

# elecworks™ Tips & Tricks

## Importing AutoCAD Electrical Drawings into elecworks™

elecworks™ has the ability to import AutoCAD®, AutoCAD® Electrical, Electrical Designer, Promis●E, Tracelec drawings whilst still maintaining the intelligence.

In this Tips and Tricks we are going to cover importing of AutoCAD Electrical data.

Create a project of the same name as the AutoCAD® Electrical project directory



Select

Select **IEC** as the template and

OK

Once created, the existing blank drawings of the newly created project can be deleted

Select the *Import/Export* ribbon tab



Select

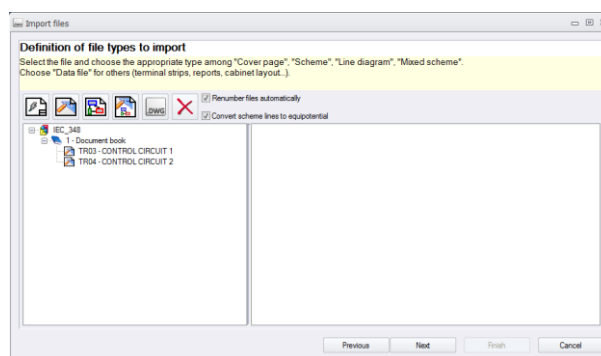
Browse to the AutoCAD® Electrical project directory and select

OK







Select the Import Configuration 'AutoCAD Electrical Import'

Select

Next

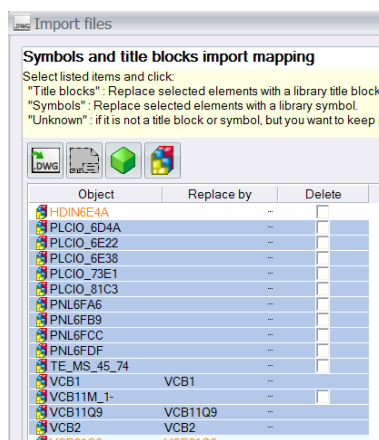
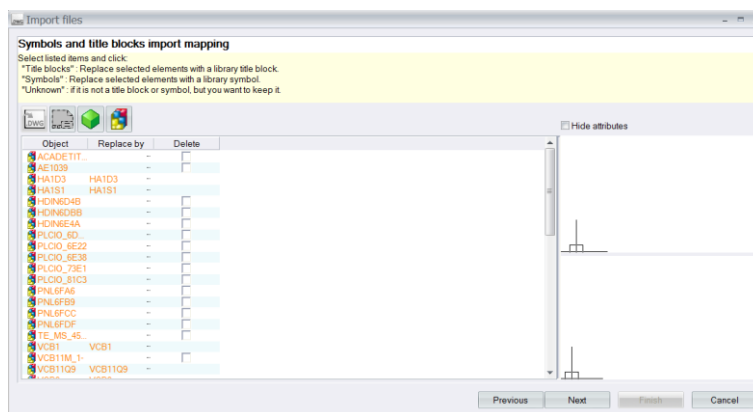



You can highlight specific drawings in the list and change the drawing type


-  Cover page
-  Schematic
-  Single Line diagram
-  Mixed single line and schematic drawing
-  Appendix or data drawing (non-intelligent)
-  Delete drawing


If drawings are shown to the right of the dialogue with a **X** then the drawing hasn't been saved correctly in AutoCAD and is corrupted.

Select



 You can highlight multiple blocks in the drawings and this command will save the blocks of each symbol in a zip file so that they can be imported and converted in your library


 Selecting this command will allow you to swap out the highlighted block with an elecworks™ title border

 Selecting this command allows you to swap out the AutoCAD® Electrical block with an elecworks™ block

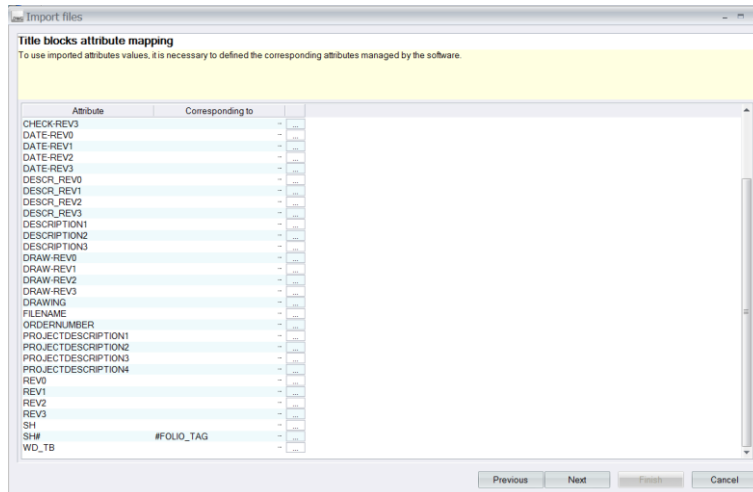
Multipole devices in AutoCAD Electrical are made up of a parent and child symbol e.g. a two pole circuit breaker is VCB1 and VCB2. When swapping this device, you can swap the parent device for a two pole circuit breaker and delete the child VCB2

