

Pulsonix Vault Evaluation Guide

#### **Copyright Notice**

Copyright © WestDev Ltd. 2000-2024 Pulsonix is a Trademark of WestDev Ltd. All rights reserved. E&OE

Copyright in the whole and every part of this software and manual belongs to WestDev Ltd. and may not be used, sold, transferred, copied or reproduced in whole or in part in any manner or in any media to any person, without the prior written consent of WestDev Ltd. If you use this manual you do so at your own risk and on the understanding that neither WestDev Ltd. nor associated companies shall be liable for any loss or damage of any kind.

WestDev Ltd. does not warrant that the software package will function properly in every hardware software environment.

Although WestDev Ltd. has tested the software and reviewed the documentation, WestDev Ltd. makes no warranty or representation, either express or implied, with respect to this software or documentation, their quality, performance, merchantability, or fitness for a particular purpose. This software and documentation are licensed 'as is', and you the licensee, by making use thereof, are assuming the entire risk as to their quality and performance.

In no event will WestDev Ltd. be liable for direct, indirect, special, incidental, or consequential damage arising out of the use or inability to use the software or documentation, even if advised of the possibility of such damages.

WestDev Ltd. reserves the right to alter, modify, correct and upgrade our software programs and publications without notice and without incurring liability.

Microsoft, Windows, Windows NT and Intellimouse are either registered trademarks or trademarks of Microsoft Corporation.

All other trademarks are acknowledged to their respective owners.

Pulsonix is a trading division 10 WestDev Ltd.

Printed in the UK. Issue date: 13/08/24

#### Pulsonix

20 Miller Court Severn Drive Tewkesbury Glos, GL20 8DN United Kingdom

Phone +44 (0)1684 296 551

Email info@pulsonix.com

Web www.pulsonix.com

Support Phone +44 (0)1684 296 570

Email support@pulsonix.com

# Contents

CONTENTS	3
Chapter 1. Getting Started	4
Introduction to the Pulsonix Vault	
Introduction to the Tutorial Additional Help	
CHAPTER 2. INSTALLING THE PULSONIX VAULT	
Installing the Pulsonix Vault	
Signing In to the Vault	. 6
CHAPTER 3. VAULT MANAGER OVERVIEW	7
Vault Manager Overview	. 7
Navigating the Vault Manager	. 7
CHAPTER 4. USING THE PULSONIX VAULT	9
Creating a new design using Vault items	. 9
Inserting Components from the Vault 1	10
Checking a design into the Vault 1	
Adding items to the Vault 1	
Update Vault Design with Vault Part 1	
Investigating items within the Vault 2	21
Creating a Search in the Vault	22
Seeing the Audit History within the Vault	
Additional Information	24

## Chapter 1. Getting Started

## **Introduction to the Pulsonix Vault**

The Pulsonix Vault is a revision/version control system which allows you to manage Pulsonix data in a company defined methodology.

The Vault can be installed locally or on a company server and provides controlled user access to a database of Pulsonix data that can be checked in/out, deleted and modified with full audit history.

### **Introduction to the Tutorial**

This tutorial is intended to familiarise you with the Pulsonix Vault features and its methodology by working through a step by step tutorial. Using a supplied Vault demo, the tutorial will show you how to setup, install and use the Pulsonix various elements within the Pulsonix Vault.

## **Additional Help**

Help is available in a number of formats; Online help available by pressing **<F1>** at any time when using Pulsonix; by emailing us at <a href="support@pulsonix.com">support@pulsonix.com</a>, or by calling us or your local distributor. You will also find the **Vault Users Guide** installed as a PDF file, this is instantly available from the Pulsonix **Help** menu and **<Online Manuals>**.

## Chapter 2. Installing the Pulsonix Vault

### **Installing the Pulsonix Vault**

To install the Pulsonix Vault, you will need to run the **Vault Server Setup** application from your Pulsonix Vault Admin installation folder.

If you are running a newer version of Windows, you may find the icon on the right appear in your recently added applications.



You should now see the Vault Server Setup dialog.

Click the **Install Demo Vault** button to install the pre-populated Vault which will be used in this tutorial.

Vault Server Setup X
Install Demo Vault Install a local Vault server pre-populated with sample data for evaluation of Vault features.
Install Vault Server Install a Vault server and empty database ready for use with your own production data.
— Pulsonix —Cancel

Once you have pressed the **Install Demo Vault** button, you will then be presented with the **Vault Demo Setup** dialog as below:

Vault Demo Setup	X
This Setup will install the database server, create a new Pulsonix Vault database, and populate it with example design and library data to make it easy for you to see how the Vault works.	2
Install Now Options	
Install and launch Postgres database server	
Create Pulsonix Vault database	
Create data file folders	
Share data file folders	
Restore sample Vault data	
Installation complete	
Pulsonix	
Close	]
	This Setup will install the database server, create a new Pulsonix Vault database, and populate it with example design and library data to make it easy for you to see how the Vault works.          Install Now       Options         Install and launch Postgres database server       Options         Create Pulsonix Vault database       Create data file folders         Share data file folders       Share data file folders         Restore sample Vault data       Installation complete         Pulsonix       Options

Click the **Install Now** button to complete the Vault Demo setup.

