Like chocolate and peanut butter: AutoCAD Electrical & Inventor are better together

Janna Spicer – Autodesk, Inc.
Randy Brunette – Autodesk, Inc.

MA2442

Want to ensure that the physical version of your electrical design has enough space in the mechanical layout? Tired of discovering too-tight bend radii during the first prototype build? Interested in keeping your mechanical and electrical designs in sync? Skeptical that any of this really works in a real-world environment? Attend this class to learn about all of this from real-life examples of projects done by your peers from around the world. And learn why AutoCAD Electrical software and Inventor software are better together—just like chocolate and peanut butter.

Learning Objectives

At the end of this class, you will be able to:

- List and describe the benefits of the electro-mechanical design workflow
- Describe the workflow capabilities of AutoCAD Electrical combined with Inventor
- Try it: Be ready to use an AutoCAD Electrical-to-Inventor workflow
- Try it: Be ready to use an Inventor-to-AutoCAD Electrical workflow

About the Speaker

Janna Spicer *received her bachelor's degree in Electrical Engineering from Washington State University, and her master's degree in Management in Science and Technology from Oregon Health Sciences University. She is now the product manager for Autodesk Product Design Suite in Lake Oswego, Oregon. Prior to that, she managed diverse engineering teams to develop new products at a leading Test and Measurement company. Recently, she was named Volunteer of the Year for FIRST Robotics in Oregon.*

Janna.Spicer@autodesk.com

Randy Brunette *has joined Autodesk as an Electrical Subject Matter Expert. Randy's duties include helping channel partners and customers through mentoring and understanding their business issues and finding solutions that solve their challenges.*
Randy has been in the design field using Autodesk products over 28 years, with experience across multiple segments of the Manufacturing industry. He has been in an Application Engineer role for 19 years.
Prior to joining Autodesk, Randy was a sole proprietor of a consulting business specializing in AutoCAD Electrical, traveling in North America and Europe providing consulting services. Randy is a top rated speaker at Autodesk Universities, Technical Academies, and seminars. He has authored AutoCAD Electrical training manuals, videos, and other materials.

Randy.Brunette@autodesk.com

Section 1: Benefits of the Electro-Mechanical design workflow

In this section you will see the benefits of the combined electrical and mechanical design workflow, as it pertains to your business and engineering needs.

1. Better Products: Explore the design options up front, and digitally determine the optimal design when you have the most flexibility – early in the design cycle.
2. Lower Costs: Don't settle for a last-minute work-around that could have been designed better, if only you had seen it coming. Detect design errors early in the process, saving costly downstream redesign and rework of built units.
3. Shorter Time to Market: Avoid last-minute design or component changes that require re-ordering and expediting components for your prototype or production builds, or worse – rework in the field.
4. Design the Way YOU Want: Start with a mechanical design in Autodesk Inventor, and add in the electrical components and wiring when you're ready. Or, enable your Electrical Engineering team to design critical elements and design the mechanics around that in Autodesk Inventor. It's up to you!
5. Complete Digital Prototyping: Create complete Digital Prototypes of your products, including all electrical and mechanical components, prior to building a single physical unit.

Section 2: AutoCAD Electrical & Inventor Workflows

In this section, you will understand the Digital Prototyping workflows specifically targeted to electrical and mechanical design engineering.

1. Design Automation
2. Automatic Error Detection
3. Extensive components libraries

Guidelines

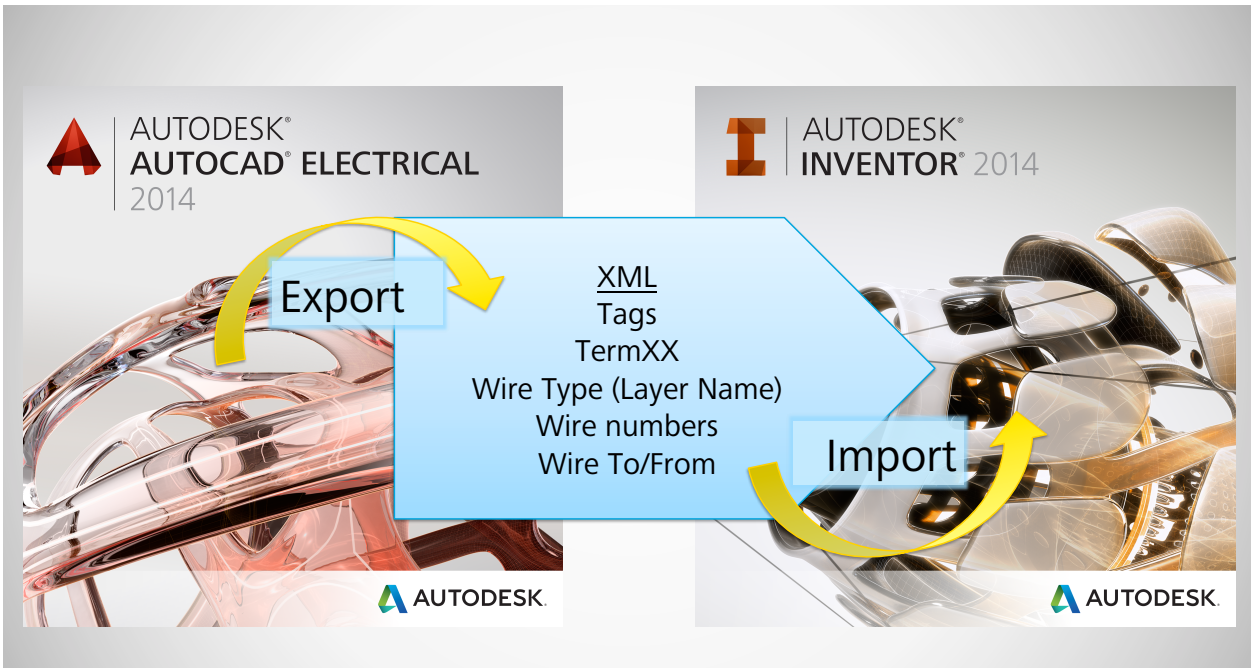
Follow these guidelines to enable smooth transfer of data between AutoCAD Electrical (ACE) and Autodesk Inventor Professional (AIP).

1. RefDes in Autodesk Inventor Professional must have the same value as the TAG attribute in AutoCAD Electrical.
2. Pin in AIP must have the same value as the TermXX attribute in ACE.
3. Wiretype (layer) used in ACE must be available in the AIP library.
4. All wires in ACE must have wire numbers.
5. All wires in ACE must be connected on both ends.

Section 3: XML Export/Import Workflow

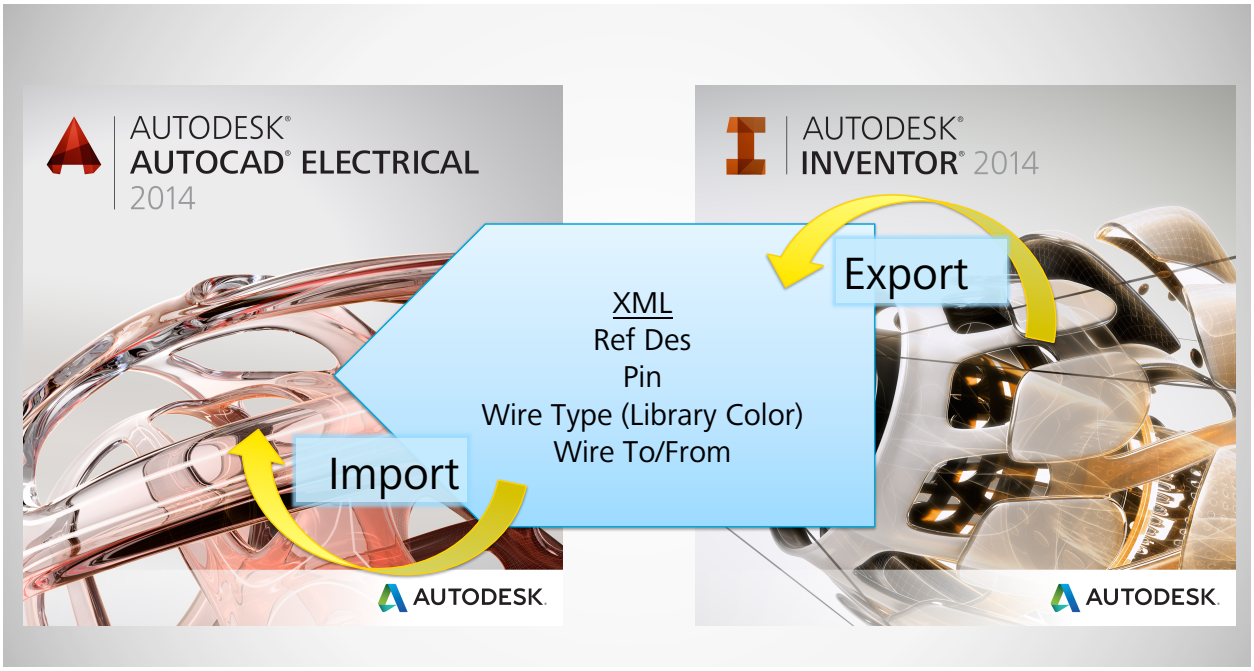
This workflow uses an XML file to transfer key data between AutoCAD Electrical and Inventor. You can start your design in either tool, and transfer data using this method whenever you are ready.