Object	Replace by	Delete
PNL6FDF	-	<input type="checkbox"/>
TE_MS_45_74	-	<input type="checkbox"/>
VCB1	TR-EL018	<input type="checkbox"/>
VCB11M_1-	-	<input type="checkbox"/>
VCB11Q9	TR-TELE003	<input type="checkbox"/>
VCB2	-	<input checked="" type="checkbox"/>
VCB21Q9	-	<input checked="" type="checkbox"/>
VCR1	TR-EL053	<input type="checkbox"/>
VDIN6DF0	-	<input type="checkbox"/>
VGND2	TR-VX004	<input type="checkbox"/>
VLT1G	TR-EL087	<input type="checkbox"/>
VMO13_1-	-	<input type="checkbox"/>
VMO14	TR-EL092T	<input type="checkbox"/>
VMS1	TR-EL053	<input type="checkbox"/>
VMS11_1-	-	<input type="checkbox"/>
VMS21	-	<input checked="" type="checkbox"/>
VMS21P	TR-EL035	<input type="checkbox"/>
VPB11	TR-EL237	<input type="checkbox"/>
VPB12ML	TR-BT011	<input type="checkbox"/>
VT0002	TR-BR001	<input type="checkbox"/>

You will note from the example above, that components with an AutoCAD® Electrical block name of ###2\* have been marked for deletion as they are defined as a child device whereas a 2 pole or 3 pole device will replace the separate symbols that make up the device in AutoCAD® Electrical

The single line diagram symbols are done differently in AutoCAD® Electrical in comparison with elecworks so these haven't been mapped. (suffixed with a "\_1-"). You could however convert these symbols in elecworks™ by exporting them out 

Once your happy with the mapping, select

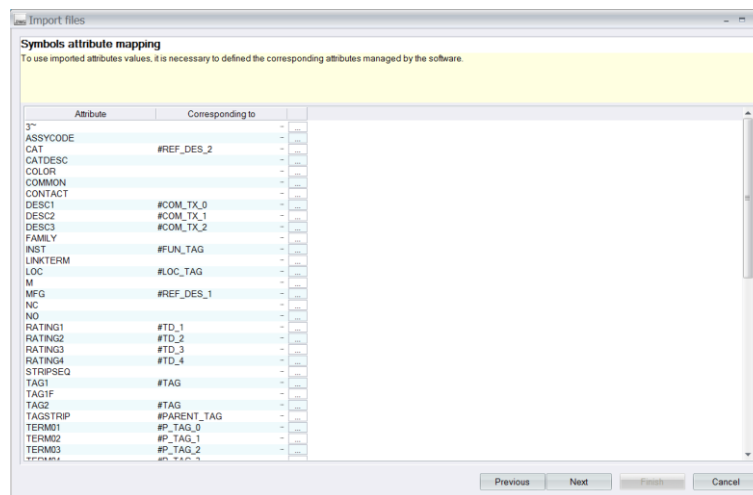


You can map the AutoCAD® Electrical attributes to the equivalent elecworks™ attributes.

Highlight the attribute from the list, select [...], select the appropriate elecworks™ attribute and then

Attribute	Corresponding to	
APPD-REV0	#REV_APPROV_BY_0	-- [...]
APPD-REV1	#REV_APPROV_BY_1	-- [...]
APPD-REV2	#REV_APPROV_BY_2	-- [...]
APPD-REV3	#REV_APPROV_BY_3	-- [...]
CHECK-REV0	#REV_VERIF_BY_0	-- [...]
CHECK-REV1	#REV_VERIF_BY_1	-- [...]
CHECK-REV2	#REV_VERIF_BY_2	-- [...]
CHECK-REV3	#REV_VERIF_BY_3	-- [...]
DATE-REV0	#REV_CREATED_ON_0	-- [...]
DATE-REV1	#REV_CREATED_ON_1	-- [...]
DATE-REV2	#REV_CREATED_ON_2	-- [...]
DATE-REV3	#REV_CREATED_ON_3	-- [...]
DESCR_REV0	#REV_NO_0_TZ_0_L1	-- [...]
DESCR_REV1	#REV_NO_1_TZ_0_L1	-- [...]
DESCR_REV2	#REV_NO_2_TZ_0_L1	-- [...]
DESCR_REV3	#REV_NO_3_TZ_0_L1	-- [...]
DESCRIPTION1	#FOLIO_TZ_0_L1	-- [...]
DESCRIPTION2		-- [...]
DESCRIPTION3		-- [...]
DRAW-REV0	#REV_CREATED_BY_0	-- [...]
DRAW-REV1	#REV_CREATED_BY_1	-- [...]
DRAW-REV2	#REV_CREATED_BY_2	-- [...]
DRAW-REV3	#REV_CREATED_BY_3	-- [...]
DRAWING		-- [...]
FILENAME		-- [...]
ORDERNUMBER	#FOLIO_NO	-- [...]
PROJECTDESCRIPTION1	#PROJECT_TZ_0_L1	-- [...]
PROJECTDESCRIPTION2		-- [...]
PROJECTDESCRIPTION3		-- [...]
PROJECTDESCRIPTION4		-- [...]
REV0	#REV_NO_0	-- [...]

