Class ID: ENR321974

Inventor and AutoCAD Electrical Working Together

Justin B. Martin
AutoCAD Electrical
Duke Energy

Kristopher Mike Myers
Autodesk Inventor
Duke Energy

Learning Objectives

- Learning how to apply your designing methods in a more realistic view
- Learn how to use the design tools to provide a clear picture to your customer for your substation design
- Learn how to provide an accurate BOM for your projects
- Learn how to modify your design with minimal effort and eliminate costly errors

Description

This session will cover the ways that Inventor software and AutoCAD Electrical software interact with each other for a much better workflow. When you use Inventor, you can see an actual representation of a component such as transformers, regulators, breakers, and so on. This lets the designer have better judgment for clearances, footprint, connections points, and more. When you incorporate AutoCAD Electrical, you can use the Inventor components converted from 3D to 2D format to have a more realistic representation. This will help the designer determine the correct wiring points on a device, as well as clearances and bend radius.

Speaker(s)

Justin B. Martin

Justin has 20 years of experience in the industry with a background of power distribution and controls design. Not knowing much about the industry but having taken drafting in high school, Justin knew that he had an interest in drafting and electricity. He attended Maryland Drafting Institute to study drafting and basic architecture but knew architecture wasn't for him. He was able land his first job in the industry with a MEP consulting firm in 2001 doing MEP drafting. While employed there, he showed a lot of interest in the electrical field and earned the opportunity to learn basic power distribution design. Over the course of his career, he continued to work hard to learn more about power distribution, automation controls, commissioning, and project managing. Justin is currently the CAD Lead within the Transmissions group at Duke Energy Carolina's West region.

Kristopher Mike Myers

Mike is a Senior CAD Operator as well as technical specialist at Duke Energy's Carolina's West region. Mike's has 20 years of experience in the Mechanical Engineering/Fabrication industry and 9 years in utilities Substation Design. Mike has been using AutoCAD since release 10, Autodesk Inventor since release 8, and Autodesk Vault since 2008. Mike has been writing code and lisp routines mostly for AutoCAD since 2002. In many cases these codes and routines were used to incorporate company's CAD standards which Mike has taken part in developing along with other SMEs.

Creating Your Necessary Blocks

For this demo being used, you will find below the steps on how to create the symbols that are needed and being used in this instructional demo.

Tip: When creating your symbols, create a .dwg file that contains all symbols needed for a drawing type. E.g. AC Schematic Layout, Panel Wiring, etc. You can organize this file to make it easier to locate the symbols you are looking for at that time. For this demo, I'm showing you just one (1) .dwg file with all the symbols.

File name: ACADE Blocks.dwg

I am going to walk you through the steps on how to create an ACADE symbol. I am going to show you how to create the transformer symbol "TRANXFMR."

Open a new ACAD drawing. In this drawing we'll build the transformer and then create it into an ACADE symbol. The videos listed below will show you the steps how I created this custom ACADE symbol.

File name: TRANSXFMR.dwg

NOTE: All videos are uploaded to YouTube. Please click name to be redirected to YouTube.

Creating Custom ACADE Symbol Step 1:

Follow the steps in the video "Creating Custom ACADE Symbol Step 1"

Creating Custom ACADE Symbol Step 2:

Follow the steps in the video "Creating Custom ACADE Symbol Step 2"

Creating Custom ACADE Symbol Step 3:

Follow the steps in the video "Creating Custom ACADE Symbol Step 3"

Creating Your Project File (WDP)

Open any ACAD drawing and follow the steps in the video listed below.

"Creating ACADE Project File"

- Before moving to the next steps, add in your drawings that are associated with your project. There is only one (1) drawing associated with this demo.