Note: If you already have a Postgres database running, press the **Options** button to set your existing database credentials.

## Signing In to the Vault

The final stage of installing the Pulsonix Vault is to activate it from within Pulsonix.

In Pulsonix, go to the **Setup** menu, click **Vault**, you will now see the **Vault Setup – Sign In** dialog. From here, enter the **User Name: 'admin'** and **Password: 'password'** from the **Sign In** tab, click **Test** to verify the connection, click **OK** to close the dialog once the connection is verified and click **OK** again to close the **Vault Setup – Sign In** dialog.

	Vault Setup	- Sign	In							×
	Sign In (	Jsers	Groups	Revision Naming	Attributes	Options	Colours			ОК
N			Prim	ary Sign-in		⊖ Alt	emate Sign-in			Cancel
$\Box \rangle$	User Nan	ne:	admin							Apply
$\square$	Password	d:	passw	ord					Show	
	ODBC Dr	river:	Postgr	eSQL ANSI					Default	Help
	Server N	ame:	localho	ost					7	
	Server Po	ort:	5432							
$\square$			Т	est	Pulsonix			×		
	Data Fol	der:	C:\Use	ers\Documents\Pu						
	Vault ID:		131208	3237909420000	Ţ	Vault su	ccessfully connec	ted —	_	
							Oł	(		

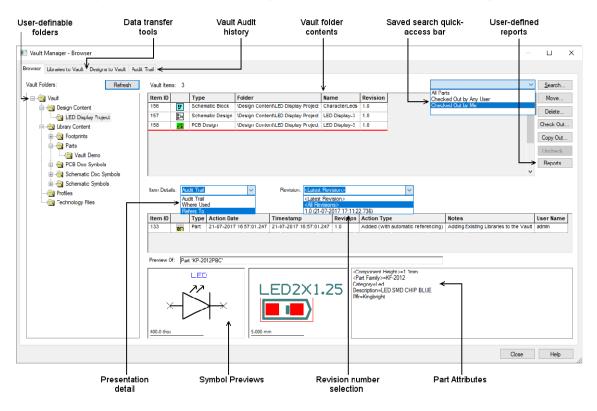
You are now ready to start using the Vault functionality.

## Chapter 3. Vault Manager Overview

## **Vault Manager Overview**

To open the **Vault Manager**, click the **File** menu and select **Vault Manager**. Within this dialog you can search and investigate all the items that are saved within your Pulsonix Vault.

The picture below shows you the major facets that are within the Vault Manager.



### **Navigating the Vault Manager**

Upon opening the Vault Manager, you will be presented with a blank dialog, as shown below, to navigate the Vault manager you can expand the folders by selecting the '+' symbol.

	Vault Manager - Browser				×
Γ	Browser Libraries to Vault Designs to Vault Audit Trail				
	Vault Folders: Refresh Vault	at items;	~	<u>S</u> earch	
	E- Vaut	m ID Type Folder Name Revision		Move	
	nation of the second s			Delete	
	Profiles			Check Out.	
	Technology Files			Copy Out	
				Uncheck	
				Reports	

Open the Design Content folder to show the content of the LED Display project. Within this folder you will see a Schematic Block, Schematic Design and a PCB Design.

Selecting the Schematic Design, you will now see specific details relating to the Schematic Design in the grid below.

-		aracterLeds 1.0		
ematic Design \Design Conte				De
	nt\LED Display Project LED	Display-3 1.0		De
Design \Design Conte	nt\LED Display Project LED	Display-3 1.0		Chec
				Cop
				Und
				Rep

## Changing the Item Details

Using the **Item Details dropdown arrow**, you can choose which information of the item you wish to inspect. You can choose from **Audit Trial** (an audit history of the selected item), **Where Used** (informs you where the Item is used, e.g. which Designs a selected Parts is used in) and **Refers To** (informs you which items are used in the selected item e.g. Which Parts are used in the selected design).

## Changing the displayed Revision

Using the Revision dropdown arrow, you can select which revision of the selected item will be displayed in the grid. You can choose from the **<Latest revision>** (Default setting), **<All Revisions>**, alternatively you can select a specific revision of an item.

## Chapter 4. Using the Pulsonix Vault

## Creating a new design using Vault items

### ► To start a new Schematic design

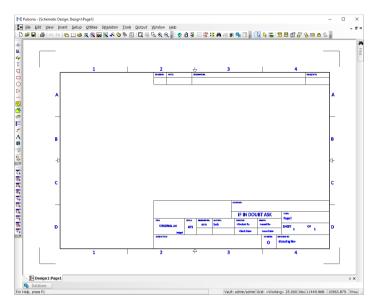
On the File menu, click New and Schematic Design from the Design tab.