Select



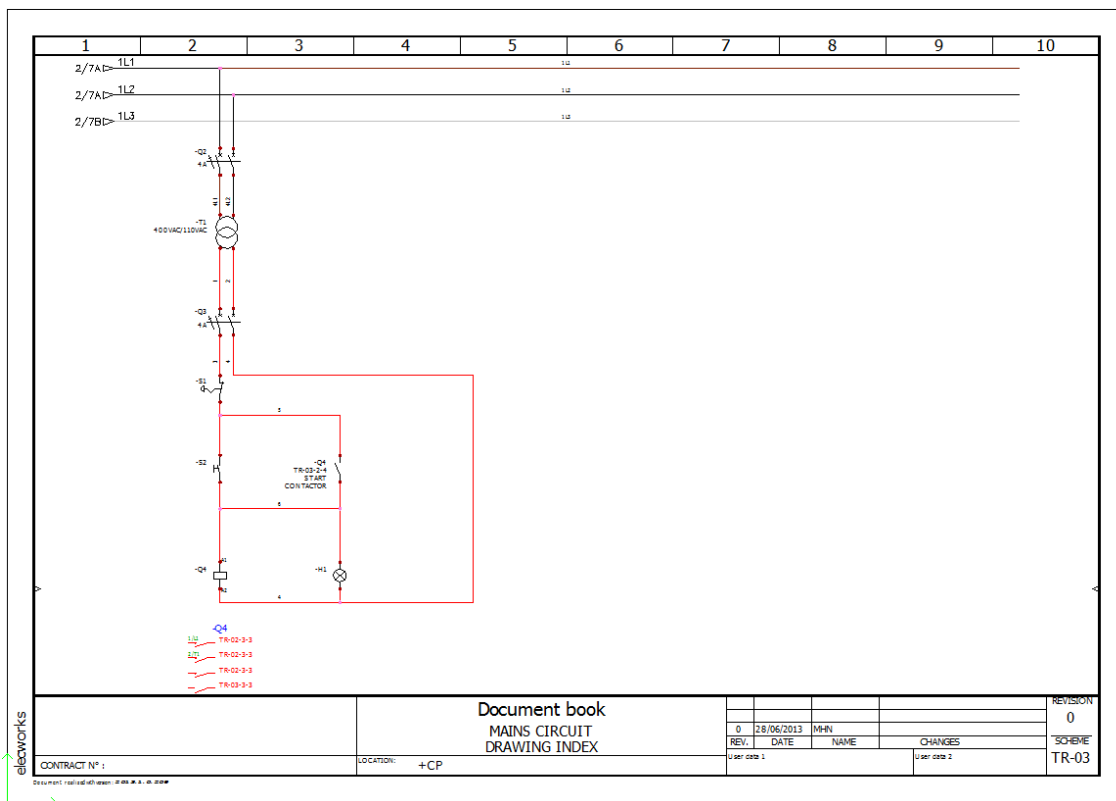
The attributes for the symbols are already premapped but the interface allows you to define any user defined attributes

Select

Elecworks saves the configuration so therefore the mapping gets better and more complete each time you import

Select  *Save current import configuration*

Select Finish



You drawings will be imported and converted

- If the equivalent symbol in elecworks is smaller than the AutoCAD® Electrical symbol you may have to stretch or extend the wire to the connection point
- A wire style can be changed through the circuit by right clicking on a wire > *Wire Style* > *Replace*
- The cross reference formula can be changed by right clicking over the project > *Configurations* > *Project*
- The AutoCAD® Electrical cross referencing in mtext can be deleted
- Origin and destination arrows can be added and the AutoCAD® Electrical ones removed

Autodesk and AutoCAD are registered trademarks of Autodesk Inc. in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders