elecworks™ Tips & Tricks

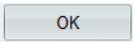
Importing AutoCAD Electrical Drawings into elecworks™

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

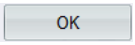
Select 

Select **IEC** as the template and 

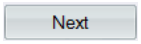
Once created, the existing blank drawings of the newly created project can be deleted

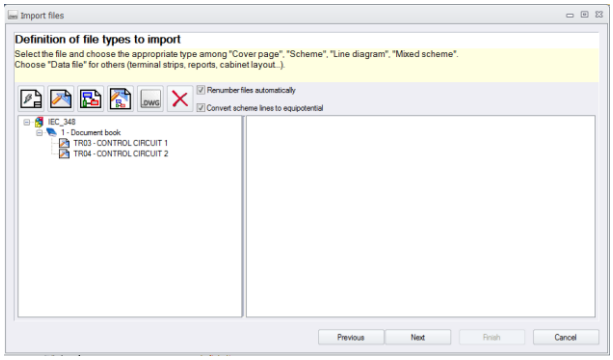
Select the *Import/Export* ribbon tab

Select 

Browse to the AutoCAD® Electrical project directory and *select* 

Select the Import Configuration 'AutoCAD Electrical Import'

Select 



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


Cover page


Schematic


Single Line diagram

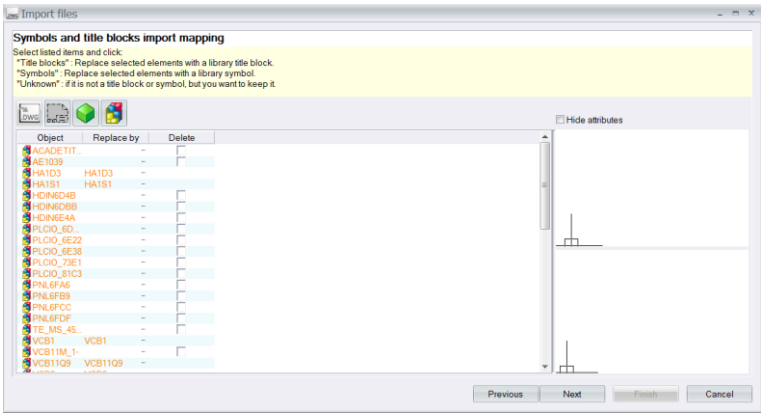

Mixed single line and schematic drawing

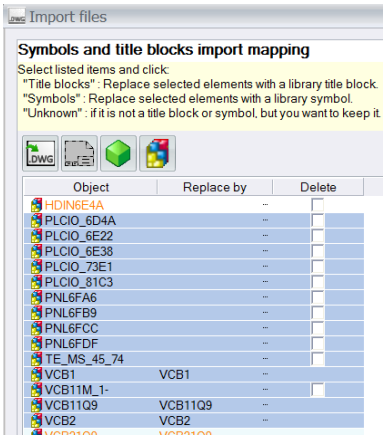

Appendix or data drawing (non-intlligent)


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 Next



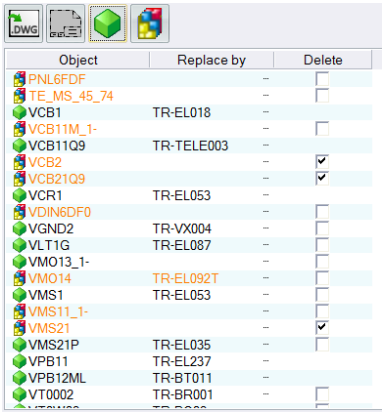


You can highlight multiple blocks in the drawings and this command will save the blocks of each symbol in a zip file so that they can be imported and converted in your library

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

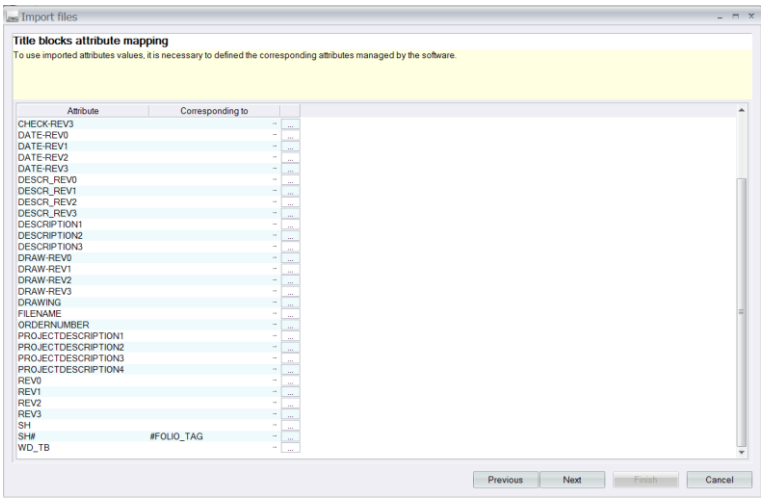


Object	Replace by	Delete
PNL6FDF		
TE_MS_45_74		
VCB1	TR-EL018	
VCB11M_1-		
VCB11Q9	TR-TELE003	
VCB2		✓
VCB21Q9		✓
VCR1	TR-EL053	
VDIN6DF0		
VGND2	TR-VX004	
VLT1G	TR-EL087	
VMO13_1-		
VMO14	TR-EL092T	
VMS1	TR-EL053	
VMS11_1-		
VMS21		✓
VMS21P	TR-EL035	
VPB11	TR-EL237	
VPB12ML	TR-BT011	
VT0002	TR-BR001	