New						
	Designs Technologies + Profiles Wizards					
PCB Design      Schematic Design      Schematic Design      PCB Footprint      PCB Doc Symbol      Schematic Doc Symbol      Schematic Doc Symbol      Part      Part      Panel Design						
	Iechnology:       Default (White · Imperial)       ~         Profile File:       A4-Landscape       ~         OK       Cancel					

On the **Technology:** drop down list, click on the small 'down' arrow to reveal the list, select **Default (White - Imperial)**, by default this will be shown in the list with a blue background to indicate it has come from your Vault.

On the **Profile File:** drop down list, click on the small 'down' arrow, select the **A4-Landscape** profile file to add an A4 drawing blank, by default this will be shown in the list with a blue background to indicate it has come from your Vault. More detail about Technology files and Profile files is available in the online Help. For now, we will progress on to adding Components to the design.

Click the **OK** button to start a new Schematic design.



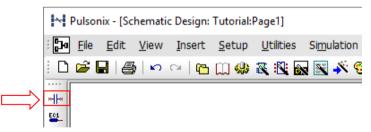
Your Pulsonix window will look like this to the left. To view the whole page, you may need to click **<A>** on the keyboard.

If you wish to start with a black background, either change it in **Colours**, shortcut **<C>** and the **Others** page or start the new design with the **Default** (Black – Imperial) Technology file.

## **Inserting Components from the Vault**

### To Insert a Component

Select the **Insert Component** button on the **SCM** toolbar (docked left side) shown circled below, or use the **Insert Component** option from the **Insert** menu. You can also use the shortcut key **Shift-C** 

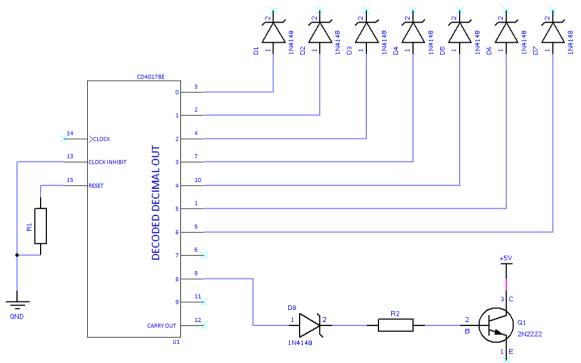


You are presented with the Insert Component dialog:

Insert Component	×
Look In: [Vault Only]	(From Vault) <u>A</u> dd
Which Parts: <u>Filter:</u> No. Pins: Apply	Cancel
Matched: 10 of 10	
Part: 1N4148	Pins: 2 Find
Desc: Diode 1N4148	Family: 1N/1N41
Eootprint: D0-35	
Name: D1	
Symbol: DIODE-1	Add to Comp <u>B</u> in
Vault Revision: <latest revision=""></latest>	Number of Copies: 1
Preview Schematic & PCB Attributes Vault	

On the initial start-up, the **Look In:** will default to [All Libraries], change the this to be [Vault Only], this will then allow you to only place Parts that have come from your Vault.

For the purpose of this tutorial, we will now make a portion of the LED Display example circuit which can be found in your Vault. As shown below:



The first Part to add is the CMOS Counter/Divider. One of the quickest ways to select the Part you want is to simply type the first letter(s) of its name.

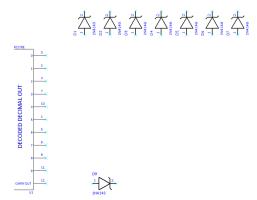
In the Part box, Type C, you will now see the CMOS Counter (CD4017BE) is filtered in the list, now click the **Add** button to insert it into the design.

	Insert Component	×
	Look In: [Vault Only]  Which Parts: Eilter: No. Pins: Apply Matched: 10 of 10 Include Connectors	(From Vault)
$\square >$	Part: CD4017BE	Pins: 16 Find
	Desc: CMOS Counter/Divider	Family: CD/CD40
	<u>N</u> ame: U1	
	Symbol: CD40178E	Add to Comp <u>B</u> in
	Vault Revision: <a href="https://www.category.com"></a>	Number of <u>C</u> opies: 1
	Pre <u>v</u> iew Schematic & PCB Attributes Vault	
		0 0 0 0 0 0 0 0 0 0
	1000.0 thou 10.000 mm	_

### Adding Diodes

We will now add the Diodes to our design. Click on the **Insert Component** button again.

Type **1N148** into the Part field. Click the **Add** button to add the Part to your design. We want to add 8 instances of this Part.



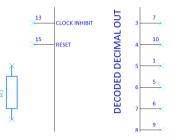
Use the 'R' key on your keyboard to rotate a component during placement.

