->System Options->Hole Wizard/Toolbox. In the Hole Wizard/Toolbox Window select "Configure"
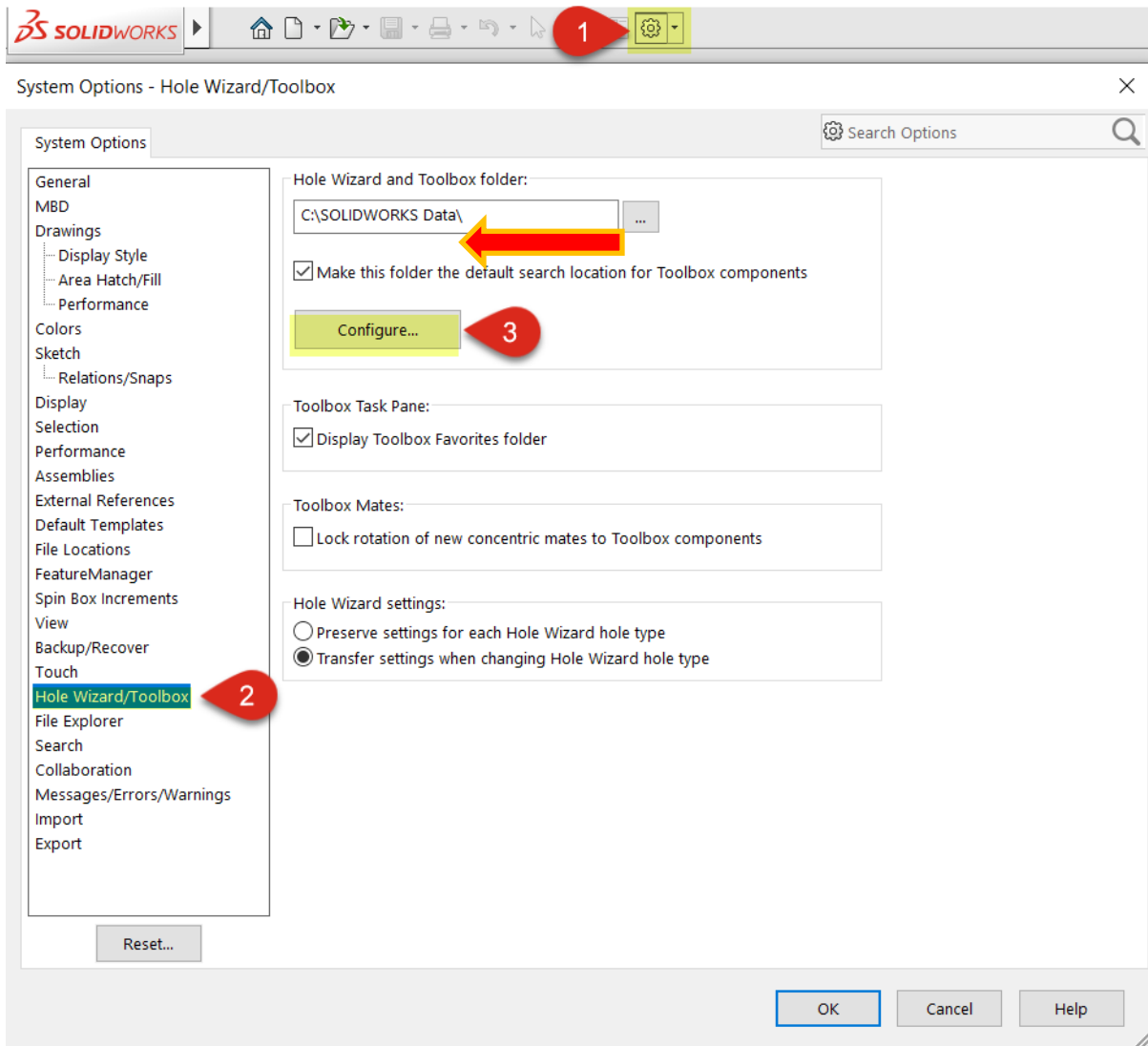
Step 2: In the Toolbox Settings Tool select the Hole Wizard link.
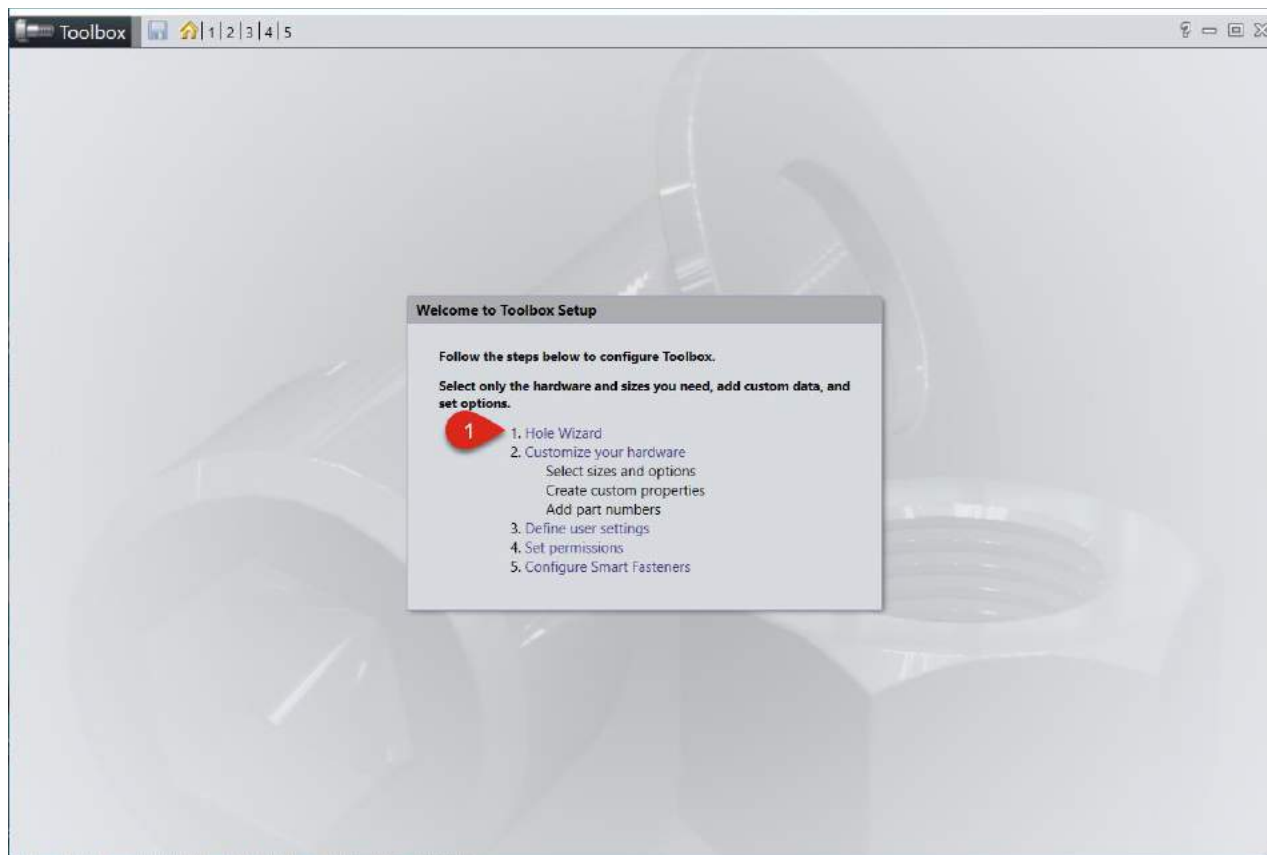
Step 3: Browse to the Standard and Hole Type to be added by using the arrows to expand each heading. Example: ANSI Inch, Straight Holes, Fractional Drill Sizes.
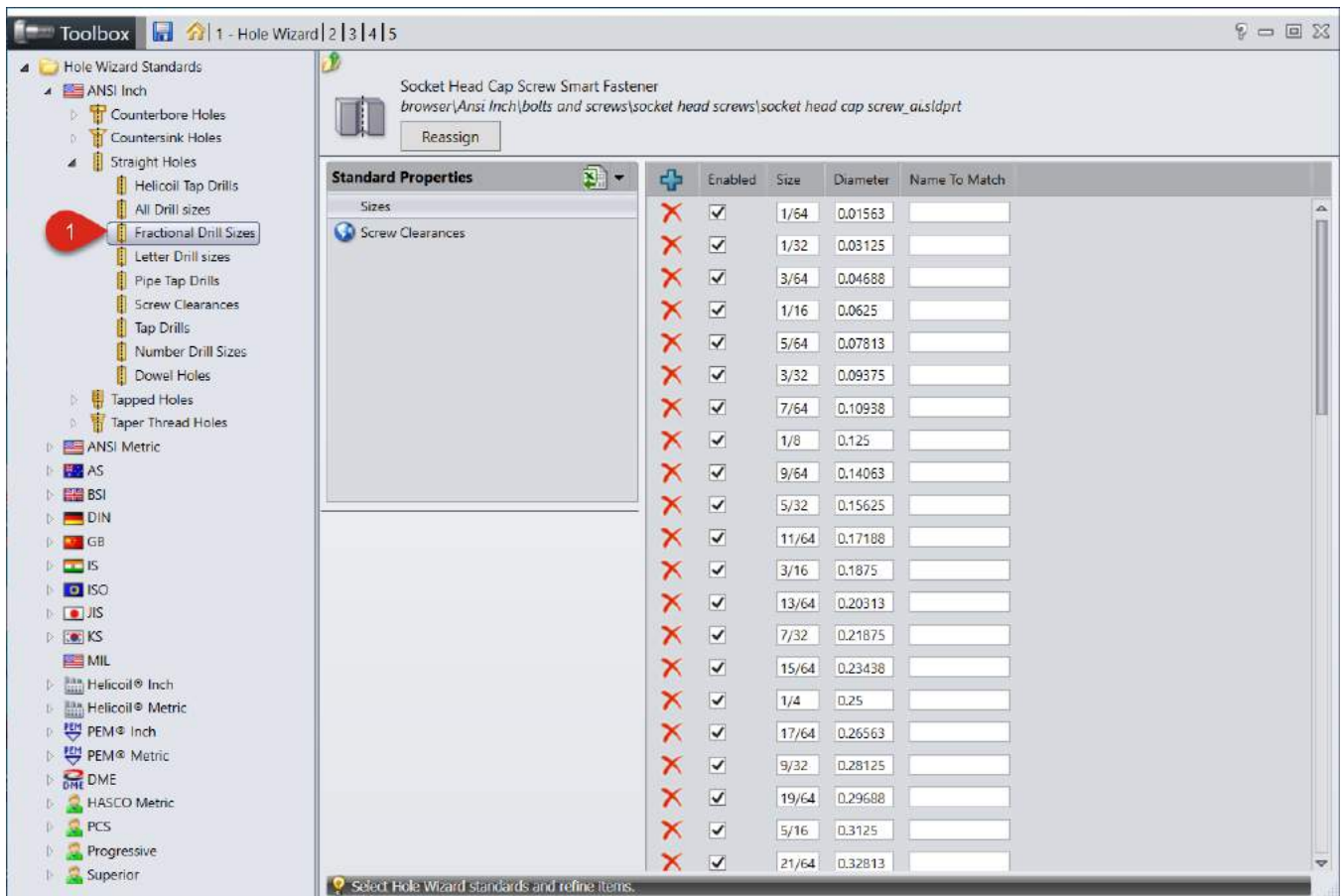
Step 4: Hit the "+" Icon to Add a Size and enter the Size Information.