AutoCAD Electrical to Inventor



Export your design (entire project, or just the active drawing) from AutoCAD Electrical as an XML file. From Inventor, import that same XML file and manually connect any components as needed in Inventor.

Inventor to AutoCAD Electrical



Export your cable and harness design from Inventor as an XML file. From AutoCAD Electrical, import that same XML file. Add wire numbers to any new wires as needed.

Note: There will be two separate XML files, one for each direction (ACE to AIP, and AIP to ACE). It is important not to mix these up during import.

Section 4: "On Your Own" Step-By-Step Instructions

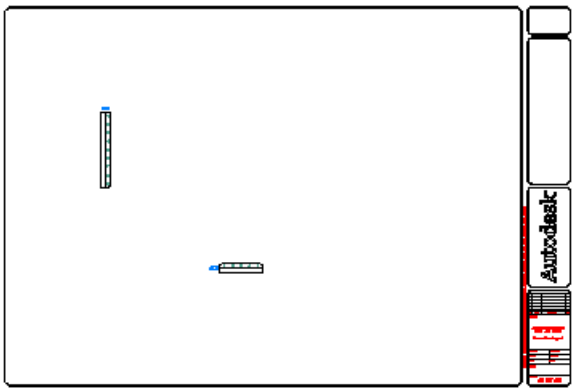
In this section, you will follow detailed instructions to perform what you saw in class.

Before you begin: Ensure you have downloaded and saved the "Electrical-Datasets.zip" provided by Autodesk University for this class to your "My Documents" folder. There are two sub-folders: "ACE Loader-E Dataset" which you'll use for the AutoCAD Electrical portions of the following exercises, and "AIP Loader-E Dataset" which you'll use for the Inventor portions.

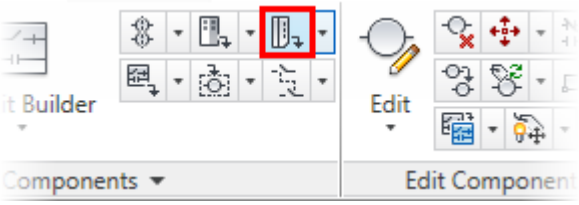
AutoCAD Electrical (ACE) - Insert Connectors

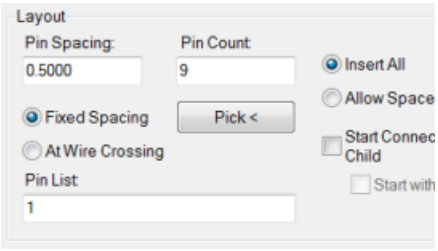
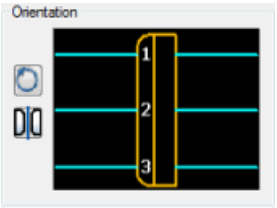
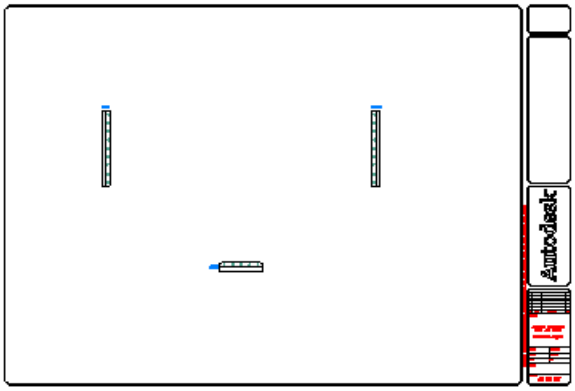
In this demonstration, you add connectors to an AutoCAD Electrical drawing. Any type of component could be inserted, as long as the guidelines are followed.

1. From AutoCAD Electrical, in the Project Manager, on the toolbar, click Open Project.
2. In the Select Project File dialog box, do the following:
   - Browse to the "ACE Loader-E Dataset" folder.
   - Select *Loader-E_2014_JIC.wdp*.
   - Click Open.
3. In the Project Manager, do the following:
   - Expand the Loader-E_2014_JIC project.
   - Expand the Inventor Interface Panel folder.
   - Right-click *Loader-E_2014_JIC_10.dwg.*
   - Click Open.



4. On the Schematic tab, Insert Components panel, click Insert Connector.

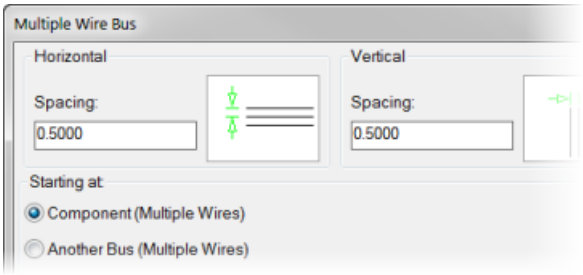


5. In the Insert Connector dialog box, do the following: • For Pin Spacing, enter **0.50.** • For Pin Count, enter **9**. • Select Fixed Spacing. • Select Insert All.	
6. Click Rotate and Flip to orient connector with plug to the left as shown. **7.** Click Insert.	
8. In the drawing, select an insertion point at **22.0,16.0**.	

ACE - Insert Wires

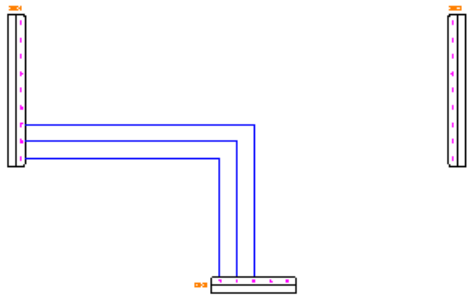
In this exercise, you add wires between the connectors. You create wire types in ACE to match existing wire types in AIP. Then you assign wire numbers to the wires. This is done to ensure compliance with Guideline #3 and #4.

1. On the Schematic tab, Insert Wires/Wire Numbers panel, click Multiple Bus.
2. In the Multiple Wire Bus dialog box, do the following:
   - For Horizontal Spacing, enter **0**.5.
   - For Vertical Spacing, enter **0.5.**
   - Select Component (Multiple Wires).
   - Click OK.



3. In the drawing, on 23X3, window and select pins 1, 2, and 3. Press ENTER.
4. At the command line, enter **T**. Press ENTER.
5. In the Set Wire Type dialog box, select 18AWG-BLU. Click OK.
6. Drag the cursor up and to the left.
7. At the command line, enter **F**. Press ENTER.
8. In the drawing, on 23X1, select pin 7.

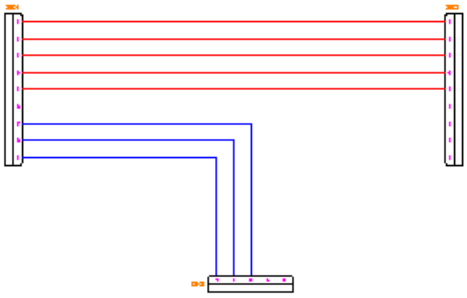
This connects the wires to pins 7, 8, and 9.



9. On the Schematic tab, Insert Wires/Wire Numbers panel, click Multiple Bus.
10. In the Multiple Wire Bus dialog box, do the following:
- For Horizontal Spacing, enter **0.5.**
- For Vertical Spacing, enter **0**.5.
- Select Component (Multiple Wires).
- Click OK.

11. In the drawing, on 23X1, window and select pins 1, 2, 3, 4, and 5. Press ENTER.
12. At the command line, enter **T**. Press ENTER.
13. In the Set Wire Type dialog box, select 16AWG-RED. Click OK.
14. In the drawing, on 23X2, select pin 1.

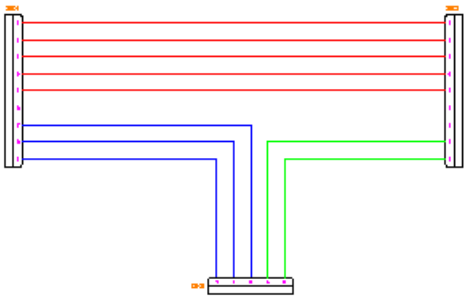
This connects the wires to pins 1, 2, 3, 4, and 5.



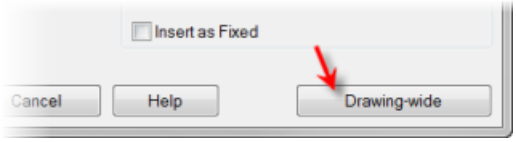
15. On the Schematic tab, Insert Wires/Wire Numbers panel, click Multiple Bus.
16. In the Multiple Wire Bus dialog box, do the following:
- For Horizontal Spacing, enter **0**.5.
- For Vertical Spacing, enter **0**.5.
- Select Component (Multiple Wires).
- Click OK.