After you have inserted the symbol, clicking escape on your keyboard will take you back to the Insert Component Dialog. Hint: Using the context menu during insert, you can select **Insert Multiple Items**, this will allow you to insert 7 instances in one process.

#### Add First Resistor

We will now add the first resistor to our design. Click on the **Insert Component** button again.

From the drop-down arrow on the Part Field, select **R 0.25W 5% MCF 10K** and add it to your design as shown in the diagram below.



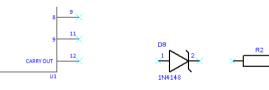
Use the `R' key on your keyboard to rotate a component during placement.

After you have inserted the symbol, clicking escape on your keyboard will take you back to the Insert Component Dialog.

#### Add Second Resistor

We now need to add the second resistor that has a different resistance value. Click on the **Insert Component** button again.

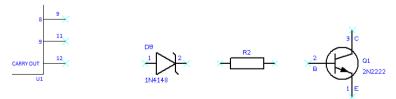
From the drop-down arrow on the Part Field, select **R 0.25W 5% MCF 330** and add it to your design as shown in the diagram below.



#### Add Transistor

The final Part we need to add the design is the Transistor. Click on the **Insert Component** button again.

Type **2N222** into the Part field. Click the **Add** button to add the Part to your design. Your design should now look something similar to the image below.



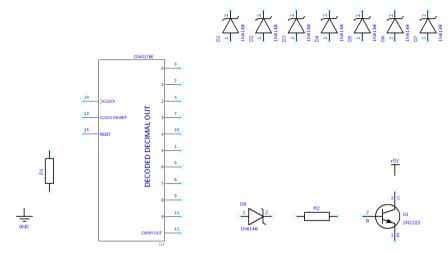
#### Adding Signal References

We now need to add Signal References to our Design. To do this, click **Signal Reference**, from the Insert Menu.

Ensure the **Look In:** is set to **[Vault Only]**. Select the **+5V Symbol** from the Symbol dropdown list and using the **Add** button, insert the symbol above the Transistor in your design. Upon adding the symbol into the design, a dialog will appear allowing you to define the Net on the Signal, leave it as +5V and click the **Use for all net type** button.

Once you have inserted the +5V Signal Reference, press the escape key on your keyboard, you will now see the **Insert Signal Reference** dialog, from here, select the **GND** symbol from the Symbol list and add it to your design below R1.

Your design should now look like the image below:

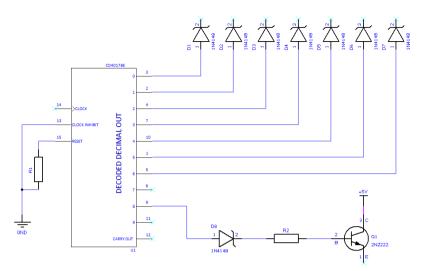


#### Adding connections to your design

To finish this example circuit, we need to add connections to this design.

To insert a connection, you can double click on a Symbol's pin which will invoke the insert connection option. Alternatively, you can use the **Insert Connection** mode, available from the **Insert** menu.

Route the Schematic so it looks similar to the example below:



Our example circuit is now finished, you can choose to click the **Save** option from the **File** menu if you would like to keep a local copy of the file prior to checking it into the Vault. If you do not choose to save the design, the Vault will automatically create a save version as you check it into the Vault for the first time.

## Checking a design into the Vault

With the example design we have just created open, click the **File** menu and select **Check into Vault**.

You will now be presented	d with the <b>Chec</b>	k Into Vault dialog.
---------------------------	------------------------	----------------------

	Check Into Vault	×
	Name: Tutorial Design	Vault Item ID:
	Default Next Vault Revision	Vault Revision:
<u> </u>	Vault Folder	
└─-/`	\Design Content\Tutorial Design	Browse
	Check In Note	
	Checking in for the first time.	
$\Box$	Close item after Check In	
	ОК	Cancel

In **Name:** type '**Tutorial Design**'. Next you need to define where the design will be saved to, set the **Vault Folder** to be \Design Content\Tutorial Design. Finally, add a **Check In Note** for the design.

When checked, the **Close item after Check In** option will close the design automatically for you.

Press the **OK** button to check the design into your Vault.

#### Adding items to the Vault

The next section of this tutorial will look at how to add new items to the Vault for the first time.

We will look to add a Surface Mount resistor to the Vault as we do not currently have one in the demo.

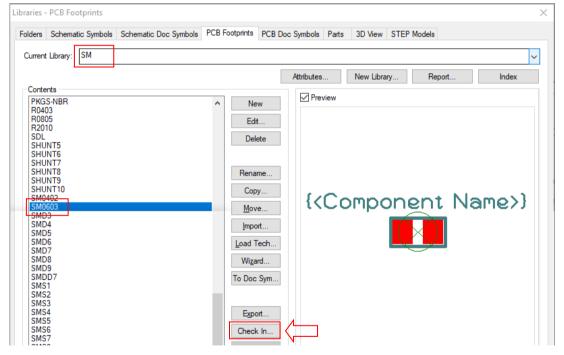
As there is already a resistor Schematic Symbol in the Vault, we will only need to add a Footprint and Part to the Vault.

#### ► Add SM0605 Footprint to Vault

From within Pulsonix, go to Setup, Libraries and click the PCB Footprints tab.

Clicking the drop-down arrow, open the **SM** library.

From within the **Contents**, select the footprint **SM0603** and click the **Check In...** button.



From within the **Check Into Vault** dialog, select the chosen **Vault Folder** and enter a **Check In Note** for the Footprint. Press the **OK** button to check the Footprint into the Vault.

	Check Into Vault	×
<u> </u>	Vault Folder	
	Library Content\Footprints\Vault Demo	Browse
N	Check In Note	
$\Box$	Checked in to Vault Demo for first time	
	Default Next Vault Revision	Check In as replacement for existing vault item
	Close item after Check In	for existing valit item
	Remove item from local library after Ch	eck In
$\Box$	OK	Cancel

Now the Footprint is in the Vault, we can look to create a Part, referencing the Schematic Symbol and PCB footprint which is form the Vault.

#### Create New Part to include Vault Items

From within the **Library Manager**, click the **Parts Tab**, select the **Resistor** Library from the drop-down arrow and click the **New** button.

Libraries -	Parts									$\times$
Folders	Schematic Symbols	Schematic Doc Symbols	PCB Footprints	PCB Doc Symbols	Parts	3D View	STEP Models			
Current	Library: Resistor									~
		—		Attrib <u>u</u> tes		Ne <u>w</u> Libra	ry Re	p <u>o</u> rt	Inde <u>x</u>	
	nts 063W SMTF 10K 063W SMTF 12		<u>^</u>	Preview	v Detai	ils				

Within the **New Part** Dialog, type the name for the new Part, this will be **R 0.25W 10K**. Now you type a **Description** for the Part, its **Name Stem** and **Pin Count**.

Next, in the **PCB View** section, we need to assign a Footprint to the Part. Using the dropdown arrow, search for the **SM0603** footprint we added to the Vault.

Finally, in the **Schematics View**, search for the Symbol **R** which is in the Vault.

New Part	×
Part Name: R 0.25W 10K OK	
Part Type:  Normal Part O PCB Only O Schematics Only Cancel Connector O Associated Part	
Provide Part Details Now (Quick Part Creation)	
Description: Thick Film Surface Mount Resistor Part Family:	
Name Stem: R Pin Count: 2	
Footprint: SM0603 {Vault}	
Schematics View  Single Symbol O Multiple Gates Gate Count:  Symbol Pin Count:	
Symbol: R {Vault} Pins: 2 Total Schematics Pins: 2 Ungated Pins: 0	
Power Pin         Ground Pin           Pin:         Net:         VDC         Pin:         Net:         GND         V	

## Adding Attributes to the Part

From within the Part Editor, we can now add Attributes to the Part, these will then be searchable from within the Vault should we need to search for them.

Type 10K in the Value field and Resistor in the Category field.

⊷ F	ulsonix - [P	art: C:\Users	\Publi	c\Docum	ients\Pu	lsonix9.1\M	asterLibra	ries\Resistor.p	al: R 0.25V
:	<u>F</u> ile <u>E</u> dit	: <u>V</u> iew ]	nsert	<u>S</u> etup	<u>T</u> ools	<u>W</u> indow	<u>H</u> elp		
: D	🖻 🖬   1		6	🖽 🏶	<b>8</b> 8	😹 📉 养	ন্য 🕐		R. Q. C
	Att	ributes							
	Manually N	ame Parts	RO	.25W 10K					
 ©*	Value		10	<					
X	Category			sistor					
<u>(</u>									
2									
<mark>са</mark>									
11									
	нчьы	Details	arts a	nd Attrib	utes / I	Pins / Gate:	s / Asso	c Parts /	
	🛛 🧾 LE	D Display-3	5.0	R 0.25	W 10K	*			
For	Help, press	F1							

If you wish to add more attributes, go to the **Insert** menu and click **Attributes**.

#### Check Part into Vault

Now the Part is complete, we now need to check it into the Vault. From the **File** menu, click **Check Into Vault**.

Select the chosen Vault Folder and enter a Check In Note for the Part.

Check Into Vault X
Name: R 0.25W 10K Vault Item ID:
Default Next Vault Revision Vault Revision:
Vault Folder           Vibrary Content\Parts\Vault Demo         Browse
Check In Note Checking in to the Vault Demo for the first time.
Close item after Check In Remove item from local library after Check In
OK Cancel

## **Update Vault Design with Vault Part**

#### ▶ Update Tutorial Design with R 0.25W 10K Resistor

We will now check the Tutorial Design out of the Vault and replace the now deemed 'obsolete' 10K Carbon Film resistor with the more modern Surface Mount version we just created.