> Note: The **Name to Match** Property is used when we change the Hole Wizard Type or Standard in the Hole Wizard Property Manager. (Straight Hole to Counter-Bored Hole) For example, if we are making a Custom ¼" Hole, we can add ¼" or M6 in the **Name to Match**. This will set the Size to use when switching the Hole Type or Standard. (If we want to bypass this function, add a "-" for the Property and delete it in the list after adding the Size.)
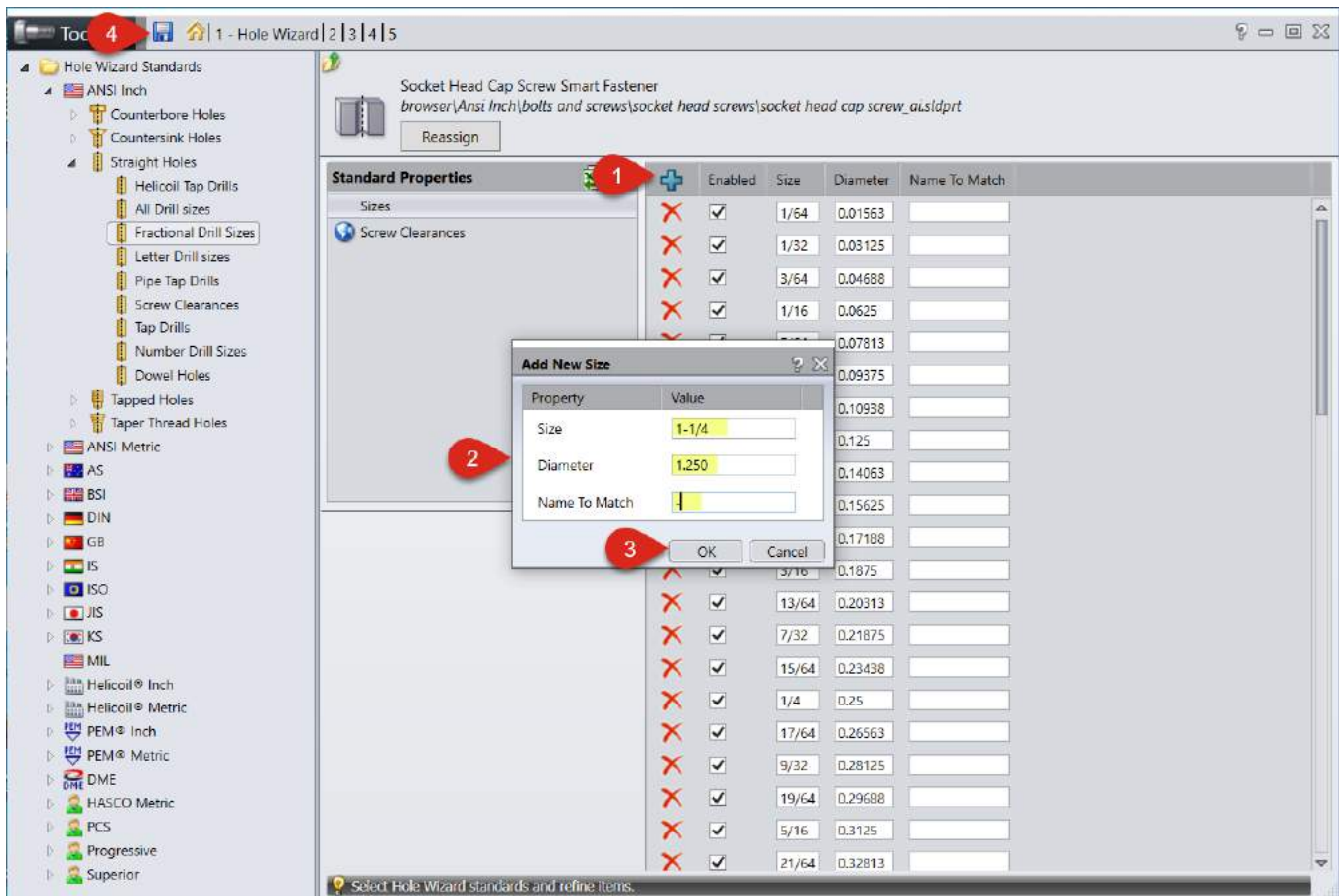
Step 5: Once all the information is entered, hit the Save Icon in the top toolbar to save the changes.