17. In the drawing, on 23X3, window and select pins 4 and 5. Press ENTER.
18. At the command line, enter **T**. Press ENTER.
19. In the Set Wire Type dialog box, select 14AWG-GRN. Click OK.
20. In the drawing, on 23X2, select pin 9.

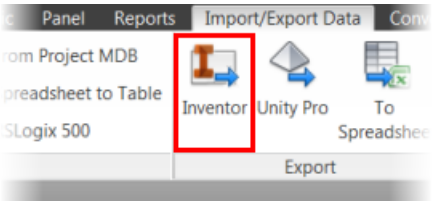
This connects the wires to pins 7 and 9.



21. On the Schematic tab, Insert Wires/Wire Numbers panel, select Wire Numbers.
22. In the Wire Tagging dialog box, click Drawing Wide.

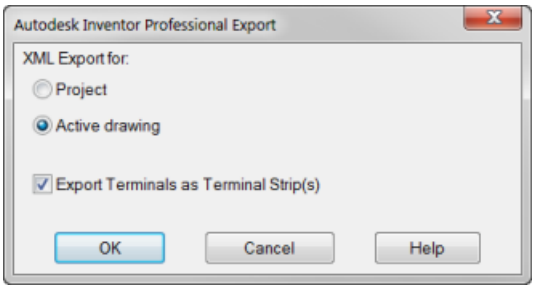


23. On the Import/Export Data tab, Export panel, click Inventor.
24. In the Autodesk Inventor Professional Export dialog box, select Active Drawing. Click OK.
25. If prompted, in the Qsave dialog box, click Always Qsave.



26. In Autodesk Inventor Professional XML File Export dialog box, do the following:
- Browse to the Documents folder.
- For File Name enter ***ACE to AIP Panel.xml***.
- Click Save.

27. If prompted, in the Qsave dialog box, click Always Qsave.
28. On the Quick Access toolbar, click Save.

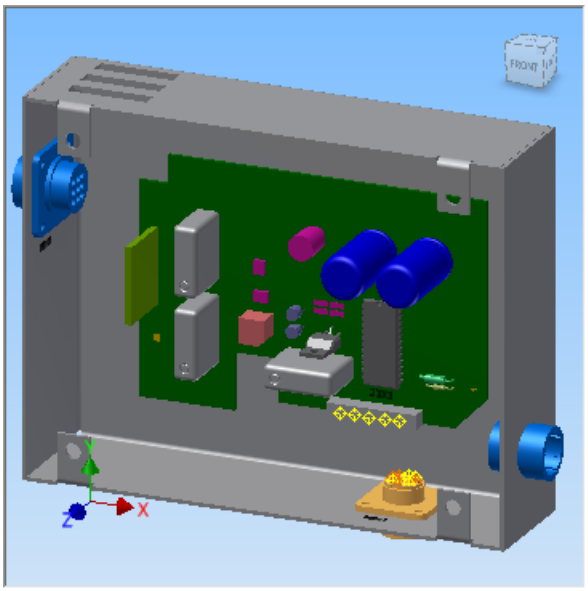
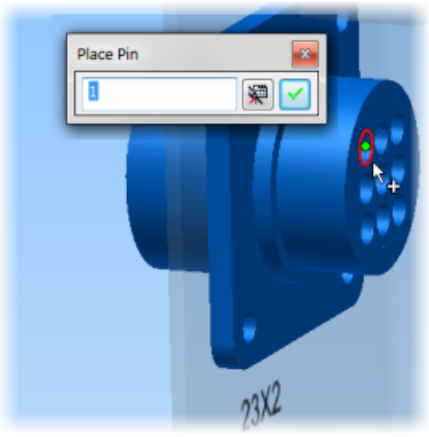


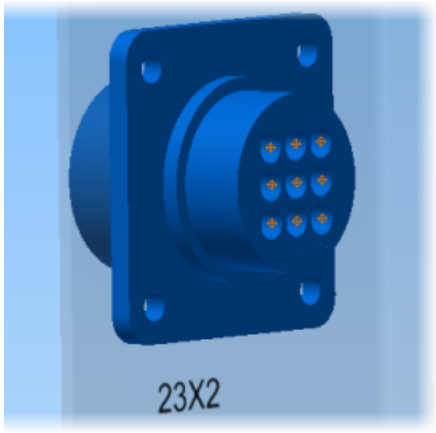
Autodesk Inventor Professional (AIP) - Create Connector

In this exercise, you open an electrical enclosure assembly in AIP containing electrical components. You create an AIP part, including Harness Pins, to represent a connector. Then you add a RefDes to match the ACE component TAG value, and add Pin values to match the ACE component pin values. These steps comply with Guidelines #1 and #2.

1. Move to Inventor.
2. On the Get Started tab, Launch panel, or in the Welcome dialog box, click Projects.
3. In the Projects dialog box, click Browse.
4. In the Choose Project File dialog box, do the following:
- Browse to the "AIP Loader-E Dataset" folder.
- Select *AIP Loader-E.ipj*.
- Click Open

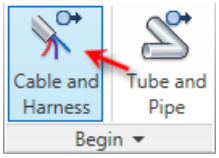
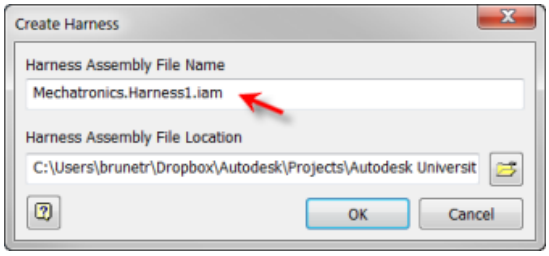
5. In the Projects dialog box, click Done.

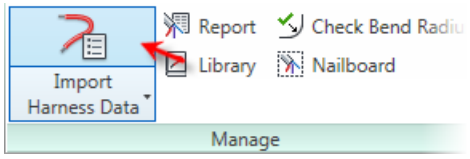
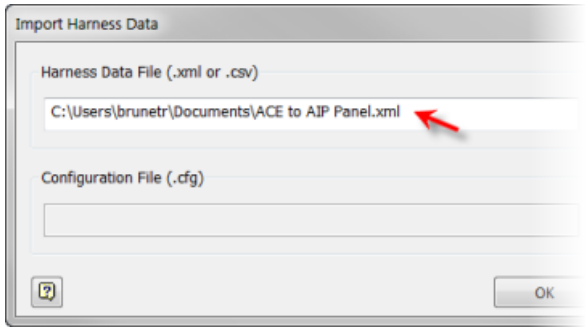
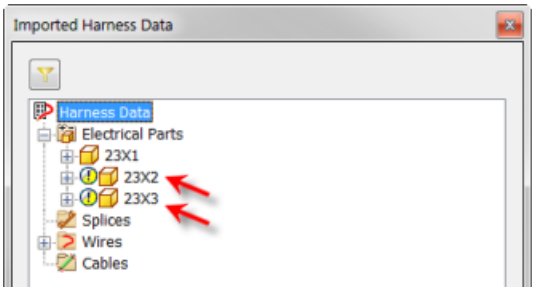
6. On the Get Started tab, Launch panel, or in the Welcome dialog box, click Open. **7.** In the Open dialog box, double-click the Workspace folder. **8.** Scroll down and select *Mechatronics.iam*. Click Open.	
9. In the assembly, double-click the blue connector in the upper left corner. **10.** On the 3D Model tab, Harness panel, click Place Pin.	
11. In the assembly, select the center of the #1 location on the connector. **12.** In the Place Pin dialog box, enter 1. **13.** Click ENTER.	

14. Working from top to down, and left to right, repeat Steps 11-13, placing in the remaining pins, entering values **2-9.** **15.** On the 3D Model tab, Return panel, click Return.	

AIP - Create Harness and Import Wires

In this exercise, you create a harness and import the wire assignments created in ACE.

