

Defining the Wire Sequence Using Toggle Connection

Electrical Designer Users

In Electrical Designer you can define the wire connection through several methods. In this month's Tips and Tricks we look at the Toggle Connection command.

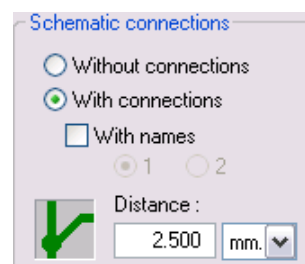
The normal IEC methodology of wire connections is a wire "T" with a connection dot at the intersection of the wires. However traditional wiring schematics show angled connections defining and showing the connection order. This is also common in automotive schematics.

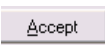

Angled connections have to be invoked through the project properties.

Select  *Edit Project*

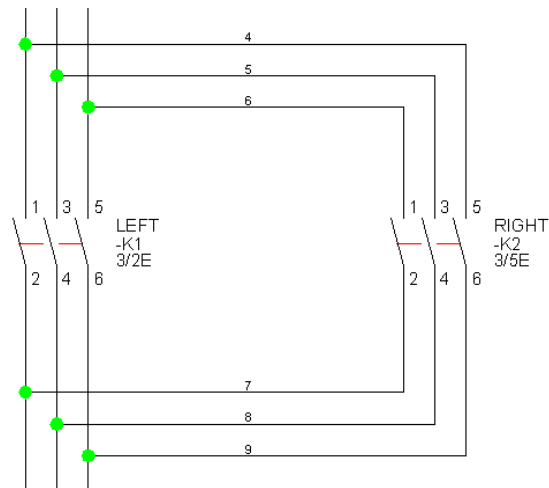
Select 


Ensure *With connections* is selected



Select  or  until the Edit Project dialogue has closed

In this example we will toggle the connections shown below:



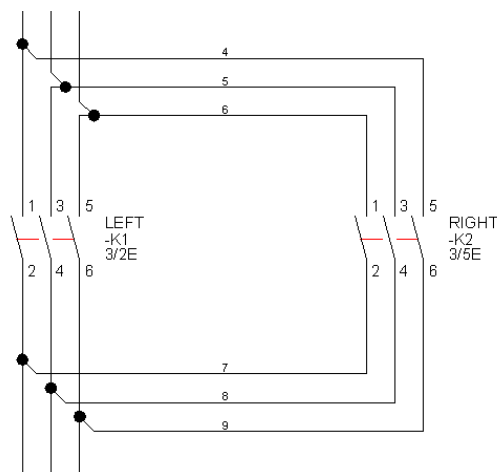
From the ED:Wires toolbar, select the command  *Toggle Connections*

Left click near to a connection dot and the connection dot will become an angled connection

Left click and repeat the *left click* until the required connection is achieved

Right click to close the Toggle Connection command. The connection dot will then appear on completion

This can be repeated until the required connections are achieved



If you have the MAX Module (Level 5), the PCP (preferred connection points) reports will show in the wire from – to lists the new defined connection order.