ACCESS / DISPLAY

TOOLBOX THREAD DISPLAY

For different preferences with performance in SOLIDWORKS, users may choose to limit certain visual attributes. This guide will show how to adjust the threads that are displayed graphically.

From User Desktop:

*Note: Before continuing, make sure to verify access to SOLIDWORKS Data folder in Windows Permissions allow for saving.

STEP 1: Browse to Start > Programs > SOLIDWORKS Tools and select **Toolbox Settings**:
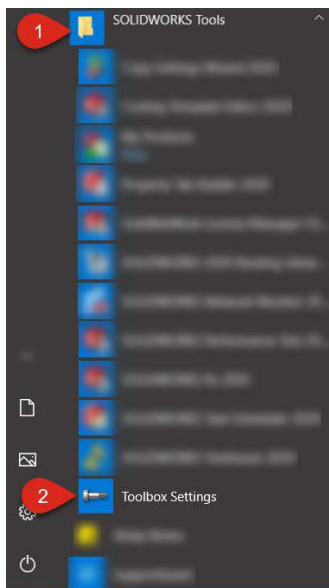
STEP 2: From main Toolbox Settings menu, select **Customize your hardware** link:

Welcome to Toolbox Setup

Follow the steps below to configure Toolbox.

Select only the hardware and sizes you need, add custom data, and set options.

1. Hole Wizard
2. Customize your hardware
   Select sizes and options
   Create custom properties
   Add part numbers
3. Define user settings
4. Set permissions
5. Configure Smart Fasteners

STEP 3: From main Toolbox Settings menu, **browse** to your desired hardware (ex. For Heavy Hex Bolt, go to ANSI Inch > Bolts and Screws > Hex Head > **Heavy Hex Bolt**):
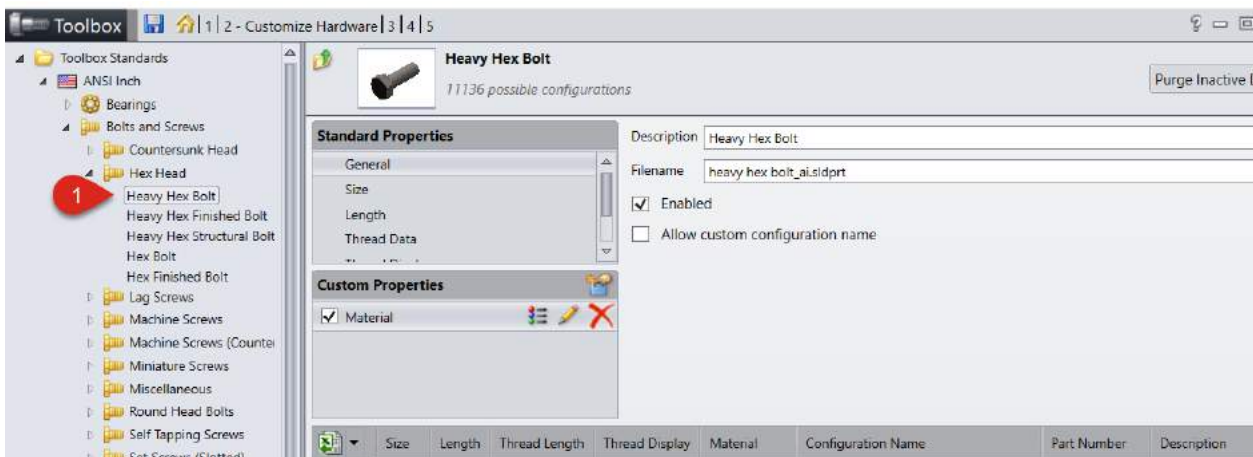
STEP 4: Choose the **Thread Display** link from Standard Properties and **deselect** your desired thread types (ex. Cosmetic and Schematic):