  Drawing: "SH.1_AC_001"

Adding Custom ACADE Symbols (Blocks) to Icon Menu
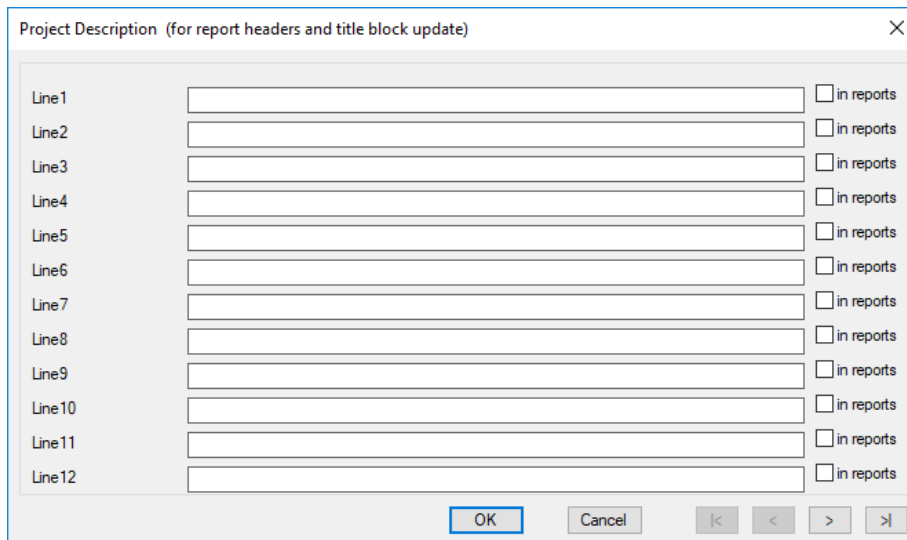
Here I am going to walk you through the steps on how to add your custom ACADE symbols that were created in the steps above to the Icon Menu.

Open any ACAD drawing and follow the steps in the video listed below.

"Adding Custom ACADE Symbols to Icon Menu"

Creating Custom Title Block Description Lines

a} In ACADE, the description lines are set to default as indicated in the screenshot below.

b} To make the default description lines custom you must first create a. WDL file.

Follow the steps in the video "Creating Custom Title Block Description Lines."

Creating & Mapping Your Title Block

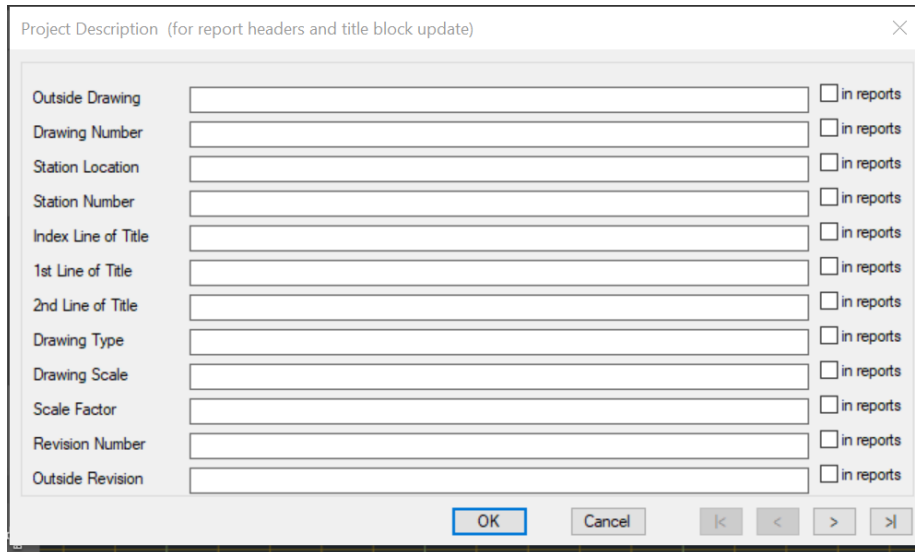a} First you will need to have your title block created. This could be your standard title block that you use currently. Some changes are needed for the title block to work with ACADE such as "mapping" your title block. I will show you how to "map" your title block in step **"c."**

b} After you have created your title block, the next step is to add in your text attributes needed for your title block information. Create your text attributes and place in your desire locations on your title block.

c} Once you have your title block and attributes created, next step is to set up and "map" your title block using the "Title Block Setup" tool in ACADE.

