

visEDATA

Tutorial

Included in this guide:

A complete guided tour of the entire visEDATA application including:

- Introduction to visEDATA (All)
- User Interface & Installation
- Project Management (Designer)
- Viewing Project Data
- Project Reporting (Proj Mngr)
- Project Collaboration & Reporting (All)



This document is for information and instruction purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in written agreements between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

RESTRICTED RIGHTS LEGEND 03/97

U.S. Government Restricted Rights. The SOFTWARE and documentation have been developed entirely at private expense and are commercial computer software provided with restricted rights. Use, duplication or disclosure by the U.S. Government or a U.S. Government subcontractor is subject to the restrictions set forth in the license agreement provided with the software pursuant to DFARS 227.7202-3(a) or as set forth in subparagraph (c)(1) and (2) of the Commercial Computer Software – Restricted Rights clause at FAR 52.227-19, as applicable.

**© 2010-2012 Mentor Graphics Corporation
All Rights Reserved**

Contractor/manufacturer is:

Mentor Graphics Corporation

8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.

Telephone: 503.685.7000 Toll-Free Telephone: 800.592.2210

Website: www.mentor.com SupportNet: <http://supportnet.mentor.com>

Send Feedback on Documentation: http://supportnet.mentor.com/doc_feedback_form

TRADEMARKS: The trademarks, logos and service marks (Marks) used herein are the property of Mentor Graphics Corporation or other third parties. No one is permitted to use these Marks without the prior written consent of Mentor Graphics or the respective third-party owner. The use herein of a third-party Mark is not an attempt to indicate Mentor Graphics as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A current list of Mentor Graphics' trademarks may be viewed at: www.mentor.com/trademarks.

Contents

INTRODUCTION TO VISEDATA	6
AN INTEGRATED ARCHIVE MANAGEMENT SOLUTION.....	7
PRIMARY USER ROLES	7
SIMPLIFIED DATA MANAGEMENT WITH VISEDATA.....	8
PURPOSE OF THIS GUIDE.....	9
WORKING TOGETHER.....	9
FURTHER EXPLORATION	9
VISEDATA PROCESS OVERVIEW.....	10
MODULE 1: USER INTERFACE & INSTALLATION.....	12
ABOUT THIS TUTORIAL	13
INSTALLING THE SOFTWARE	13
INTRODUCTION TO THE USER INTERFACE	14
<i>Main Window</i>	14
<i>Archive Navigator Window</i>	15
VISEDATA TOOLBARS.....	18
<i>visEDATA Main Toolbar</i>	18
<i>Working Folder Toolbar</i>	18
<i>PCB Toolbar</i>	18
<i>Schematic Toolbar</i>	18
ORGANIZING YOUR VAULT	19
<i>LAB: Setting visEDATA Options</i>	19
<i>LAB: Creating a New Vault</i>	20
<i>LAB: Setting Archive Navigator Options/Filters</i>	21
CREATE A FOLDER STRUCTURE TO MANAGE PROJECTS.....	22
<i>LAB: Create Folders</i>	22
MODULE 2: VISEDATA PROJECT MANAGEMENT	24
ARCHIVING A PROJECT.....	25
<i>LAB: Adding Project Containers</i>	25
<i>LAB: Archiving a Project for the first time</i>	27
<i>LAB: Viewing the Project Vault Contents</i>	29
	3

<i>LAB: Managing archive templates</i>	30
EDITING A PROJECT	33
<i>LAB: Making a Schematic Change (Invoking DxDesigner)</i>	33
<i>LAB: Making a Layout Change (Invoking Pads Layout)</i>	33
CREATING A SUBSEQUENT ARCHIVE	34
<i>LAB: Sequential Archive</i>	34
<i>LAB: Retrieving a Project from the Vault (Branching)</i>	35
PROJECT/OBJECT SEARCHING (ALL)	37
<i>LAB: Project Searching</i>	37

MODULE 3: VIEWING PROJECT GRAPHICAL DATA (ALL)..... 39

GRAPHIC NAVIGATION BASICS	39
<i>LAB: Schematic Graphics</i>	40
<i>LAB: Layout Graphics</i>	42
COMMON GRAPHICAL ACTIONS	44
<i>Viewing Design Item Information</i>	44
<i>Redrawing the Display</i>	44
<i>Controlling Layer Display</i>	44
<i>Toggling Component and Pin Net Labels</i>	44
<i>LAB: Layout Reports</i>	46
CROSS-PROBING	49
<i>LAB: Selecting the Schematic and Layout Previews</i>	49
<i>LAB: Activating the Cross-Probe Function</i>	50
<i>LAB: Cross-Probing Between Views</i>	50

MODULE 4: PROJECT REPORTING (PROJ MNGR)..... 53

NETLIST REPORT	53
<i>LAB: Schematic to Layout Netlist Data Report</i>	54
.....	55
DATA REPORT	55
<i>LAB: Schematic Comparison Data Report</i>	55
<i>LAB: Layout Comparison Data Report</i>	56
GRAPHICAL REPORT	58
<i>LAB: Comparing Two PCB/Layout Views</i>	58

MODULE 5: COLLABORATION/REDLINING (ALL) 62

<i>LAB: Creating Topics & Issues</i>	<i>63</i>
<i>LAB: Creating Markups & Views</i>	<i>64</i>
<i>LAB: Design Elements and Redline/Markup.....</i>	<i>68</i>
<i>LAB: Saving the Redline to the Vault.....</i>	<i>69</i>
CONCLUSION	71

Introduction to visEDATA

In this section:

- Introduction - Design Archive Management
- visEDATA Process Overview
- Flow Graphic
- visEDATA Highlights

Welcome to the visEDATA Tutorial. Our goal is to introduce you to a design archive management solution and assist you in understanding how to apply simple archive usecases to your design process.

visEDATA is specifically designed to provide an integrated and efficient method of automatically creating, viewing, reviewing, comparing, and restoring project backups/archives. It is a focused solution for individual users or small workgroups that want to easily create, manage, & review their project design backups in a database.

We invite you to use this guide to fully explore both the power and ease of use of visEDATA.

An Integrated Archive Management Solution

During a typical design cycle there is often the need to create multiple backups of the project data and then retrieve that data later for review &/or updates or modifications. Additionally, users like to create a snapshot of the design to perform various what if scenarios such as:

- Changing constraints and running simulation analyses
- Analyze different placement/routing strategies
- Design reviews – Preserve record of important design milestones
- Personal security – What happens if the system/network crashes? How do I retrieve a known good state?
- Release to manufacturing – The design is complete

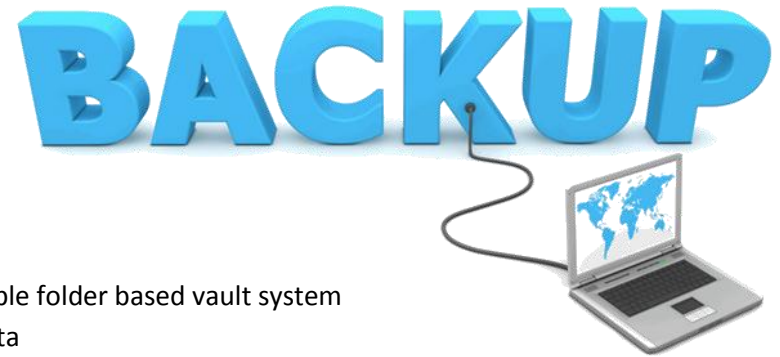
With visEDATA you are able to save time & costs by providing an integrated & efficient method of automatically creating & restoring Archives. This allows the engineering community to accurately describe archive purpose for quick & easy identification and retrieval. This seamless environment allows users to track & manage all archives associated with the Mentor Graphics design/project. By providing these features, visEDATA can minimize potential costly rework(s) by clearly identifying archive content.

**Why do we
need backups?**

**Who can benefit from
Archive Management?**

Primary User Roles

The value of data management is that it can provide information to a multitude of users with different roles within your design community. visEDATA specifically provides value to Designers, Project Managers, and general consumers of PCB design data such as Manufacturing Engineers.



For Designers, visEDATA specifically provides:

- Archive searching - search for existing data to create new projects from
- Schematic & PCB Comparison - understand complex engineering changes during the work-in-progress design process

For Project Managers, visEDATA provides:

- Project Management - create templates and organize & index projects in a simple folder based vault system
- Project Reporting - generate progress and comparison reports of the design data

For the General PCB Consumers:

- Simple Interface - access complex design data using a simple graphical interface
- Collaboration & Redlining - graphical preview with redlining to communicate design intent/changes

Simplified Data Management with visEDATA

visDATA was created to reduce the overhead of manual backup creation by providing a simple vault-based storage mechanism. It increases productivity by efficiently creating, indexing, and restoring archives within a vault that is transparent to the user. The user has the ability to quickly and easily search the contents of the vault based on archive names and user-defined descriptions. These elements can be restored or compared against, which significantly reduces the risk of error when reworking or starting a new project from existing design data.

How does visEDATA simplify data management?

Upon creation of an archive, visEDATA will automatically generate the Schematic and Layout views based off of Mentor Graphics' powerful collaboration format. This format provides robust graphical preview, redline and markup, and report of project data and reduces costs by eliminating the need to consume licenses of the authoring tools. This collaborative environment can provide a simple interface to a design community that may not be familiar with the engineering tools and ensures that exact design review intent is accurately captured, available, and traceable during later reviews.

Purpose of This Guide

This Tutorial will introduce you to the major features and capabilities of this application and help you to understand how it works together with your design environment to unify and simplify your design process. We walk through the entire workflow from vault creation, configuration, archive template creation, project creation & archive, collaboration, and reporting providing tips on how to manage design archives and collaboration more efficiently. The goal is allow you to work with your own PCB project data and understand how visEDATA can bring value to your environment immediately

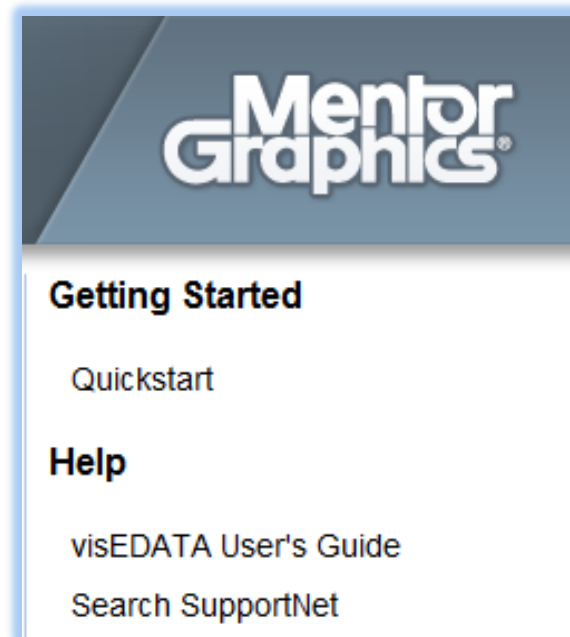
Working Together

This attention to workflow means that we will also show you how to use the application together with real-world design examples. Along the way, we also introduce you to many of the great new features of visEDATA, highlighting some of the more interesting techniques with steps and lots of illustrations. To see which features a section covers, scan the ***In this section*** list at the beginning of each section.

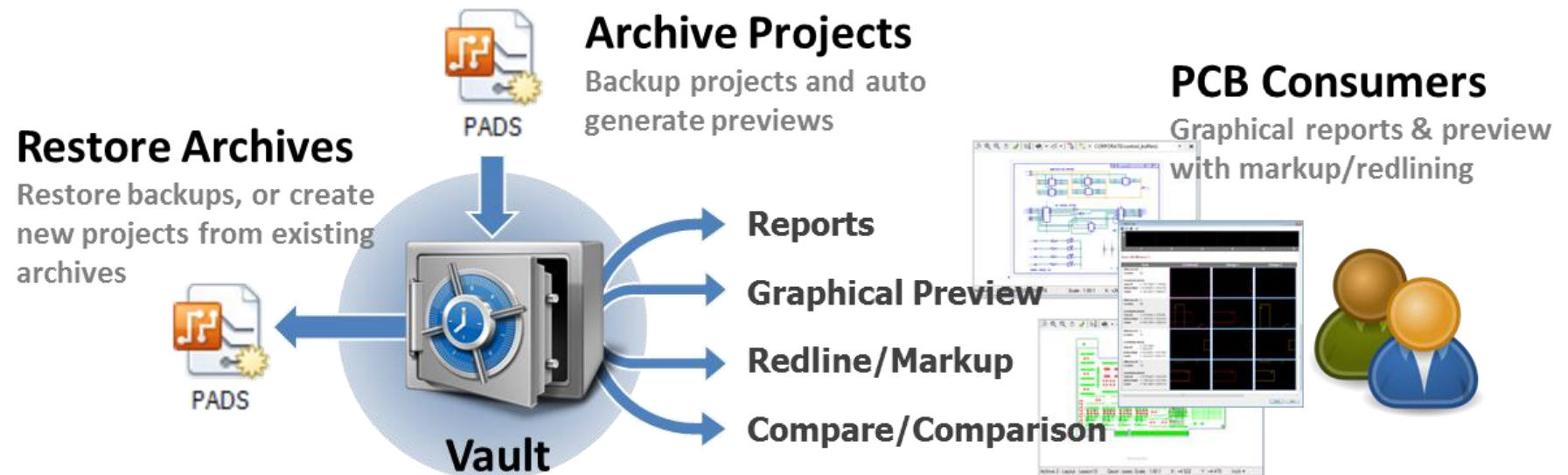
Further Exploration

Finally, remember that this document is a Tutorial, not a comprehensive user guide. Your most complete source of detailed feature information is the Help in each application. In addition, take time to examine the resource information accessible through the Info Hub link within each application. There you'll find information on a variety of **PADS** resources that will help you get started, get informed, and get inspired while using **visEDATA** with the **PADS ES Suite**.

We hope this Tutorial helps you see some of the exciting possibilities available to you with visEDATA. Enjoy!



visEDATA Process Overview



End of section. This page intentionally left blank.

User Interface & Installation

In this section:

- About this Tutorial
- Installing the software
- Understanding the Interface
- Creating a vault
- Creating project folders
- Setting vault Options

Installing visEDATA will allow you to manage your own vault or access a pre-existing vault. Within a vault you can create multiple folders to manage your project structure and set basic vault options such as colors and fonts.

In this module you will be introduced to the user interface and access a vault for the first time.

About This Tutorial

This Tutorial will allow someone with limited to no data management experience the opportunity to review several features and capabilities of visEDATA. Keep in mind this presentation is a snapshot of the full range of features visEDATA provides.

There is a set of tutorial design data installed with visEDATA, it is located here:
c:\MentorGraphics\visEDATA_1.0\SDD_HOME\visEDATA_Tutorial\visEDATA_Tutorial.zip

Please unzip the file to: C:**visEDATA_Tutorial**

The Tutorial will utilize a directory on your computer **C:\visEDATA_vault** (which you will create) along with existing design data that can be copied from the C:**visEDATA_Tutorial** directory or from your own personal PADS designs. We are able to do this because of the simplicity of the visEDATA environment and minimal setup & configuration requirements.



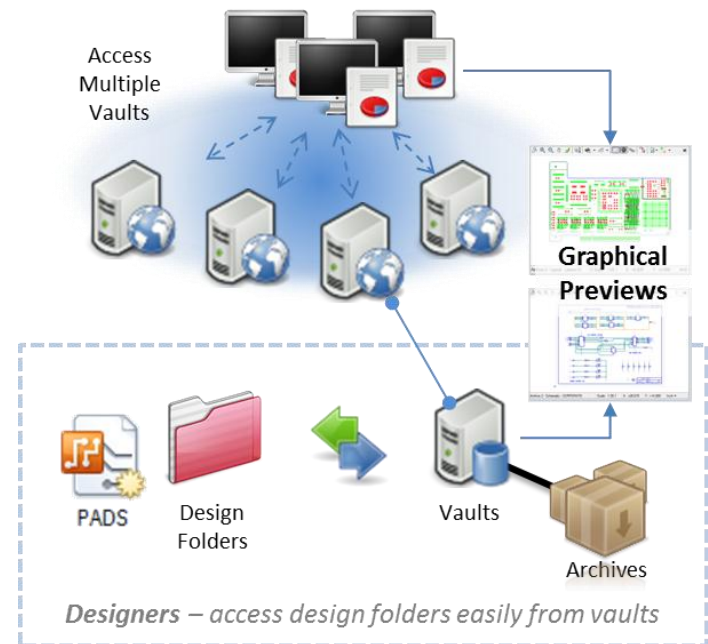
If you are going to use your own PCB data, you should make a copy of this information so that you can fully experience the data management software use cases.

This Tutorial is intended to be used with the **PADS PCB** software, this particular Tutorial has Lessons dedicated to interfacing with PADS Layout PCB software.

Installing the software

The visEDATA installation process is as straightforward as the user interface. Upon installing the application on your system, you will be able to start creating vaults, or optionally select an existing visEDATA vault on your file system. When you choose to create a vault (this Tutorial), the system will automatically install and configure an environment that will manage the archived design data, index the archive name & description, and provide access to the project working folders.

Collaborators - do not require design tools



Introduction to the User Interface

Main Window

Vault Navigator

Layout View

Redline Toolbars

Discussion Windows

Working Folder

Schematic View

HTML Report

1

2

3

4

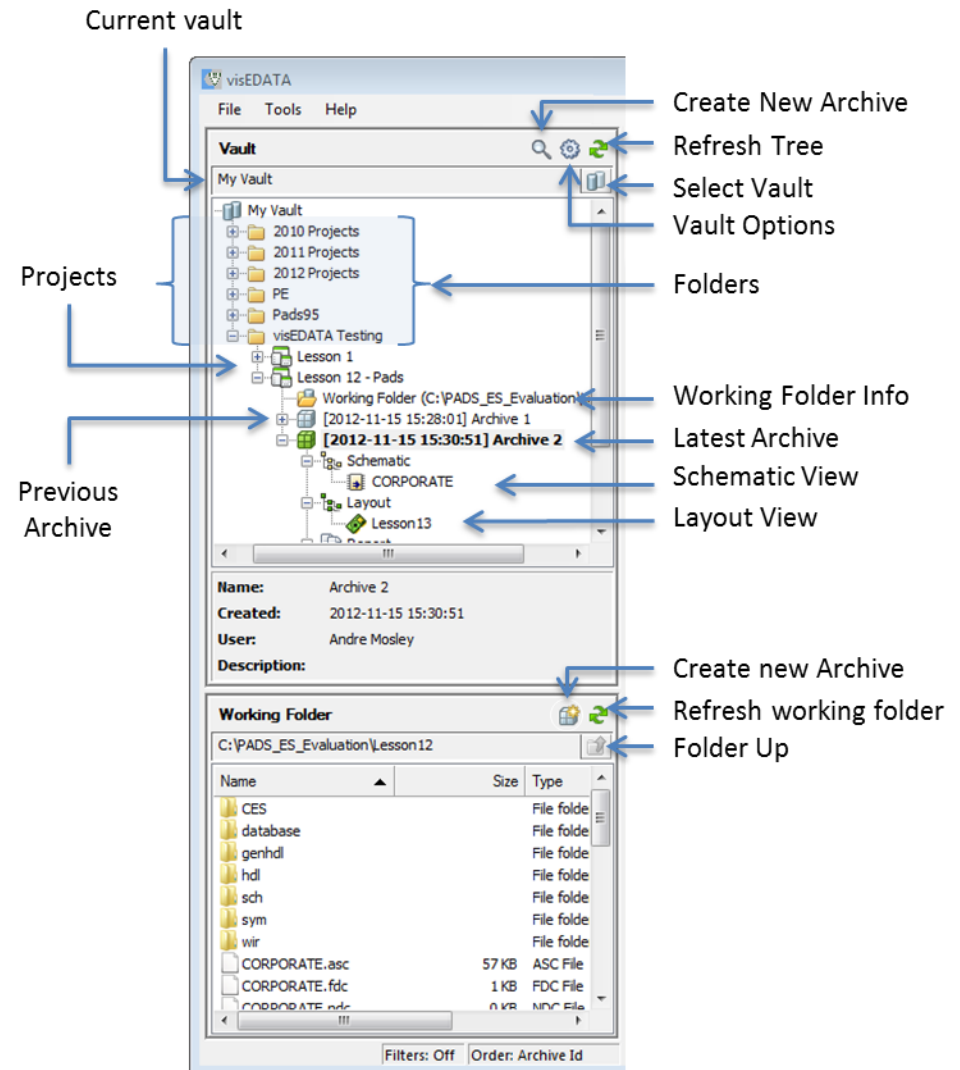
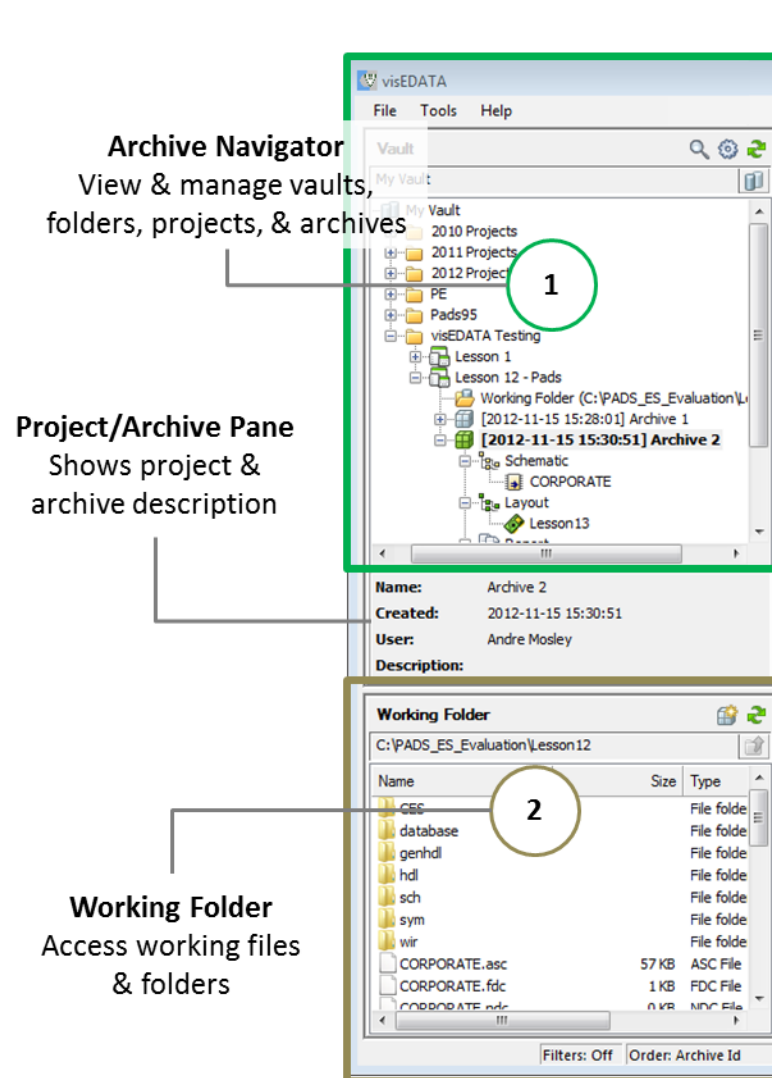
5

6

The screenshot displays the visEDATA software interface. On the left, the 'Vault Navigator' shows a tree structure of projects and folders, with a 'Working Folder' pane below it. The central area is divided into two views: 'Layout View' (top) showing a PCB layout and 'Schematic View' (bottom) showing a circuit diagram. To the right, there are 'Redline Toolbars' and 'Discussion Windows'. A 'Data Compare' window is also open, showing a table of differences between two designs.

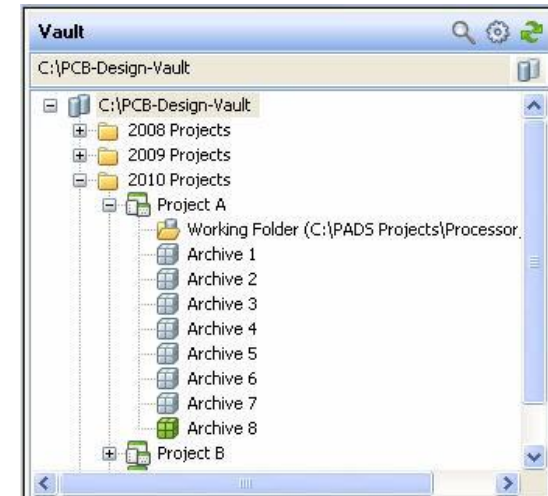
Compare	Design 1	Design 2	Design 2 Number of Differences	Report
PCB Component Differences	123	126	3	View
Component Attribute Differences	0	0	0	N/A
Unique Test Points	76	1229	1153	View
Via Differences	20	15716	15696	View
Drill Hole Differences	696	2	2	View
Unique Traces	0	70	70	View
Layer Differences	208	1306	1098	View
Unique Net Names	2086	947	1139	View
Net Pin Differences	0	0	0	N/A
Unique Net Pins	0	0	0	N/A
Placed Test Probe Differences	0	0	0	N/A
UnPlaced Test Probe Differences	0	0	0	N/A
Unique Tooling Holes	0	0	0	N/A
Fiducial Differences	0	0	0	N/A

Archive Navigator Window



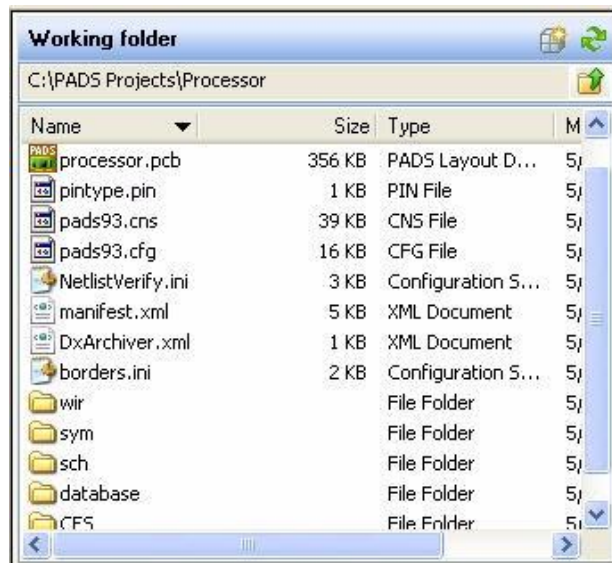
Use the visEDATA Vault view to:

- Open schematic and layout archives of a design for viewing and redlining.
- View design and collaboration data information for layout and schematic.
- Compare netlist, graphical, and data information between archives of a design.
- Organize archives into three levels of storage—vaults, folders and projects.
- Store successive numbered archives of a project with identifying data, including a text description.
- Restore any project archive from the vault to a working directory, make changes, and store the modified project as a new archive.
- Create archive templates to simplify customized archiving.