*Note: To speed up performance in large assembly environments, choosing only Simplified is recommended.
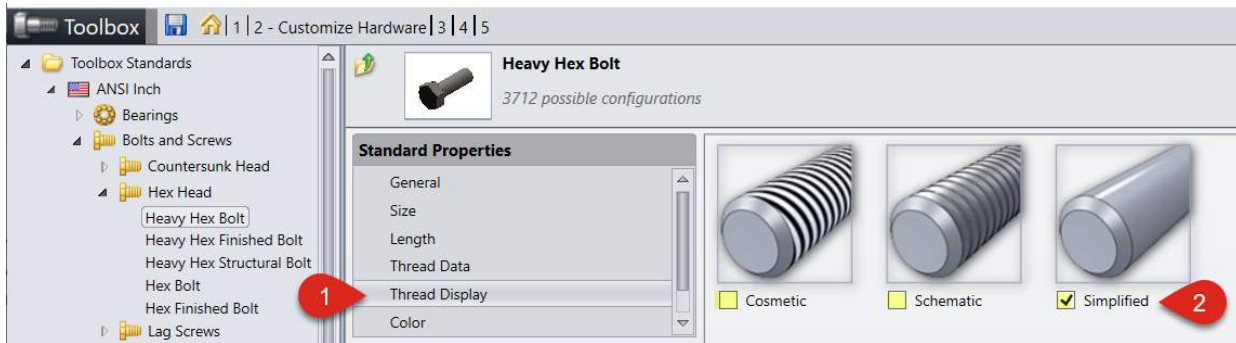
STEP 5: Select the **Save** icon to secure your changes upon exit:
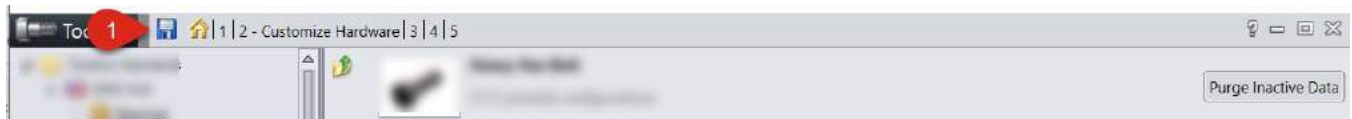
DISALLOWING AN ENTIRE TOOLBOX HARDWARE STANDARD

STEP 1: From main Toolbox Settings menu, select **Customize your hardware** link (Start > Programs > SOLIDWORKS > **SOLIDWORKS Tools**):
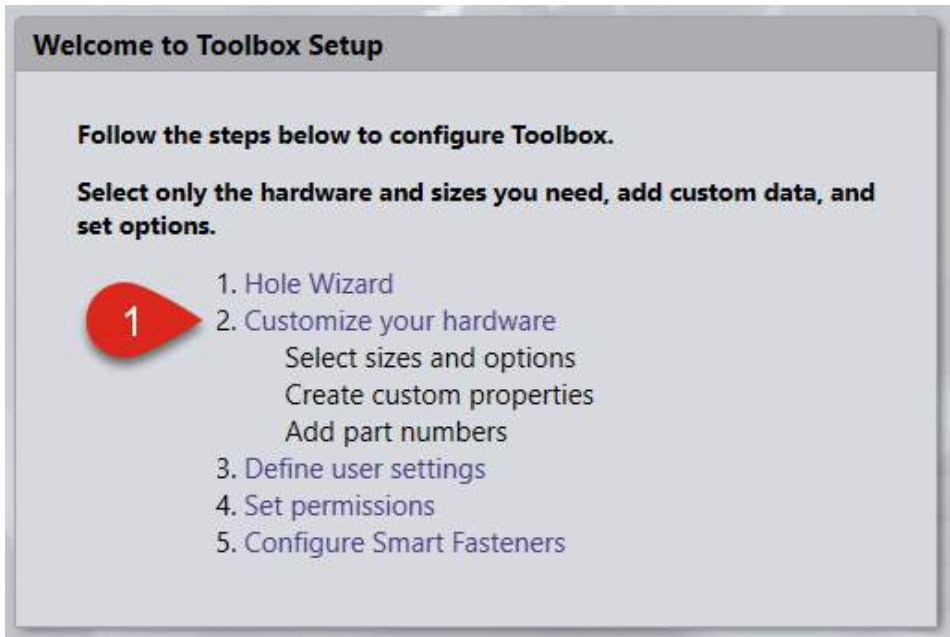
STEP 2: From the main standards menu, **click** on the hardware standard to be disabled (ex. AS, CISC):

*Note: By disabling a standard, it is removed from selection in SOLIDWORKS and can reduce load time.
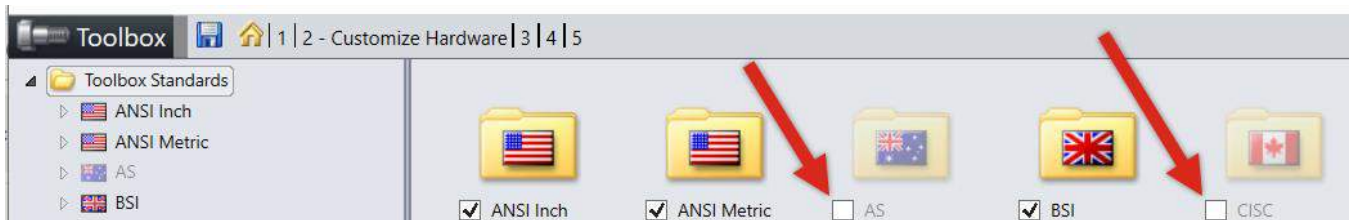
STEP 3: Select the **Save** icon to secure your changes upon exit:
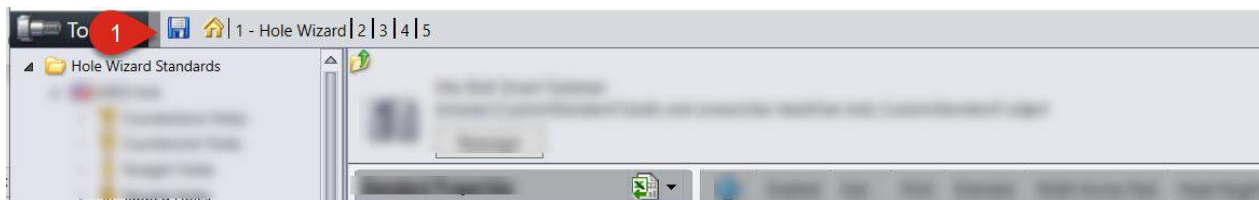
Disallowing a Toolbox Fastener

This guide shows how to disable certain fastener types and eliminate them from appearing during use in SOLIDWORKS.

From User Desktop:

STEP 1: **Browse** to Start > Programs > SOLIDWORKS Tools > **Toolbox Settings**:

**STEP 2:** From main Toolbox Settings menu, select **Customize your hardware** link:

STEP 3: Browse through the folder tree on the left to select Heavy Hex Bolt, for example (ANSI Inch > Bolts and Screws > Hex Head > **Heavy Hex Bolt**):
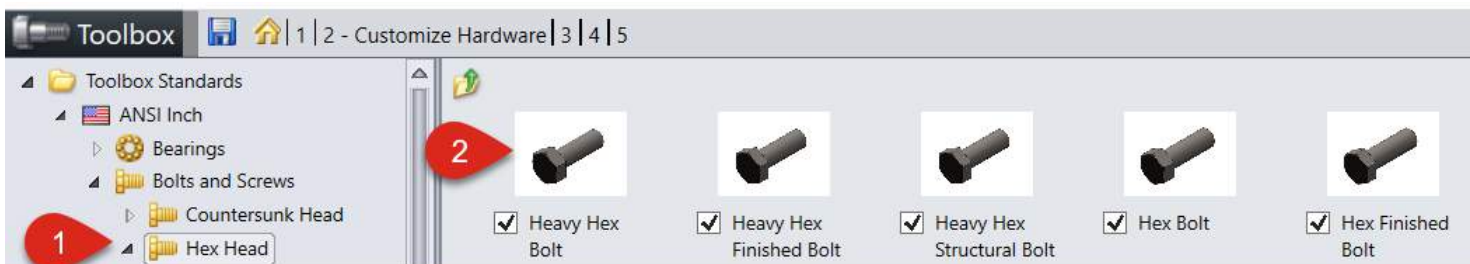
STEP 4: Select **Size** and **uncheck** ½" size (for example). This will remove this size from availability in Toolbox

from SOLIDWORKS. See Alternate figure below to show step for disallow of the entire Fastener type:
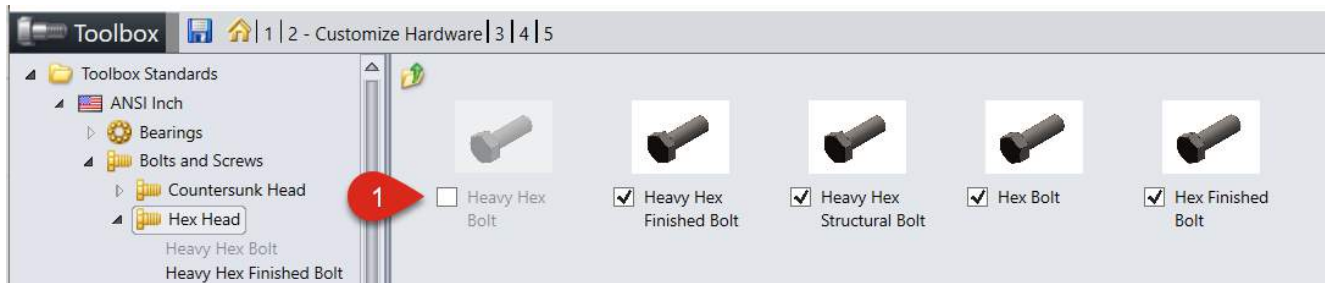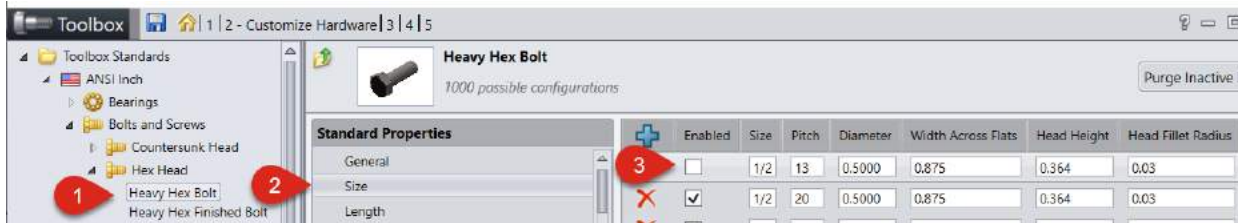
Alternate Figure

STEP 5: Once all preferred changes have been made, select the **Save** icon to secure changes:
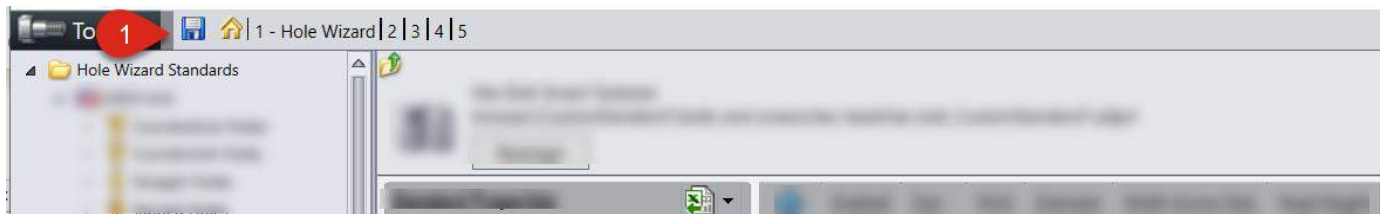
DISALLOWING A HOLE WIZARD SIZE

This guide shows how to disable certain hole sizes and eliminate them from appearing during use of the SOLIDWORKS Hole Wizard.