Follow the steps in the video "Title Block Mapping & Set-up"

Note: After you have completed your title block mapping and you are still seeing the default description lines, close your ACADE project located in Project Manager and open it again. After doing this, you should see your custom description lines.

Creating Wire Types

Open SH.1_AC_001.dwg drawing and follow the steps in the video listed below. I've included a screenshot of the finished wire type list you will need for this demo. Create the remaining wire types based off this list.

Follow the steps in the video "Creating Wire Types"

Create/Edit Wire Type ✕

	Used	Wire Color	Size	Layer Name	Wire Numbering	USER1
1		BLACK	4c12AWG	1F_4c12	Yes	
2		BLACK	2c12AWG	1K_2c12	Yes	
3		WHITE	12AWG	A_4c12	Yes	
4		GREEN	12AWG	B_4c12	Yes	
5		BLACK	12AWG	C_4c12	Yes	
6		WHITE	12AWG	CA_4c12	Yes	
7		GREEN	12AWG	CB_4c12	Yes	
8		BLACK	12AWG	CC_4c12	Yes	
9		BLACK	12AWG	CG_2c12	Yes	
10		WHITE	12AWG	CN_2c12	Yes	
11		RED	12AWG	CN_4c12	Yes	
12		BLACK	12AWG	G_2c12	Yes	
13		BLACK	100kV	H_A_PHASE_100kV	Yes	
14		BLACK	100kV	H_B_PHASE_100kV	Yes	
15		BLACK	100kV	H_C_PHASE_100kV	Yes	

Option
☐ Make All Lines Valid Wires
[Import...]

Layer
Layer Name Format: %C%S
[Color...] [Linetype...] [Lineweight...]
[Add Existing Layer...] [Remove Layer]
[Mark Selected as Default]   1F_4c12

[OK] [Cancel] [Help]

Create/Edit Wire Type ✕

	Used	Wire Color	Size	Layer Name	Wire Numbering	USER1
7		GREEN	12AWG	CB_4c12	Yes	
8		BLACK	12AWG	CC_4c12	Yes	
9		BLACK	12AWG	CG_2c12	Yes	
10		WHITE	12AWG	CN_2c12	Yes	
11		RED	12AWG	CN_4c12	Yes	
12		BLACK	12AWG	G_2c12	Yes	
13		BLACK	100kV	H_A_PHASE_100kV	Yes	
14		BLACK	100kV	H_B_PHASE_100kV	Yes	
15		BLACK	100kV	H_C_PHASE_100kV	Yes	
16		WHITE	12AWG	N_2c12	Yes	
17		RED	12AWG	N_4c12	Yes	
18		BLACK	12.5kV	X_A_PHASE_12.5kV	Yes	
19		BLACK	12.5kV	X_B_PHASE_12.5kV	Yes	
20		BLACK	12.5kV	X_C_PHASE_12.5kV	Yes	
21						

Option
☐ Make All Lines Valid Wires
[Import...]

Layer
Layer Name Format: %C%S
[Color...] [Linetype...] [Lineweight...]
[Add Existing Layer...] [Remove Layer]
[Mark Selected as Default]   1F_4c12

[OK] [Cancel] [Help]

Modifying Wire Number Options

Open SH.1_AC_001.dwg drawing and follow the steps in the video listed below.

 "Modifying Wire Number Options"

Once you have all your wires assigned, your list should look like the screenshots below.

Modifying ACADE Symbol

Open drawing "3-PHASE CT" and follow the steps in the video listed below. In this example we're going to modify our "TRANXFMR" Symbol to add in the 3-Phase CT on the transformer.

 "Modifying ACADE Symbols"

Building the AC Schematic Demo Drawing

Now that we have all our ACADE symbols, wires, wire layers, title block, etc. all setup, it's time to start building our AC Schematic layout that is used in this demo. For this, you can look at the presentation demo video and follow along.

Exporting and Importing ACADE Data for Inventor

Now that you have all your devices and wires associated with your layout, it is time to export the ACADE data for Inventor. For this you can look back at the demo presentation to learn how to export and import the XML and .csv data from ACADE to Inventor and from Inventor to ACADE.

Autodesk Inventor Portion Demo

The Inventor data set contains all files needed to construct the transformer, conduit, trench, and panel with switches. First you will need to Unzip the data set to the desired folder and change the project file to: *487657_SAMPLE.ipj*. From this point you should be able to open the assemblies with all parts and sub-assemblies (44 components) constrained.

Note: the control cabinet inner and outer doors, inert gas system cabinet, low voltage surge arrester bracket, and pressure relief pipe were removed for clarity and not included in the data set. It is also important to note that these are Inventor 2018 files. The software used was Autodesk Inventor Professional 2018.

Components and files of note:
- *487657_SAMPLE.iam* – top level assembly.
- *487657_SAMPLE-CONTROL CABINET.iam* – assembly containing the cabinet and conduit sub-assemblies.
- *487657_SAMPLE-PANEL ASSEMBLY.iam* – assembly containing the panel and concrete foundation.
- *487657_SAMPLE-XFMR Terminal Blocks.iam* – assembly containing all four terminal blocks that are housed in the Control Cabinet on the transformer.
- *487657_SAMPLE-DITCHBOARD ASSEMBLY.iam* – Assembly containing the Trench or Ditch Board sub-assemblies.
- *487657_SAMPLE-SWITCH A17.ipt* thru *487657_SAMPLE-SWITCH A22.ipt* – part files of the switches mounted on the panel.
- *Insulator bitmap 2.png* – image for the style to simulate the radiator fins.
- *ACADE XML_COMPLETE.xml* – The XML file exported from ACADE with electrical components and wire information.
- *Wire Run List.cfg* – configuration file used to create the CSV file ACADE will read to create a wire list BOM. *This CFG and others can be downloaded from Autodesk as well.*

Adding Pin Locations to Inventor Components

In this step, we will be adding the pins to the Inventor components so that when we import the XML from ACADE, we'll be able to assign those objects and properties to our Connectors.
Files used in this step are:
487657_SAMPLE-XFMR Terminal Blocks.ipt
487657_SAMPLE-SWITCH A17.ipt
487657_SAMPLE-SWITCH A18.ipt
487657_SAMPLE-SWITCH A19.ipt
*487657_SAMPLE-SWITCH A20.ipt*
487657_SAMPLE-SWITCH A21.ipt
487657_SAMPLE-SWITCH A22.ipt

Tip: you can open these directly from the main assembly (487657_SAMPLE.iam) instead of file explorer. Just right click the part and select open. You can also right click the component and select edit to modify the part inside of the assembly.

In each of the files we will place pins on the terminal blocks and switches using the following diagrams for guidance.


Pin call outs for the Terminal Blocks

Pin call outs for A17 thru A22
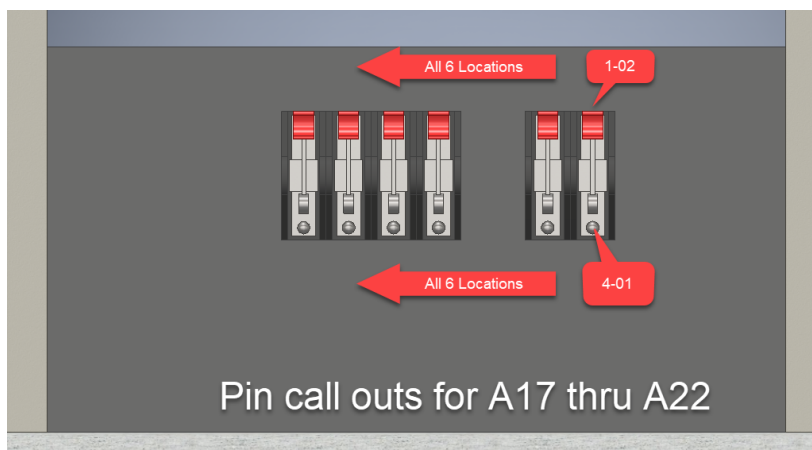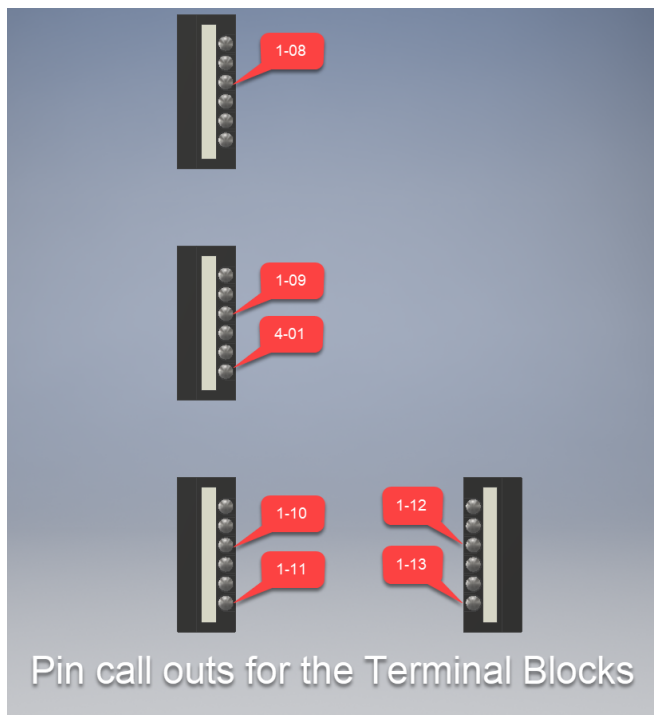
Follow the steps in the video "Placing pins on components" for all connectors specified above.

Creating the 1F_4c12 (4 wire) and 1K_2c12 (2 wire) cable styles

In this step we will be adding these 2 cable styles to our library. The file used in this step is: *487657_SAMPLE.iam*