From within the **Vault Manager**, open the Tutorial Design folder, select the Tutorial Schematic Design and click the **Check Out...** button. Add a note in the **Check Out Note** section and press **OK**.

Vault	Item ID		Туре	Folder	2	Name	Revision				Mo
vauit ⊷⊖ Design Content	207	50	Cabanadia			Name		_			IVIC
LED Display Project		010		Check Out			×				De
Tutorial Design				Vault Item:							Chec
Library Content				Type:	Schematic Design						
E Gotprints				10.	207						Cop
E- Parts				ID:	207						Unc
				Name:	Tutorial Design						Rep
E - CB Doc Symbols				Folder:	\Design Content\Tutor	del Desdar					
🗄 🍓 Schematic Doc Symbols		_		Folder:	pesign content (ruto	lai Design					
🗄 🍓 Schematic Symbols	Item Deta	ails: Re	efers To	Revision:	<latest revision=""></latest>		~				
🔄 Profiles	Item ID		Туре						Revision	Timestamp	
🔄 Technology Files	121	÷	Schematic I	Check Out M	lote				1.0	21-07-2017 16:55:41.917	
	128	80	Part	Replacing	Carbon Film resistor wit	h Surface Mount re	sistor	F 330	1.0	21-07-2017 16:56:59.689	
	130	80	Part						1.0	21-07-2017 16:57:00.370	
	132	80	Part						1.0	21-07-2017 16:57:00.962	
	136	-	Part	Settings				= 10K	1.0	21-07-2017 16:57:02.071	
				Open F							

Once checked out, you will notice the Tutorial Design has changed its background colour in the **Vault Manager**. This indicates the item has been checked out by a user. Changing the **Item Details:** to display the **Audit Trail** allows you to see the action that has occurred and by whom.

/ault Manager - Browser										>
wser Libraries to Vault Designs to Vault Au	dit Trail									
ault Folders: Refresh	Vault Item	s: 1						~	Search	
∋- 🔄 Vault	Item ID			Folder		tevision			Move.	
Design Content	207	Sc Sc	chematic Design	\Design Content\Tutorial De	sign Tutorial Design 1	.0			Delete.	
									Check Or	ut
🖶 🔄 Library Content									Copy Ou	.t.
Footprints     General Parts									Uncheck	k.
Vault Demo									Reports	
I - CB Doc Symbols										
Grant Schematic Doc Symbols     Grant Symbols	Item Deta	ls: Audit T	Trail	~ Revision	n: <latest revision=""></latest>		~			
	Item ID	Ту	rpe	Action Date	Timestamp	Revision	Action Type	Notes	User Nar	m
	207	Sc Sc	chematic Design	28-07-2017 13:01:52.861	27-07-2017 10:01:14.9	6 1.0	Checked out	Replacing Carbon Film resistor with Surface Mount resistor	admin	1
	207	De So	chematic Design	27-07-2017 10:01:15.351	27-07-2017 10:01:14.9	6 1.0	Moved	\Design Content\Tutorial Design	admin	7
	207	Do Sc	chematic Design	27-07-2017 10:01:14.916	27-07-2017 10:01:14 9	6 10	Added (with automatic referencing)	Checking in for the first time	admin	

## Replace Resistor in the Tutorial Design

Select the component R1 in your design, right click and select **Replace Part** from the Context Menu.

From within the Change Part dialog, click the **change** button and select the R 0.25W 10K Resistor we added to the Vault earlier in this tutorial. Click the **Replace** button to update the design with the new Part.

		3	
	Replace Part	×	
	Replace This:		
	Part: R 0.25W 5% MCF 10K	~	
	Desc: Carbon Film Resistor	Pins: 2	
	Family: R 0.25W		
	Footprint: <any footprint=""></any>		
	Replace With:		
	Part: R 0.25W 10K	Change Database	
	Desc: Thick Film Surface Mount Resistor	Pins:	
N N	Family: R 0.25W	Alternate	
	Footprint:	~	
Í			
Ţ	Which Components:	Replace Options:	
	Look for components to replace in:	□ Keep attribute styles & positions	
	○ <u>W</u> hole Design ○ Current <u>P</u> age	Keep local attribute values Replace nets assigned to ungated pins	
	Selection Only Current Block	Update predefined nets on symbol pins	
<u> </u>		Update name stem	D8
GND		Replace gate symbols with defaults     Show Report after Replace	1 2
	<u>Replace</u>		1N4148

We can now check the design back into the Vault. From the **File** menu, click **Check Into Vault**, add a **Check In Note** and press **OK** to finish.