1. On the Environments tab, Begin panel, click Cable and Harness.	
2. In Create Harness dialog, for New Harness Assembly File Name, enter **Mechatronics.Harness1.iam**. **3.** Click OK.	
4. With Harness1 selected for editing, in the Browser, do the following: • Right-click 360124:1 and select Harness Properties. • In the Part Properties dialog box, for RefDes, enter **23X1**. Click OK. ***Note:*** *23X1 now matches the TAG value from the ACE component. Other components will be labeled in a later step.*	

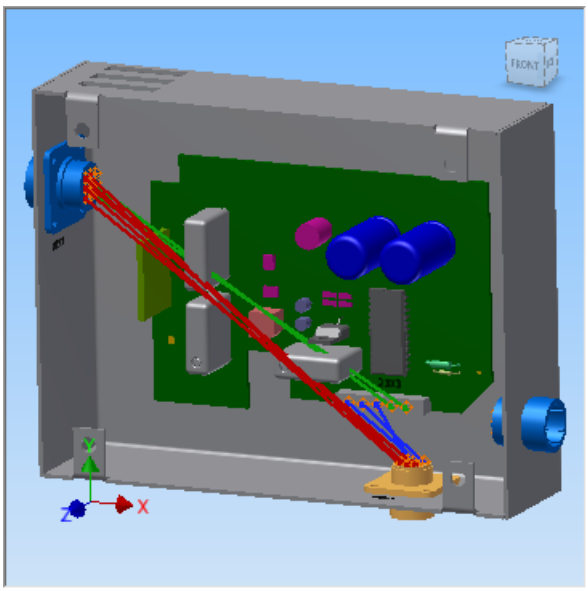
5. On the Cable and Harness tab, Manage panel, click Import Harness Data.	
6. In the Import Harness Data dialog box, do the following: • Click Browse. • In the Select Wire List Data File dialog box, browse to the *My Documents* folder → Electrical-Datasets → AIP Loader-E Dataset → Saved XML Files • Select *ACE to AIP Panel.xml.* Click Open. • Click OK.	
7. In the Imported Harness Data dialog box, expand the Electrical Parts category. ***Note:*** *23X2 and 23X3 show errors.*	

8. Right-click 23X2, click Issue Description.

This describes the issue that has occurred.

9. In the Imported Harness Data dialog box, click Close.
10. Right-click 23X2, click Assign to An Existing Electrical Part.
11. In the drawing, select the 23X2 connector you edited.
12. In the Select Electrical Part dialog box, click OK.
13. Right-click 23X3, click Assign to An Existing Electrical Part.
14. In the drawing, select the strip connector, 23X3. Click OK.
15. Expand and review the Wires category. Click OK.
16. In the Imported Harness Data dialog box, click Close.

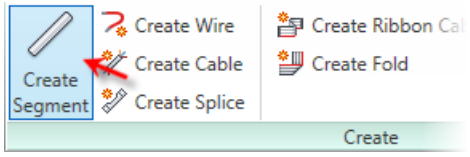
Wires are added to the assembly.



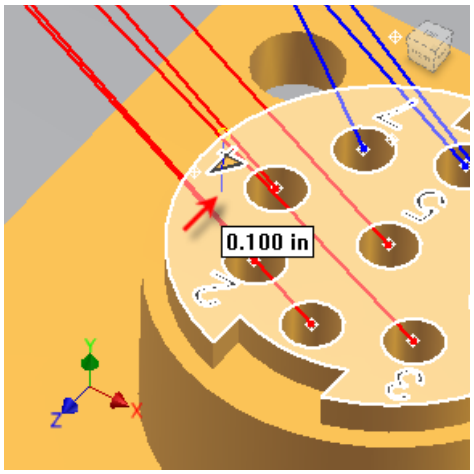
AIP - Create Segments

In this exercise, you create segments. Segments are the routings for the wires. Segments behave similar to wire looms.

1. On the Cable and Harness tab, Create panel, click Create Segment.

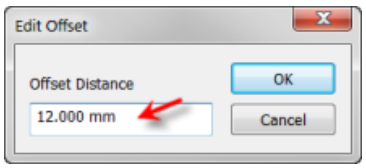


2. In the assembly, select the top surface of the 23X1 connector.

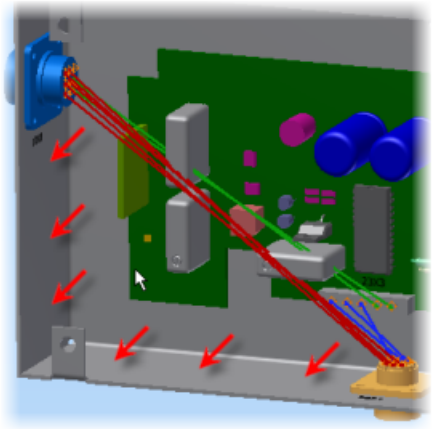


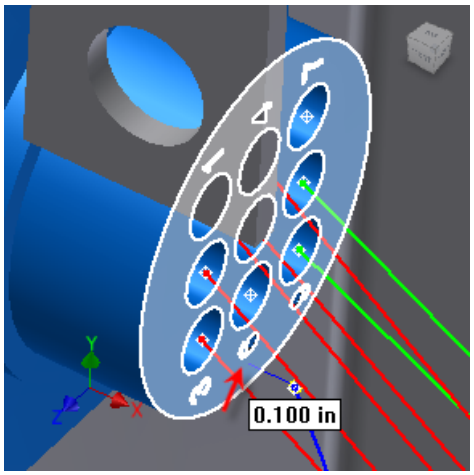
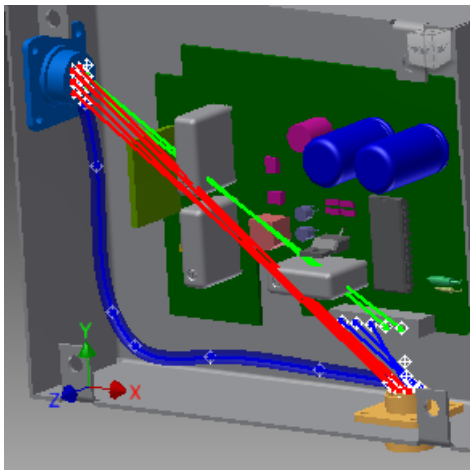
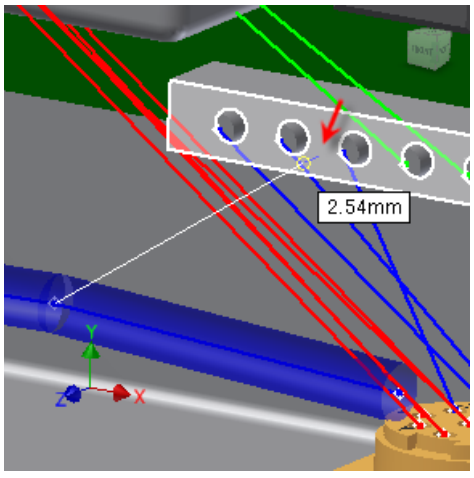
3. Right-click anywhere. Select Edit Offset.

4. In the Edit Offset dialog box, for Offset, enter **12.0**. Click OK.



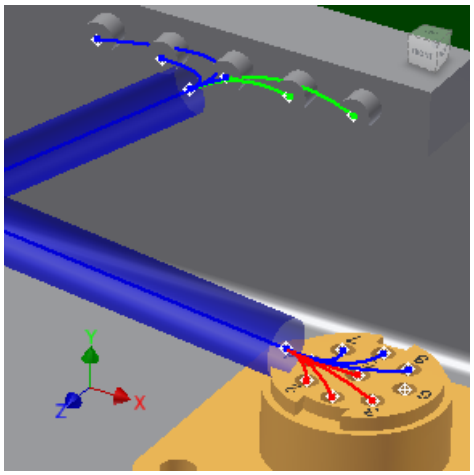
5. Insert workpoints for the segment from X1 to X2, selecting the sides of the enclosure for placement of the workpoints.



6. For last point, change offset back to **2.54**. Select top surface of 23X2.	
7. Right-click and select Finish.	
8. On the Cable and Harness tab, Create panel, click Create Segment. **9.** On the newly created segment, select first workpoint. *(closest to 23X1)* **10.** Select near the middle of the top surface of 23X3. **11.** Right-click and select Finish.	

12. On the Cable and Harness tab, Route panel, click Automatic Route.
13. In the Auto Route dialog box, select All Unrouted Wires. Click OK.

Note: *All un-routed wires are automatically routed through the nearest segment.*

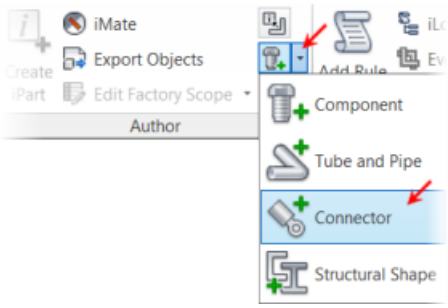


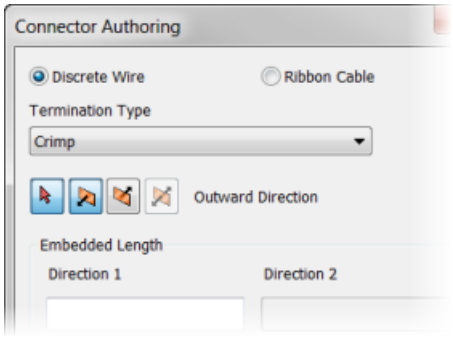
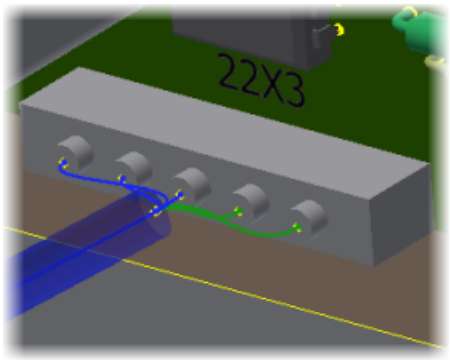
AIP - Author Connector Part

Notice the wires start directly on the pin connect and use the most direct path to the segment connection point. You can control the direction of the wire when leaving the pin connection by authoring the part.

In this exercise, you author a connector, establishing the wire connection direction.

1. In the Browser, expand PCB:1.
2. Double-click Connector:1 to activate it for editing.
3. On the Manage tab, Author panel, on the Component flyout, click Connector.

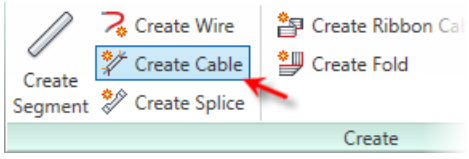


4. In the Connector Authoring dialog box, do the following: • Select Discrete Wire • For Termination Type, from the drop down list, select Crimp. **5.** In the assembly, select the top face of the Connector part. **6.** Verify that the arrow points out from the top face. If not, flip its direction using the direction buttons. **7.** In the Connector Authoring dialog box, click OK. **8.** In the Authoring Result dialog box, click OK.	
9. On the Manage tab, Return panel, click Return. *Notice how the flow of the wires is now perpendicular to the pin face.*	

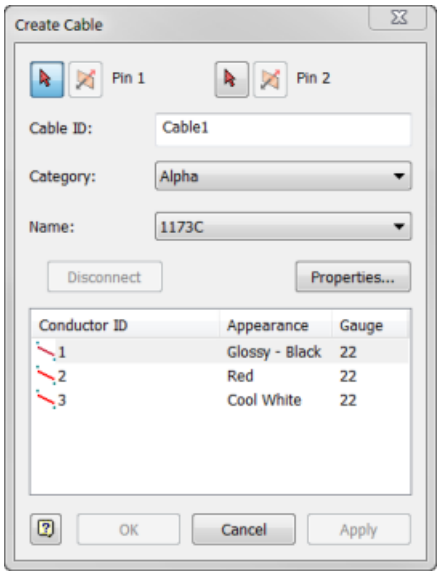
AIP - Create Cable

In ACE, only individual wires were created, and exported to the XML file.

In this exercise, you delete several of the individual wires and replace them with a wire cable.

1. In AIP, in the assembly, zoom up to parts 23X1 and 23X3. **2.** Right-click over one of the three blue wires. Click Delete. **3.** Repeat step 2 for the other two blue wires. **4.** On the Cable and Harness tab, Create panel, click Create Cable.	

5. In the Create Cable dialog box, do the following:
- For Category, select Alpha.
- For Name, select 1173C.

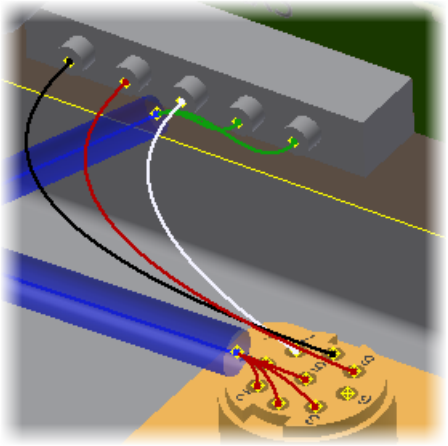


6. Select the Glossy-Black color style, and in the drawing, do the following:
- For Pin 1, select 23X3:Pin 1.
- For Pin 2, select 23X1:Pin 9.

7. Select the Red color style, and in the drawing, do the following:
- For Pin 1, select 23X3:Pin 2.
- For Pin 2, select 23X1:Pin 8.

8. Select the Cool White color style, and in the drawing, do the following:
- For Pin 1, select 23X3:Pin 3.
- For Pin 2, select 23X1:Pin 7.

9. In the Create Cable dialog box, click OK.



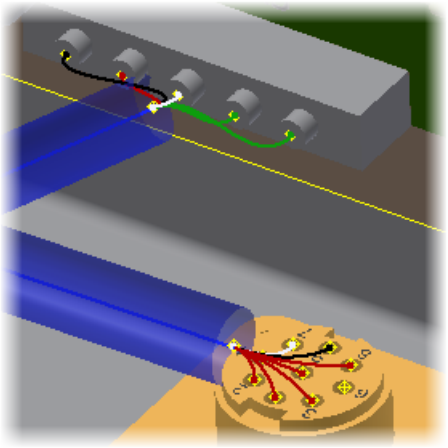
10. On the Cable and Harness tab, Route panel, click Route.
11. In the Route dialog box, select Wires.
12. In the drawing, select any one of the cable wires.

Notice how all cable wires are selected.

13. In the Route dialog box, select First Segment.
14. In the drawing select the segment nearest the 23X1 plug.

In the Route dialog box, the Second Segment is selected automatically.

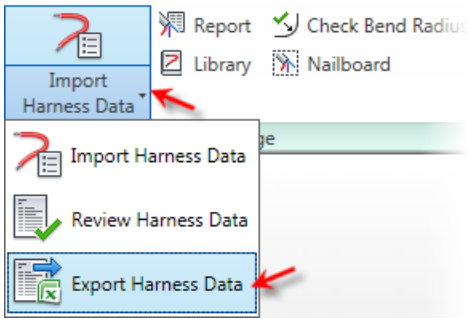
15. In the drawing, select the second segment nearest the 23X3 connector.
16. In the Route dialog box, click OK.



AIP - Export to ACE

In this exercise, you export the modified wire harness to an XML file. Later you import this file into ACE.

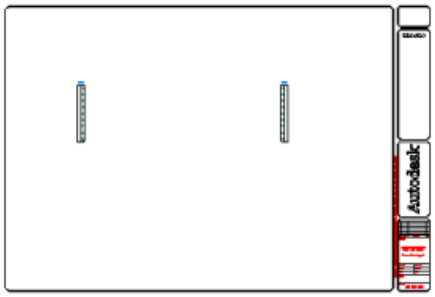
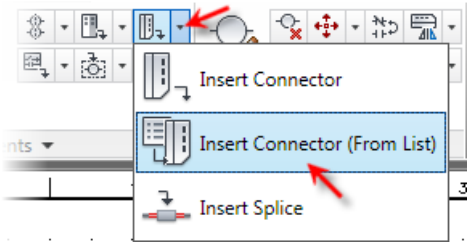
1. On the Cable and Harness tab, Manage panel, Import Harness Data flyout > click Export Harness Data.



2. In the Export Harness Data dialog box, browse to the *My Documents* folder. **3.** For Name, enter ***AIP to ACE Panel.xml***. Click Save. **4.** In the Autodesk Inventor warning dialog box, click Yes. **5.** In the Cable & Harness dialog box, click OK. **6.** On the Quick Access toolbar, click Save. **7.** In the Save dialog box, click OK.	

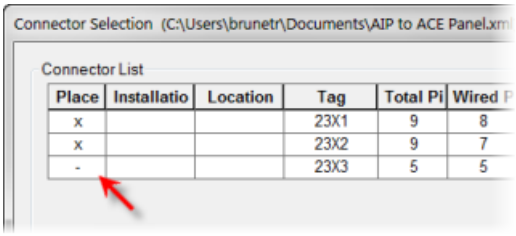
ACE - Import Inventor Data and Insert Connectors and Wires

In this exercise, you insert connectors into a new ACE drawing file. Then you import the AIP XML file you just created. The imported wires automatically connect to the proper connector pins, and automatically route through the drawing.

1. Move back to AutoCAD Electrical. **2.** In drawing *Loader-E_2014_JIC_10.dwg*, erase all wires, wire numbers and the connector 23X3.	
3. On the Schematic tab, Insert Components panel, Insert Connector flyout > click Insert Connector From List.	

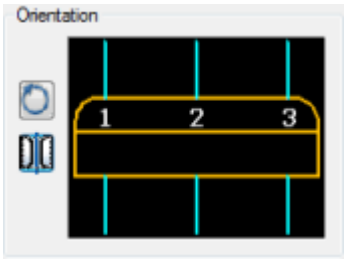
4. In the Autodesk Inventor Professional Import File Selection dialog box, do the following:
- Browse to the My Documents folder.
- Select *AIP to ACE Panel.xml*.
- Click Open.

Note: *The Place column is showing that 23X1 and 23X2 already exist in the project, but 23X3 does not.*



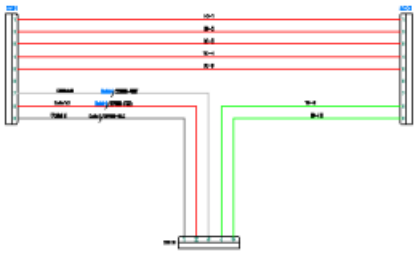
5. In the Connector Selection dialog box, do the following:
- In the grid, select 23X3.
- For Pin Spacing, enter **0.5.**
- Click Rotate and Flip to orient connector with plug to the top as shown.
- Click Insert.

6. In the drawing select an insertion point at **13.0,7.0.**



7. In the Connector Selection dialog box, click Wire It.

8. Click Cancel



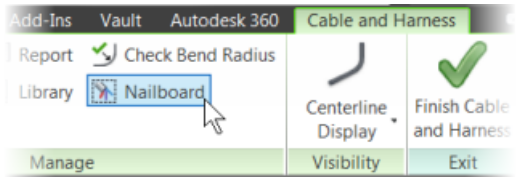
AIP - Create and Annotate Nailboard

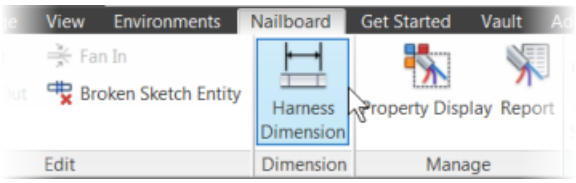
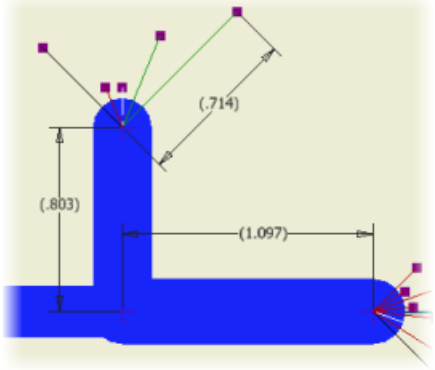
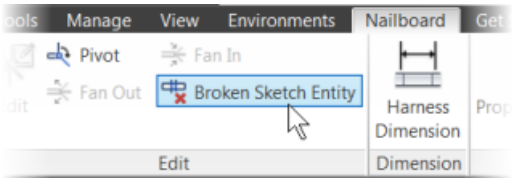
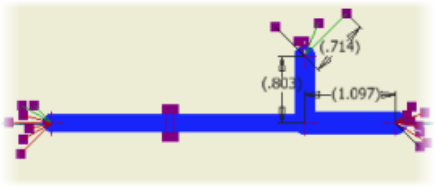
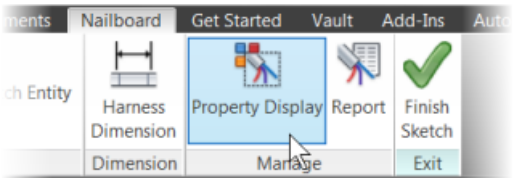
In this exercise, you move back to Autodesk Inventor Professional and create a nailboard drawing of the harness. You add dimensions, labels, and a Bill of Material table to the drawing.

1. Move back to Autodesk Inventor Professional.

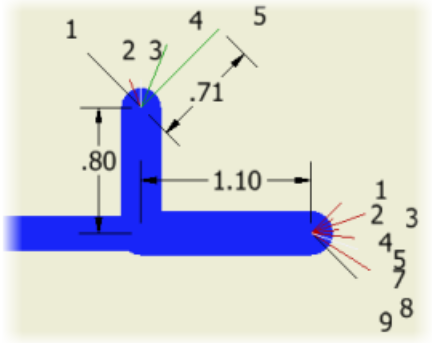
2. Select Harness1 for editing

3. On the Cable and Harness tab, Manage panel, click Nailboard.

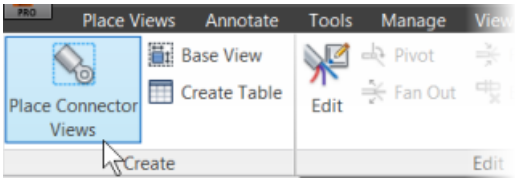


4. In the Open Template dialog box, select Standard.dwg. Click OK. ***Note:*** *The Sketch tab is active, but does not have the Nailboard commands available.* **5.** On the Nailboard tab, Dimension panel, click Harness Dimension.	
6. In the drawing, add several dimensions, selecting workpoints for dimension placement. **7.** In the Harness Dimension Warning dialog box, select Do Not Show Message Again In This Session. Click OK. ***Note:*** *Dimensions can be added to segments and individual wires.*	
8. On the Nailboard tab, Edit panel, click Broken Sketch Entity.	
9. In the drawing, select two points on the long segment. **10.** In the Edit Break dialog box, enter **4.** Press ENTER. **11.** ***Note:*** *The value entered is the distance* removed *from the sketch.*	
12. On the Nailboard tab, Manage panel, click Property Display.	

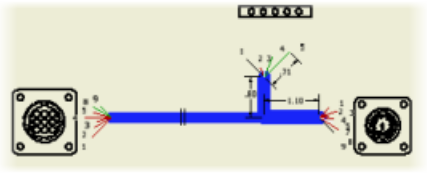
13. In the Property Display dialog box, select All Pins.
14. From the list, select Pin Name. Click OK.
15. In the drawing, select a location for the annotation from the wire end.
16. On the Nailboard tab, Exit panel, click Finish Sketch.



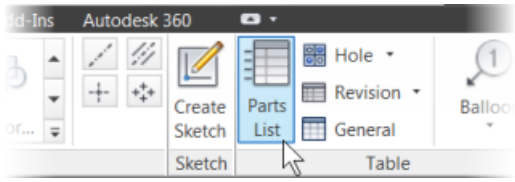
17. On the Nailboard tab, Create panel, click Place Connector Views.



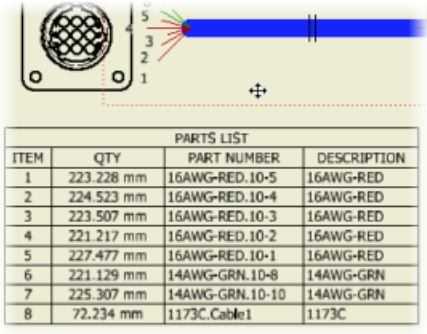
18. In the Place Connector Views dialog box, click OK.



19. On the Annotate tab, Table panel, click Parts List.
20. In the Parts List dialog box, for Select Document, click Browse.



21. In the Open dialog box, browse to Mechatronics>AIP>Cable and Harness.
22. Select Mechatronics.Harness1.iam. Click Open.
23. In the Parts List dialog box, click OK.
24. In the BOM View Disabled dialog box, click OK.
25. In the drawing, select an insertion point in the lower left corner.
26. If prompted to update assembly, click OK.



PARTS LIST

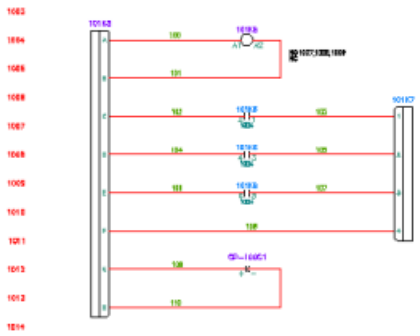
ITEM	QTY	PART NUMBER	DESCRIPTION
1	223.228 mm	16AWG-RED.10-5	16AWG-RED
2	224.523 mm	16AWG-RED.10-4	16AWG-RED
3	223.507 mm	16AWG-RED.10-3	16AWG-RED
4	221.217 mm	16AWG-RED.10-2	16AWG-RED
5	227.477 mm	16AWG-RED.10-1	16AWG-RED
6	221.129 mm	14AWG-GRN.10-8	14AWG-GRN
7	225.307 mm	14AWG-GRN.10-10	14AWG-GRN
8	72.234 mm	1173C.Cable1	1173C

ACE - Export Distributed Component, Terminal, and Terminal Strip Harnesses

In this exercise, you move back to AutoCAD Electrical and export three harness files that will be used to demonstrate several different interface workflows relating to distributed components and terminals.

1. Move back to AutoCAD Electrical.
2. Expand the Inventor Interface Loader folder.
3. Right-click *Loader-E_2014_JIC_11.dwg*. Click Open.

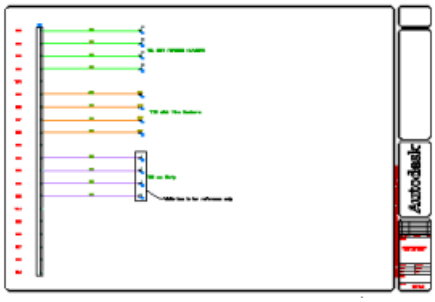
Note: *This schematic contains a relay coil (parent) with distributed (child) contacts.*



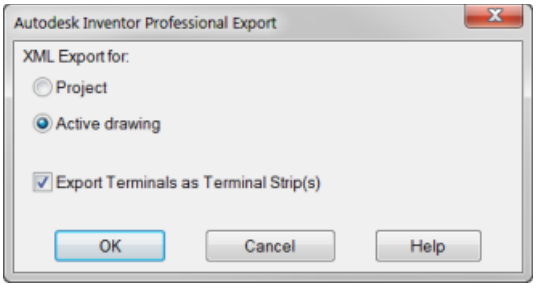
4. On the Import/Export tab, Export panel, click Inventor.
5. In the Autodesk Inventor Professional Export dialog box, select Active Drawing. Click OK.
6. In the Autodesk Inventor Professional XML File Export dialog box, for File name, enter ***ACE to AIP Distributed.xml.***
7. Click Save.

8. Open drawing *Loader-E_2014_JIC_12.dwg*.

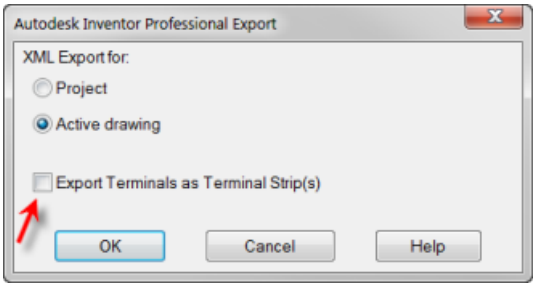
Note: *This schematic contains three terminal strips with different examples of schematic terminals, including terminals with terminal numbers, and terminals with wire numbers.*



9. On the Import/Export tab, Export panel, click Inventor.
10. In the Autodesk Inventor Professional Export dialog box, select Active Drawing. Click OK.
11. In the Autodesk Inventor Professional XML File Export dialog box, for File name, enter ***ACE to AIP Strip.xml***.
12. Click Save.



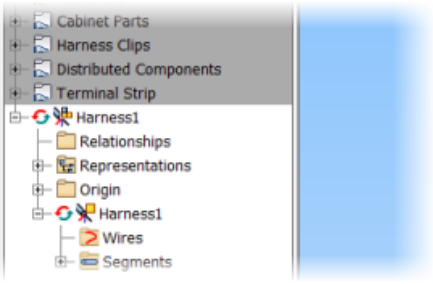
13. On the Import/Export tab, Export panel, click Inventor.
14. In the Autodesk Inventor Professional Export dialog box, select Active Drawing.
15. Deselect Export Terminals as Terminal Strip(s).Click OK.
16. In the Autodesk Inventor Professional XML File Export dialog box, for File name, enter ***ACE to AIP Terminals.xml***.
17. Click Save.



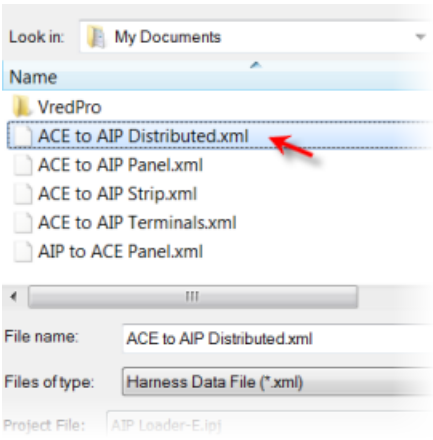
AIP - Import Distributed Harness and Edit Wire Paths

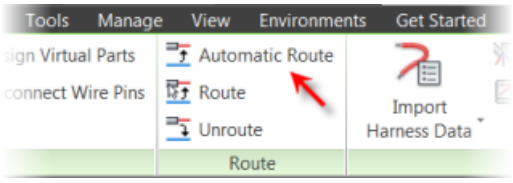
In this exercise, you move to Autodesk Inventor Professional and import the Distribute wire harness. Once wires are imported and routed, you edit the wire paths using segments, authoring a component, and by adding work points to the wires.

1. Move back to Inventor. On the Quick Access toolbar, click Open.
2. In the Open dialog box, browse to *AIP Loader-E Dataset>Workspaces>Electrical System*.
3. Select *Enclosure Assembly.iam*. Click Open.
4. Activate Harness1 for editing.



5. On the Cable and Harness tab, Manage panel, Import Harness Data flyout, click Import Harness Data.
6. In the Import Harness Data dialog box, for Harness Data File, click Browse.
7. In the Select Wire List Data File dialog box, browse to the *My Documents* folder.
8. Select *ACE to AIP Distributed.xml.* Click Open.
9. In the Import Harness Data dialog box, click OK.

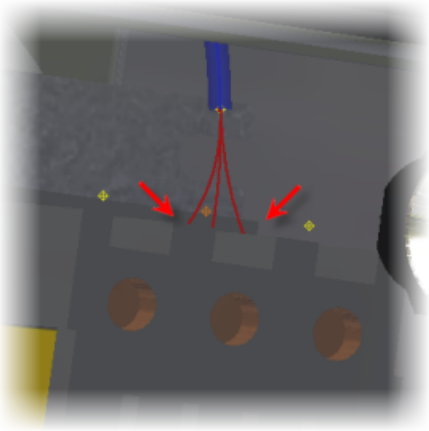


10. In the Imported Harness Data dialog box, expand Electrical Parts and Wires. *Notice that no warnings appear for the Electrical Parts or the Wires.* **11.** Click OK. **12.** In the Imported Harness Data dialog box, click Close.	
13. On the Route panel, click Automatic Route. **14.** In the Auto Route dialog box, select All Unrouted Wires. Click OK.	

AIP - Route Wires With Segments

In this exercise, you add three segments to the existing segment routing. These additional segments will each add more detail and control over the path of the wires entering the relay.

1. Zoom into the top portion of the relay as shown (near the bottom right corner of the enclosure). Notice the paths of the three wires run through the relay component.

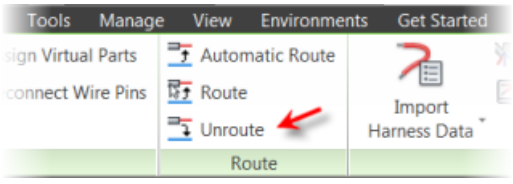


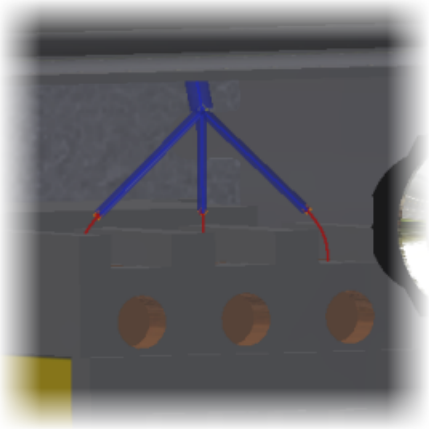
2. On the Cable and Harness tab, Create panel, click Create Segment.
3. In the assembly, select the work point at the end of the existing segment.
4. Select the left-most work point on the component.
5. Right-click anywhere in the drawing. Click Finish.

Note: *These work points were added to the component, not to the harness assembly.*

6. Repeat Steps 2-5 for the other two segments as shown.

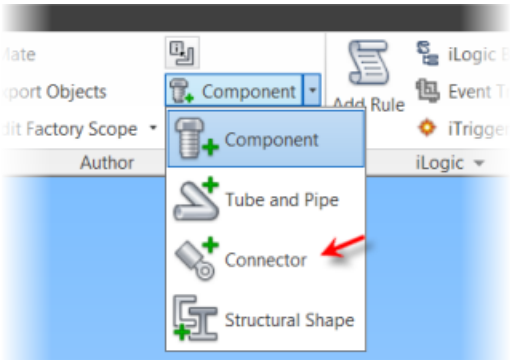
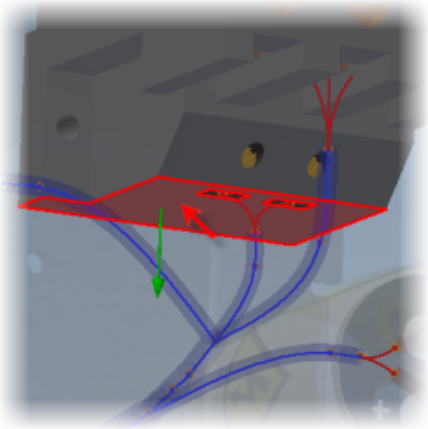
7. On the Route panel, click Unroute.
8. In the drawing, select the three red wires.
9. In the Unroute dialog box, click OK.



10. On the Route panel, click Automatic Route. **11.** In the Auto Route dialog box, select All Unrouted Wires. **12.** Click OK. *Notice how the segments changed the routing of the wires into the relay.*	

AIP - Route Wires With Authoring

In this exercise, you Author a component, and add a control direction that determines the initial path of a wire connected to the component. This is one of the steps in adding a component to the Component Library.	
1. Zoom to view the two coil wires on the bottom side of the relay as shown. *Notice how the wires curve directly from the component pins into the segment.*	

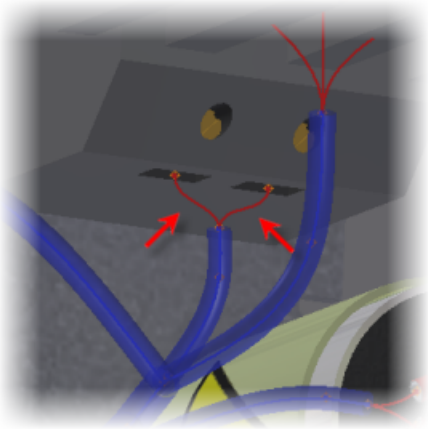
2. In the Browser, expand the folder Distributed Components. **3.** Activate 3136 7022 83:1 for editing.	
4. On the Manage tab, Author panel, Component flyout, click Connector.	
5. In the drawing, select the front face of the relay component as shown. *An arrow appears indicating the initial direction of the wire when attaching to the component.* **6.** In the Connector Authoring dialog box, click OK. **7.** In the Authoring Result dialog box, click OK.	