1. Archive Navigator

Use this window to view and manage vaults, folders, projects and archives, and to open schematics and layouts for collaboration. The vault allows you to see a combination of Work-In-Process project locations (Working Folder) along with their available archives which include the associated schematic, layout, and reports. For simplicity, the latest archive is colored in **Green** and bold letters (as per latest installs)

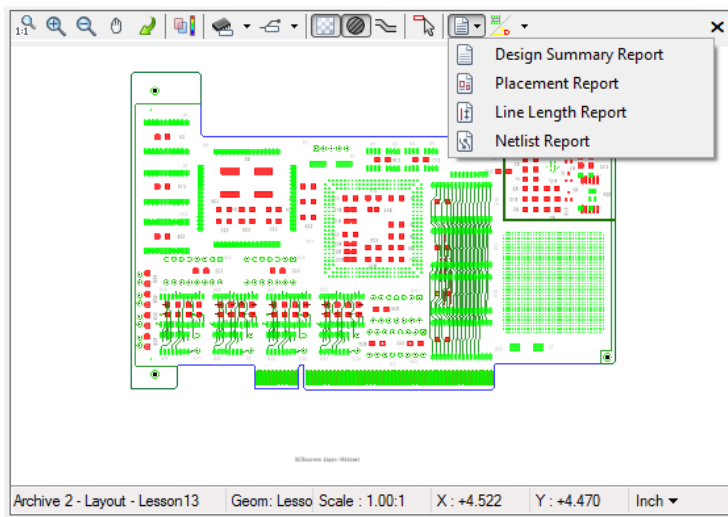


2. Working Folder

Use this window to view and work with the files and folders in the current working folder for the selected project. As an item is selected in the Vault View, this window display is updated to reflect the files and folders within the associated project's working folder. This view allows immediate access to the project as if you were to navigate to it on your filesystem. This view allows you to edit the most recent PCB design objects directly.

3. PCB Graphic View

Use this window to view the PCB layout and add graphical redline notes. You can zoom the view and toggle the display of various board items, the draw priority is automatically set to give the most important items the highest level of visibility. Additionally within this view, you can generate several PCB reports and initiate redlining of the Layout elements.



4. Schematic Graphic View

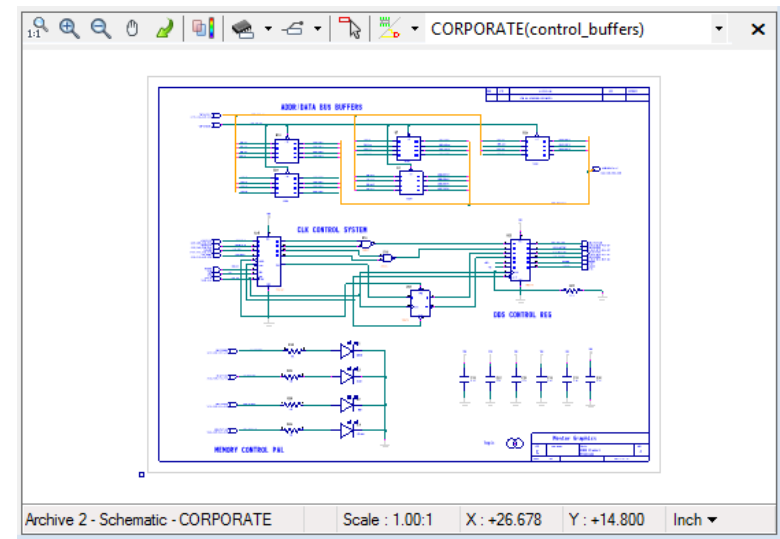
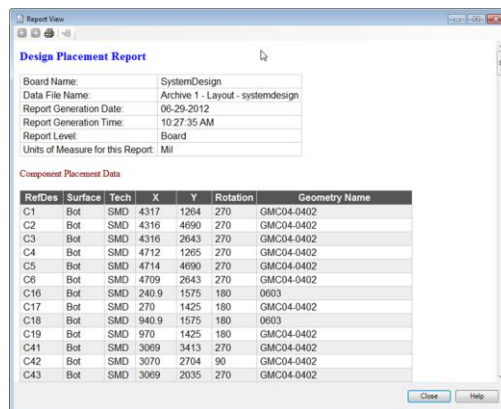
Use this window to view the schematic and add graphical redline notes. You can zoom the view and toggle the display of various board features, the draw priority is automatically set to give the most important items the highest level of visibility. Additionally within this view, you can select other schematic sheets and initiate redlining of the Layout elements.

5. Discussion Windows

Use these windows to define redline and collaboration data, discussions, topics, and issues. Post-it notes can be added to the graphical display and associated with one or more design items.

6. HTML Reports

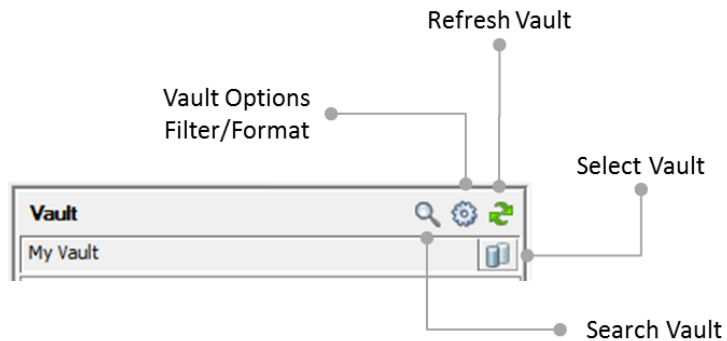
visEDATA allows you to generate HTML reports from the Layout Graphic and also HTML compare reports from different schematics and layout archives. These reports will be displayed in an external HTML window and the report can be printed or saved to the vault alongside the respective archive. Some reports are multi-level and contain links.



visEDATA Toolbars

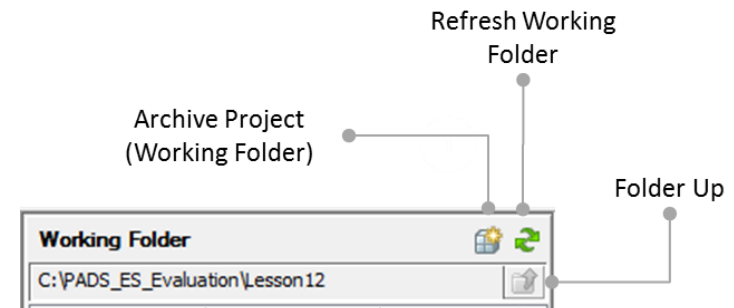
visEDATA Main Toolbar

Available from the Vault Navigator



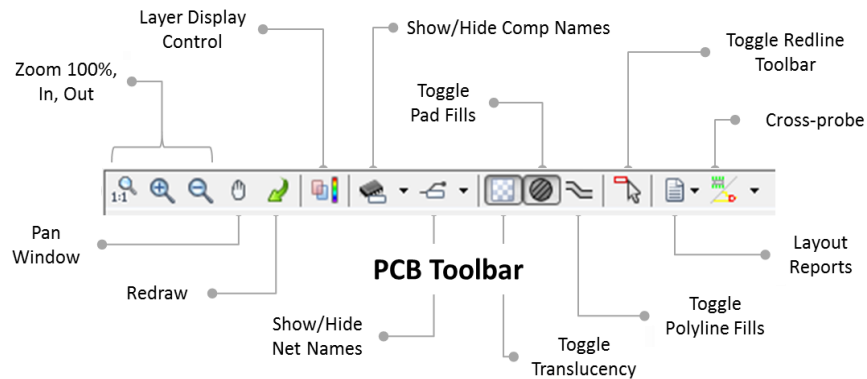
Working Folder Toolbar

Available from the Working Folder



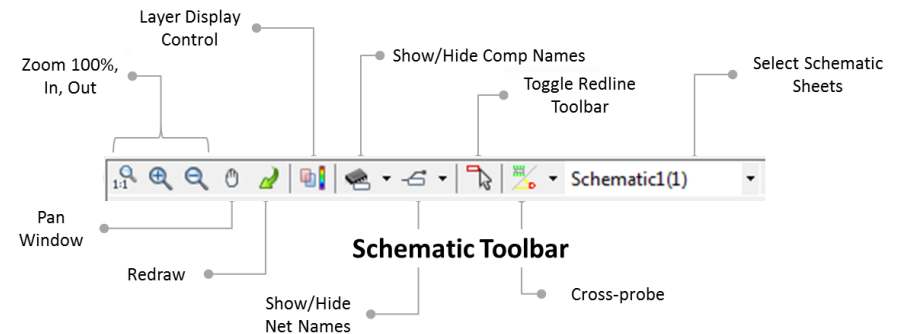
PCB Toolbar

Available when a Layout Graphic window is displayed



Schematic Toolbar

Available when a Schematic Graphic window is displayed



Organizing Your Vault

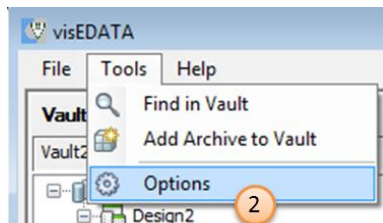
You can organize your vault storage to suit your situation. If you have only a few projects, you can store them at the top level of the vault. If you have a larger number of projects, you may want to create folders within your vault to organize them. If you have a multitude of projects, you may want to create multiple vaults, each containing multiple folders containing multiple projects.

To begin archiving a PCB project you must:

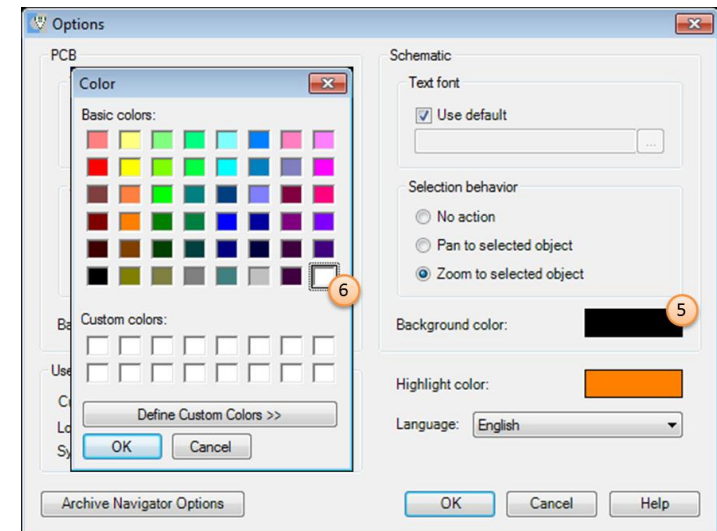
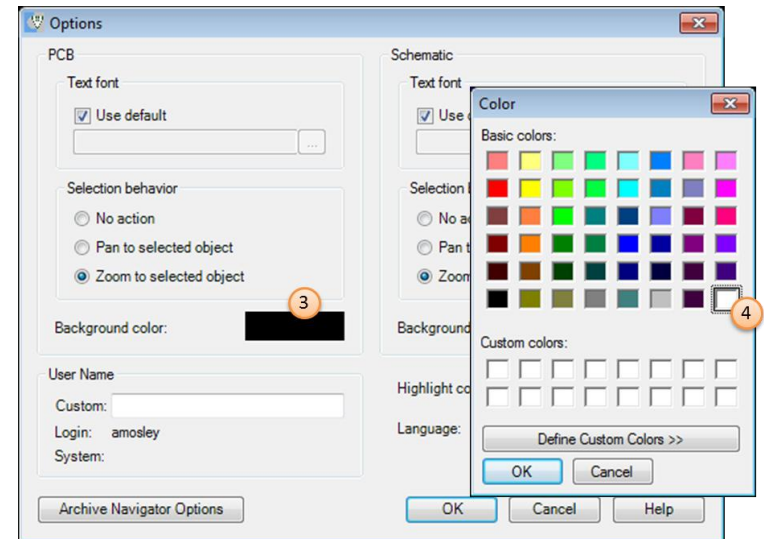
- Create a vault
- Create an empty Project container in the vault
- Vault a project
- Accessing the project & working folder from the vault
- Adding redline & markup

LAB: Setting visEDATA Options

1. Invoke visEDATA from the start menu **Start > All Programs > Mentor Graphics SDD > visEDATA 1.0 > visEDATA 1.0**
2. Select **Tools > Options** from the menu




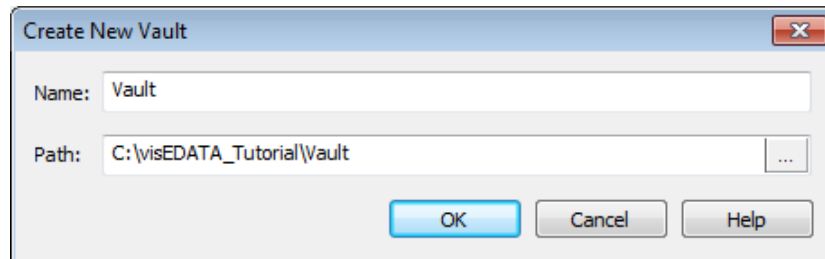
3. Select the PCB Background Color **Black**
4. Change it to **White** and click **OK**
5. Select the Schematic Background Color **Black**
6. Change it to **White** and click **OK**
7. Click **OK** until you close the dialogs



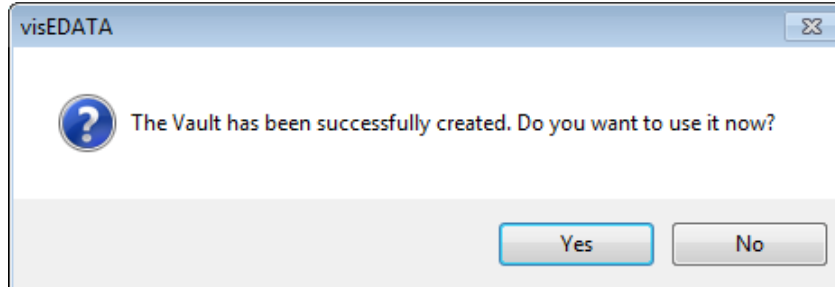
LAB: Creating a New Vault

Tip: Design projects that you want to collaborate on must first be stored as an archive in a vault.

1. In the Vault View (or Startup screen), click the **Select Vault** button ()
2. In the Select Vault dialog box, click **Create New Vault**. 
3. In the Create New Vault dialog, enter the name **visEDATA Tutorial** and set the path to the **C:\visEDATA_Tutorial\Vault** folder, note a new Vault can only be created in an empty folder.



4. and click **OK**.
5. Click **Yes** when asked *Do you want to use it now?*



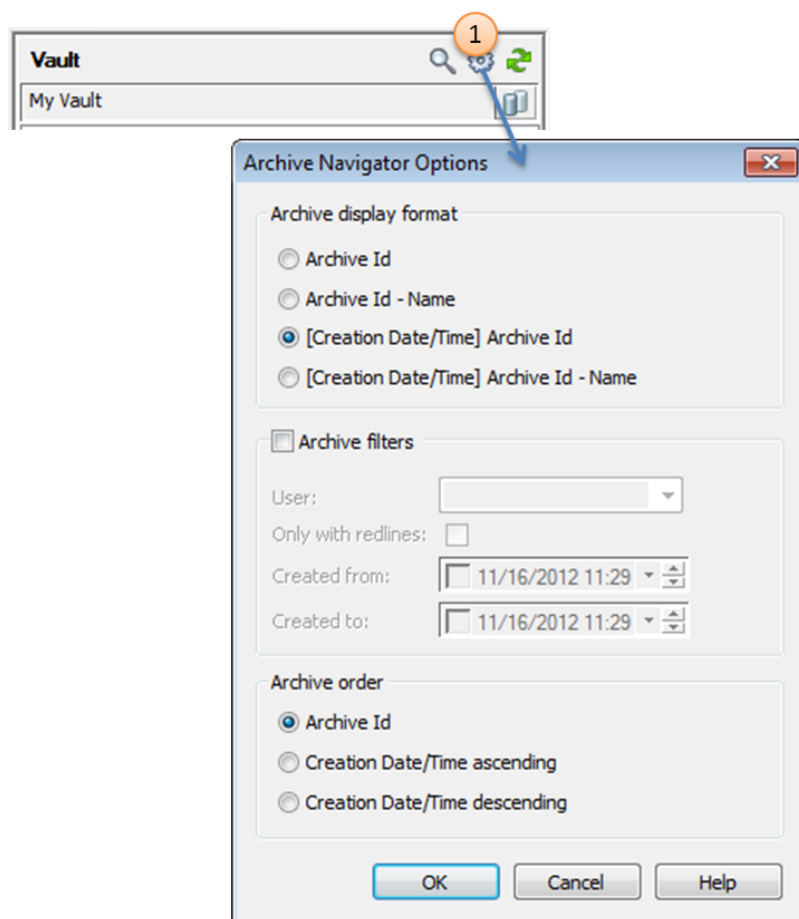
6. Click **OK** to access the vault

NOTE: Selecting an Existing Vault - You can optionally select an existing vault if it has been created by another visEDATA user. To select an existing vault you can simply select the vault container to access it. The name will automatically be displayed based on the vault settings.

LAB: Setting Archive Navigator Options/Filters

Tip: Now that a vault has been created, you can manage the options of the archive display format, archive filters, and archive order.

1. Select the options button (⚙️)
2. Notice the options for archive display, filters, and archive order as displayed in the associated table
3. Click **OK**



Name	Description
Archive display format	Specifies how archives should be listed in the vault view tree, as follows: <ul style="list-style-type: none">• Select Archive Id to display only the assigned archive ID (Archive 1, Archive 2, ...).• Select Archive Id - Name to display the archive ID and the optional archive Name.• Select [Creation Date/Time] Archive Id to display the archive creation date & time and the archive ID.• Select [Creation Date/Time] Archive Id - Name to display the archive creation date & time, the archive ID, and the optional archive Name.
Archive filters	Specifies which archives should be displayed in the Vault view, as follows: <ul style="list-style-type: none">• User drop-down list—Select a user to display all archives whose User attribute contains that user name. If no user is selected, archives with any user name are returned.• Only with redlines—Check to display only archives that have redlines.• Created from—Select this checkbox and specify a date/time to display all archives created after the specified date/time.• Created to —Select this checkbox and specify a date/time to display all archives created up to the specified date/time.
Archive order	Specifies the order in which archives are displayed in the Vault view, as follows: <ul style="list-style-type: none">• Archive Id—Orders by assigned archive ID (Archive 1, Archive 2, ...).• Creation Date/Time ascending—Orders by creation date/time, most recent at the bottom.• Creation Date/Time descending—Orders by creation date/time, most recent at the top.

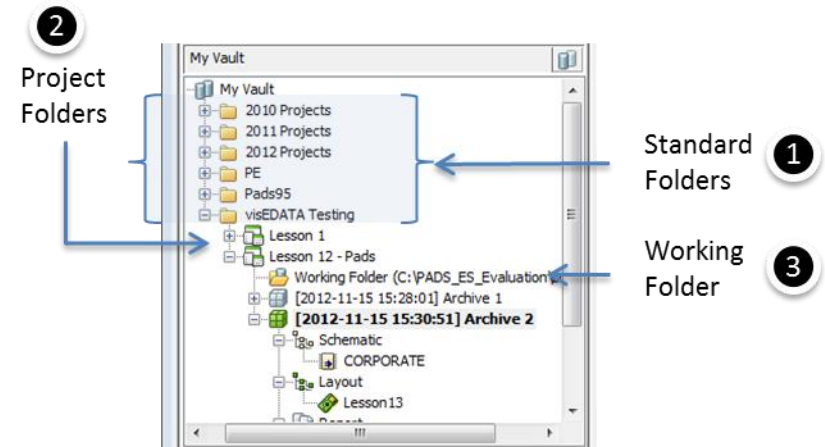
Create a Folder Structure to Manage Projects

There are three types of folders that are displayed and managed within visEDATA:

- (1) - Standard Folder (can contain Project Folders or other Containers)
- (2) - Project Folder (displays the project name in the vault view), and
- (3) - Working Folder (folder on disc of the active project)

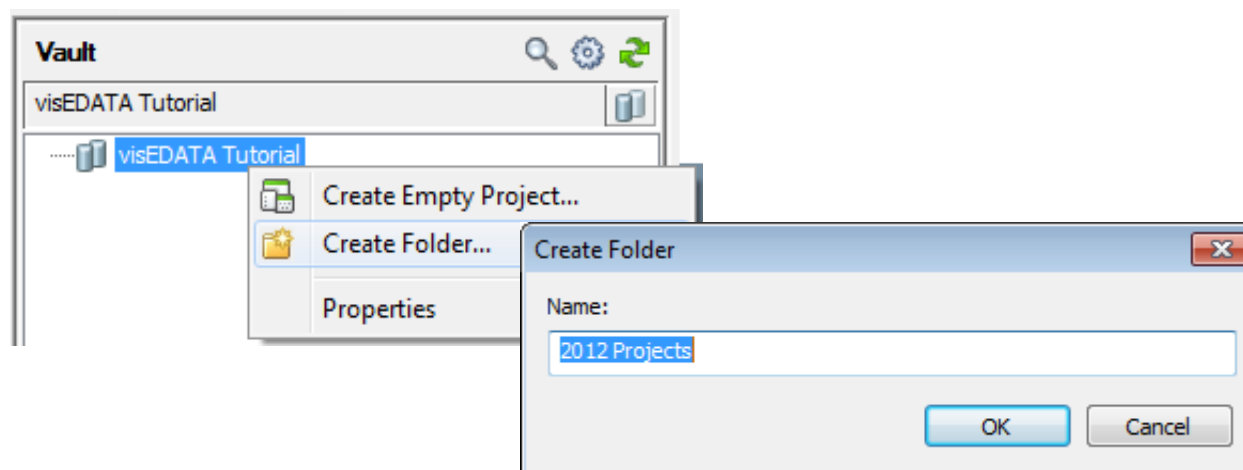
Prior to managing Project Folders and Working Folders, a folder structure must be created to partition your information.

Tip: Creating project folders allows you to partition the different projects that you want to manage with visEDATA. You can create as many multi-level folders as you like.

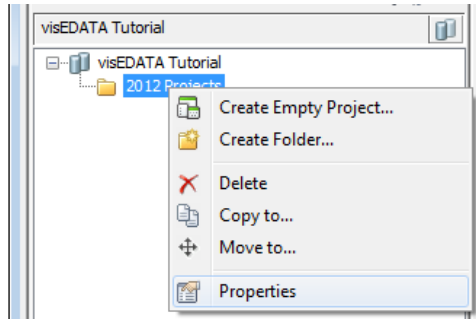


LAB: Create Folders

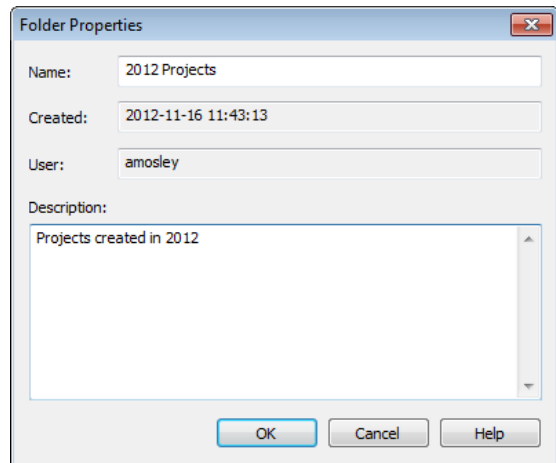
1. Select the **visEDATA Tutorial** Vault in the Vault Navigator
2. Click **RMB > Create Folder ...**
3. Enter **2012 Projects** for the Name and click **OK**



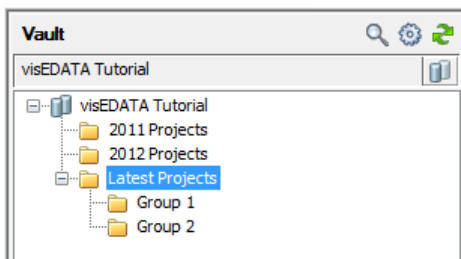
4. Select the new folder and click **RMB > Properties**



5. Set the description of the folder to be “**Projects created in 2012**”



6. Continue creating folders with the **RMB > Create Folder ...** command until your vault looks like the following image



visEDATA Project Management

In this section:

- Archiving a project with optional templates
- Viewing vault contents
- Editing a project
- Creating subsequent archives
- Retrieving a project from the vault
- Searching for projects in the vault
- Project Reporting & Comparison

Managing project data with visEDATA is as simple as creating a project definition and then archiving the design with optional templates. Templates allow you to pre-configure the naming convention, description, schematic .prj, pcb file(s), and additional folders/files to archive.

Once a project is archived, you can continue to make design edits and create subsequent archives. This allows the user to report on the design and even view design changes from different archives.

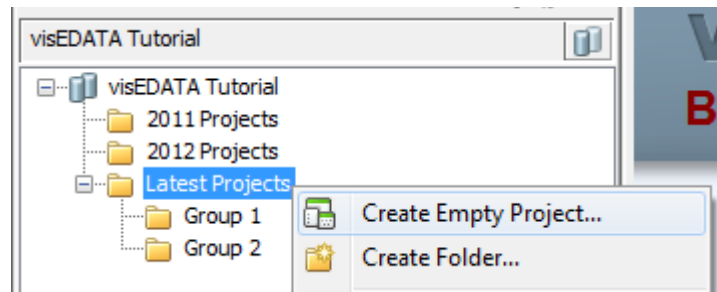
Archiving a Project

In order to work with a project in visEDATA, you must first create the project definition and set the working folder of the project. Additionally, you can create archive templates for your specific project to define rules on how your specific project should be bundled.

