

SOLIDWORKS Electrical Data Recovery without SQL

Disclaimer: This is only a partial recovery. This process should only be used if the data could not be recovered through stitching back together or if the SQL cannot be recovered.

If you have the data from the ProgramData\SOLIDWORKS Electrical folder **some** data can be recovered! Without the SQL, items that CANNOT be restored are the Manufacturer Part Manager and the Cable Reference Manager.

Titleblocks

Start with the recovery of the Titleblocks

- Open SOLIDWORKS Schematic
- Open Library Tab
- Open Titleblock Manager
- Select the Import DWG Files icon
- Select either Add files or Add a folder – depending on the selection desired
- Browse to the location of wanted Titleblocks
 - By default, Titleblocks are located under
C:\ProgramData\SOLIDWORKSElectrical\Titleblock
 - This location may vary depending on where the data might be located for the recovery
- Select Open in the dialog window
- Select Next – a summary will appear of the process that will take place
- Fields can be changed prior to import – like Class selection, Description, Library, etc. These fields can also be changed later after import.
- Select Next
- Lastly, you will not need to save the settings for the import – select Do Not Save
- Press Finish

Your Titleblocks have now been recovered and put back into the software using the collaborative service.



Symbols

Continue with the Schematic Symbols, Line Diagram Symbols, and Cabinet Footprints. They all fall into the same folder. It is not easy to decipher until the symbols are imported. Once done, they can then be reassigned to the proper symbol type.

- Open SOLIDWORKS Schematic
- Open Library Tab
- Open Symbols Manager
- Select the Import DWG Files icon
- Select either Add files or Add a folder – depends on selection desired
- Browse to the location of wanted Symbols
 - By default, symbols are located under C:\ProgramData\SOLIDWORKSElectrical\Block
 - This location may differ depending on where the data might be located for the recovery
- Select Open in the dialog window
- Select Next – a summary will appear of the process that will take place
- Fields can be changed prior to import – like Class selection, Description, Library, etc. These fields can also be changed later after import.
- Select Next
- Lastly, you will not need to save the settings for the import – select Do Not Save
- Press Finish

Your Symbols have now been recovered and put back into the software using the collaborative service. These symbols do not have any metadata -this information will need to be added back into each symbol.

Important: Do not forget to change the type of symbol! (e.g., Schematic, Line Diagram, Cabinet Footprint, Connection Label, Passive Symbol)

Description, Library, and Classification selections will also need to be designated.

Project Templates

Now the libraries have been updated, we will proceed to the Project Template.

- Open SOLIDWORKS Schematic
- Open Project Manager
- Select the UNARCHIVE icon
- Browse to the location of the project templates
 - By default, project templates are located under C:\ProgramData\SOLIDWORKSElectrical\ProjectTemplates
 - This location may differ depending on where the data might be located for the recovery
- The template does not need to be opened once it is unarchived. Simply browse to the ALL PROJECTS tab to HIGHLIGHT the newly created project.

- Select SAVE AS TEMPLATE in the Program Manager window
- Rename template as desired.
- Project in the Project Manager Window can now be deleted.

When creating new projects from the Project Manager, the selection for the added templates can be used.

Projects

Lastly, we can process the legacy projects.

These projects originally had metadata that was managed by the SQL. Without the SQL, that metadata has been lost (project name, contract number, project description, locations, location descriptions, and more). However, if you are at least wanting to capture the actual schematic creation then some of the design can be restored.

These are a bit more difficult to handle.

- Open File Explorer
- Browse to the location of the projects
 - By default, projects are located under C:\ProgramData\SOLIDWORKSElectrical\Projects
 - This location may differ depending on where the data might be located for the recovery
- Each project has a few subdirectories – in most cases – Drawings, SOLIDWORKS, Titleblocks, XMLConfig

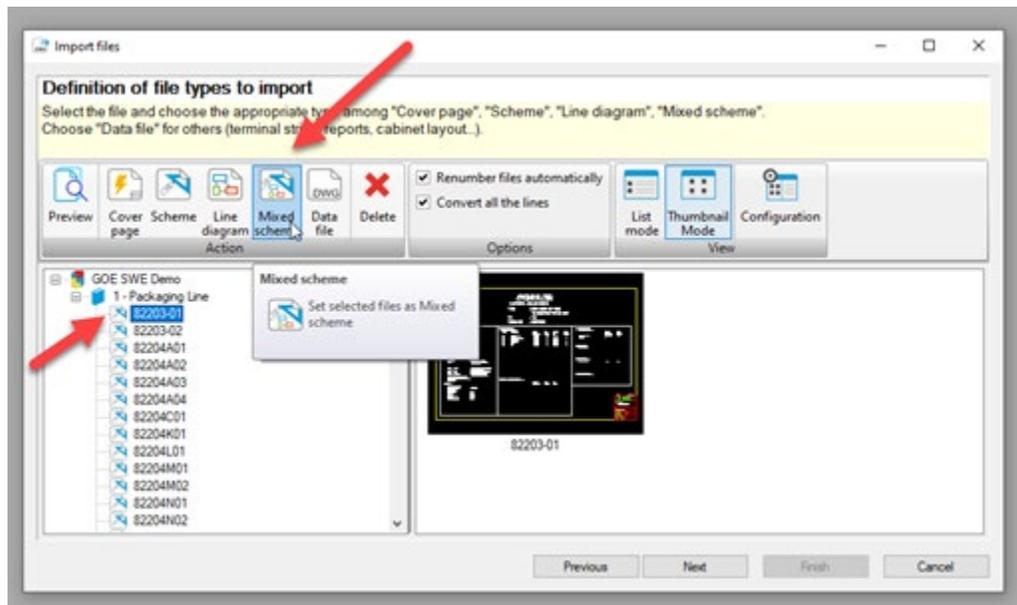
These project folders are numbered. The numbers are related to the SQL and the order in which they were created. Working from the largest numbered project backward to the lowest number directory will be the easiest since the largest number was the last project that was created prior to the loss of your data.

- Make a working directory where you can **COPY the files** from inside of the .../Projects/(Project Number)/Drawings directory and paste them.
- Rename the drawing extension from **.ewg to .dwg**. If a notification happens during this rename process, select YES.

Now that these files are prepped for use, we can proceed to bring them into the SOLIDWORKS Schematic software.

- Open SOLIDWORKS Schematic
- Open Project Manager
- Create a NEW project using whichever project template is required
- Delete the existing sheets inside of the Book of your project
- Select the Import/Export Tab
- Select the Import DWG Files
- Browse to the newly created directory where the copied and renamed files exist from the original project folder
- Use <no configuration>
- Select which book of your project the drawings will land into

The image below is the window that will then show up for the files that have been selected for Import. There is an icon next to the name of the file. This icon can then be changed to a Line diagram, Mixed Scheme but as default show as Scheme. Do not import any reports (terminal strip drawings, reports, design rule checks). These files, when highlighted, will show a preview on the right-hand side of the dialog box.



- Select Next
- Symbol and Titleblock import mapping then appears
 - These attributes and Titleblocks should be recognized by the software since those items are already in the software libraries
- Select Next
- Symbol attribute mapping will appear – again these items should be automatically recognized by the software
- Select Next
- Import options appear
- Leave the merge lines check box off
- Select Next
- Do not save the configuration
- Select Finish

Your files have now been added as drawings. These drawings will need to have the Properties filled in. There will be no component manufacturer parts assigned since this information was lost with the SQL. You will need to rearrange the drawings to put them in the order you desire.

Again, this is not a FULL recovery! It is only partial to at least retrieve the available information.

Lastly, **regular ENVIRONMENTAL ARCHIVES are the best and safest way to protect yourself and your company from lost data!**