In the Cable and Harness environment, select Library Library. Then use the following info to create both cable styles. Once they have been created these cable styles can be used in later projects as well.
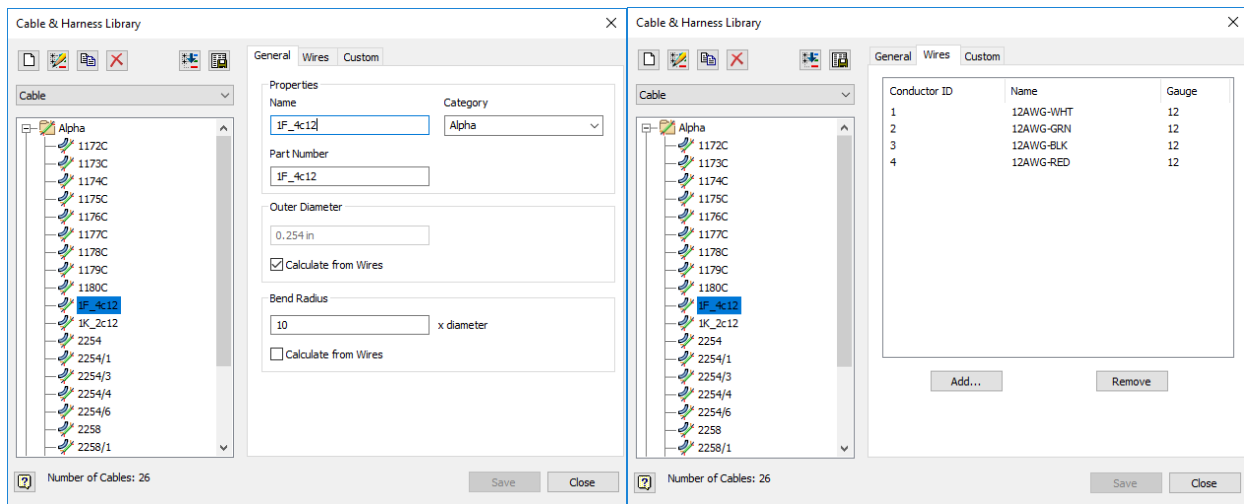
1F_4c12 cable data constructed
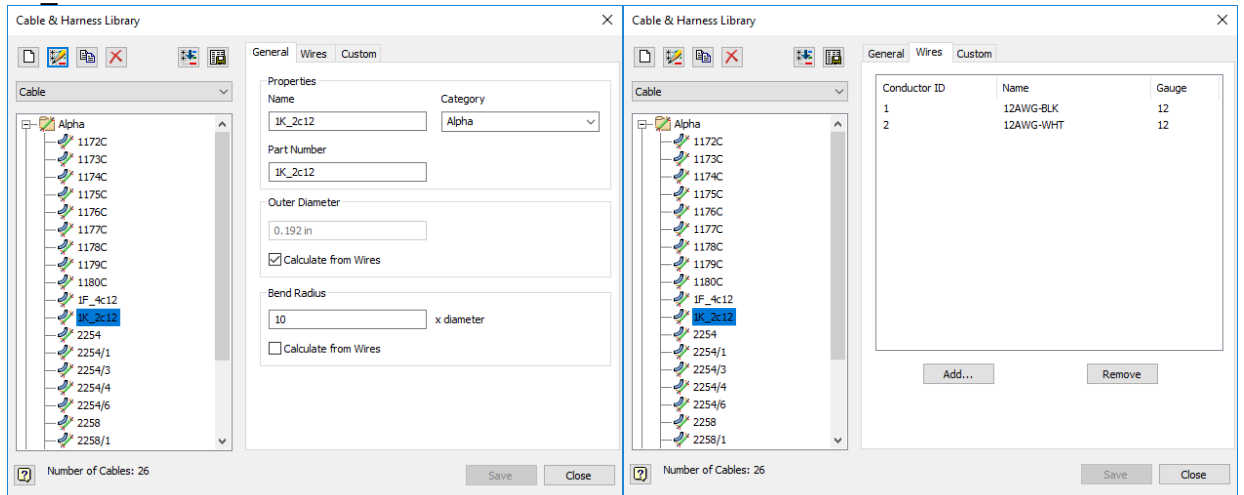
1K_2c12 cable data constructed

Follow the steps in the video "Creating Cables in the Library"

## Importing the XML from ACADE and assigning components

In this step we will be importing the XML and assigning the electrical parts to the Inventor components with the equivalent pins. We will be working exclusively in the Cable and Harness environment. The files used in this step are:
487657_SAMPLE.iam
ACADE XML_COMPLETE.xml
487657_SAMPLE-XFMR Terminal Blocks.ipt
487657_SAMPLE-SWITCH A17.ipt
487657_SAMPLE-SWITCH A18.ipt
487657_SAMPLE-SWITCH A19.ipt
487657_SAMPLE-SWITCH A20.ipt
487657_SAMPLE-SWITCH A21.ipt
*487657_SAMPLE-SWITCH A22.ipt*