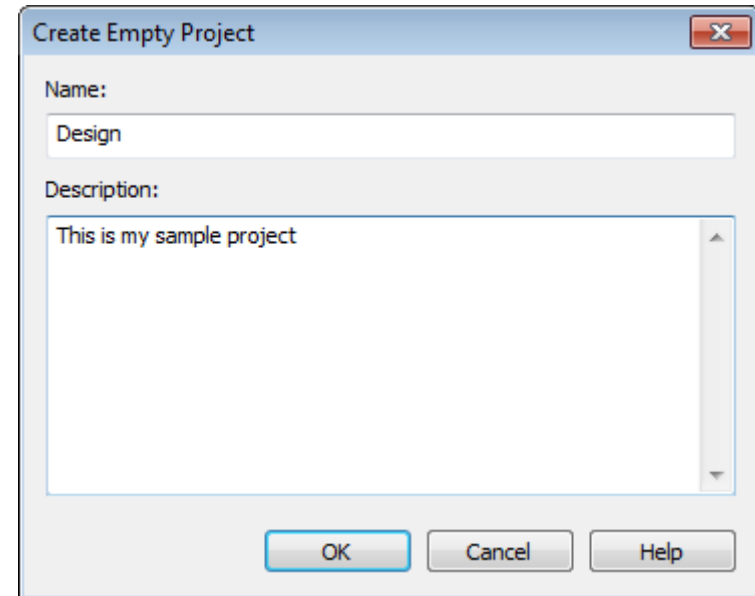
Once this is set, visEDATA can now manage the project archives and collaboration while allowing the user an interface for working with and extracting different views of the project.

LAB: Adding Project Containers

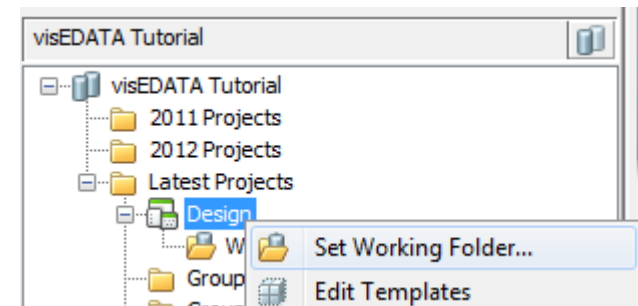
1. In the Vault view, select the Latest Projects folder and click **RMB > Create Empty Project**.



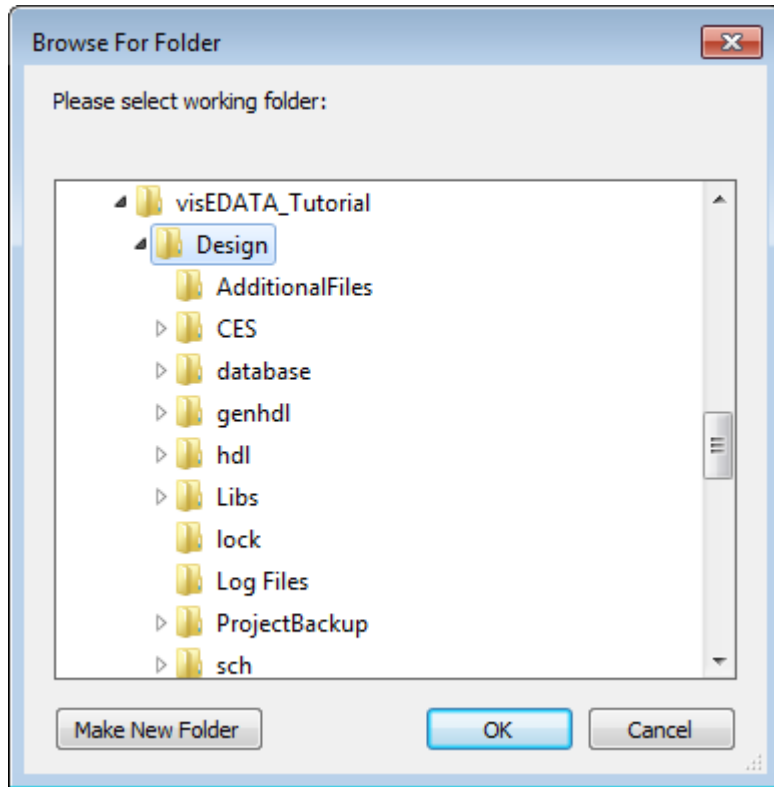
2. Type **Design** for the name and **This is my sample project** for the new project description,.



3. Click **OK**
4. Right-click on the new project and click **Set Working Folder**.

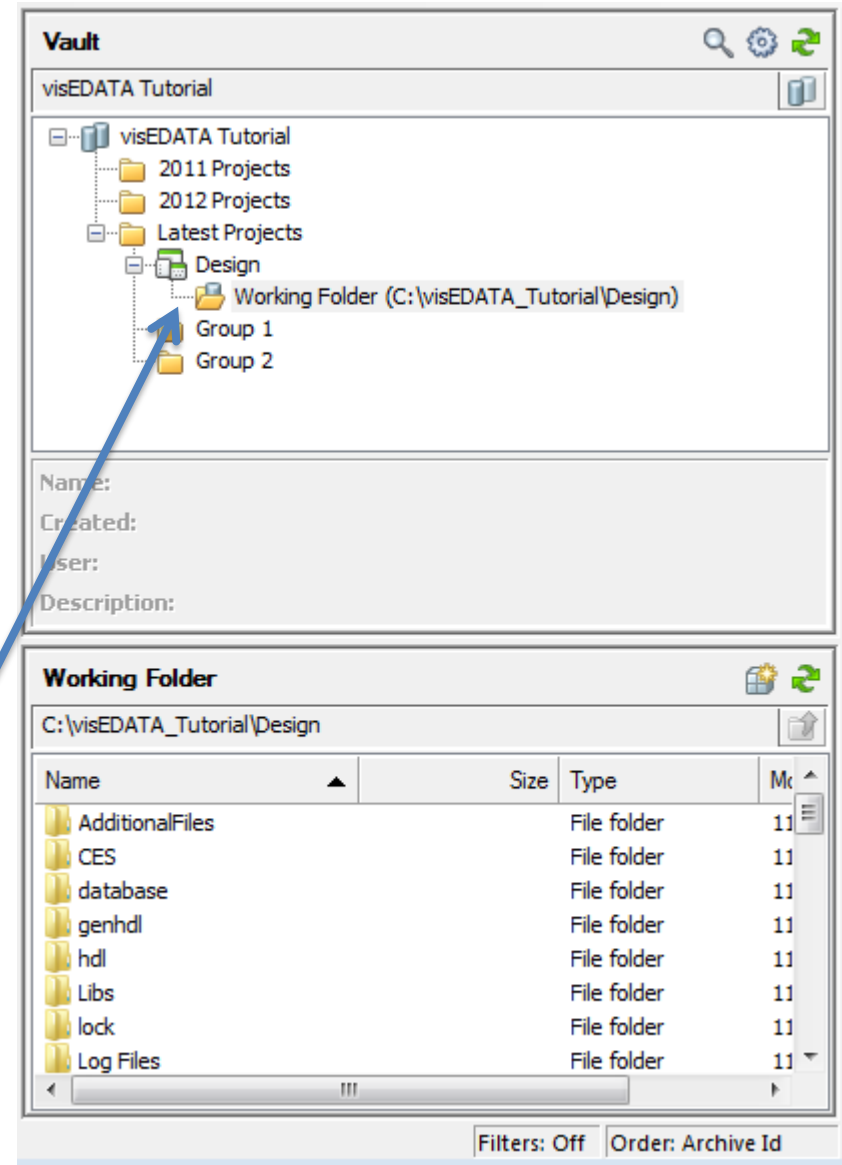


5. In the Browse for Folder dialog box, select the **C:\visEDATA_Tutorial\Design** folder.



6. Click **OK**.
7. Now select the Design in the Vault directory as shown in the image to the right.

NOTE: You will see that the Working Folder displays your project



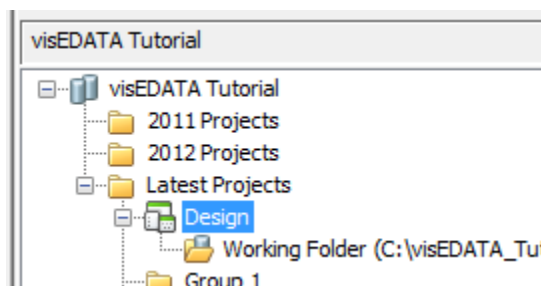
LAB: Archiving a Project for the first time

Tip: When you add an archive to the vault, it is an archive of the current contents of the working folder and the default name will be Archive 1

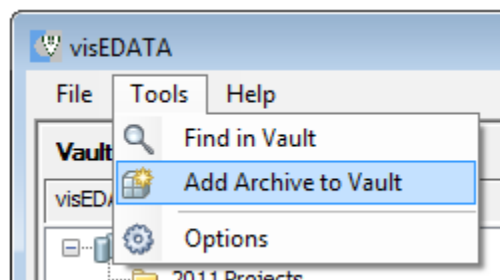
An archive can be


- Front end only (Schematic: DxD)
- Layout only (PCB: PADS)
- Front end and layout (Schematic and associated PCB design)

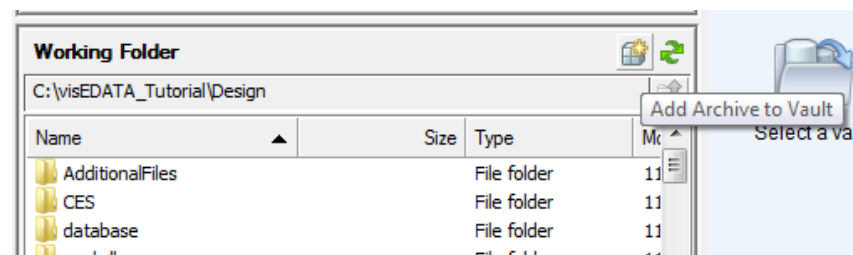
1. Select the Design in the Vault window



2. Select **Tools > Add Archive to Vault** from the File menu

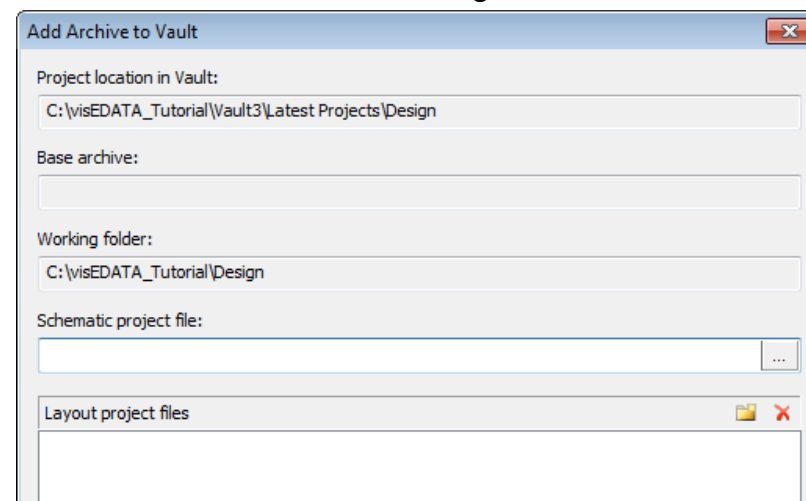


NOTE: You can also select the, click the **Add Archive to Vault** button {  } in the **Working Folder** window as shown below

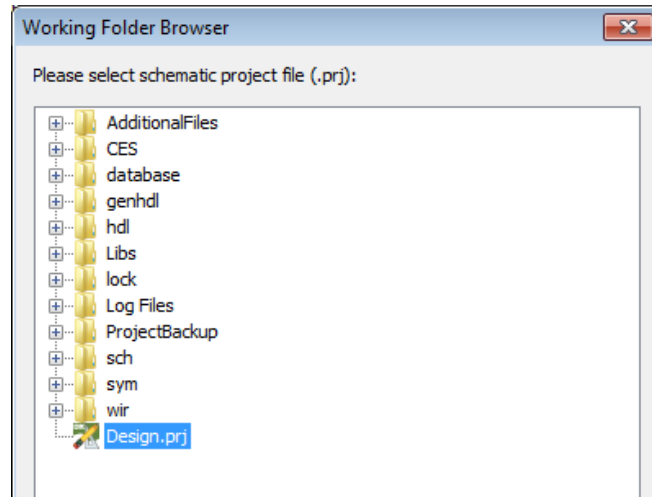



NOTE: The Add archive to Vault button is available only when a project or archive is selected in the Vault tree.

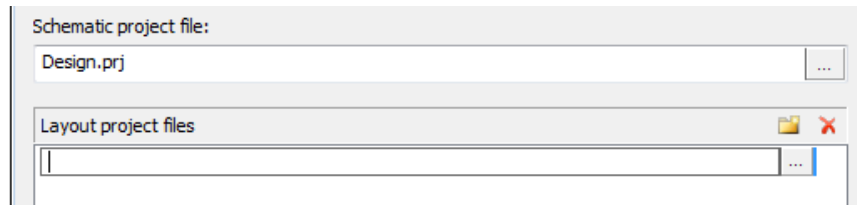
3. The **Add Archive to Vault** dialog box will be shown:



4. In the Schematic project file box, browse for the **C:\visEDATA_Tutorial\Design\Design.prj**



5. Click **OK** to close the dialog
6. In the Layout project files box, click the **New (Insert)** button {  }

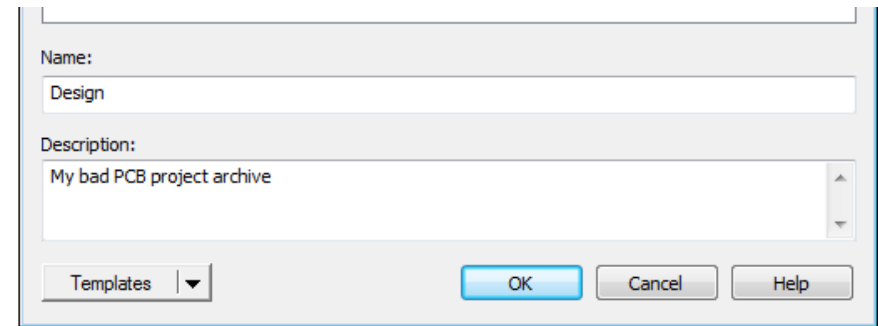


7. Select the ellipse { ... } button and browse for the **Design_bad.pcb** file.
8. Click **OK**



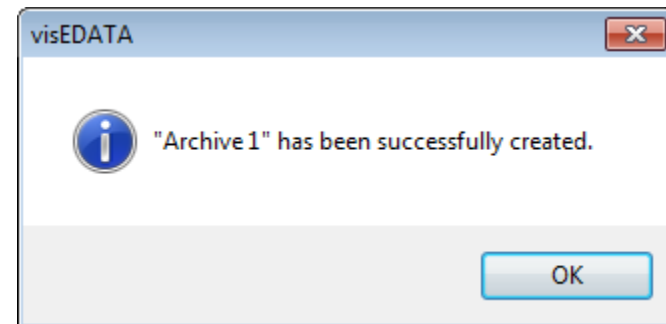
NOTE: The Additional files and folders box, allows you to add non-Schematic and non-PCB files/folders to the archive

9. In the **Name** box, type **Design**.
10. In the **Description** box, type **My bad PCB project archive**



11. Click **OK** to archive the design

NOTE: After several seconds, you will notice a new Archive 1 has been added to your project folder.



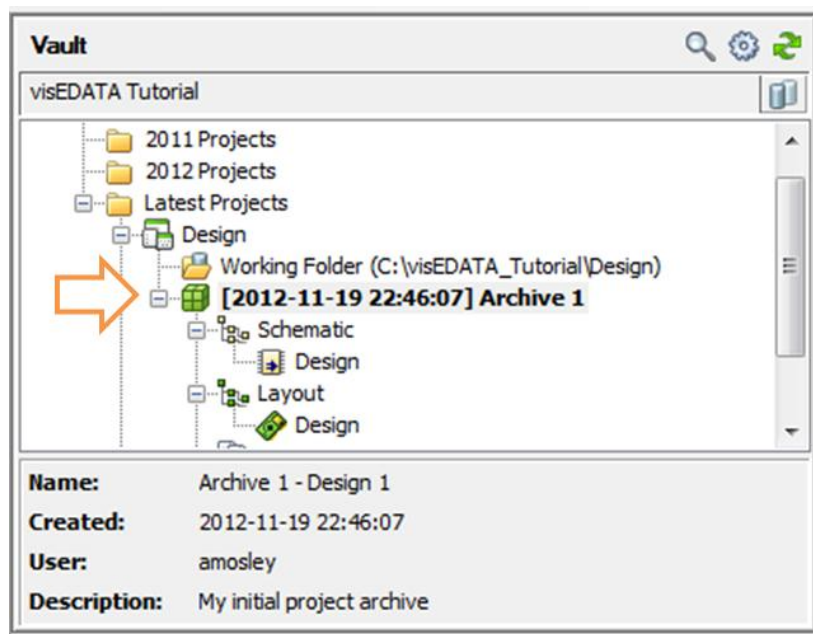
12. Click **OK** to close the dialog

LAB: Viewing the Project Vault Contents

Once a project is archived, the **Schematic** and **Layout** graphics are automatically created and associated to the project in the vault view if a schematic and/or PCB is associated. These items allow the user to view the graphic preview, create reports (compare), and add collaboration/redlining.

Tip: Double-clicking a project Schematic or Layout object opens a design file stored in the vault for viewing and redlining in visEDATA. If you want to open a PCB or schematic file for editing in PADS Layout or DxDesigner, double-click on its name in the **Working Folder** area.

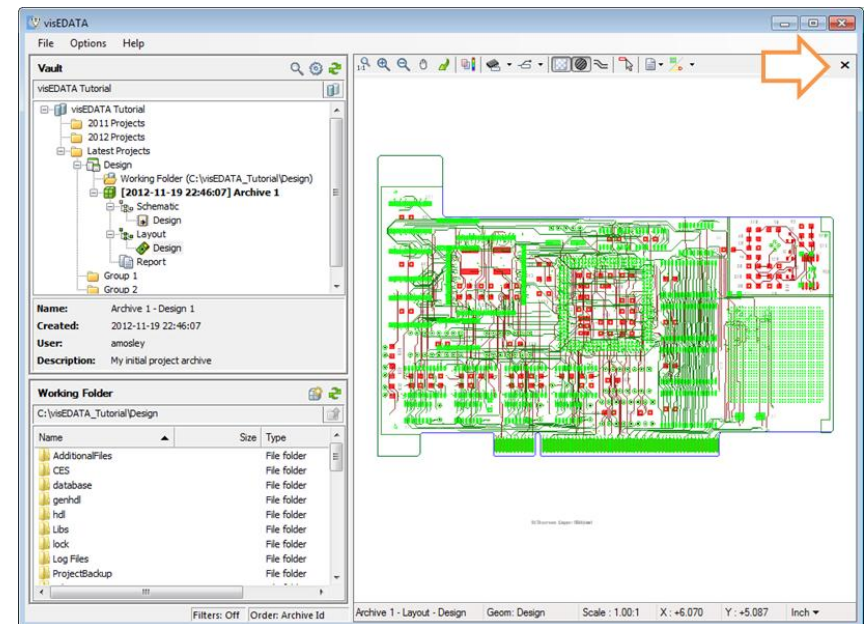
1. Expand the **Archive 1** node to see the Schematic and Layout nodes



2. **Double-click** the Design graphic under the Layout node to see a preview

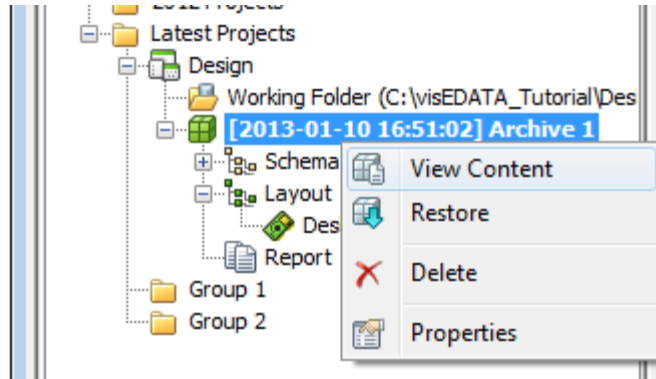


3. Close the Layout Preview, but selecting the **(X)** in the Layout window

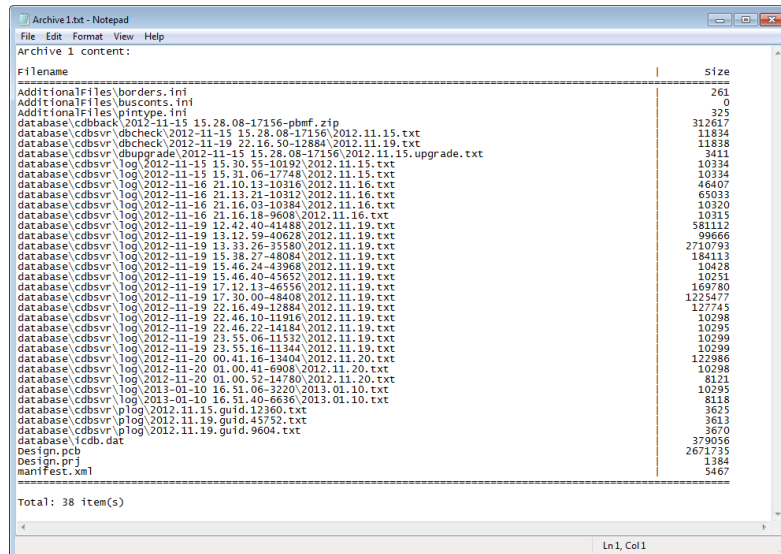


NOTE: We will review the PCB and Schematic previews in a later section.

4. Click on the **Archive 1** and Select **RMB > View Content**



NOTE: This will show you a list of files that are stored in the Archive within a text editor.



5. Click **File > Exit** to close your Notepad window

LAB: Managing archive templates

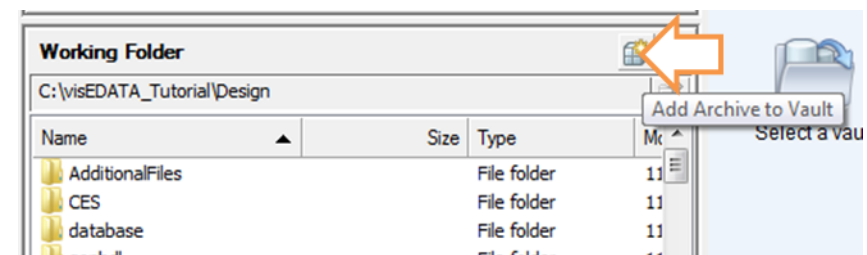
When you are creating an archive, you can use an archive configuration template to set all the fields in the **Add Archive to Vault** dialog box for a specific project. Use the Templates dialog box to select a template for creating a new archive, to create a new template, or to edit an existing template.

An archive template specifies:

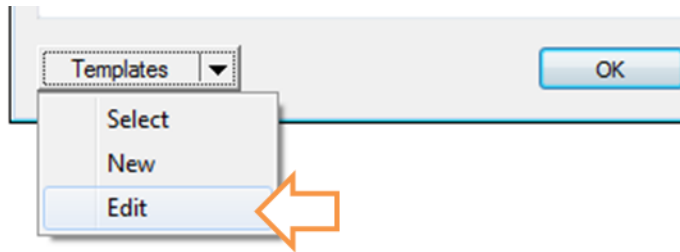
- The schematic project (.prj) file
- Zero or more layout project (.pcb) files
- Zero or more additional files to archive
- A name and text description to identify the archive in the vault (optional)

Tip: When specifying filenames in the Templates dialog box, you can enter the variable "`<projectname>`"; this variable will be replaced by the name of the Vault Project when you use the template. For example, if you enter "`<projectname>.pcb`" in the Layout project files field, when you use this template to archive the Processor project, "Processor.pcb" will appear in the Add Archive to Vault dialog box; but when you use it to archive the Timer project, "Timer.pcb" will appear.