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 seperate 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 havent been mapped. (suffixed with a "_1-"). You could however convert these symbols in elecworks™ by exporting them out 

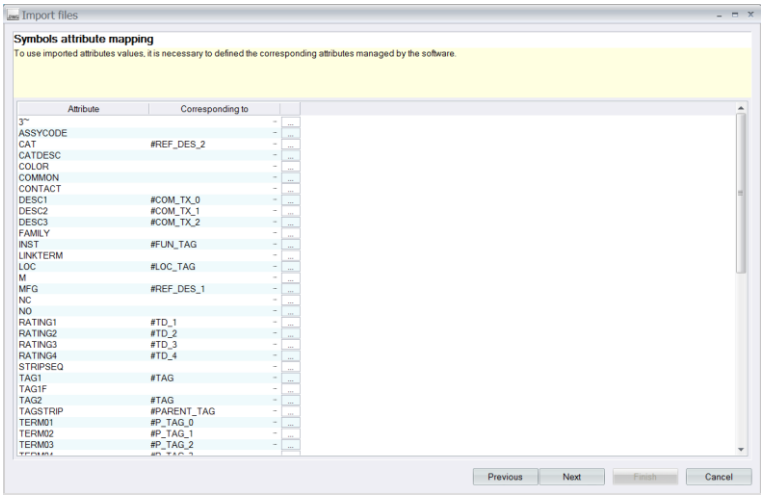
Once your happy with the mapping, *select* Next



You can map the AutoCAD® Electrical attributes to the equivelent elecworks™ attributes.
Highlight the attribute from the list, select […], select the apprpriate elecworks™ attribute and then Replace

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	

Select Next



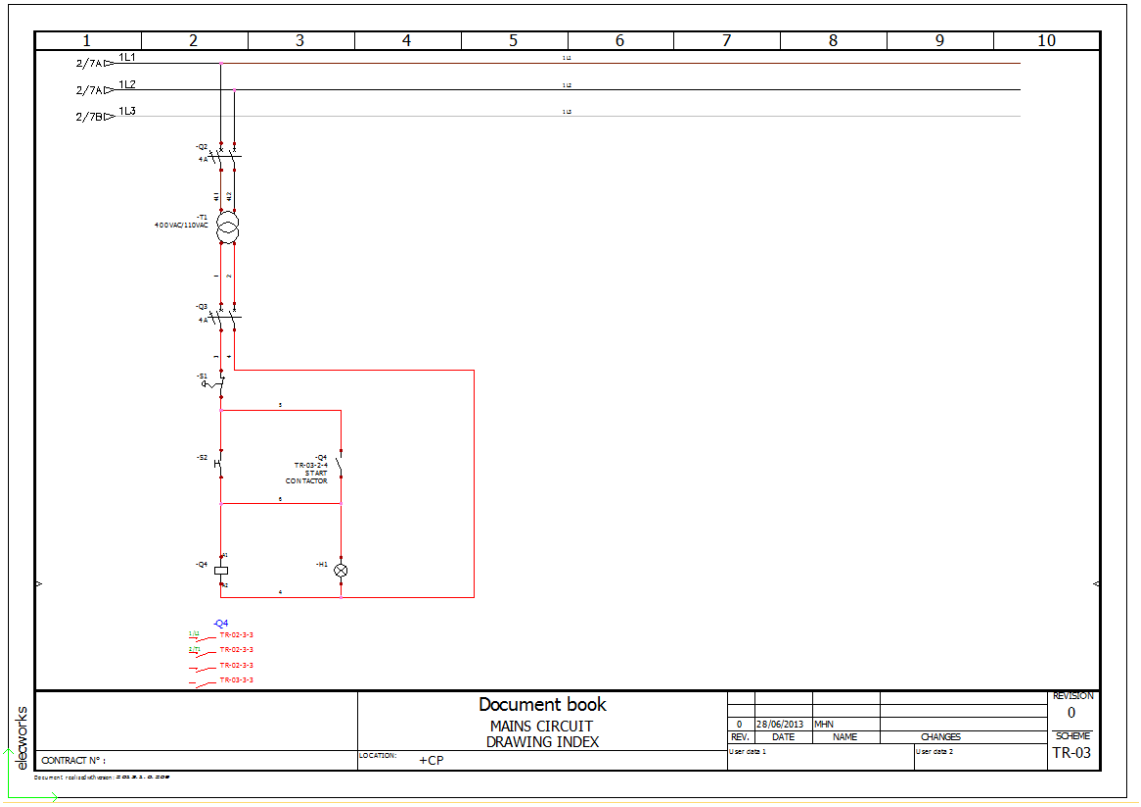
The attributes for the symbols are already premapped but the interface allows you to define any user defined attributes

Select Next

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 ciruit 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