Go to the Environments tab and select "Cable and Harness". When the Create Harness dialog box becomes active, give the Harness a unique file name. Such as, *487657_SAMPLE.Harness3-1.iam*. accept the default File Location.
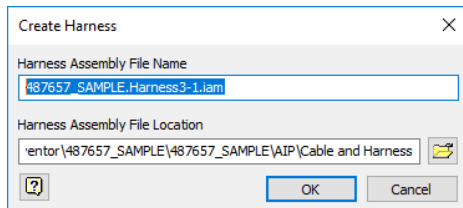
In the context tab "Cable and Harness", in the Manage Panel, select the flyout that has Important Harness Data. Notice there are three options here. We will use two of them in this exercise. Select Import Harness Data and browse to where the *ACADE XML_COMPLETE.xml* is located (In the Data Sets folder under Workspace) and select OK.
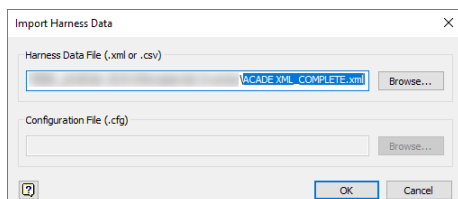

Then, under Electrical Parts in the Imported Harness Data browser, right click and select "Assign to an existing Electrical Part" for parts A17, A18, A19, A20, A21, A22, XMFR1, and the component without a description. The other parts were not created for the demo and will remain with issues. This is not an error. It is just not included.
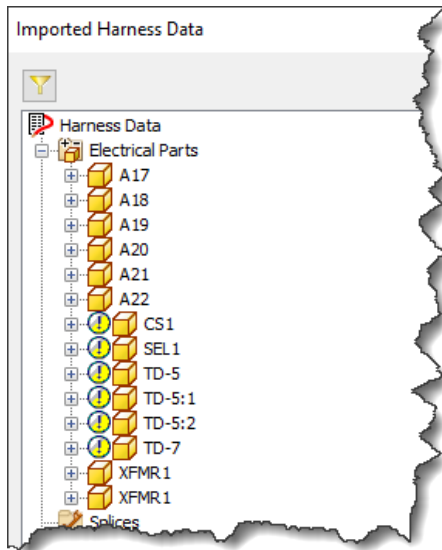
Follow the steps in the video "Assigning Electrical Parts to Inventor Components"

With the Imported Harness Data browser still available (or selecting Import Harness Data and selecting the same XML again), select the cable nodes without names then right click and select Choose from Cable & Harness Library for 1F_4c12 (4 wire) and 1K_2c12 (2 wire).
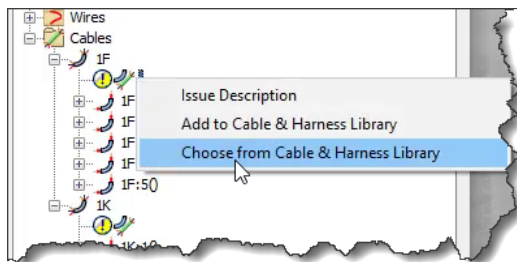

Assigning the cable within the Imported Harness Data browser

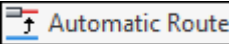Follow the steps in the video "Choosing Cables to assign to the Imported Cables"

Building the Harness, "Autorouting", and fine tuning the wire

In the next step we will be creating the cable harness that the cables will be routed through. Afterward, we will use Automatic Route to let Inventor route cables through the closest possible harness – which will be the one we just created.

In the Cable and Harness environment, select Create Segment. Then using any number of points, pick points in the Control cabinet, centers of the Conduit, and finally points through the trenching. Select Finish from the right click menu.
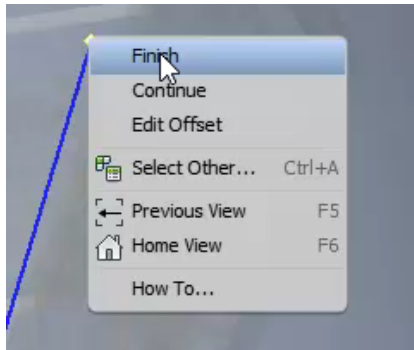
Tip: Creating work points within the separate assemblies can create a more precise, easy to select, route.

With the harness completed select [Automatic Route] and select at least 1 wire from each cable to get all 6 wires. Then, select Apply or OK to route the cables through the harness.

Follow the steps in the video "Building the Harness and Autorouting the cable"

In a further step, a connector (the switches for instance) can have a face chosen for the wire to leave perpendicular to. That is, a direction that the wire connects to the pin.