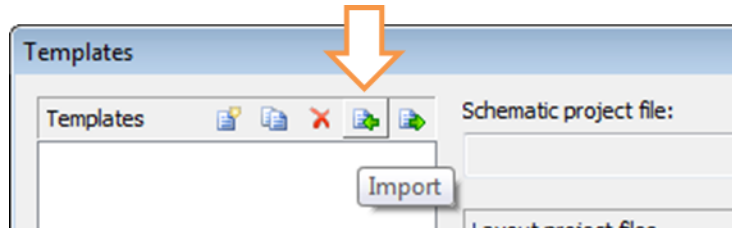
1. In the Working Folder view, click the **Add Archive to Vault** button



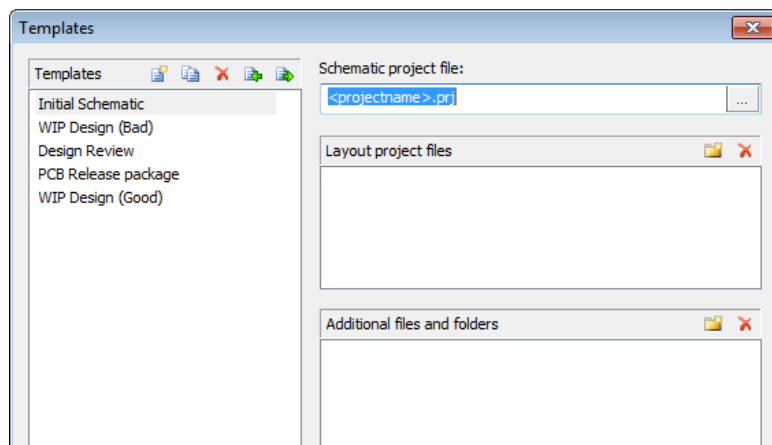
2. Now select the **Templates** button and choose **Edit**



3. Click the **Import** button to import templates from disk



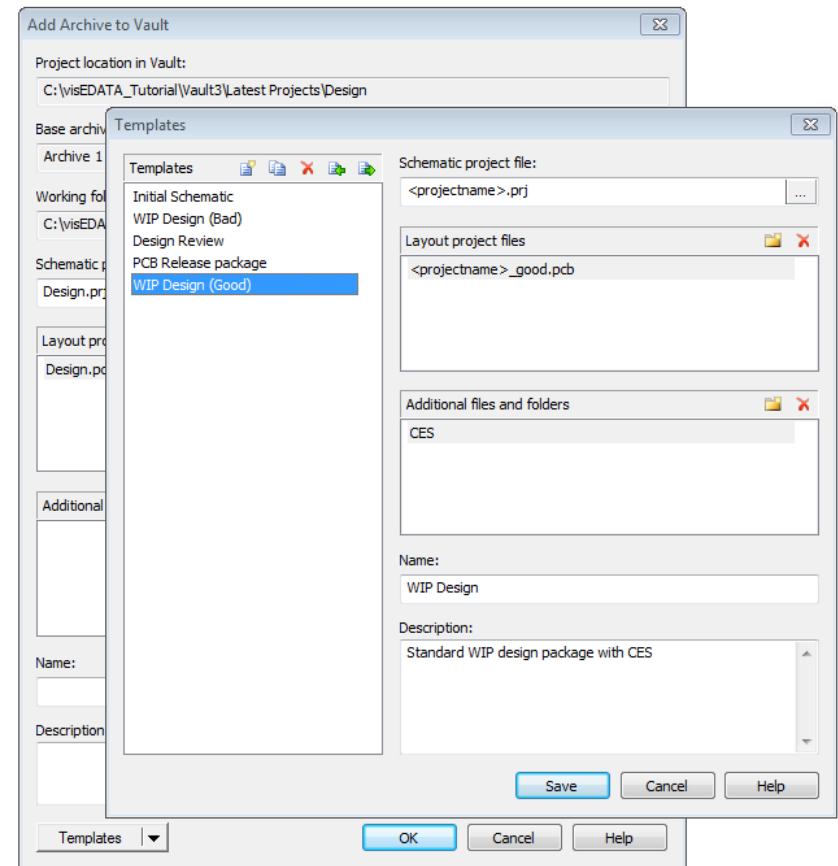
4. Navigate to the **C:\visEDATA_Tutorial\Templates\Example.templates** folder and click **Open**



NOTE: You can create templates and export them to disk for use by other projects

5. Select the **WIP Design (Good)** Template

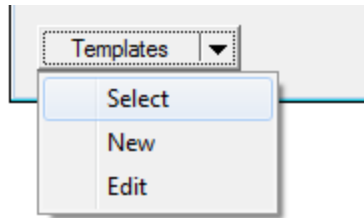
NOTE: You will see that the archive dialog is filled in based on the template chosen and the CES folder is in the Additional files and folders section



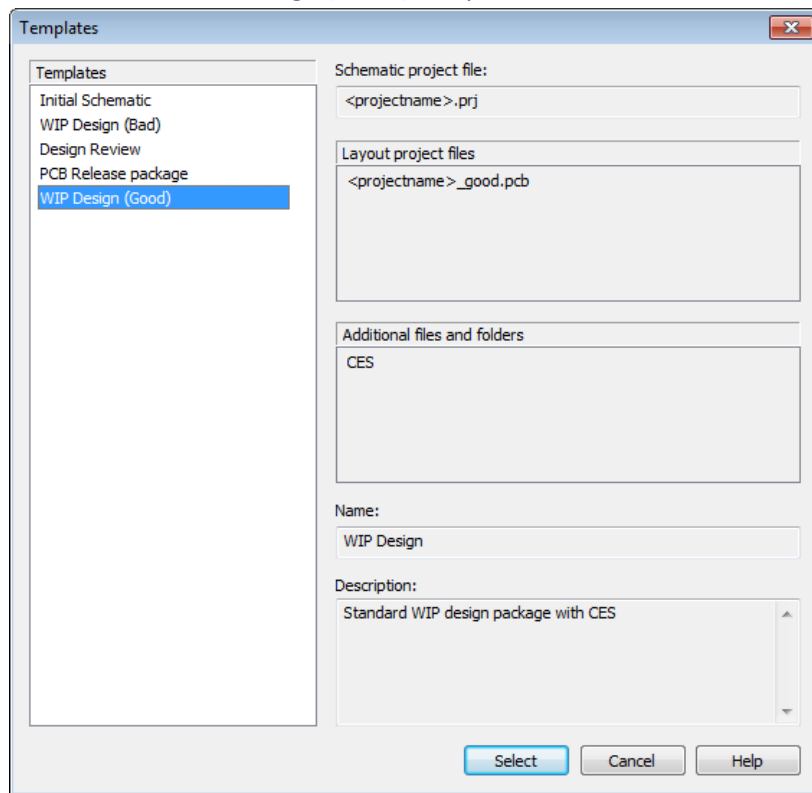
6. Click **Save**

NOTE: This will not save the templates so that you can access them for your design.

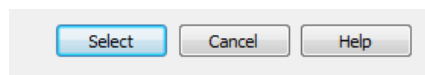
7. Within the **Add Archive to Vault** dialog select **Templates > Select**



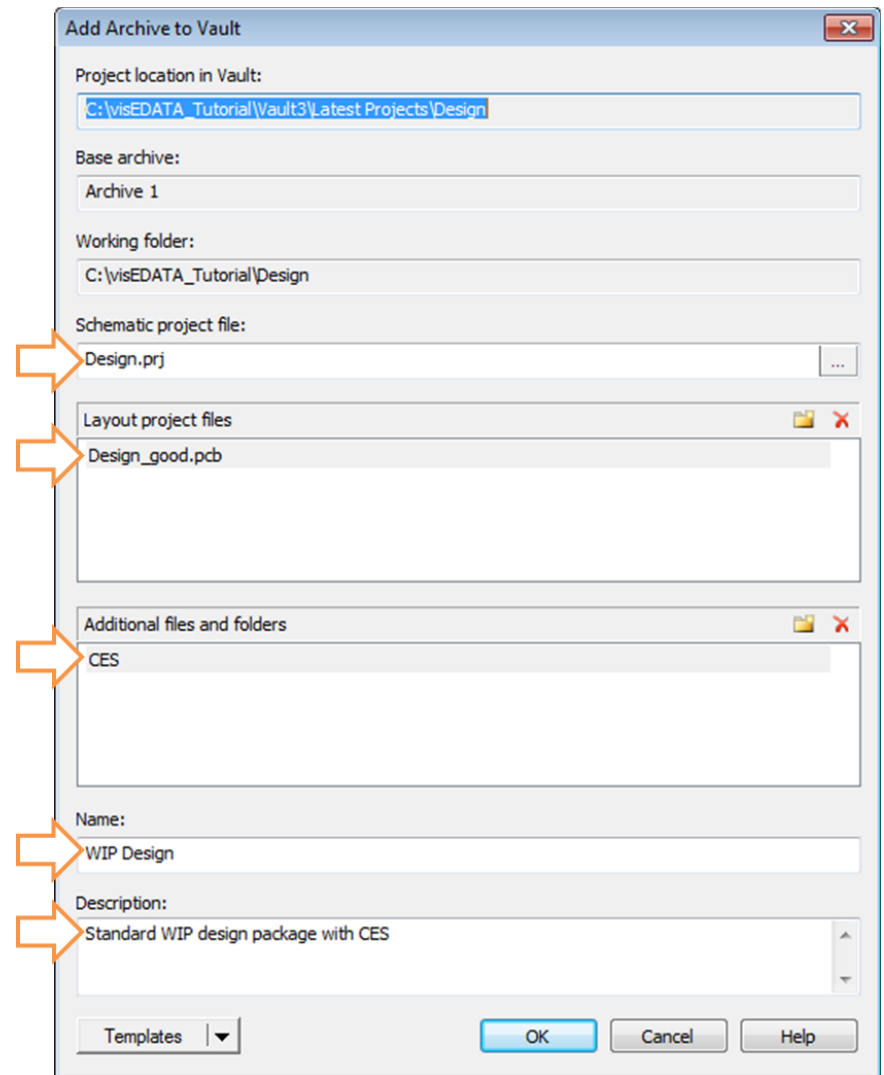
8. Select the **WIP Design (Good)** template



9. Click **Select** to choose the template



NOTE: The Add to Archive dialog will not contain the template *Schematic project file*, *Layout project files*, *Additional files and folders*, *Archive Name*, and *Archive Description* auto-filled.



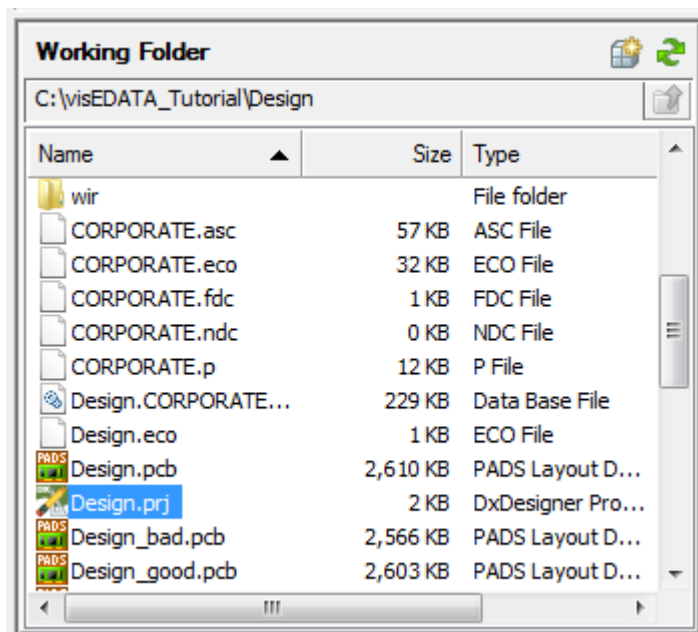
10. Click **OK** to Vault the "Good" PCB

Editing a Project

Now that the project has been archived, you can continue to make subsequent changes by accessing the project within the “Working Folder”. By selecting any node within the vault view, the Working Folder window will update based on the project selected. From this location, the user can immediately access the project and perform design edits prior to the next archive.

LAB: Making a Schematic Change (Invoking DxDesigner)

1. Select the **Design** project in the Vault Navigator
2. In the Working Folder window, double-click the **Design.prj** file



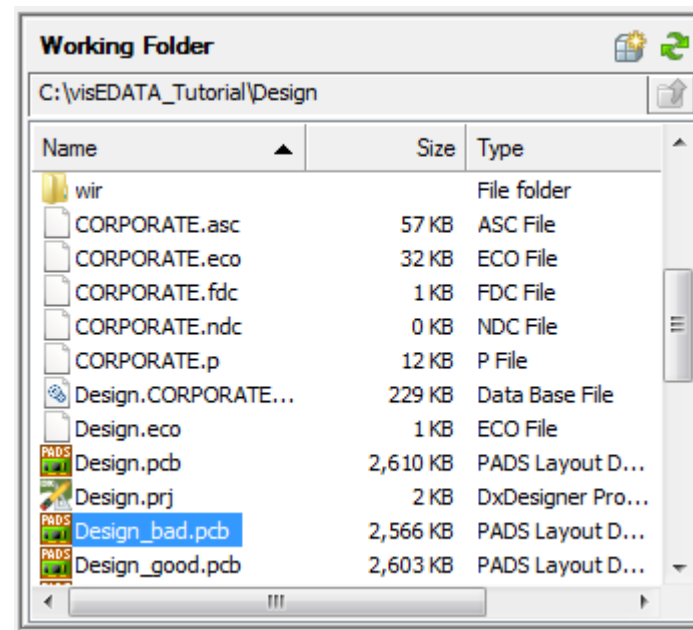
NOTE: This will open up DxDesigner on your project. Editing a schematic is similar to opening the project directly from a directory. Any changes

made will not show in the visEDATA vault until it is archived by the designer. This allows the designer to control the archive process.

3. Click **File > Exit** to close DxDesigner

LAB: Making a Layout Change (Invoking Pads Layout)

4. Open the visEDATA window
5. In the Working Folder window, double-click the **Design_bad.pcb** file



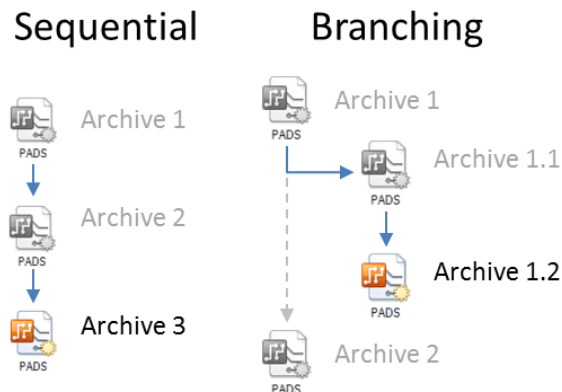
NOTE: This will open up Pads Layout on your project. Editing the layout is similar to opening the PCB directly from a directory

6. Do **NOT** make any changes
7. Click **File > Exit** to close Pads Layout

Creating a Subsequent Archive

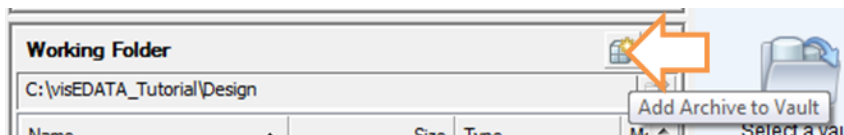
When a project is first vaulted, the default name is set to **Archive 1**. Upon subsequent “sequential” updates and archives, visEDATA will automatically increment the archive name to **Archive 2**, and then **Archive 3**.

visEDATA also allows for “branching” which allows a user to restore a previous archive and begin naming relative to the restored archive. For example, if **Archive 1** is restored and then re-archived, the name becomes **Archive 1.1** and then **Archive 1.2**.



LAB: Sequential Archive

1. In the Working Folder view, click the **Add Archive to Vault** button.



2. In the Add Archive to Vault dialog box click **OK**.

The 'Add Archive to Vault' dialog box contains the following fields and controls:

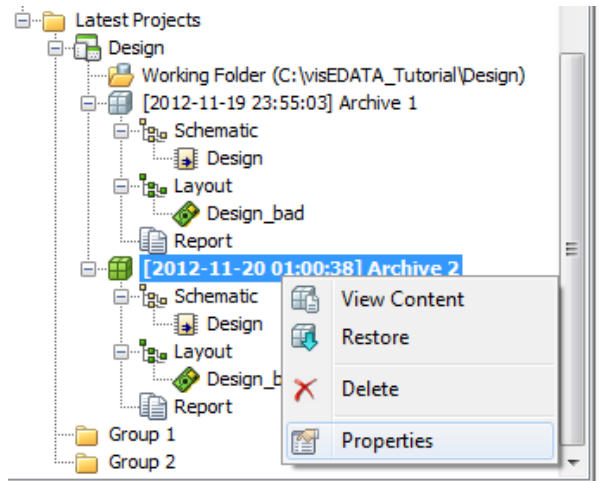
- Project location in Vault:** C:\visEDATA_Tutorial\Vault\Latest Projects\Design
- Base archive:** Archive 1
- Working folder:** C:\visEDATA_Tutorial\Design
- Schematic project file:** Design.prj
- Layout project files:** Design_bad.pcb
- Additional files and folders:** (Empty list)
- Name:** My second archive
- Description:** This is my second archive of my design
- Buttons:** OK, Cancel, Help
- Templates:** A dropdown menu.

NOTE: The system will remember the selections made during the last archive and will pre-fill in your selections.

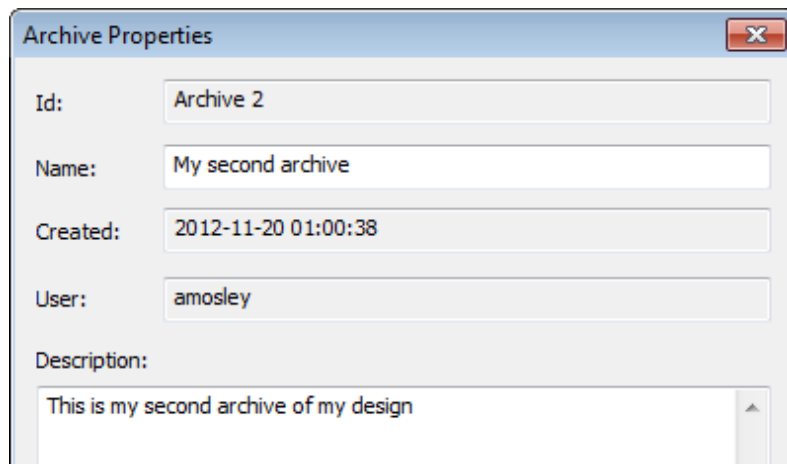
3. Click **OK** once “Archive 2” has been successfully created

NOTE: You will notice Archive 2 was created and is now active (Green)

4. Select Archive 2 and click **RMB > Properties**



NOTE: Notice that you can change the Name and Description of the archive, but the **ID**, **Created**, and **User** are all locked



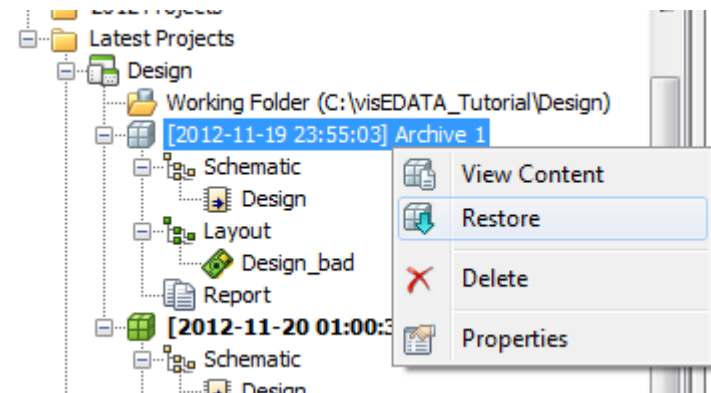
5. Click **OK** to close Archive Properties window

LAB: Retrieving a Project from the Vault (Branching)

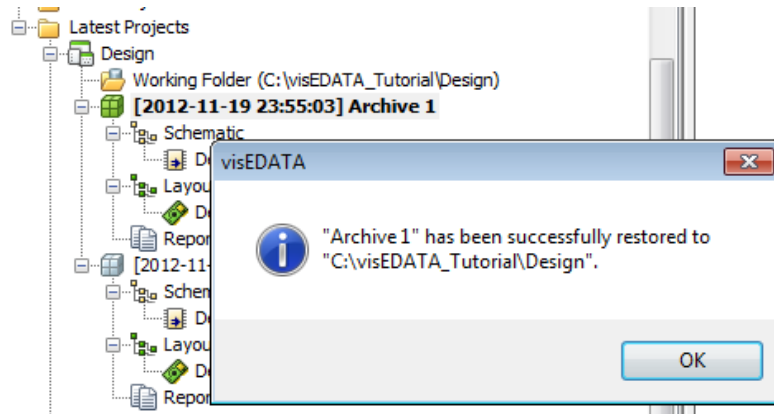
- Many times a new PCB project is just a minor alteration of an existing project. You can extract an archive from a project to create a new object or restore an archive for recovery or branching purposes.
- When restoring an Archive to a current working directory, all common files will be overwritten in the local directories
- Any additional files that were not included in the Archive will remain in the current working directory
- You can select any archive in the vault and restore it to the working folder
- Restoring previous archives creates “Branching”

Procedure

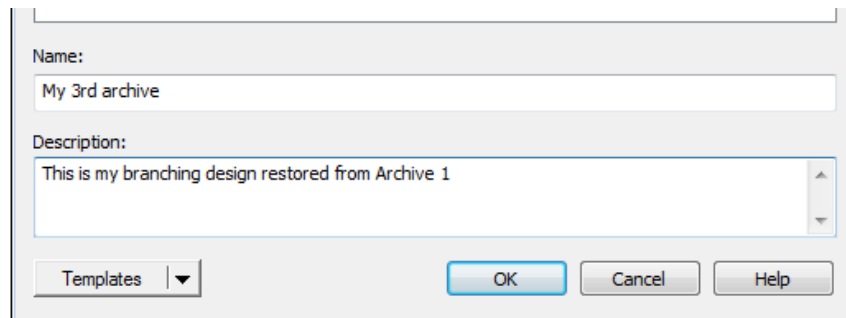
1. In the Vault view, **Select Archive 1**
2. Click **RMB > Restore**
3. When you are prompted to confirm the restore operation, click **Yes**.



NOTE: The archive is restored to the working folder, and the *Archive 1* icon turns green to indicate that it is the currently restored archive.

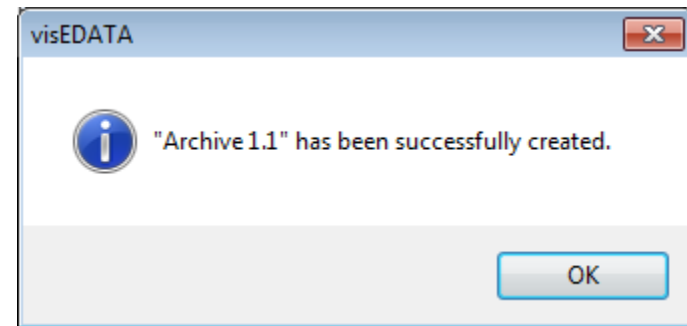


3. Click **OK** to close the dialog
4. In the Working Folder view, click the **Add Archive to Vault** button.
5. In the Add Archive to Vault dialog box change the Name to **This is my branching design restored from Archive 1**



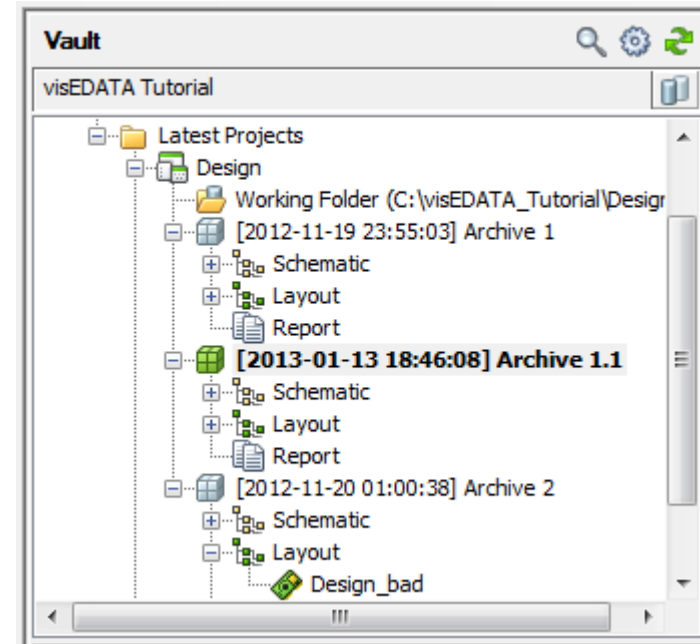
6. Click **OK**

NOTE: Now Archive 1.1 has been created

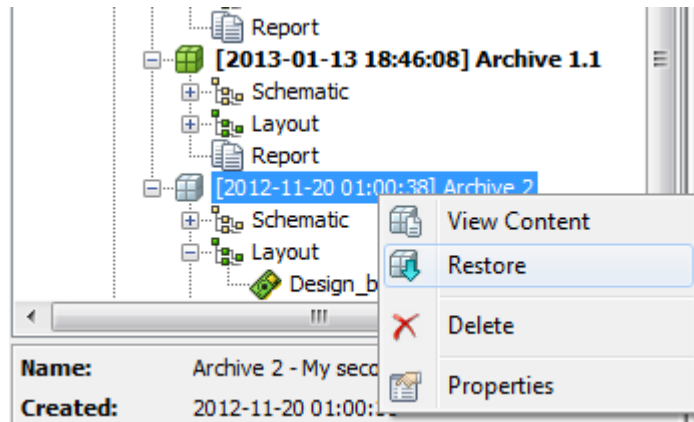


7. Click **OK**

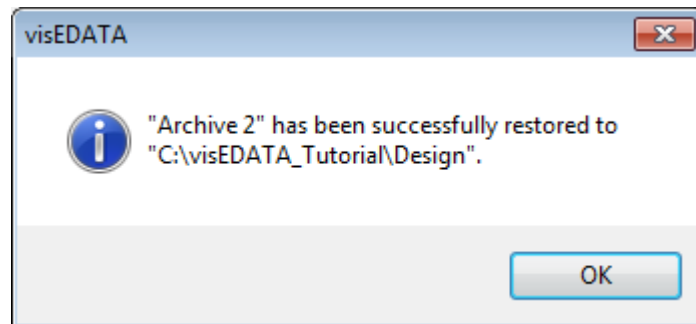
NOTE: Archive 1.1 will now be the default archive (bold)



8. Select **Archive 2**
9. Click **RMB > Restore**



10. Click **Yes** to the warning dialog
11. Click **OK** to close the note dialog that that archive has been restored



Project/Object Searching (All)

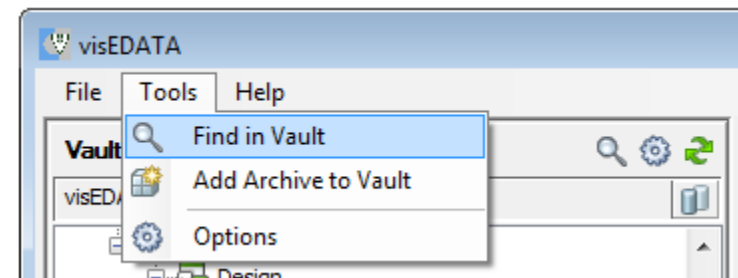
visEDATA contains a simple searching interface that allows finding objects (Projects, Archives, and Folders) based on the user, description, if it contains redline information along with the date the object was created. Once an object is found, it can be selected within the search results window and immediately accessed in the Vault Navigator.

Tips:

- Searches are not word-based; a search for the string "ic" will find Iceland, picnicking, and alphanumeric.
- Wildcards are not supported.
- Searches are not case-sensitive.
- Leaving a field empty returns items with any value.
- Items must match all fields to satisfy the search criteria.
- Both the Name and ID attributes of items are searched for the search string specified in the Name field.

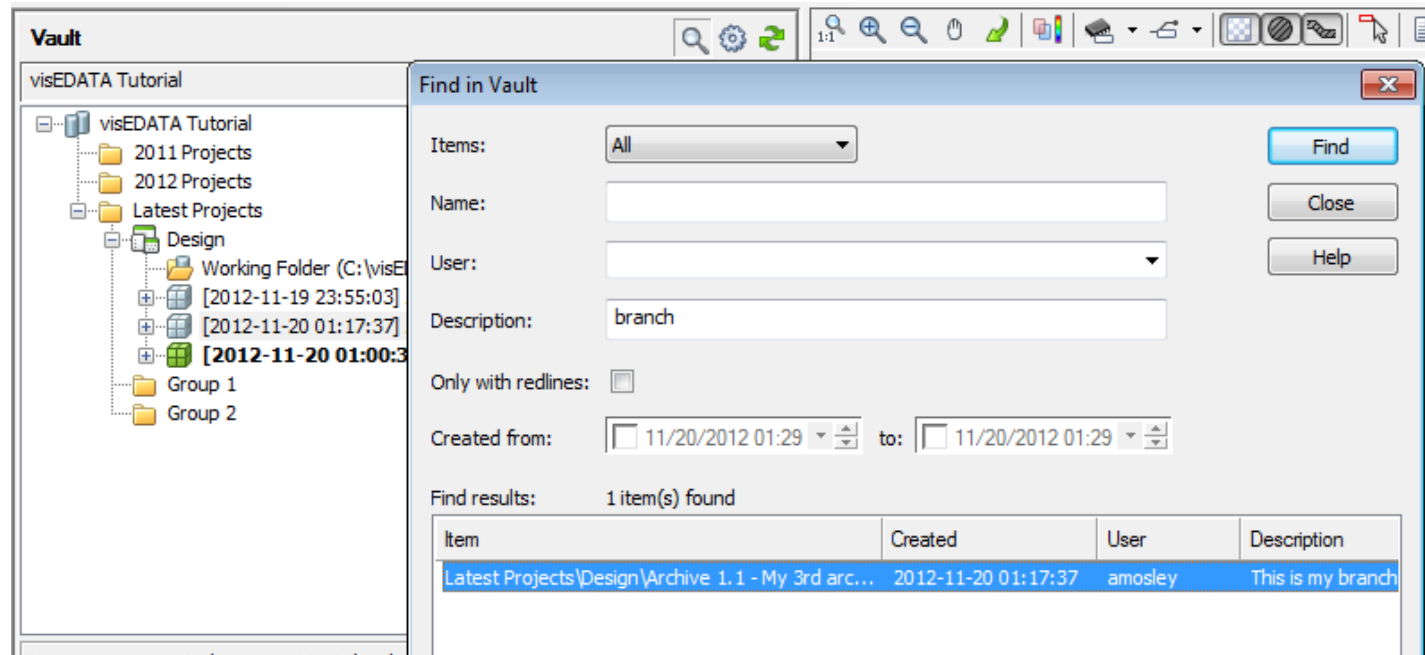
LAB: Project Searching

1. In the Vault view, Click the **Find in Vault** button () or select **Tools > Find in Vault**



2. In the Find in Vault Dialog Box, from the **Items** list select **Projects**, Click **Find**.

3. In the Find in Vault Dialog Box, from the **Items** list select **Archives**, Click **Find**
4. In the Description field type “branch” to find the archived you used for branching, Click **Find**
5. **Double-click** the item in the results list to have it highlighted and selected in the Vault tree.



Module 3: Viewing Project Graphical Data (All)

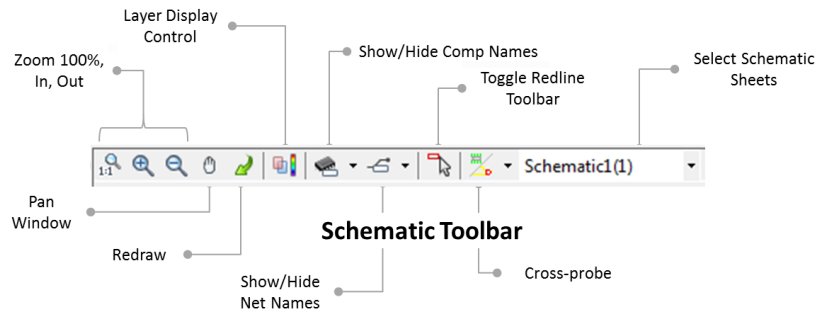
When viewing archive graphics, you can have at most 1 schematic and 1 layout view open at a time. visEDATA will allow you to choose any schematic archive and any layout archive to view, but if you are attempting to cross-probe, the schematic and layout must be from the same archive.

Graphic Navigation Basics

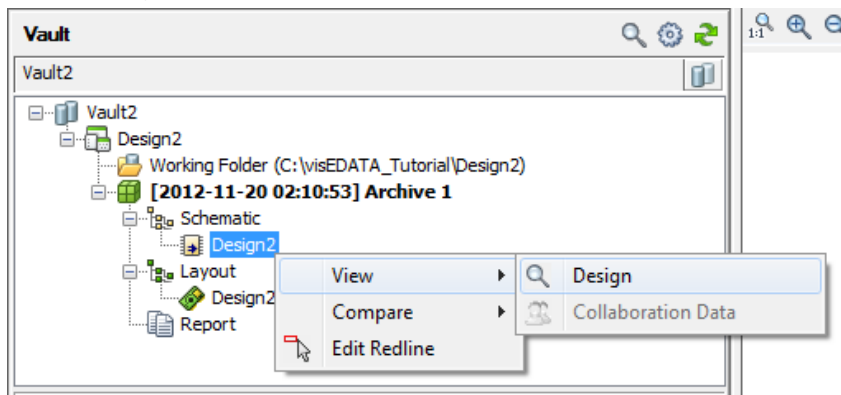
- Click and drag with the left mouse button to define an area, then release the button to zoom in on that area.
- Use the mouse wheel to zoom in and out at the cursor location.
- Use the Zoom In, Zoom Out, and Zoom 1:1 buttons, or the Page Up and Page Down keyboard keys, to zoom in and out at the center of the viewing area.
- Use the LMB to draw a box around an area to zoom
- Placing the mouse on a net will show the net name and net handle as a tooltip
- Placing the mouse on a pin will show the pin, pin number, and net name
- Placing the mouse on symbol will show the ref designator
- Refresh - redraw the Layout or Schematic viewing window. Redrawing does not change the position of the design in the viewing window
- Layer Display Control - change the layers
- Managing layers - change layer colors,
- Component Name control (Angle, Size, Color)
- Net Name Control (Angle, Size, Color)

LAB: Schematic Graphics

- To change the schematic sheet displayed in the Schematic view area, select a sheet from the Sheet dropdown list in the Schematic toolbar.



- In the main visEDATA window, click the **Select Vault** icon
- Navigate to the **C:\visEDATA_Tutorial\Vault2** directory, and Click **OK**
- Select the **Design2 > Archive 1 > Schematic > Design 2** object project in the Vault view window
- click **RMB > View > Design** to open the graphic view (or double-click)

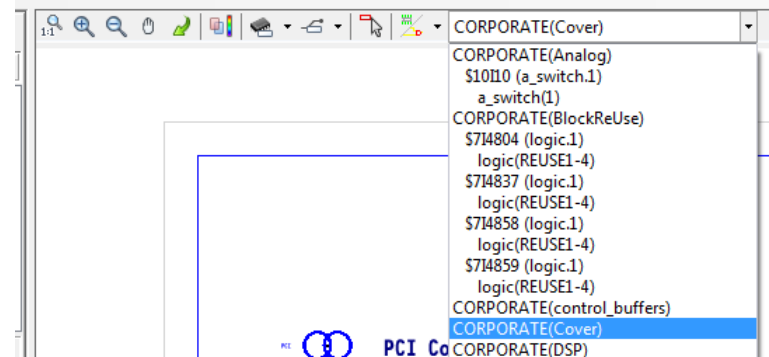


- Select the **Manage Layers** icon { }

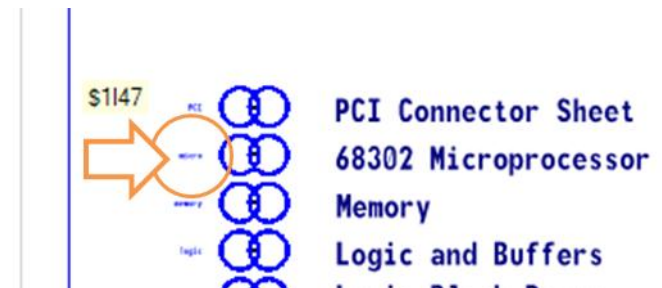
NOTE: You can turn layers on/off and change their colors, and even save the color schemes

- Click **Close** to close the Display Control window
- Select the **Schematic Sheets** dropdown
- Choose the **CORPORATE(Cover)** sheet


NOTE: Hierarchical blocks are shown as indented

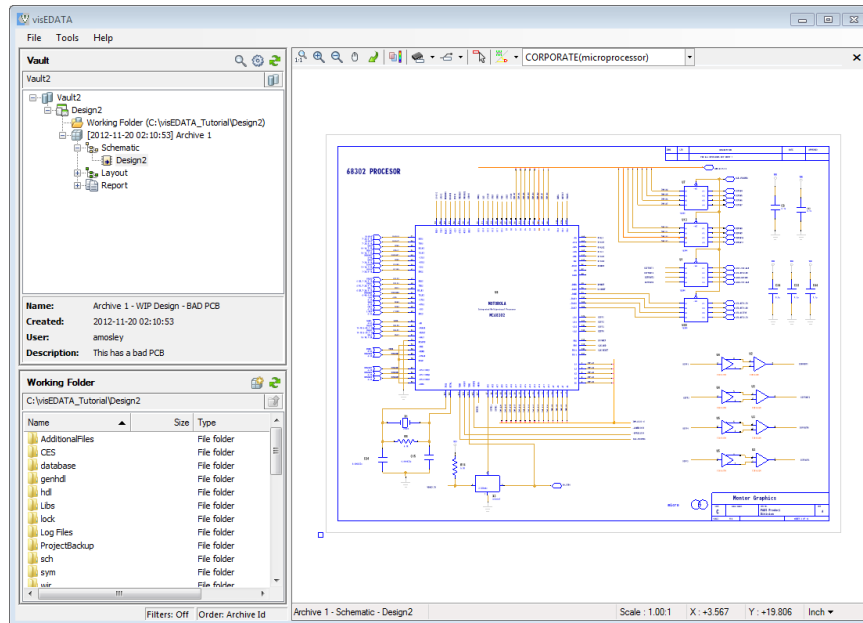


- Click on the link next to the 68302 Microprocessor text



NOTE: The link will take you directly to the Microprocessor sheet that was added on the schematic


10. Click the Zoom 1:1 icon {  } to see the entire sheet



11. Play with the Zoom & Pan functionality to move around the Schematic


a. Make sure the Pan icon is **NOT** selected



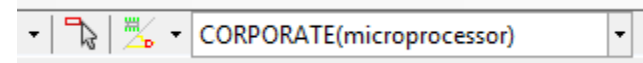
b. Click the Zoom In/Out {  } Page Up/Page Down/Draw a Box around an area

c. Click the **Pan icon** {  }

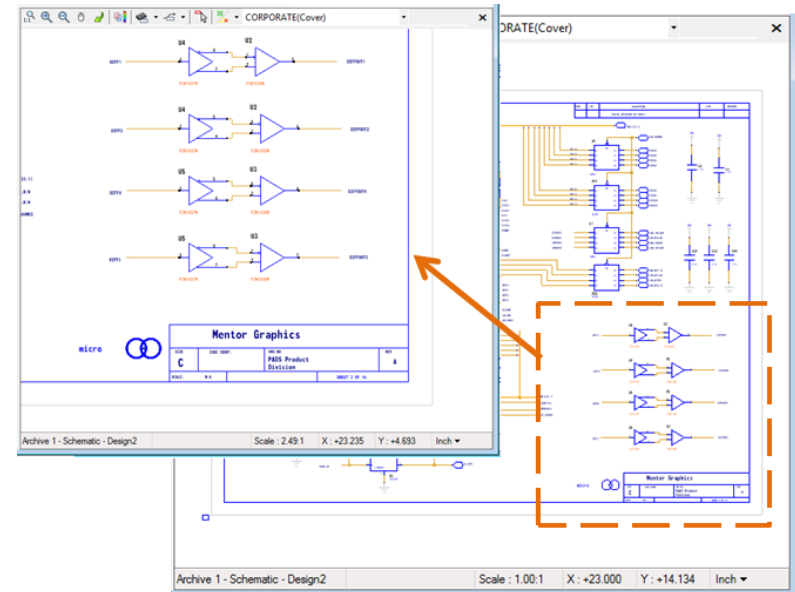
d. Click and hold the LMB to drag around the schematic


e. Depress the Pan icon {  }


12. Select the CORPORATE(microprocessor) schematic



13. Zoom into an area of the schematic

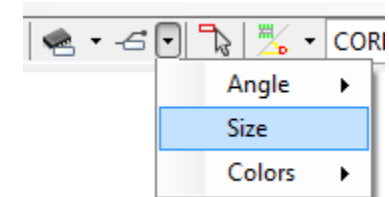


14. Toggle the **Show Comp Name** button {  } on and off
NOTE: You will notice that the component Ref Designators are turned on/off

15. Toggle the **Show Net Name** button {  } on and off

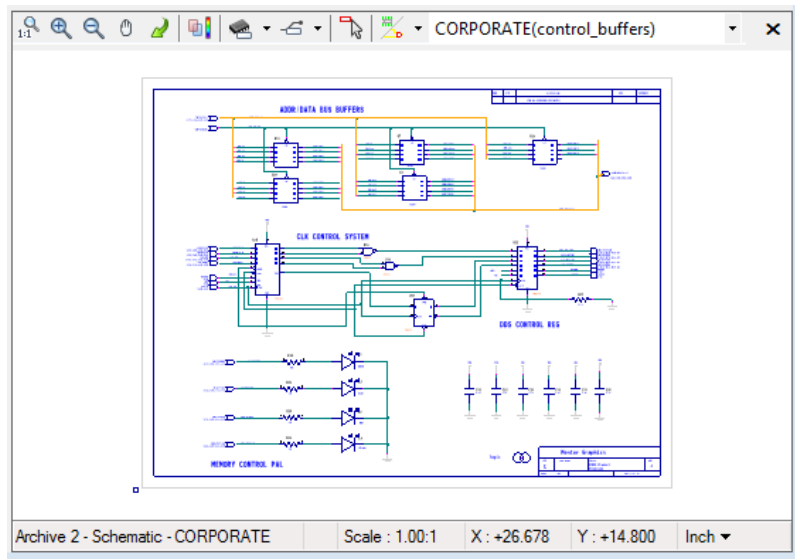
NOTE: You will notice that the Net Name ID's are turned on/off

TIP: By selecting the arrow next to the Show buttons, you can change the **Angle**, **Size**, and **Color** of the text



16. Select the **CORPORATE(control_buffers)** sheet

17. Click the **Zoom 1:1** button {} to return to the full view

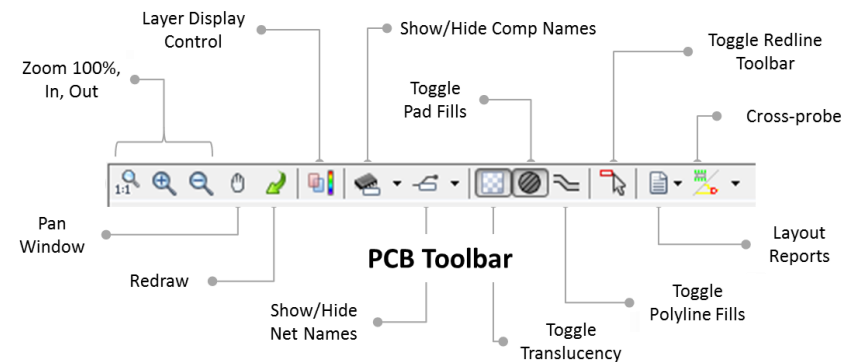


18. Close the Schematic graphic by clicking the {X} in the upper right hand corner

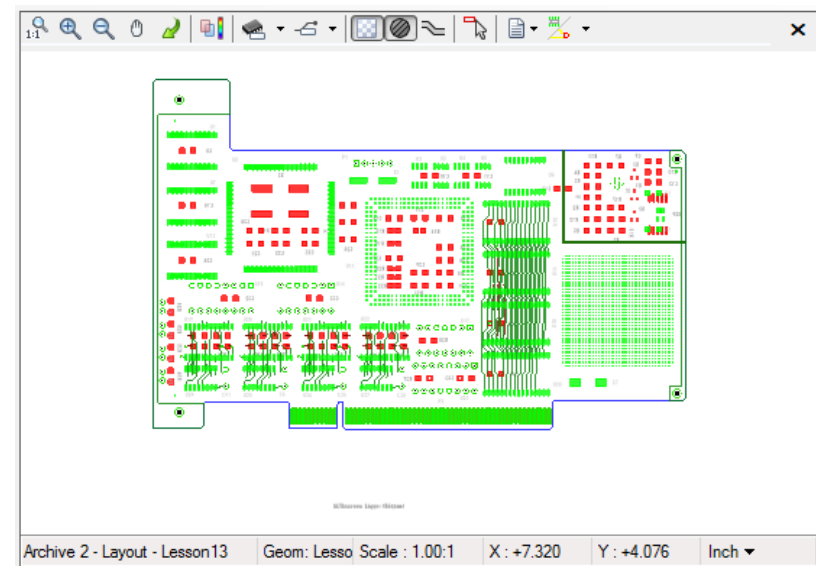
LAB: Layout Graphics

- **Manage Layers** - allows changing of Color, Draw Order, and Visibility
- **Translucency** - lets you view through multiple solid areas on the board. This option is useful if you are leaving polyline fills turned on.
- **Pad Fills** - use this to provide a cleaner view of the padstack data, or to reduce the number of fills drawn to the screen.

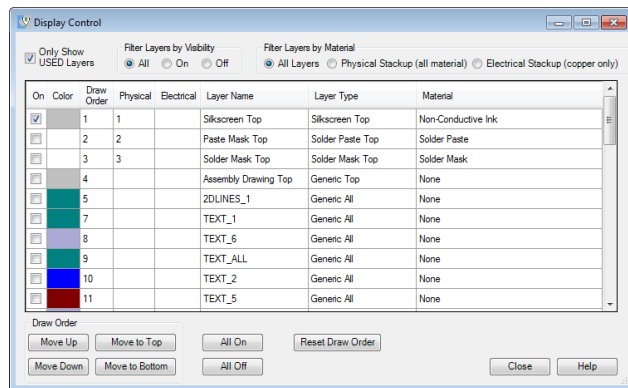
- **Polyline Fills** - use this to clean up the view if you are inspecting components only, or if your circuit board contains many flooded plane areas.



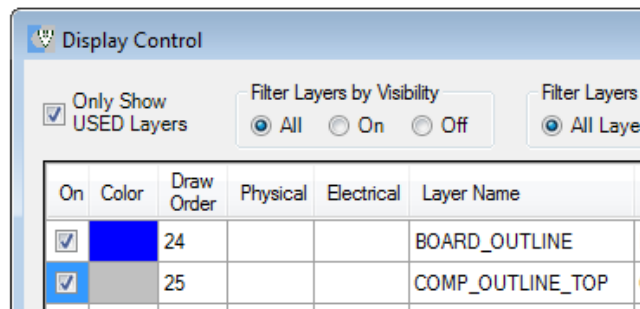
1. Select the **Design2 > Archive 1 > Layout > Design2** object project in the Vault view window
2. click **RMB > View > Design** to open the graphic view (or double-click) Select the **Layout Design** and **double-click** (or click **RMB > View > Graphic**) to open the graphic view



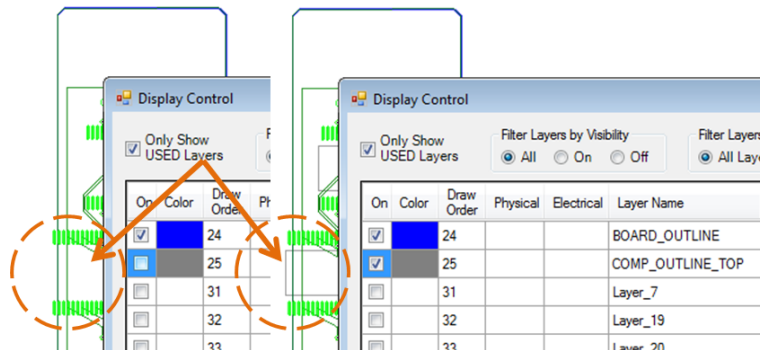
3. Select the **Manage Layers** icon { }



4. Toggle the **COMP_OUTLINE_TOP** (Draw Order 25) to show the top components outline

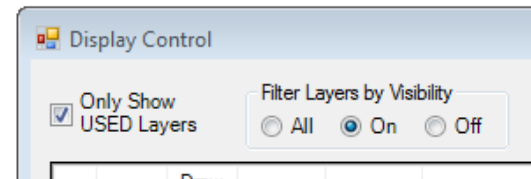


5. Continue to turn the label on/off - eventually leave it **ON**



NOTE: The layer controls will immediately show your changes

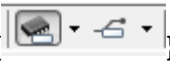
- Turn **ON** the **COMP_OUTLINE_BOTTOM** Layer (Draw Order 46)
- In the **Filter Layers by Visibility** section click the **On** radio button



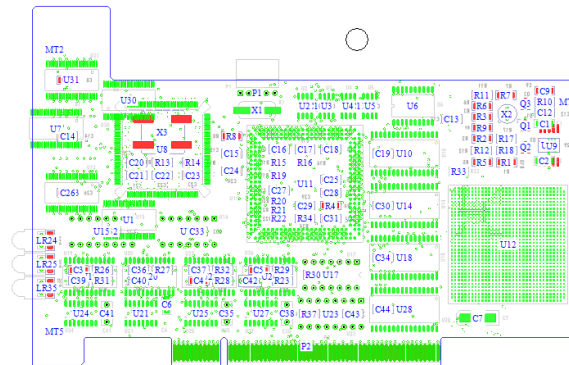
- Toggle the **Component Side Layer 1** (Draw Order 15) - leave it on
- Toggle the **Solder Side Layer 6** (Draw Order 40) - leave it on


NOTE: You can change the layer colors to fit your preferences

10. Click **Close** to close the Display Control window

- Toggle the **Show Comp Name** button {  } on and off

NOTE: You will notice that the component Ref Designators are turned on/off



- Toggle the **Show Net Name** button {  } on and off

NOTE: You will notice that the connected Net Name ID's are turned on/off

Common Graphical Actions

Viewing Design Item Information

Position the cursor over a component, component pin, via, etch, or other item to display information about the item, as shown in the following figure. Item selection is prioritized based on data types. From highest to lowest priority, the item selection priority is:

- Component pin
- Component
- Trace
- Via
- Other Inserts
- Other Polys

Redrawing the Display

Click the Redraw button to redraw the Layout or Schematic viewing window. Redrawing does not change the position of the design in the viewing window.

Controlling Layer Display

All design files are composed of multiple layers, representing different features on the PCB Layout board. For example, the pads / electrical contact areas for components on the top and bottom of the board are on two different layers, in two different colors.

You can control which layers are displayed, their order in the display from top to bottom, and their colors.

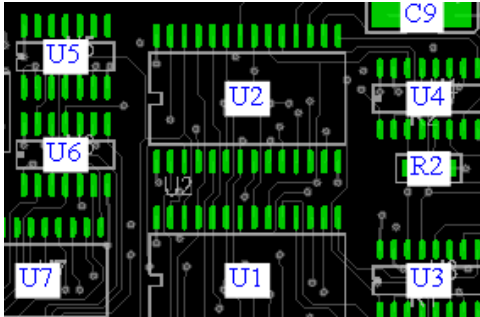
Procedure

1. In the PCB or Schematic viewing area, click the Display Control button .
2. In the Display Control Dialog Box for PCBs or Display Control Dialog Box for schematics, make the appropriate settings, and click Close.

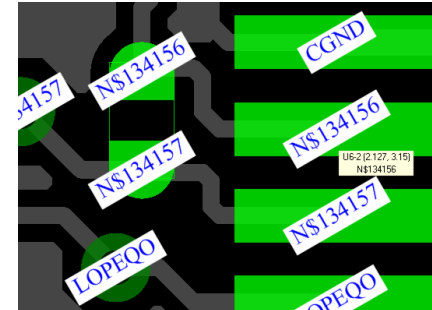
Toggling Component and Pin Net Labels

Click the Show Comp Names button or Show Net Names button to show/hide labels identifying each component or net on the board. Some care is taken to show only labels for parts on visible surfaces as opposed to component labels for both surfaces.

Component Labels Toggled On



Pin Net Labels Toggled On & Angled



Setting Label Display Attributes

- Click the **dropdown arrow** next to the **Show Comp Names** button or **Show Net Names** button to set the angle of display, default desired size, and text / background colors of component labels/.



Toggling Translucency

- Click the **Translucency** button to toggle translucency, which lets you view through multiple solid areas on the board. This option is useful if you are leaving polyline fills turned on.

Toggling Pad Fills

- Click the **Show Pad Fills** button to toggle pads/aperture fills. You can use this to provide a cleaner view of the padstack data, or to reduce the number of fills drawn to the screen.

Toggling Polyline Fills

- Click the **Show Poly Fills** button to toggle Polyline Fills. You can use this to clean up the view if you are inspecting components only, or if your circuit board contains many flooded plane areas.

Toggling the Redline and Collaboration Toolbars

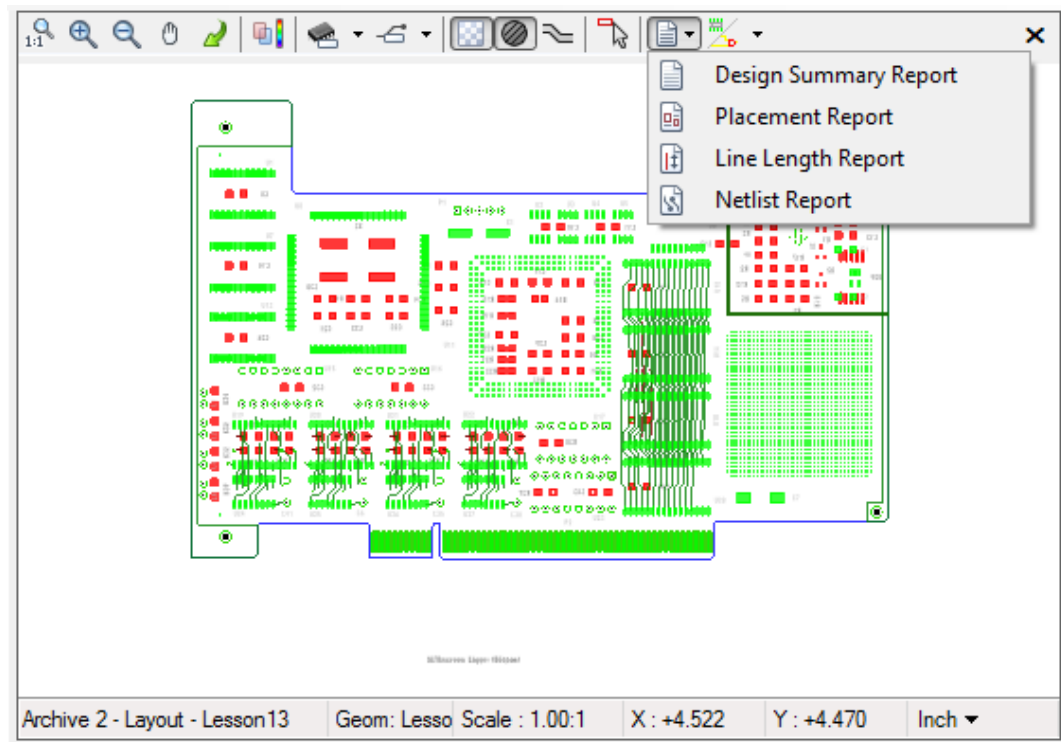
- Click the **Redlining** button to toggle the display of the The Redline Toolbar and the A Collaboration Tree, which enables you to share design information with others, is built into visEDATA.. When you click the Redlining button, the two toolbars toggle on or off together.