From User Desktop:

STEP 1: **Browse** to Start > Programs > SOLIDWORKS > SOLIDWORKS Tools > **Toolbox Settings**:
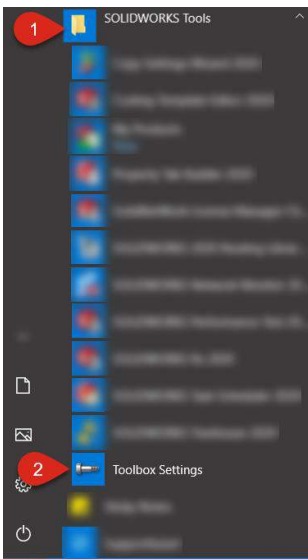
STEP 2: From main Toolbox Settings menu, select **Hole Wizard** link:
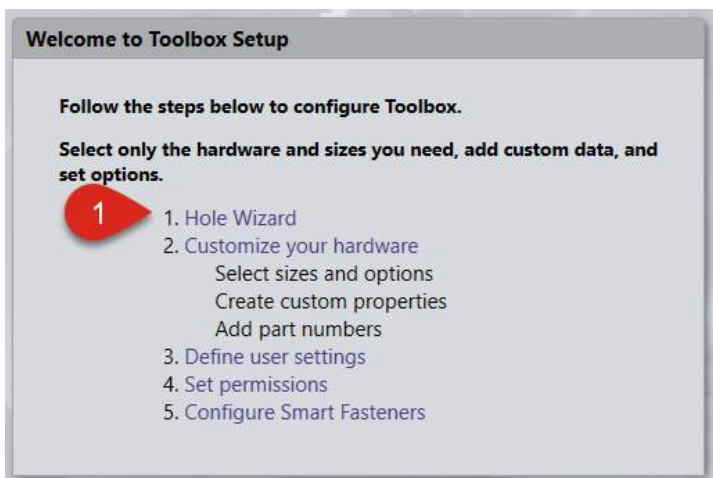
STEP 3: From the left file tree, **click through** to Hex Screw, for example (ANSI Metric > Counterbore Holes > **Hex Screw**) and **uncheck** the boxes for ¼" 20 pitch and ¼" 28 pitch. These are then removed from the SOLIDWORKS user interface. For disabling an entire size, see Alternate figure below:
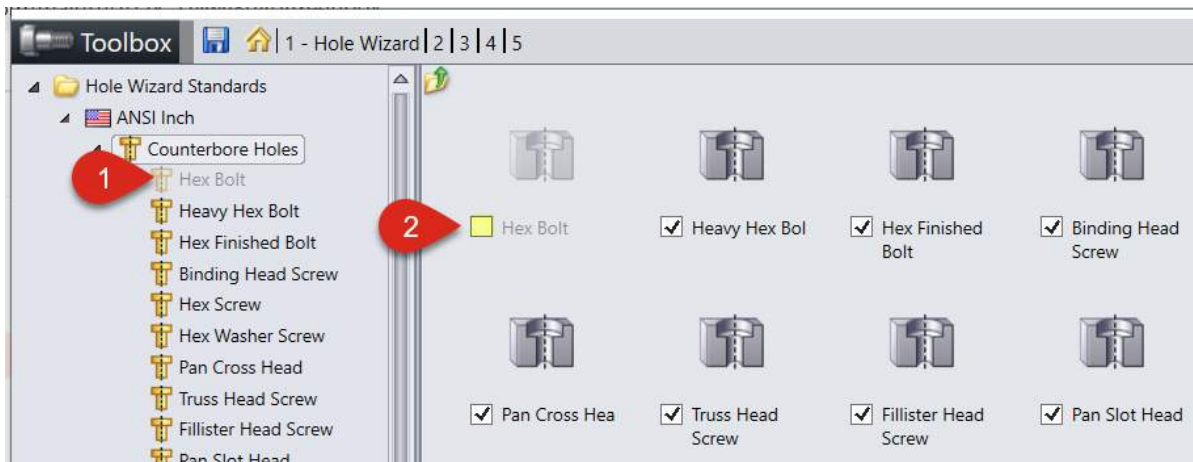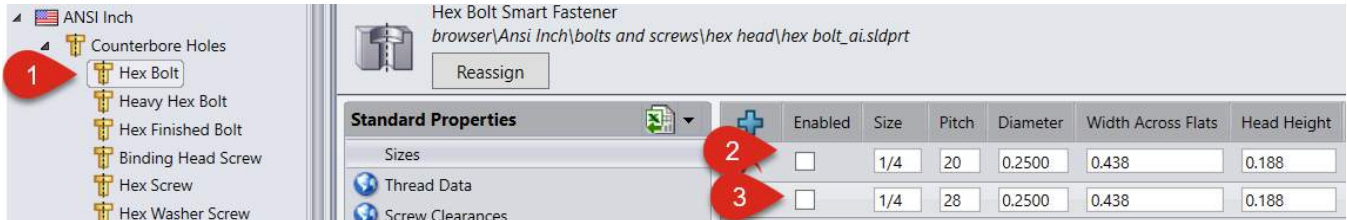
Alternate Figure

STEP 4: After desired changes have been made, click the **Save** icon before exiting:
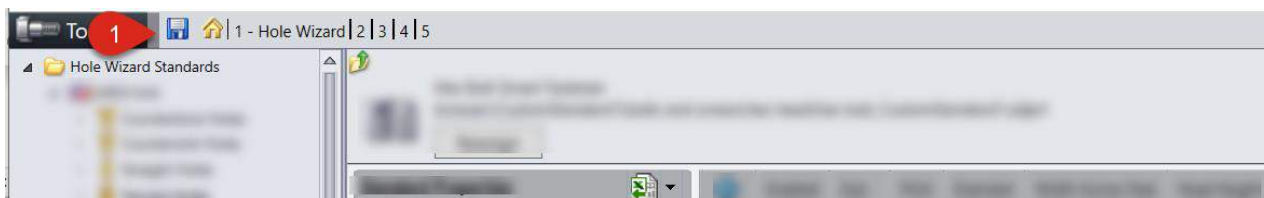
Permission Settings Overview

This section will explain the various options available to limit user's permissions in the Hole Wizard and Toolbox Database. These settings will help ensure that changes are not accidentally made by the end users. Choose "Set permissions" from the Toolbox Setup dialog to get started.