Activating a connector from the browser, or alternatively opening it, and selecting the Manage tab. In the Author panel select [Connector] Authoring. Once the Connector Authoring dialog is active, you select a face for the direction of the wire. A green arrow on the face denotes the facing. Once you are satisfied, select OK. Returning to the assembly, you will notice wire has changed geometry. You will need to do this to all six of the switches.
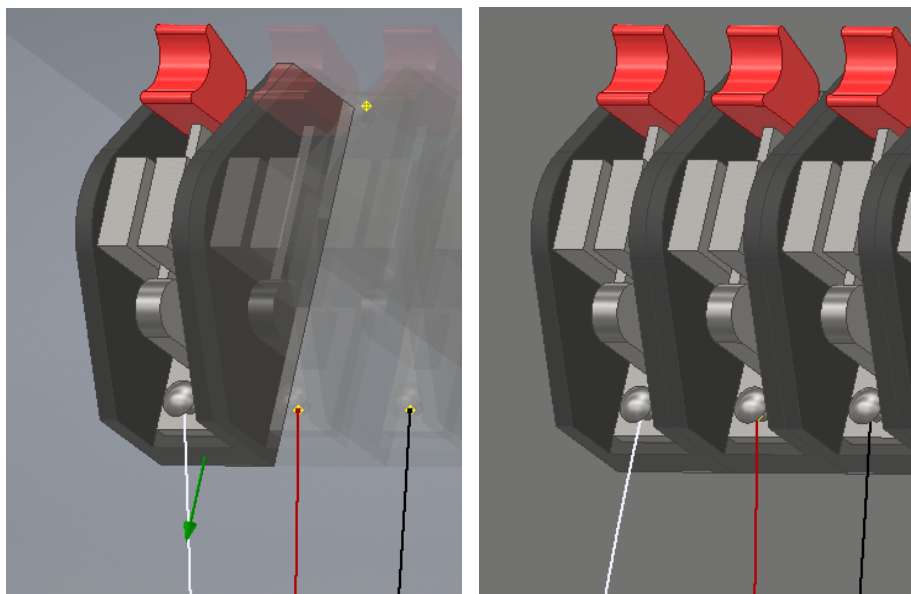
Follow the steps in the video "Connector Authoring for Wire Facing"

Exporting the XML from Inventor

Still within the Cable and Harness environment, our next step will involve us exporting our XML for AutoCAD Electrical to read from. If we had added more cables or wires to the assembly, they would have been communicated through the XML as well.

In the Cable and Harness environment, select the flyout next to Import Harness Data. Then, select Export Harness Data. Give the XML a file name and save it in a location where it can be easily retrieved for usage in ACADE.
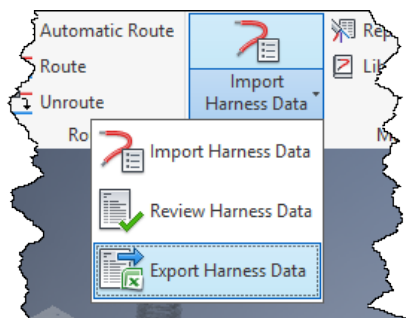
Building the CSV file from the "Wire Run List.cfg" file.

Finally, in our last step we will be creating a CSV file from a configuration file that uses the cables we created in our assembly to build a wire list with important data that the CFG file determines.

In the Cable and Harness environment, in the manage panel, select Report. When the Report Generator dialog becomes active, select "Create Report" from the available options. Then, select "Add File To List", when the Create Reports interface comes up. Then locate the *Wire Run List.cfg* file and select open. Back in the Create Reports dialog box select OK. This will create the CSV (*487657.Harness4-1_PartsList_1* in our case) file ACADE can then import.

Once this is complete, we can select Finish Cable and Harness to return us back to our top level assembly.

Follow the steps in the video "Building the CSV file from the Wire Run List CFG file"