LAB: Layout Reports

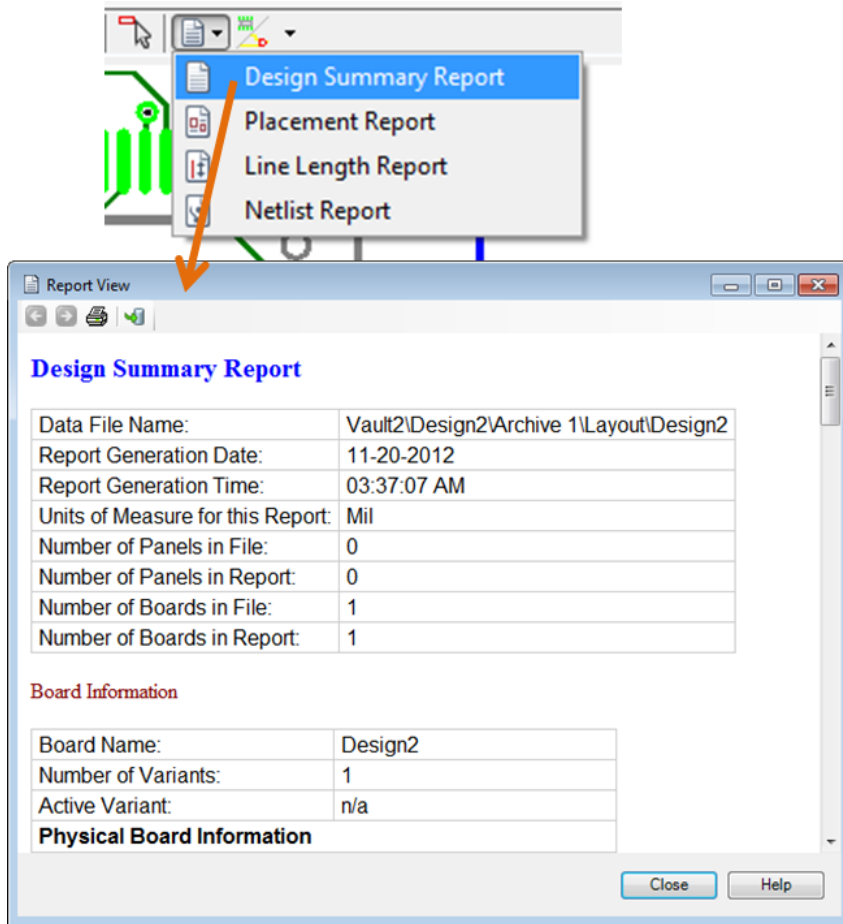
Within the graphics view of visEDATA, you can use the Reports button to create reports from data loaded into visEDATA. Clicking on the button, or on the dropdown arrow to the right of button, reveals the Reports menu.

The options in the Reports menu are:

- **Design Summary** - Lists general information about boards, drills, vias, nets, components, land patterns, etc.
- **Placement** - Lists component placement data.
- **Line Length** - Lists line lengths and widths.
- **Netlist** - Lists nets, pins, and connections.

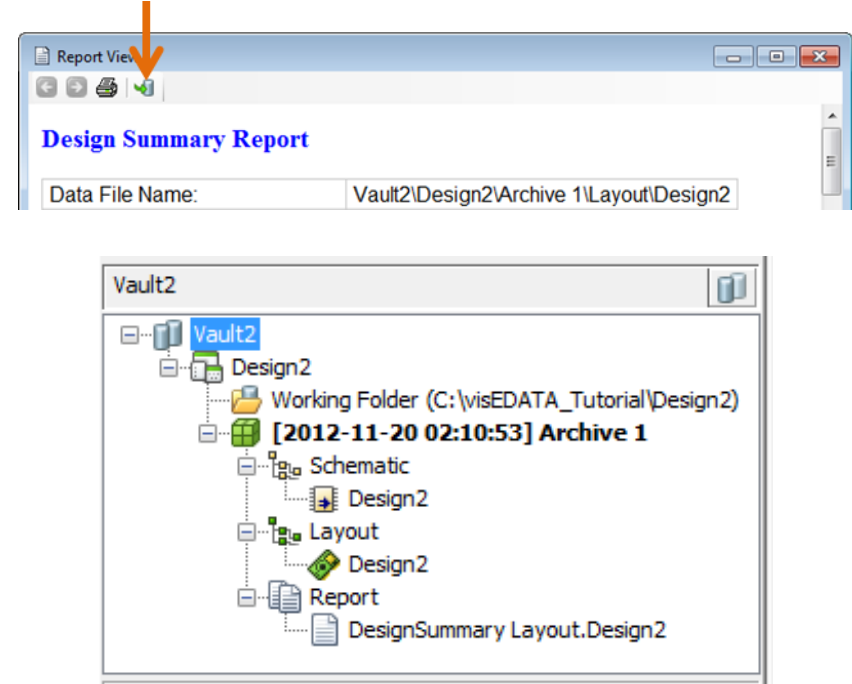


1. Click the **Report Icon > Design Summary Report**



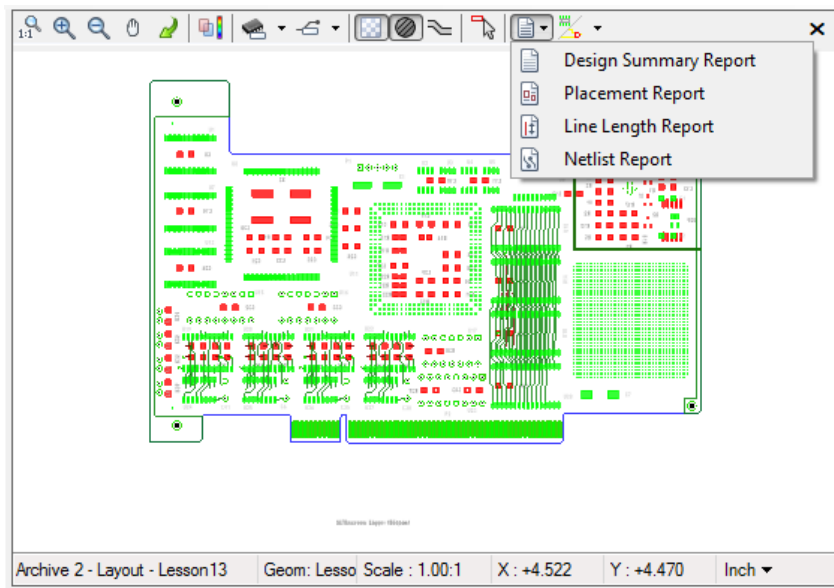
2. View the report information
 - a. Can you find the number of layers? *HINT: Board Information section*
 - b. Can you find the number of total components? *HINT: Component Information section*
 - c. Which geometry has the largest pin count? *HINT: Land Pattern Information*

3. Click the **Save report to vault** icon



NOTE: Notice the report is *auto-named* **DesignSummary Layout.Design2** and saved in the archive node in the vault view

4. Click **Close** to close the report
5. Create & View the other reports
 - a. **Report Icon > Placement Report**
 - b. **Save the Reports** to the Vault
 - c. Click **Close** to close the report
 - d. Repeat for each report (Line Length Report & Netlist Report)



6. Click the **Zoom 1:1** button to see the entire PCB
7. Do **NOT** close the preview window

Cross-Probing

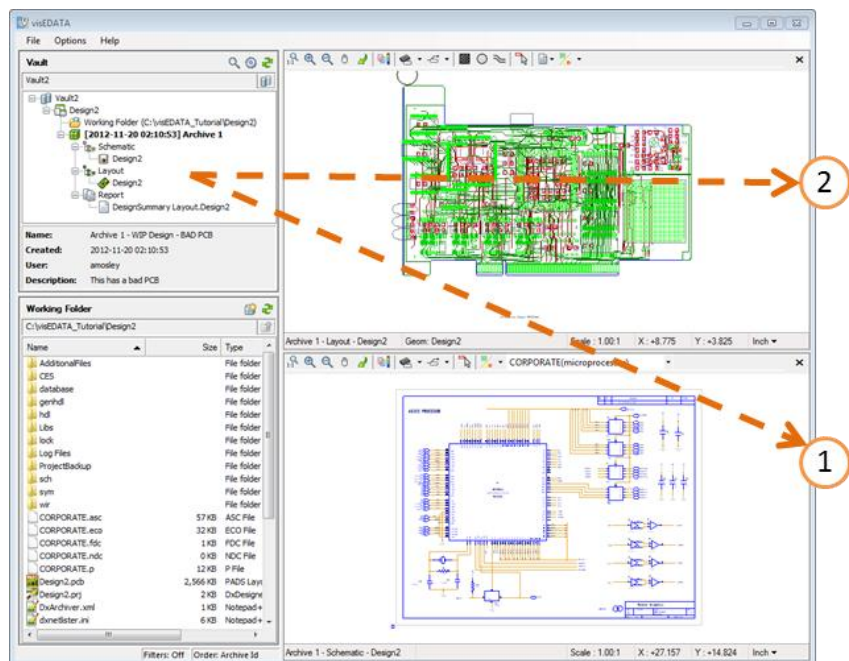
Cross-Probing is the process of synchronizing a Schematic and Layout graphical preview so that you can select an element in one, and auto highlight and zoom in another. To cross-probe, you must have a Schematic and Layout graphic from the **same** archive opened.

When activated, the system will automatically attempt to link the objects PCB and Schematic component & nets and report any mismatches. You can imagine they will most likely not be 100% in sync if the front-end or back-end has not been synchronized. After this sync process, the user is presented with a summary of the Components & Nets that have by linked together and if they want to continue. The user can immediately choose to view a detailed log file of the synchronization that will document any unmatched items such as unused pins or mechanical elements that may not have a schematic representation.

NOTE: Schematic and Layout views must be opened from the same archive in order to cross-probe.

LAB: Selecting the Schematic and Layout Previews

1. In the Vault View, select the **Archive 1 > Schematic > Design2** object and **Double-Click** it to open it
2. Select the **Archive 1 > Layout > Design2** object and **Double-Click** it to open it



LAB: Activating the Cross-Probe Function

1. In either the Schematic or Layout preview, select the **Cross-Probe** button

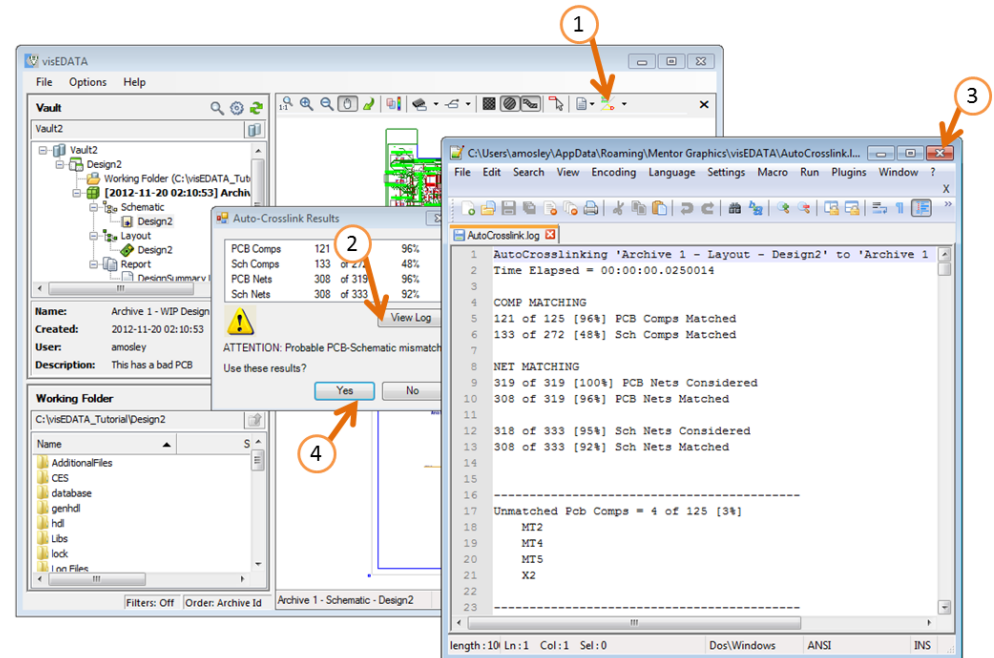



NOTE: Upon selection the PCB and Schem is analyzed for mismatches. If there are mismatches, you can approve the results to continue the process.

2. Click the **View Log** button to see the differences

NOTE: Some component may not have been placed, are mechanical, or are not for packaging (e.g. sheet borders)

3. Close the report window, **Click (X)** in report window
4. Click **Yes** to use these results

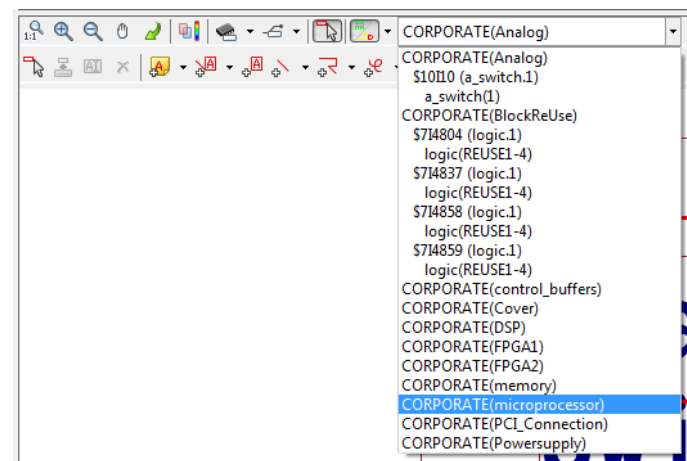


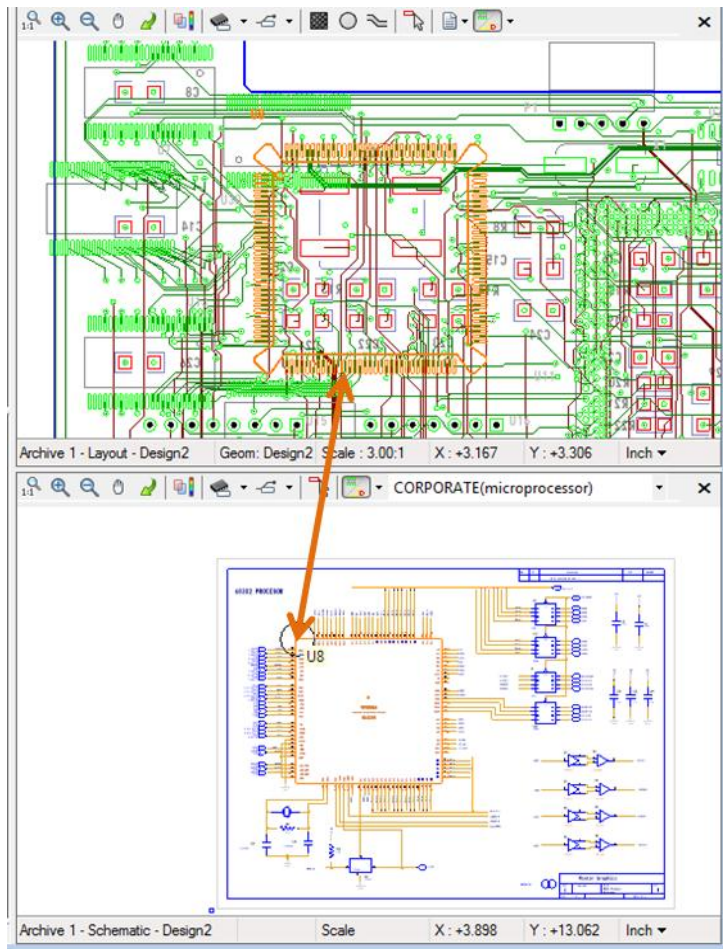
NOTE: Once in cross-probe mode, the icon will stay depressed {  }

LAB: Cross-Probing Between Views

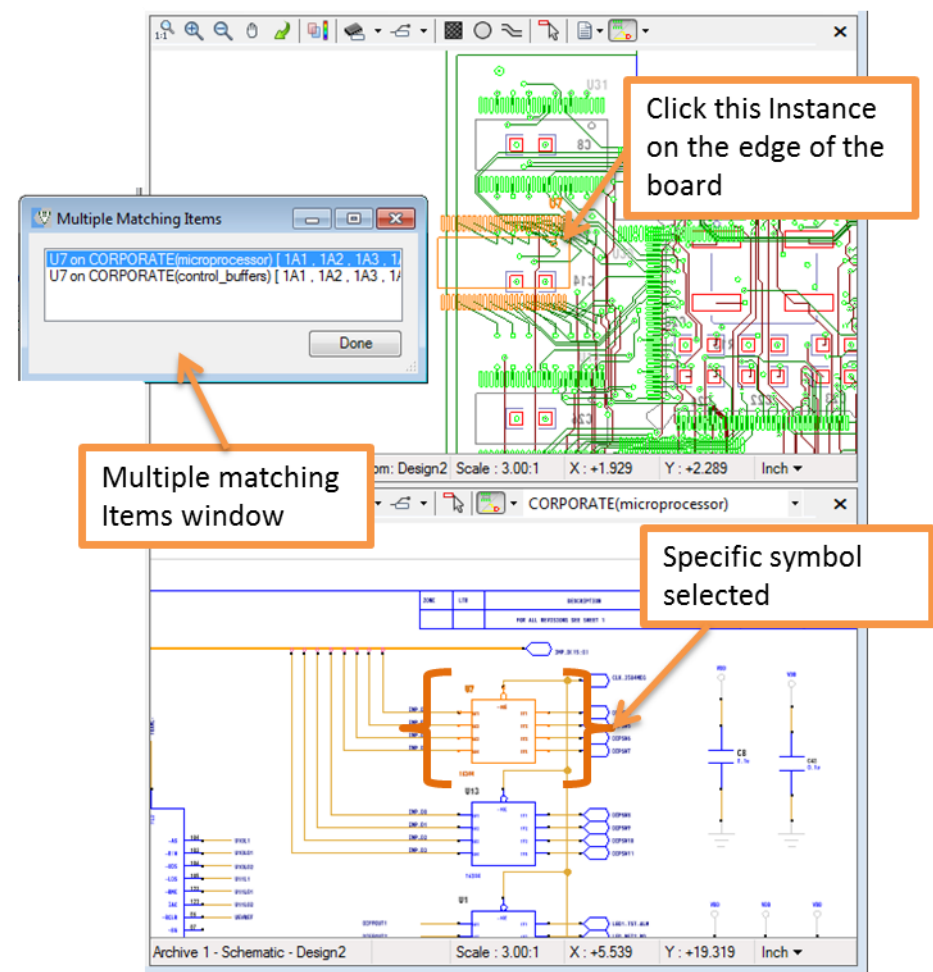
1. In the Schematic preview, select the **CORPORATE(microprocessor)** sheet\ as shown in the image to the Right.
2. Zoom into the schematic
3. Select the **border** of one of the symbols

NOTE: The Layout view will automatically zoom to the instance as shown in the image below



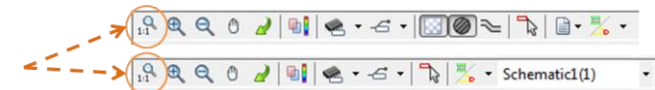


Cross-Probe Schematic to Layout

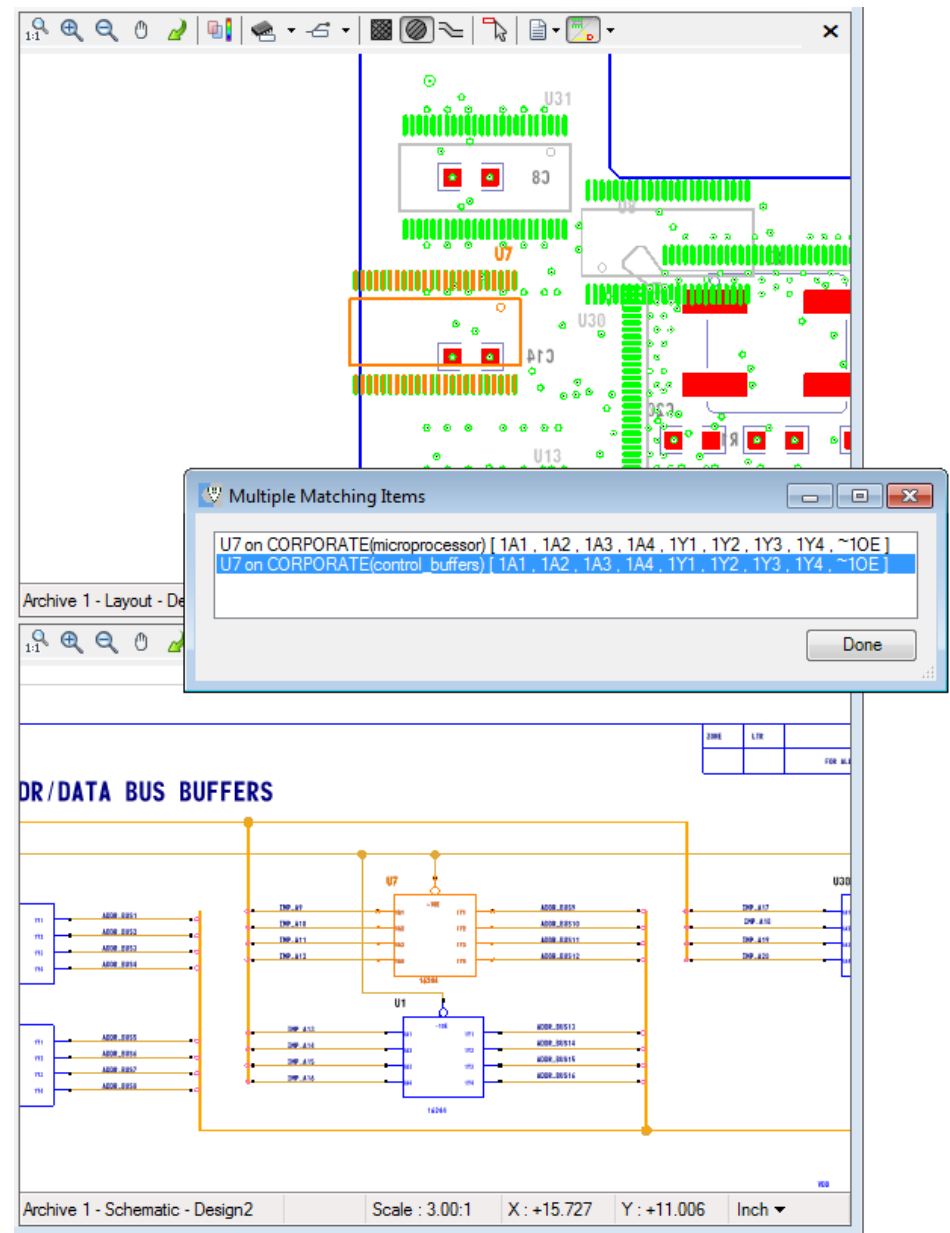


Cross-Probe Layout to Schematic

4. Click the **Zoom 1:1** icon in **each** of the graphic previews
 5. In the Layout graphic, select the instance that is on the edge of the board
- NOTE: Since this instance has two gates, the **Multiple Matching Items** window will be displayed
6. **Select the first item** to have it highlighted in the schematic



7. **Select the second item** to have it highlighted in the schematic
8. Click **Done** to close the Multiple Matching Items window
9. In either window, attempt to select a Net (Schematic) or Trace (Layout) and notice how visEDATA cross-highlights



Module 4: Project Reporting (Proj Mngr)

visEDATA allows you to generate HTML reports from the Layout Graphic and also HTML compare reports from different schematic and layout archives. These reports will be displayed in an external HTML window and the report can be printed or saved to the vault alongside the respective archive using an automatic & simplified naming convention. Some reports are multi-level and contain links.

TABLE: Available Design Comparison Reports between Archive Data

When Comparing:	You Can Compare this Data
2 schematics	Netlists Data
2 layouts	Netlists Data Graphical
A schematic and a layout	Netlists

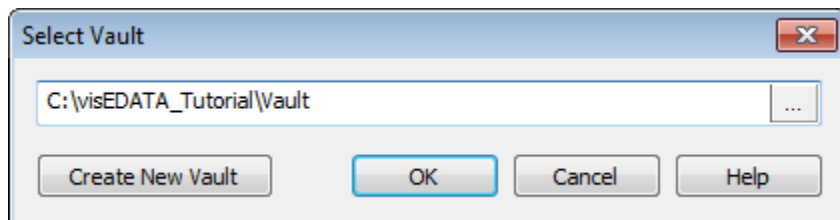
Netlist Report

A Netlist comparison report can be generated between any design object and will identify differences with the following:

- Unique Components
- Unique Component Pins/Net Names
- Unique Net Names
- Netname Differences with Identical Connectivity
- Component-Pin Connectivity Differences

LAB: Schematic to Layout Netlist Data Report

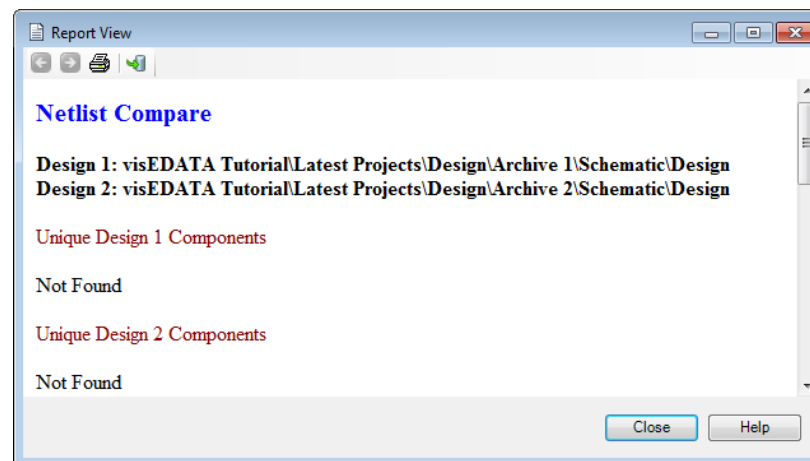
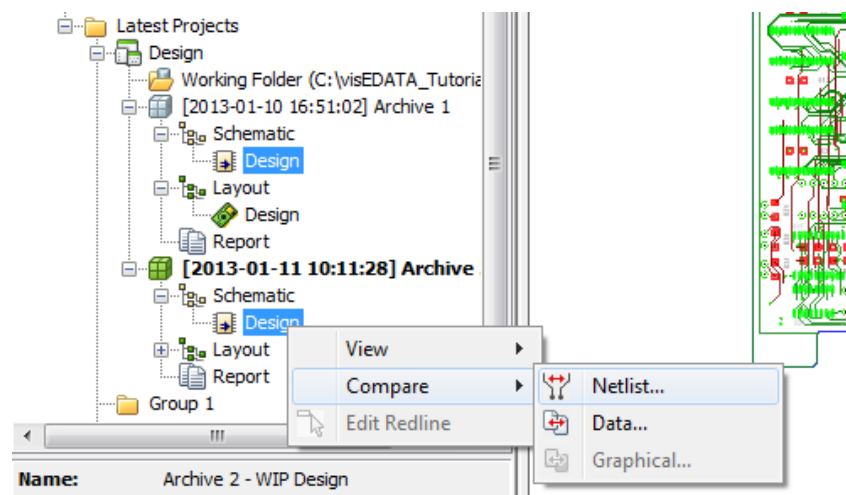
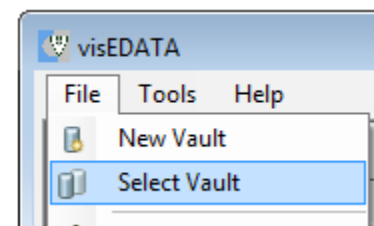
1. Click the **File > Select Vault** from the application menu
2. Navigate to your original vault **C:\visEDATA_Tutorial\Vault** directory



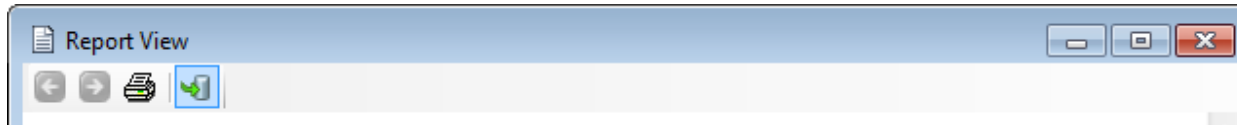
3. Click **OK**
4. Select the Schematic > Design from **Archive 1** and the Layout > Design good from the **Archive 2** design
5. Click the **RMB > Compare > Netlist**

NOTE: The report is broken down into multiple sections to outline the netlist differences.