Setting the Password

In order to change the permission settings you will first need to set a password along with a security question and answer. Select "Change/Create" to set the password or to change it later.
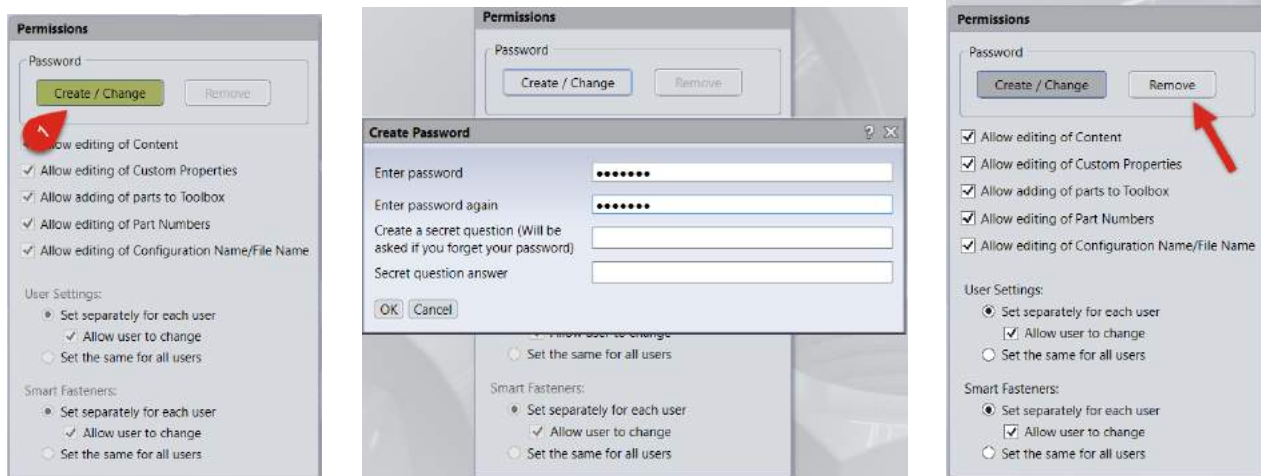
After the password has been set if you ever need to make changes to the permissions, or if you need to have full rights to make changes to toolbox components, you will need to log in by clicking the link along the top.
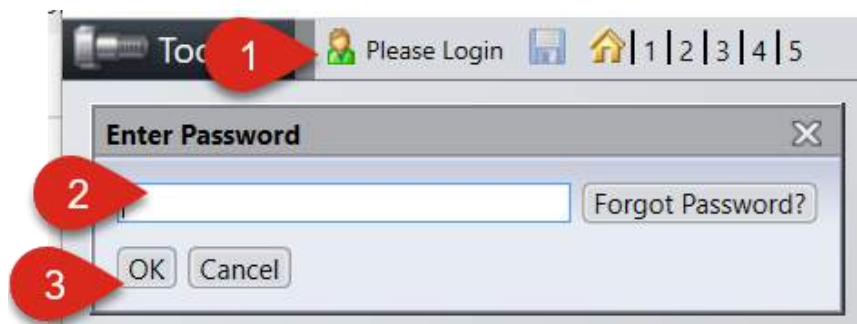
Permission Settings Overview

Allow editing of content:

With this option de-selected users who start up the Toolbox Settings tool will be unable to turn on or off any Standards, Folders, Components, and Sizes. Only after logging in with the set password can changes be made. Additionally the Descriptions for the various Sizes cannot be changed and the "Allow custom configuration name" cannot be toggled. All of these settings will be greyed out.
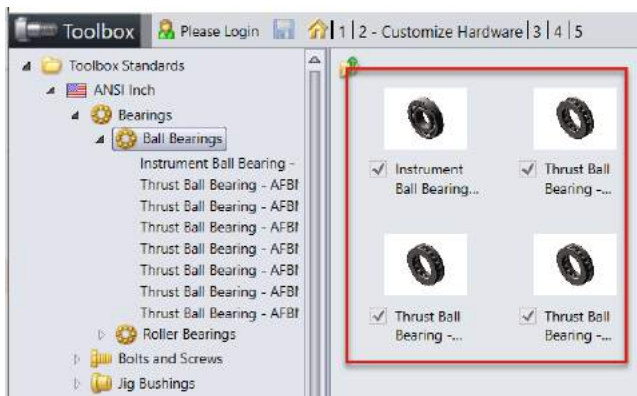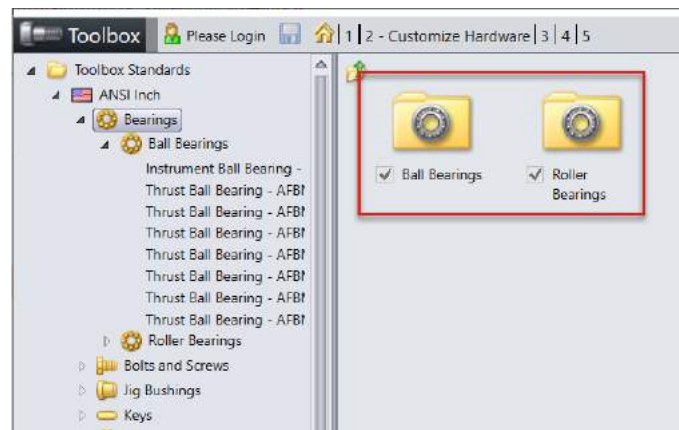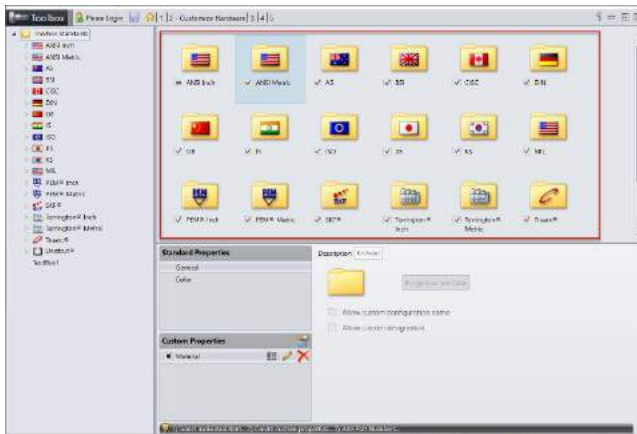
Allow editing of Custom Properties:

With this option de-selected users who start up the Toolbox Settings Tool will be unable to toggle, change, or add Custom Properties associated to each Toolbox component (outlined on page 28).