	Check Into Vault	×
	Name: Tutorial Design Vault Item ID: 207	
	Default Next Vault Revision Vault Revision: 1.0	
	Vault Folder	
	\Design Content\Tutorial Design Browse	
<u> </u>	Check In Note	
$\Box$	Replaced Carbon Film resistor with Surface Mount resistor.	
	Close item after Check In	
	Remove item from local folder after Check In	
	OK Cancel	

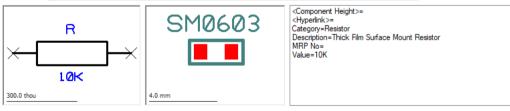
#### Investigating items within the Vault

Now that we have more than one revision of the Tutorial Design in the Vault, we can start to see an audit history of what actions have happened and by whom.

If we change the **Item Details**: to **Refers To**, and the Revision: to **<Latest Revision>**, we will see the design now contains the new resistor we have added to replace the 'obsolete' Carbon Film resistor.

ltem Detai	ils: Ref	fers To	~	Revision: <latest revision=""></latest>	<		
Item ID		Туре	Folder		Name	Revision	Timestamp
147		Schematic Technology	\Technology	Files	Default (White - Imperial)	1.0	21-07-2017 16:57:45.921
192	÷	Schematic Doc Symbol	\Library Content\Schematic Doc Symbols\Vault Demo		A4-Border-Landscape	1.0	26-07-2017 09:47:30.294
200		Schematic Profile	\Profiles		A4-Landscape	1.0	26-07-2017 09:55:32.113
211		Part	\Library Con	tent\Parts\Vault Demo	R 0.25W 10K	1.0	28-07-2017 12:52:24.095
					•		

#### Preview Of: Part 'R 0.25W 10K'



We now need to if the 'obsolete' Carbon Film resistor is used in any other designs and will then be able to update those designs with the new Surface Mount version accordingly.

#### Locate Carbon Film Resistor

To locate the Carbon Film Resistor we can change the Design **Revision**: to be **1.0**, as we know this design revision uses the old resistor. Select the Resistor in the Item Details section, right click and select **Show 'Where Used'** from the **Context Menu**.

all         Description         R 0.25W 5% MCF 10K         1.0         21-07-2017 16:5           136         an         Part         \Library Content\Parts\Vault Demo         R 0.25W 5% MCF 10K         1.0         21-07-2017 16:5           138         an         Part         \Library Content\Parts\Vault Demo         Show 'Where Used'         21-07-2017 16:5           147         Schematic Technology         Technology Files         Show 'Refers To'         21-07-2017 16:5	Item ID		Туре	Folder	Name	Revision	Timestamp
138       an       Part       Library Contenti/Parts\Vault Demo       Show 'Where Used'       21-07-2017 16:5'         147       Image: Schematic Technology       Technology Files       21-07-2017 16:5'         192       Image: Schematic Doc Symbol       Library Content\Schematic Doc Symbols\Vault Demo       A4-Border-Landscape       1.0         192       Image: Schematic Doc Symbol       Library Content\Schematic Doc Symbols\Vault Demo       A4-Border-Landscape       1.0         Preview Of:       Part 'R 0.25W 5% MCF 10K'       Category-Resistor/Carbon Film       26-07-2017 09:4'         Category-Resistor/Carbon Film       Description=Carbon Film Resistor       2-01/C	132		Part	\Library Content\Parts\Vault Demo	CD4017BE	1.0	21-07-2017 16:57:00.962
147       Image: Schematic Technology       Technology Files       Image: Schematic Technology       21-07-2017 16:51         192       Image: Schematic Doc Symbol       Library Content/Schematic Doc Symbols/Vault Demo       A4-Border-Landscape       1.0         Preview Of:       Part 'R 0.25W 5% MCF 10K'       Chart Family>=R 0.25W       Category=Resistor/Carbon Film         Description=Carbon Film Resistor       Description=Carbon Film Resistor	136	30	Part	\Library Content\Parts\Vault Demo	R 0.25W 5% MCF 10K	1.0	21-07-2017 16:57:02.071
147       Schematic Technology Viechnology Files       21-07-2017 16:5         192       Schematic Doc Symbol Ulibrary Content/Schematic Doc Symbols/Vault Demo   A4-Border-Landscape   1.0       26-07-2017 09:4         Preview Of:       Part 'R 0.25W 5% MCF 10K'       26-07-2017 09:4         Category=Resistor/Carbon Film       Description=Carbon Film Resistor	138		Part	\Library Content\Parts\Vault Demo			21-07-2017 16:57:02.868
192       Image: Schematic Doc Symbol       \Library Content\Schematic Doc Symbols\Vault Demo   A4-Border-Landscape   1.0       26-07-2017 09:41         Preview Of:       Part 'R 0.25W 5% MCF 10K'        26-07-2017 09:41         Category-Resistor/Carbon Film         26-07-2017 09:41         Description=Carbon Film Resistor	147		Schematic Technology	\Technology Files	Show 'Refers To'		21-07-2017 16:57:45.921
Preview Of: Part 'R 0.25W 5% MCF 10K'  Carbon Film Category-Resistor/Carbon Film Description=Carbon Film Resistor Description=Carbon Film Resistor	192	÷	Schematic Doc Symbol	\Library Content\Schematic Doc Symbols\Vault Demo	A4-Border-Landscape	1.0	26-07-2017 09:47:30.294

The Vault Manager will now 'jump' to the **Item Details** table for the Resistor Part and list all the designs it is used in.