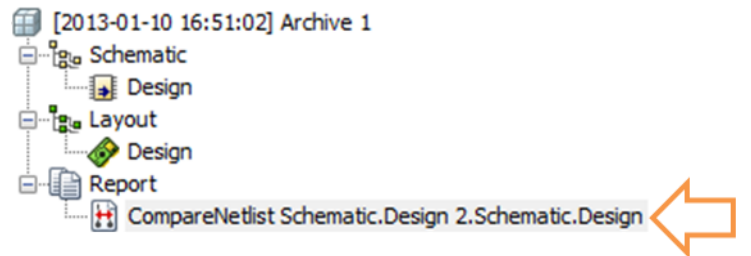
- Unique Design 1 Components
- Unique Design 2 Components
- Unique Design 1 Component Pins
- Unique Design 2 Component Pins
- Unique Design 1 Net Names
- Unique Design 2 Net Names
- Netname Differences with Identical Connectivity
- Component-Pin Connectivity Differences



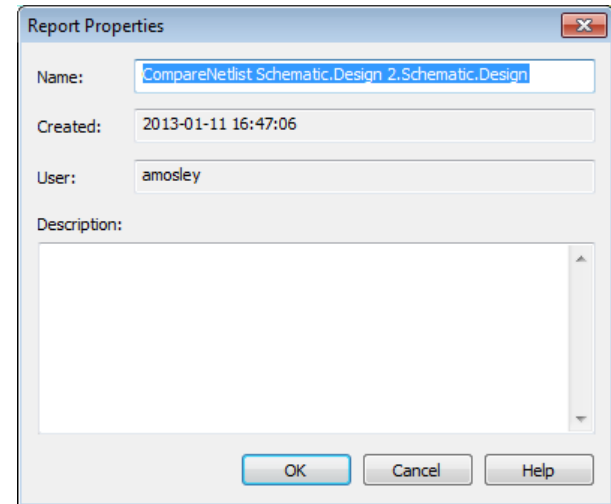
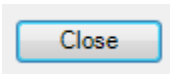
6. Click the **Save report to Vault** button



NOTE: You will notice that the report is **Auto-named** and associated to the schematic in the Vault (see image below). You can modify the name and description of a report by selecting it in the vault and select **RMB > Properties** (see image to the right)



7. Click the **Close** button to close the window



OPTION: You can select any combination of *Schematic* and/or *Layout* views and generate Netlist Comparison Reports

Data Report

A Data Report comparison can be performed between two *Schematic* archive items for a **Component Attributes** comparison, or two *Layout* archive items for a **full** comprehensive report as shown below.

LAB: Schematic Comparison Data Report

1. Select the *Schematics* from the **Archive 1** and **Archive 2** designs
2. Click the **RMB > Compare > Data**

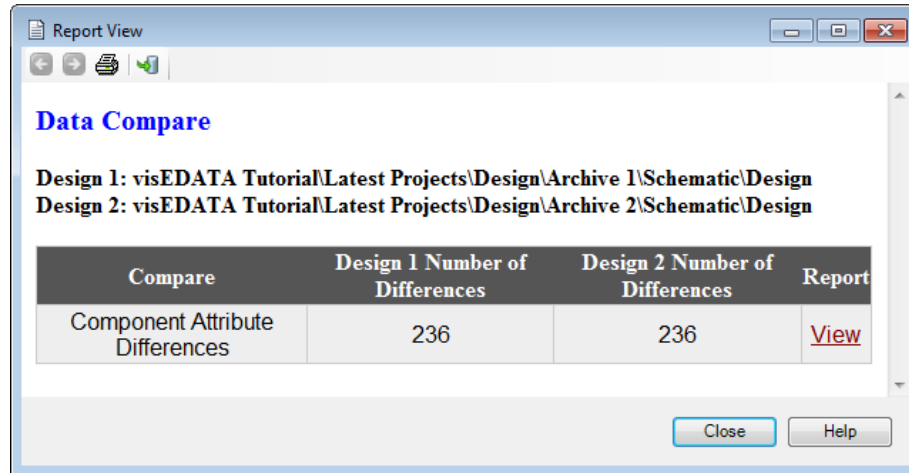
NOTE: Only the **Component Attributes** is available

Data Compare compares the following items:

- Component Placement
- Component Attributes
- Component Geometry
- Pin Geometry
- Test Point
- Via
- Drill hole
- Traces
- Layers
- Net Names
- Net Pins
- Keywords
- Placed Probes
- Unplaced Probes
- Tooling Hole
- Fiducial
- Generic Component
- Composite Component



3. Click **OK** to generate the report

NOTE: The report overview window will open,



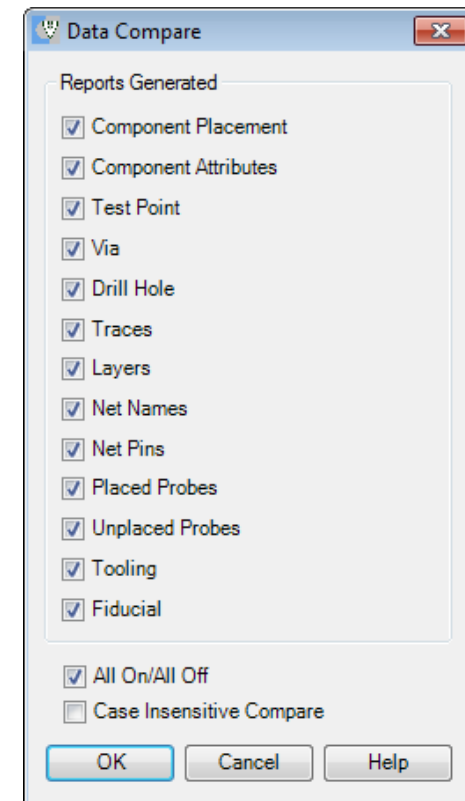
4. click **View** within the report window to see the detailed report

NOTE: The detailed report will open within the same window.

5. Click the **Save report to Vault** button 
6. Click the **back arrow**  to return to the main report window
7. Click the **Close** button to close the window

LAB: Layout Comparison Data Report

1. Select the **Archive 1 > Layout > Design_bad** and the **Archive 2 > Layout > Design_good** layouts within the vault navigator
2. Click the **RMB > Compare > Data**
NOTE: Unlike the Schematic, the Layout comparison contains a multitude of report options as shown in the image to the right.
3. Click **OK** to generate the report



4. The report overview window will open, click **View** to see the detailed differences

Data Compare

Design 1: My Vault\visEDATA Testing\TEST\Archive 1\Layout\Design1
Design 2: My Vault\visEDATA Testing\TEST2\Archive 1\Layout\Design2

Compare	Design 1 Number of Differences	Design 2 Number of Differences	Report
PCB Component Differences	2	2	View
Component Attribute Differences	0	0	N/A
Unique Test Points	0	0	N/A
Via Differences	51	1091	View
Drill Hole Differences	51	1091	View
Trace Differences	0	0	N/A
Unique Trace Elements	574	0	N/A
Layer Differences	5	0	N/A
Unique Net Names	0	0	N/A
Net Pin Differences	0	0	N/A
Unique Net Pins	0	0	N/A
Placed Test Probe Differences	0	0	N/A
UnPlaced Test Probe Differences	0	0	N/A
Unique Tooling Holes	0	0	N/A
Fiducial Differences	0	0	N/A

PCB Component Differences

Design 1: My Vault\visEDATA Testing\TEST\Archive 1\Layout\Design1
Design 2: My Vault\visEDATA Testing\TEST2\Archive 1\Layout\Design2

PCB Component Differences

Design Ref	Des	Geom Name	X	Y	Surface	Place	Bottom	Mirror	Rotation	Scale
1	U1	SOP48	450	3300	Top	no	no	no	270 degrees	1
2	U1	SOP48	450	3300	Top	no	no	no	180 degrees	1
1	U7	SOP48	375	2655	Top	no	no	no	180 degrees	1
2	U7	SOP48	450	2640	Top	no	no	no	180 degrees	1

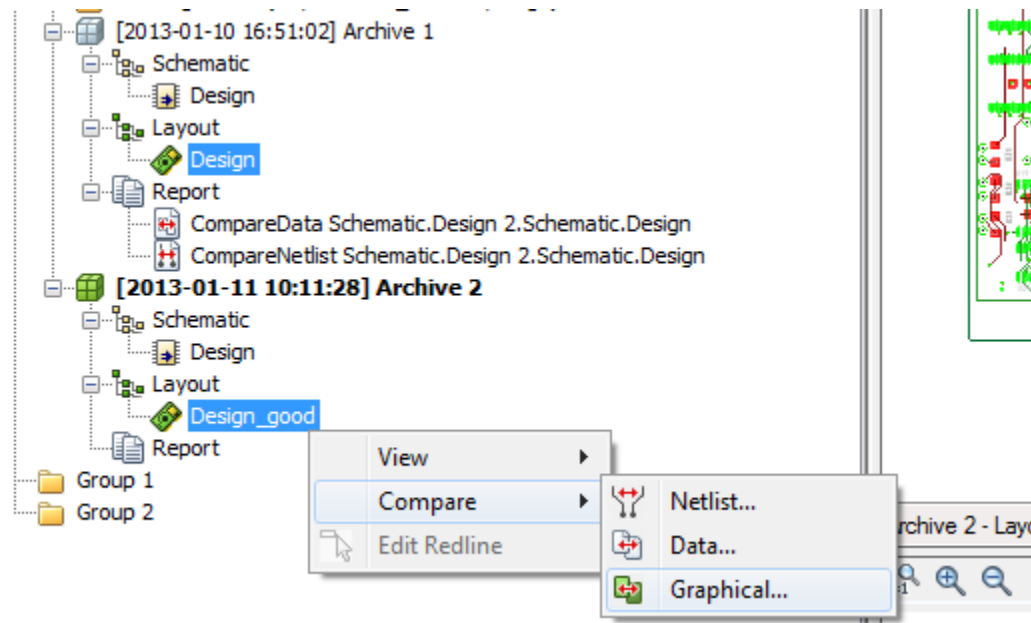
5. Click the **Save report to Vault** button
6. Click the **Close** button to close the window

Graphical Report

LAB: Comparing Two PCB/Layout Views

A Graphical Report comparison can only be performed between two *Layout* archive items.

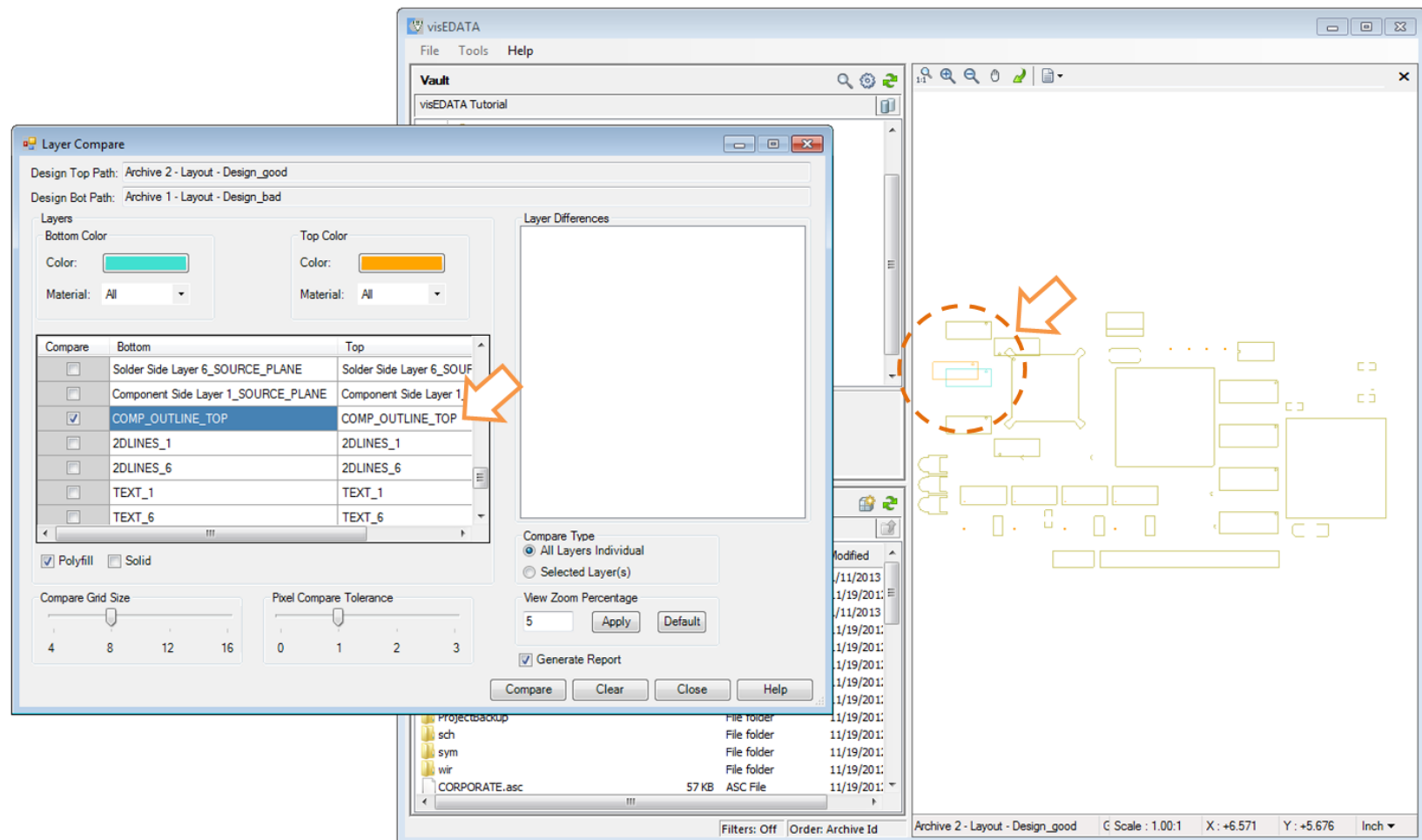
1. Select the **Archive 1 > Layout > Design_bad** and the **Archive 2 > Layout > Design_good** layouts within the vault navigator
2. Click **RMB > Compare > Graphical**



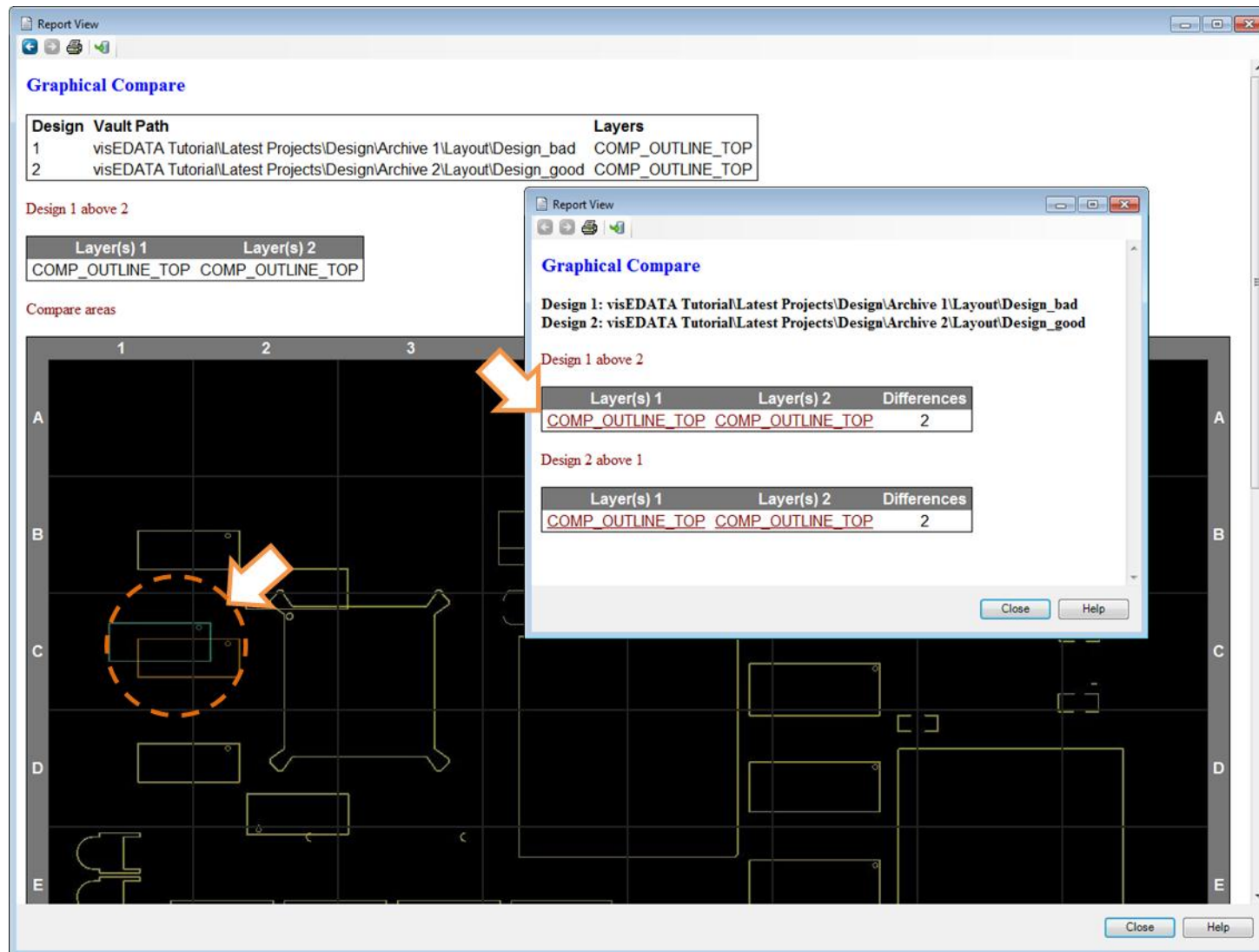
NOTE: In the **Layer Compare** dialog box, you can *interactively* select which layers you want to compare and see them in the visEDATA window.

3. To see an example of a layout difference, *scroll down* and select the and **COMP_OUTLINE_TOP** layer

NOTE: You will notice the differences are color-coded based on the designs (**Top/Bottom**) that were selected.

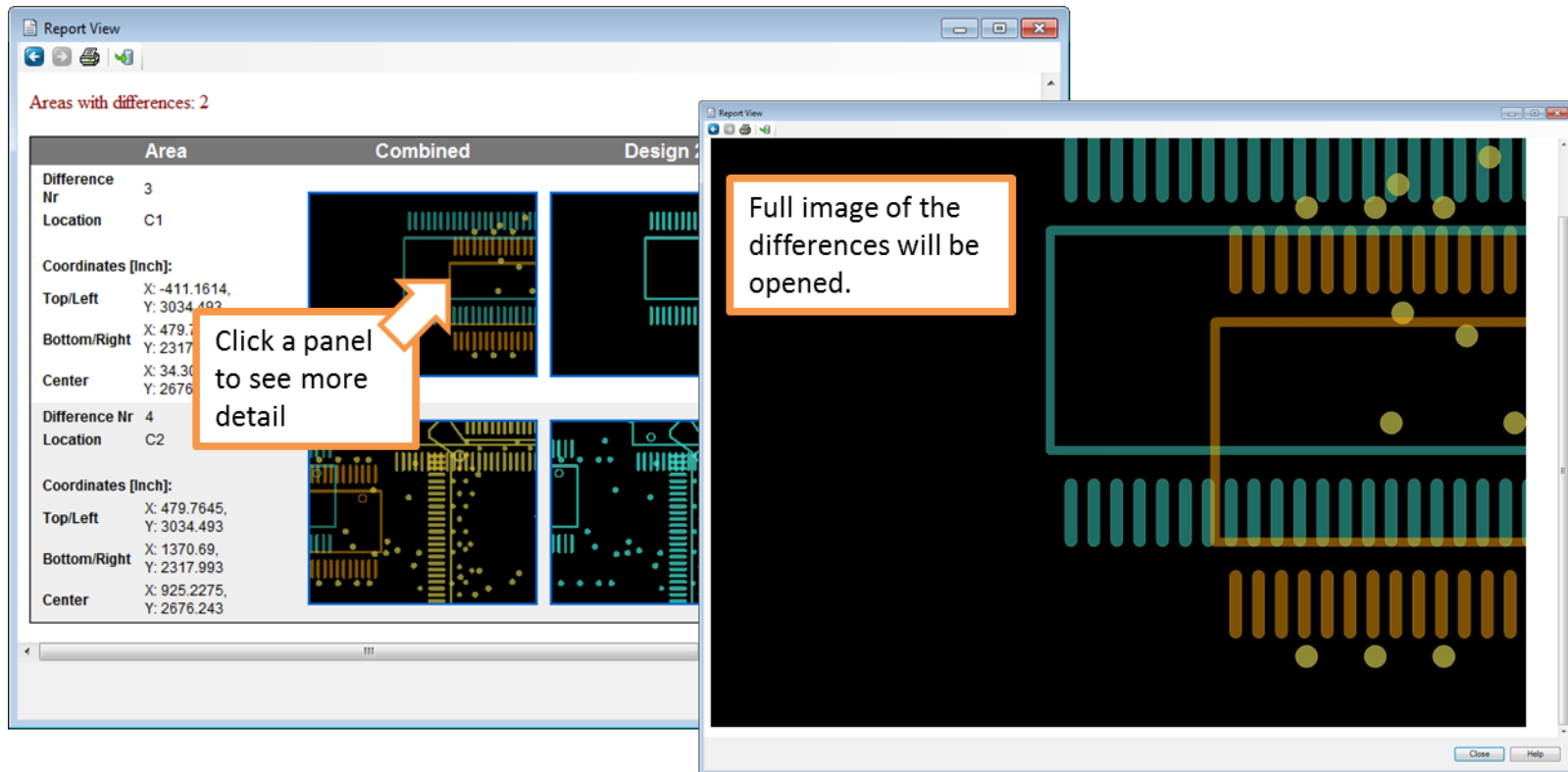


- Click the **Compare** button to see a “hierarchical” report of the comparison



- Click the **Save report to Vault** button 

5. Click the **Close** button to close all of the report windows
6. In the Layer Compare dialog box, you can “explore” other reports by selecting other layers like the **PADLAYER_TOP** to generate reports similar to the following:



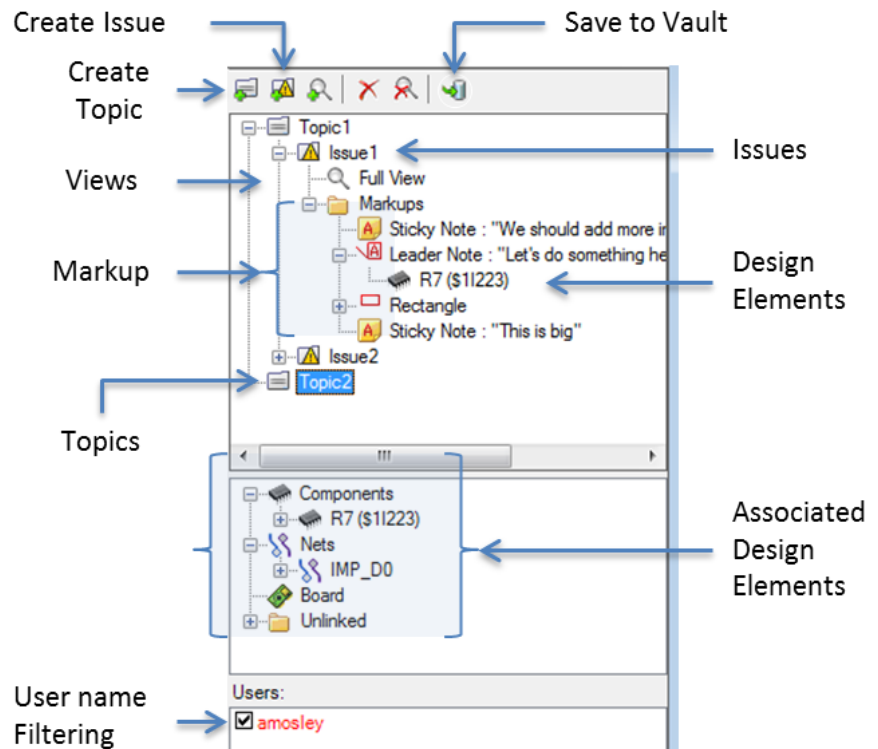
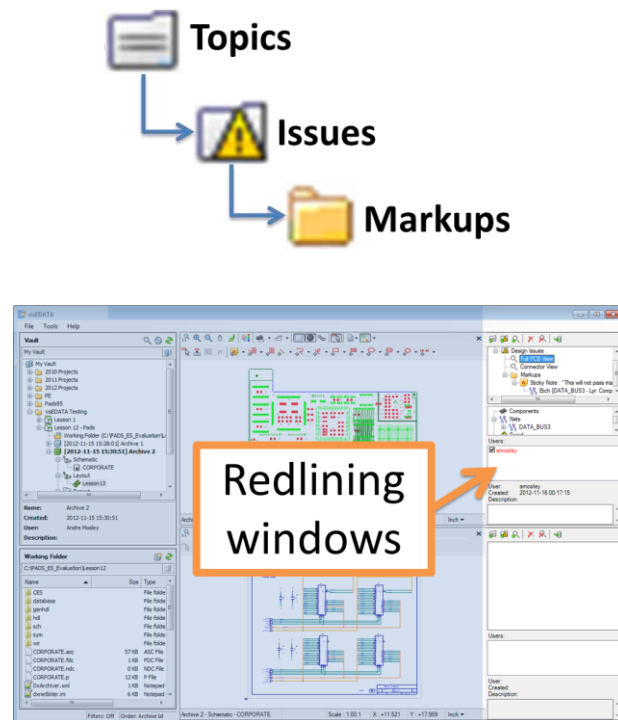
NOTE: Clicking one of the image panels will display a larger view of the differences

7. Click the **Close** button to close all of the report windows
8. Click **Close** on the Layer Compare dialog

Module 5: Collaboration/Redlining (All)

To redline a design, you must first select an archive within the vault and then select either the Schematic or Layout views. Within those views, the user can toggle the Redline Toolbar and add a multitude of redline options including notes, views, line items, and measurements to communicate issues. This input is automatically associated to a user and optionally to specific design elements making it easy to filter this information.