Allow adding of parts to Toolbox:

With this option de-selected users who start up the Toolbox Settings Tool will be unable to add their own custom components to the Toolbox (outlined on page 18).
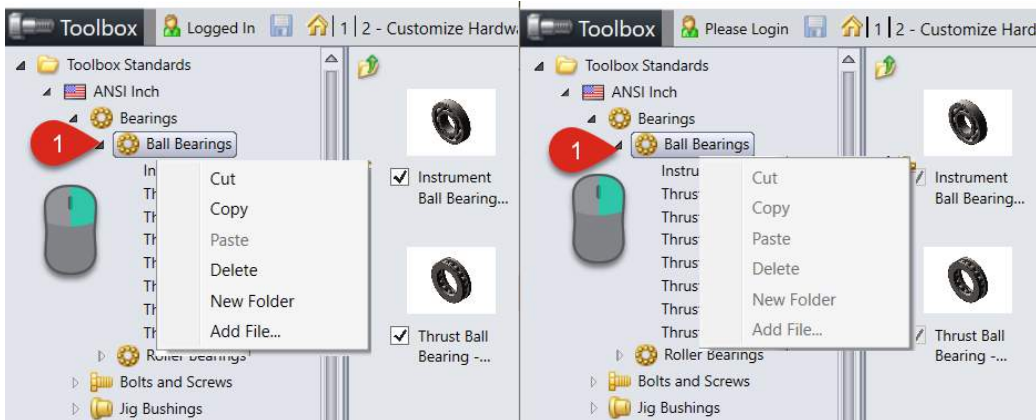
Allow editing Part Numbers:

With this option de-selected users who start up the Toolbox Settings Tool will be unable to change the values under the Part Number column while browsing the list of sizes. The cell will be greyed out.

Cage	Show Balls	Configuration Name	Part Number	De
No cage	6	AFBMA 12.2 - 0.0250 - 0.1000 - 0.0312 - 6,SI,NC,6		
No cage	6	AFBMA 12.2 - 0.0250 - 0.1000 - 0.0312 - 6,DE,NC,6		
Add cage	6	AFBMA 12.2 - 0.0250 - 0.1000 - 0.0312 - 6,DE,AC,6		
No cage	Full	AFBMA 12.2 - 0.0250 - 0.1000 - 0.0312 - Full,SI,NC,Full		
No cage	Full	AFBMA 12.2 - 0.0250 - 0.1000 - 0.0312 - Full,DE,NC,Full		
Add cage	Full	AFBMA 12.2 - 0.0250 - 0.1000 - 0.0312 - Full,DE,AC,Full		
No cage	8	AFBMA 12.2 - 0.0400 - 0.1250 - 0.0469 - 8,SI,NC,8		
No cage	8	AFBMA 12.2 - 0.0400 - 0.1250 - 0.0469 - 8,DE,NC,8		
Add cage	8	AFBMA 12.2 - 0.0400 - 0.1250 - 0.0469 - 8,DE,AC,8		
No cage	Full	AFBMA 12.2 - 0.0400 - 0.1250 - 0.0469 - Full,SI,NC,Full		
No cage	Full	AFBMA 12.2 - 0.0400 - 0.1250 - 0.0469 - Full,DE,NC,Full		

Allow editing of Configuration Name/File Name:

With this option de-selected users who start up the Toolbox Settings Tool will be unable to modify the file name associated to each Toolbox component. The field will be greyed out.

Additionally when users insert a Toolbox Component into their assembly they will not be allowed to change the Configuration name.
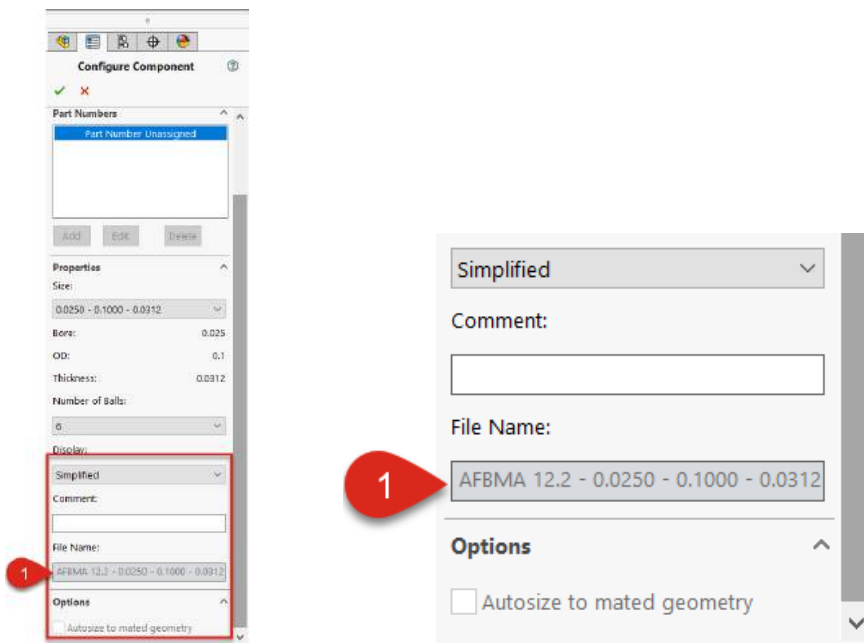
User Settings:

With the option "Set separately for each user" each user may have a different set of selections for the User Settings. If the "Allow user to change" option is selected the user may adjust the User Settings if they wish. Alternatively if "Set the same for all users" is active then all of the User Settings will be greyed out and the users will be unable to change the settings without the password.

Smart Fasteners:

With the option "Set separately for each user" each user may have a different set of selections for the Configure Smart Fastener settings. If the "Allow user to change" option is selected the user may adjust the Smart Fastener settings if they wish. Alternatively if "Set the same for all users" is active then all of the Smart Fastener settings will be greyed out and the users will be unable to change the settings without the password.

Welcome to Toolbox Setup

Follow the steps below to configure Toolbox.

Select only the hardware and sizes you need, add custom data, and set options.

1. Hole Wizard
2. Customize your hardware
   Select sizes and options
   Create custom properties
   Add part numbers
3. Define user settings
4. Set permissions
5. Configure Smart Fasteners

Smart Fasteners

Washer sizes
Show washer sizes that match fastener criteria
○ Exact match
○ Greater than tolerance   > 20   %
● Unrestricted

Automatic fastener change
☑ Change fastener length to ensure minimum thread engagement

2   Threads beyond a nut

1.5   x diameter into a tapped hole

☑ Change stack components when fastener size is changed

Fastener to use with Non-Hole Wizard holes
Socket Head Cap Screw
C:\SOLIDWORKS Data\browser\Ansi Inch\bolts and screws\socket head screws\socket head cap screw_ai.sldprt