We can see that the Carbon Film resistor is also used in the LED Display Schematic and PCB Designs. These would need to be checked out and updated before the next iteration is produced.

157 📴 Schematic Desig 158 🔂 PCB Design	n \Design Content\LED Display Project			
158 Transformer PCB Design				17 17:08:14.212
	\Design Content\LED Display Projec	Show 'Refers	o' 7-20	17 17:11:22.736
207 🕞 Schematic Desig	n \Design Content\Tutorial Design	Tutorial Design 1.0	27-07-20	17 10:01:14.916
R X[	→ -×		ry=Resistor/Carbor tion=Carbon Film F	

## Creating a Search in the Vault

We will now look to create a Search from within the Pulsonix Vault, these can be saved for easy access in future.

From within the Vault Manager, Click the Search... button.

You will now be within the **Search in Vault** dialog, change the **Find:** to be **Schematic Design** and press the **Apply** button.

t Folders: Refresh	Vault Item	s: 2							✓ Search
ault 🗧	Item ID			Folder	Name	Revision			Move
🖨 🍓 Design Content	157	Ba		\Design Content\LED Display Project					Delete
	207	₿ <del>]</del> le	Schematic Design	Design Content/Tutorial Design	Tutorial Design	1.3			Check O
E G Library Content	Search i	n Vault	*					×	Copy Or
Footprints     Arts									Unched
Vault Demo	Finds	Sche	ematic Design	✓ Clear Load.	. Save	Sav	ve As	~	Report
I									
🗄 🔄 Schematic Doc Symbols		Item	ID	$\sim$ Equal to $\sim$					
B Schematic Symbols		Item	ID	Equal to					
		Item	ID	Equal to					
		Chec	ked Out By: <any< td=""><td>User&gt; ~</td><td>Show Check</td><td>ed-out State</td><td>•</td><td></td><td></td></any<>	User> ~	Show Check	ed-out State	•		

Now click the **Save As** button, and type **Vault Schematic Designs** for the name of your search and press **Save**.

	😽 Save As							×
	← → ✓ ↑ ≪ Pulsonix Vault			~ Ū	Search Checked Out Files		Q	
	Organize 🔻 Ne	w fold	er					?
	🖈 Quick access	^	Name	^		Date modified	Туре	
	🔜 Desktop 🖈			No items match your search.				
	👆 Downloads	*						
	Documents	*						
	Pictures	*						
	iCloud Drive	*						
		~	<					>
$\Box$	File <u>n</u> ame:	Vault	Schematic Designs					~
	Save as <u>t</u> ype:	Vault	Find Settings (*.vfs)					$\sim$
	∧ Hide Folders					<u>S</u> ave	Cancel	

Going back to the Vault Manager, you will now see the Search is available from the drop-down arrow for ease of access and use in the future.



Note: You could create quick searches to show All items currently checked out of the Vault, or All items checked of the Vault by user x.

## Seeing the Audit History within the Vault

When you wish to see recent activity within the Vault, you can use the Audit Trail tab to see what actions have occurred, when they occurred and by whom.

#### Create Audit Trail Query

Enable the **Filter By:** section, tick the **Latest:** box and enter **10** in the value section. Now select the **entries** radial button.

Tick the **Where:** box and select **Action Type**, Equal to and Added in the drop-down boxes.

Now press the **Apply** button to see the results of your query.

🔳 Vault Manager - Audit Trail				
Browser Libraries to Vault Designs to Vault Audit Trail				
☑ Filter By:				
☑ Latest: 10				
Where: Action Type				
Apply				
Clos		Help		
Clos	-	Help		

From this search, we can see two items have been added to the Vault, the time and date they were added, by whom, what the item was and the note that accompanied the check in.

📧 Vault Manager - Audit Trail		$\times$
Browser Libraries to Vault Designs to Vault Audit Trail		
28-07-2017 12:52:24.095 211 28-07-2017 10:56:01:580 210 Date & Time Item Item Action Number	 	

This completes our brief run-through of the Pulsonix Vault.

### **Additional Information**

There are more aspects to Pulsonix Vault which are not covered in this document. Further information can be obtained in the Online Help using the <F1> key at any time, or from the Pulsonix Users Guide supplied as a PDF file under the **Help** menu and **Online Manuals** or on our web site.

While evaluating Pulsonix or once you've purchased it, technical support is provided in the form of telephone support from our office or from your local sales office, or by email.

Head office support +44 (0)1684 296 570

Head office email support <a>support@pulsonix.com</a>

Web site www.pulsonix.com