Redline information is organized by **Topics**, Topics contain **Issues**, and issues contain **Views** and/or **Markups**.



NOTE: Once a Topic or Issue is saved to the vault it cannot be renamed. Additionally, the Schematic or Layout view associated to it will have a new icon to represent markup information is available. A Topic or Issue cannot be deleted if it contains markup from another person.

Icon	Redline	Control
	Select Redline	NA
	Associate Redline	NA
	Edit Text	NA
	Delete Redline	NA
	Sticky Note	Color, Text Value
	Leader Note	Color, Text Value
	Text	Text Value
	Line	Color, Thickness, Arrows

Icon	Redline	Control
	Polyline	Color, Thickness, Arrows
	Freehand	Color, Thickness, Arrows
	Rectangle	Color, Thickness
	XOut Rectangle	Color, Thickness
	Ellipse	Color, Thickness
	XOut Ellipse	Color, Thickness
	Cloud	Color, Thickness
	Dimension	Color, Thickness

NOTE: If collaboration has been added to the views, then a “redline” icon will be denoted on the archive, Schematic, or Layout.



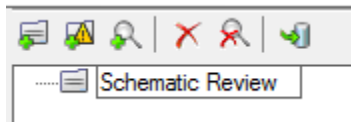
If you do not name your Topic and/or issue, “**Issue 1**” will automatically be created within the **Topic 1** node. You can optionally create and/or modify issues within the discussion area to fit your collaboration needs.


TIP: When adding a markup for the first time, Topic 1 will automatically be created. If you have multiple Topics within your Discussion Window, you will be prompted to select a Topic and Issue before proceeding.

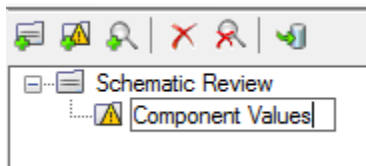
LAB: Creating Topics & Issues

TIP: To modify the name of a Topic, Issue, or Markup/Redline, click **F2** on the item

1. Open the Schematic Graphic by selecting **View > Design** or by **Double-clicking** the **Design**
2. Click on the **Redlining** icon
3. Click the **Add Topic** icon and change the name to **Schematic Review**



4. Select the **Schematic Review** Topic and lick the **Add Issue** icon 
5. Change the issue to **Component Values**

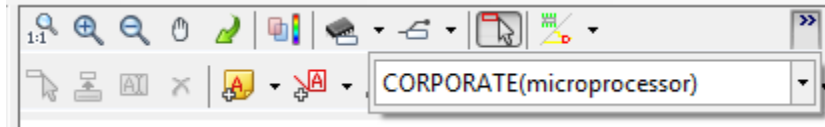



LAB: Creating Markups & Views

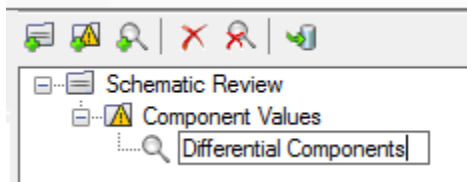
LAB: Adding a View


NOTE: A view is a location as opposed to a specific schematic - e.g. use "Sheet Border"

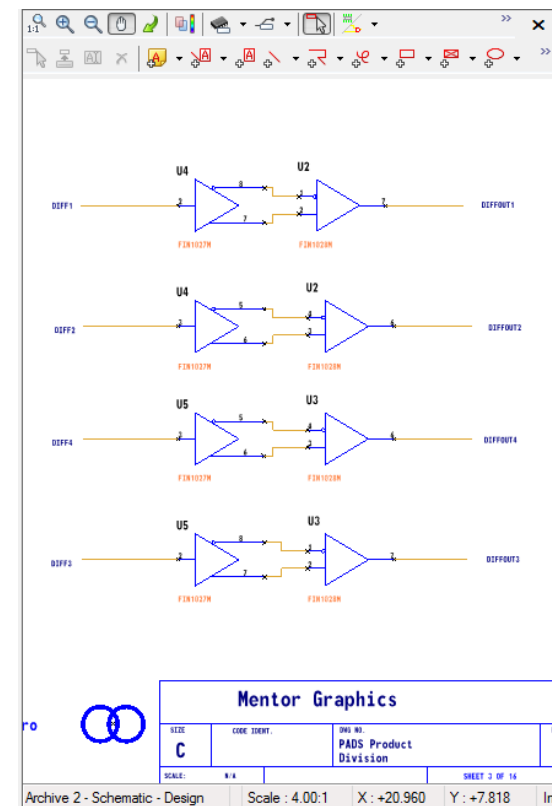
1. In the Schematic Graphic select the **Corporate(microprocessor)** sheet



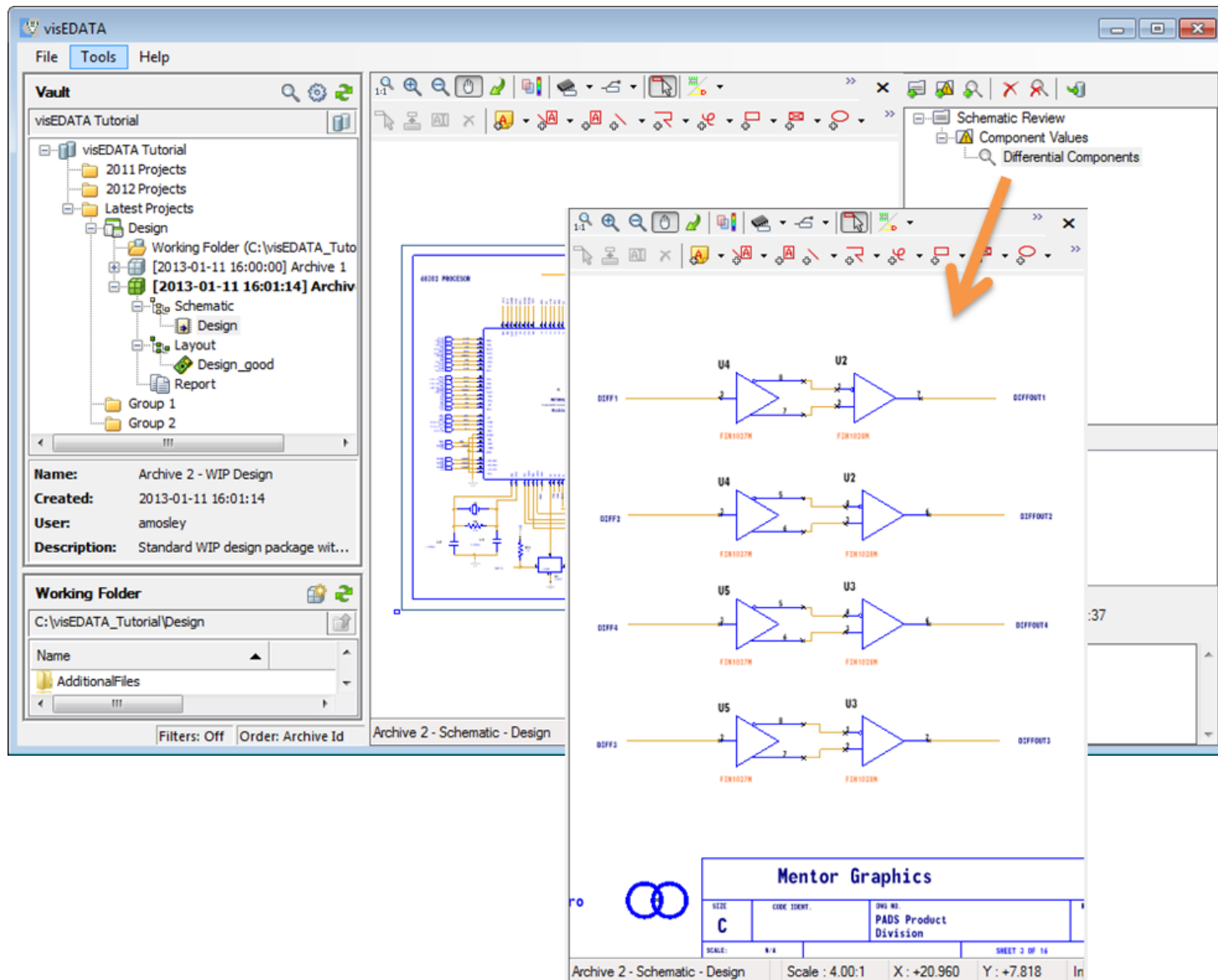
2. Select the **Zoom In** icon and pan to the location as shown on the right →
3. Select the **Add View** icon  in the
4. Change the name of the view to be **Differential Components**



5. Zoom out of the schematic by selecting the **Zoom 1:1** icon ()

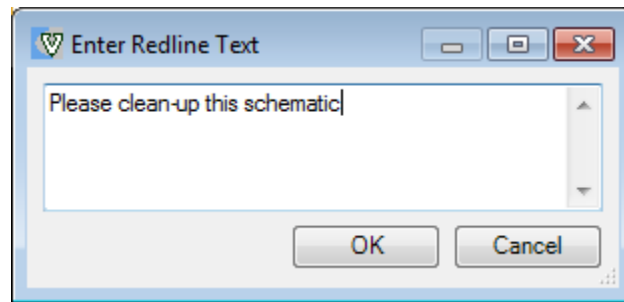


6. Now select the **Differential Components** view to have the sheet view set to the redline markup setting

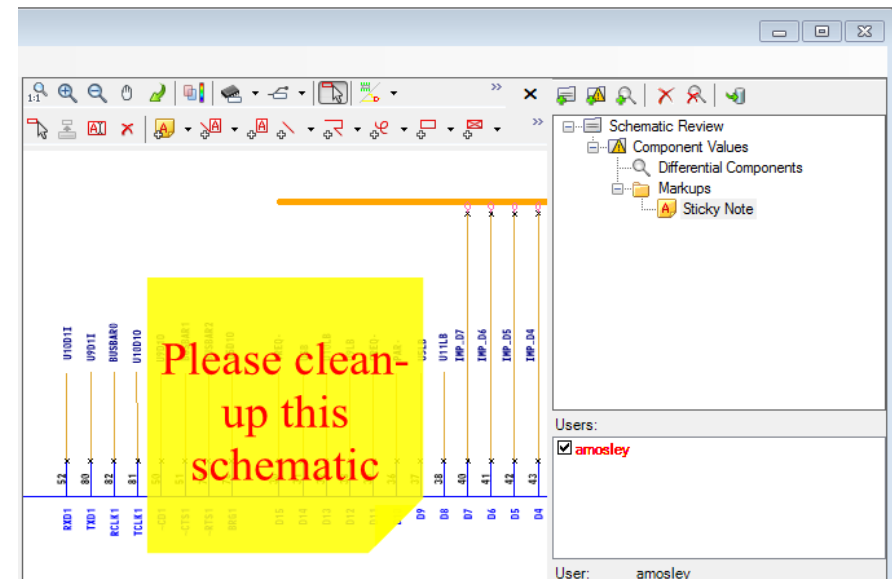


Creating a Sticky Note

7. Select the **Sticky Note** redline option
8. **Draw a box** within the schematic to create the note
9. Enter Redline Test **"Please clean-up this schematic"**





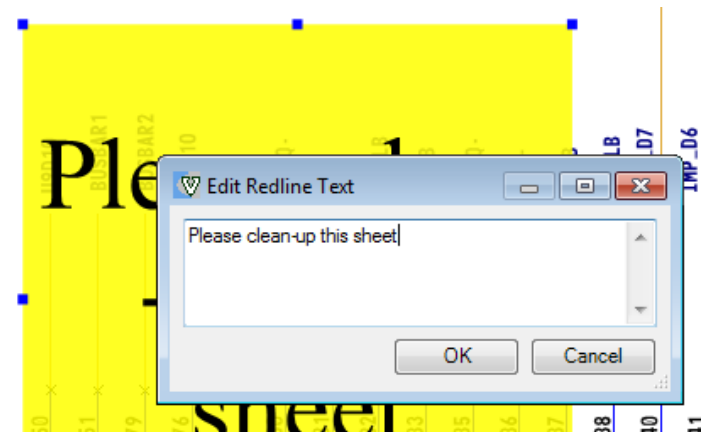
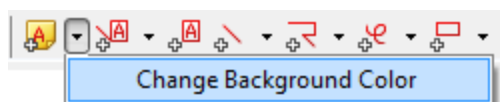
10. Click **OK** to create the Sticky Note



NOTE: You will notice that the markup is added to the previous issue as **Sticky Note** to the previous issue (**Differential Components**) you were working on.

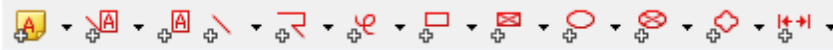
Modifying Existing Redline/Mockups

11. To change a mockup, select the **Select Redline** icon ()
12. Click on the **Sticky Note** (or other redline) that you've created
13. Click the **Edit Redline text** icon ()
14. Change the text to **"Please clean-up this sheet"**
15. Click **OK** to save the changes
16. Select the **center** of the Sticky Note to **drag it to another location**
17. Select the **exterior handles** to **resize** the sticky note
18. Select the Sticky Note **drop-down** icon to change the background color of the sticky note



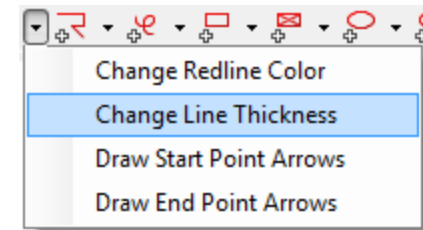
Experimenting with other Redline/Mockups options


19. In the schematic graphic **zoom out** and **experiment** with the other redline options




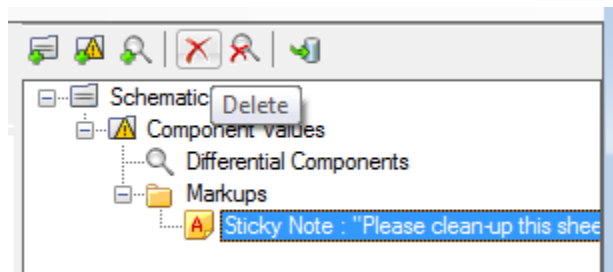
NOTE: Each markup has specific options that can be modified such as

- Color
- Thickness
- Text, and
- Arrow Points if applicable)



20. To delete any markup, select the markup and then select the **delete markup** icon ()

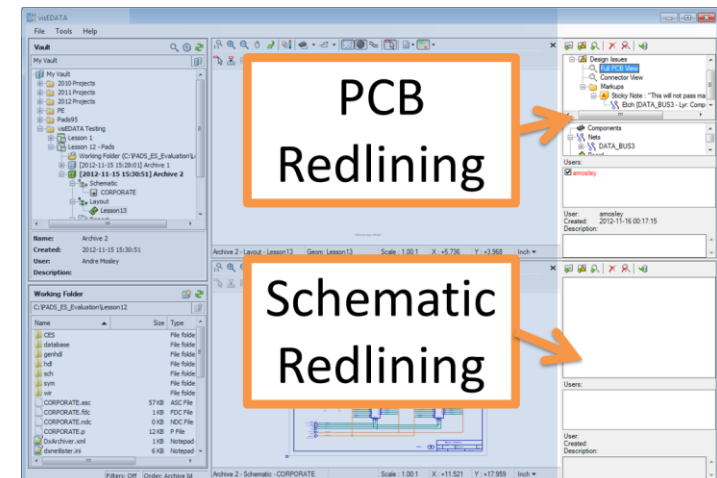
NOTE: You can also select the markup in the Redline window and click the **delete** icon () – As a precaution, the delete key can **NOT** be used to delete the markup.



PCB/Layout Redlining

NOTE: You can experiment with similar usecases for the Layout Redline. You will notice a specific redline window will be added to that view to manage the Topics, Issues, and associated markup..

1. OPTIONAL: Open the PCB Redline window and add Topics and Issues
2. OPTIONAL: Experiment with redline/markup similar to the Schematic markup



LAB: Design Elements and Redline/Markup

When creating a markup for your design, you can optionally associate the markup to a design elements including Components, Nets, and Boards. This capability allows greater accessibility and reference within the design and upon associating a design element, it will immediately be shown in the Discussion window and allow cross-highlighting to the schematic and layout views.

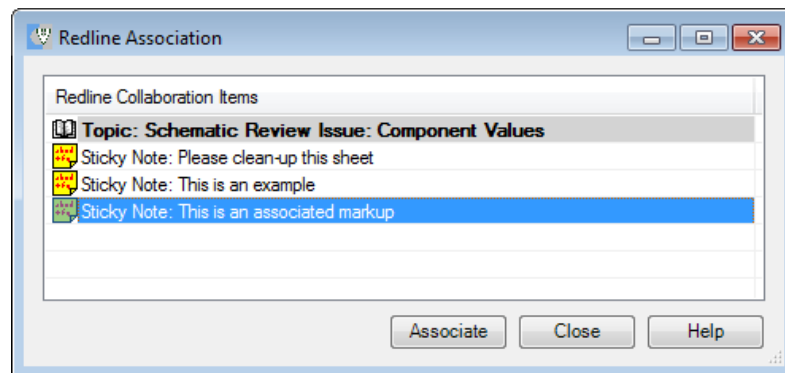
TIP: For selecting design elements, select the border of the element (i.e. not the symbol graphic center)

Associating a design element

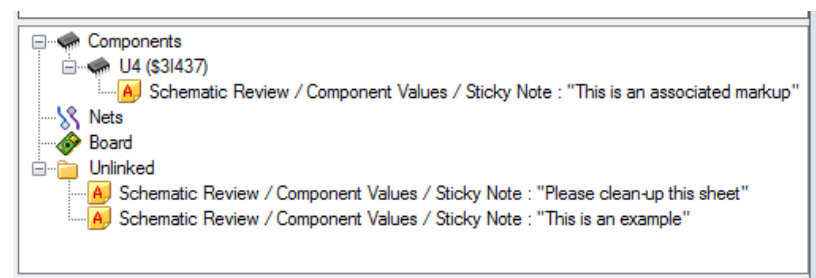
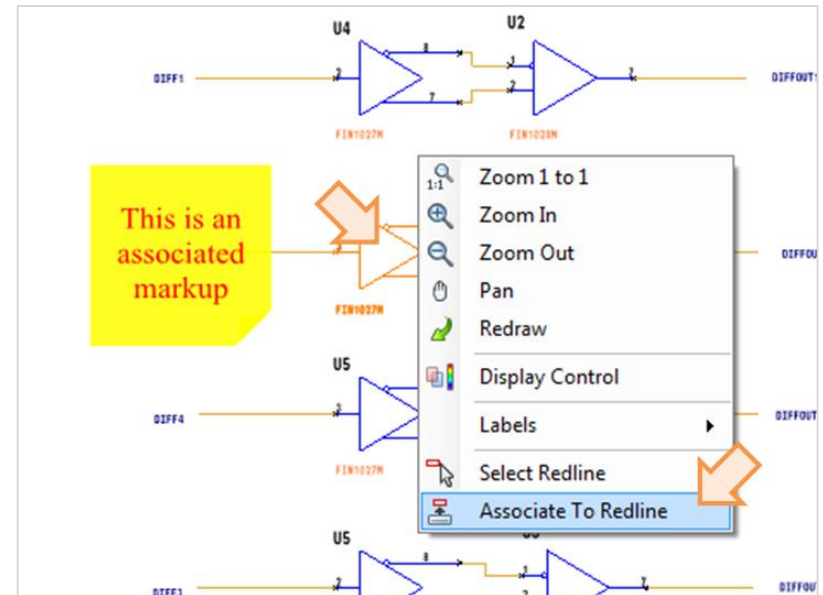
1. Open the Schematic graphic view - Select the CORPORATE(microprocessor) sheet if it is not already selected
2. Add a **Sticky Note** to the schematic
3. Make sure the Select Redline icon is not pressed and the pan icon is not pressed



4. Zoom into the schematic and select the border of any symbol to highlight it
5. Click the RMB > **Associate to Redline**




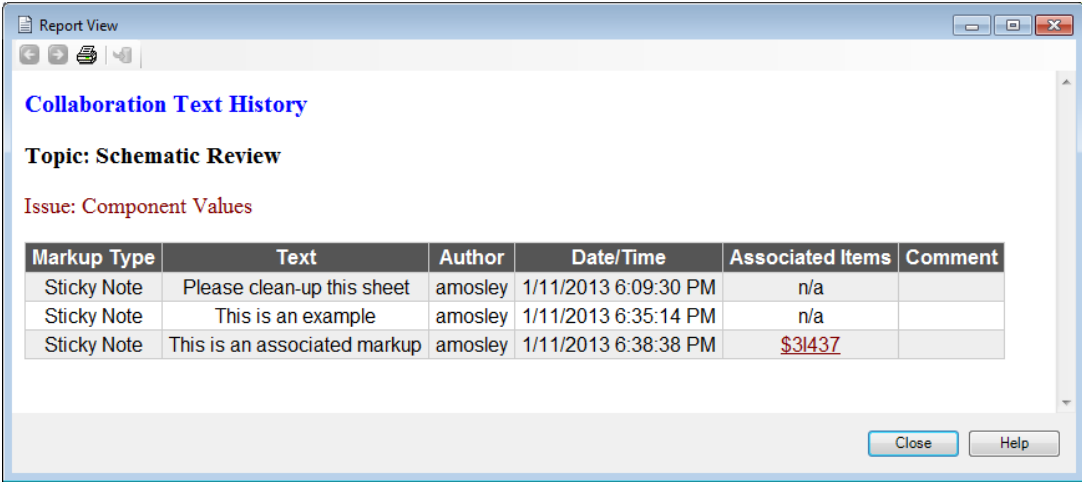
6. Select a redline and click **Associate**
7. You will now see the relationships in the Redline window



LAB: Saving the Redline to the Vault

Once you are done adding redline to the Schematic and/or PCB, you must save it to the vault to allow other users to access it. If you do not save the redline data to the vault, all changes will be removed.

1. In the schematic graphic redline window, select the Save to Vault icon ()
2. Click **OK** to save the design to the vault
3. Once completed, a Collaboration Text History report will be displayed to show all of the Redline that was added



Report View

Collaboration Text History

Topic: Schematic Review

Issue: Component Values

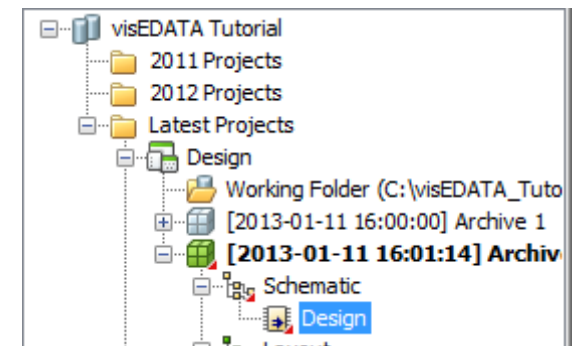
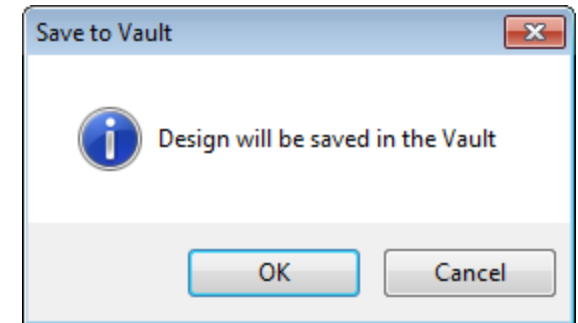
Markup Type	Text	Author	Date/Time	Associated Items	Comment
Sticky Note	Please clean-up this sheet	amosley	1/11/2013 6:09:30 PM	n/a	
Sticky Note	This is an example	amosley	1/11/2013 6:35:14 PM	n/a	
Sticky Note	This is an associated markup	amosley	1/11/2013 6:38:38 PM	\$3I437	

Close Help

NOTE: Within the report, you can cross-highlight to the associated items

4. Click **Close** to close the report window
5. Within the Vault window you will notice that the icons change to represent that redlines are present
6. Close the visEDATA window

NOTE: This completes the tutorial



End of section. This page intentionally left blank.

Conclusion

This Tutorial has introduced you to some of the power and flexibility available in visEDATA, but that is just a beginning. Obviously, this Tutorial has only scratched the surface and presented you with an abbreviated view into the full spectrum of the archive capabilities of the product. If you would like to learn more about the many features and functions of this application, there is a wealth of knowledge to experience and discover in the extensive resources that are available.

Additional resources Include:

- Extensive Help files
- User Manuals
- The InfoHub

For more information or to find your local sales representative, please visit:

<http://www.mentor.com/>

LAST PAGE:



Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.
Telephone: 503.685.7000
Toll-Free Telephone: 800.592.2210
Website: www.mentor.com