8. On the Manage tab, Return panel, click Return.

Notice how authoring changed the direction of the wire paths.

Note: *At this time, a component can have only one wire direction path set. For example, the wires routed through the individual segments first head in the authored direction.*

9. In the Browser, collapse the display of the Distributed Components folder.

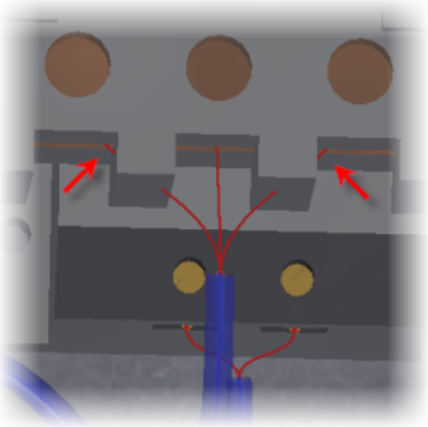


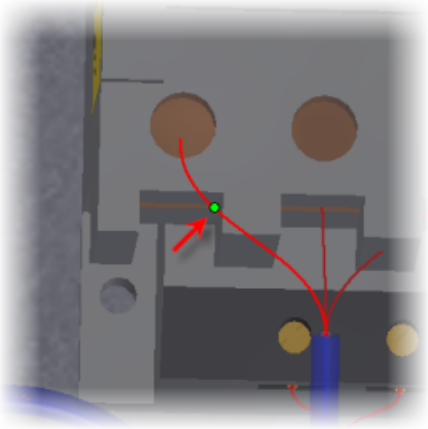
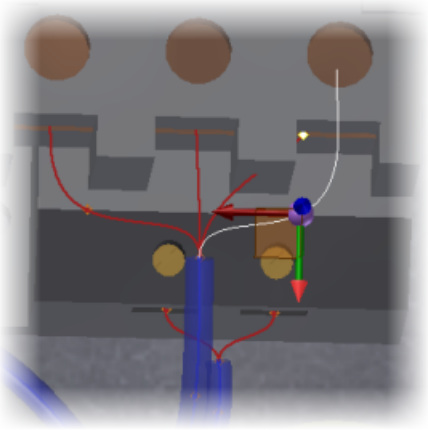
AIP - Route Wires With Workpoints

In this exercise, you add work points onto the wires and then move the work points in 3D space to control the path of the wires.

1. Zoom to the relay to see the three wires entering the top front of the relay as shown.

For this exercise, it is important to be able to see the small segments of the left and right wires when they appear in the openings.



2. In the drawing, right-click on the left wire. Select Add Points. **3.** In the drawing, select a location on the segment of wire as it appears in the component opening as shown. ***Note:*** *If you pick a point on the wire that is hidden by the component, it will be difficult to select the point in the next steps.* **4.** Right-click anywhere in the drawing area and click Finish. **5.** Repeat Steps 2-4 for the right wire.	
6. Right-click over the point you added to the left wire. Click 3D Move/Rotate. **7.** Select the arrow head on the green arrow and drag away from the component until the wire path looks approximately as shown. **8.** On the glyph, click OK. **9.** Repeat steps 6-8 for the right wire.	

AIP - Import Terminal Harness

In this exercise, you import the Terminal harness that includes wiring for terminals that were exported from AutoCAD Electrical as individual terminals.

You will discover the challenges produced in this scenario. You will explore various options and methods to correct the errors caused by importing to different types of terminal assemblies in Autodesk Inventor Professional.

Reminder:

TS1 is a terminal strip with terminal numbers assigned to the terminals.

TS2 is a terminal strip with wire numbers assigned to the terminals.

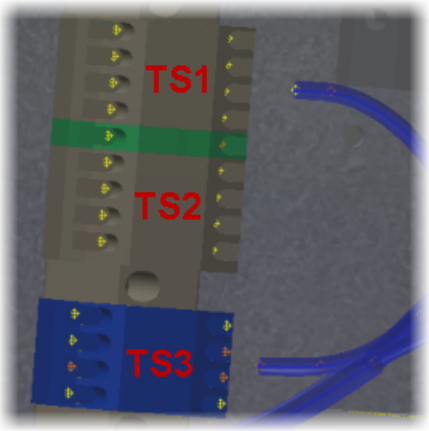
TS3 is a terminal strip with wire numbers assigned to the terminals. TS3 is created in Autodesk Inventor Professional with the four terminals combined into a single terminal strip.

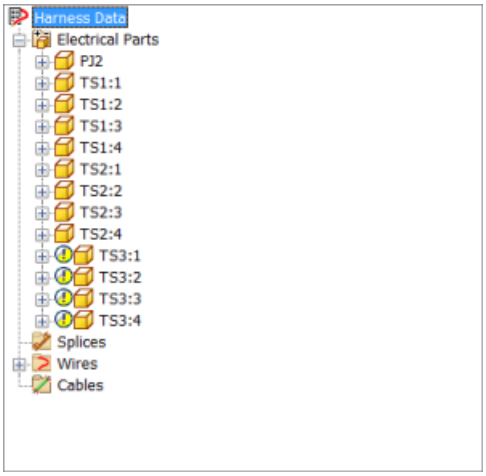
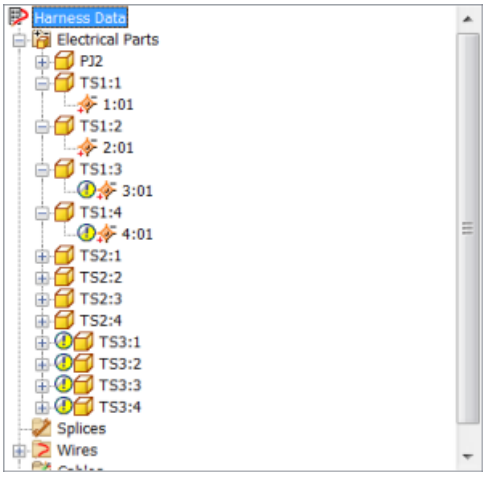
In Autodesk Inventor Professional, the same terminal components are used for TS1 and TS2.

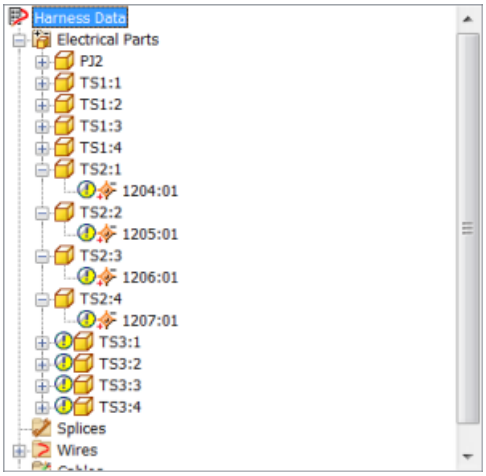
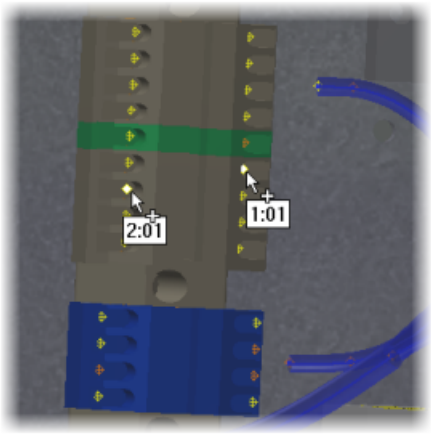
NOTE: *A current bug in the XML output from AutoCAD Electrical assigns the terminal number as the pin number. This produces several of the challenges you will see in the exercise.*

1. In the assembly, zoom into the three terminal strips as shown.

Note: *The red labels shown are for identification purposes only and do not appear in the drawing.*



2. On the Cable and Harness tab, Manage panel, click Import Harness Data. **3.** In the Import Harness Data dialog box, for Harness Data File, click Browse. **4.** In the Select Wire List Data File dialog box, browse to the My Documents folder. **5.** Select *ACE to AIP Terminals.xml.* Click Open. **6.** In the Import Harness Data dialog box, click OK.	
7. In the Imported Harness Data dialog box, expand Electrical Parts. ***Note:*** *TS1 and TS2 terminal strips appear without warning markers. TS3 has warnings for the four terminals.*	
8. In the Imported Harness Data dialog box, expand the four TS1 terminals. *Notice each terminal has only one pin connection. Pins 1 and 2 appear without warnings on two separate terminals. Pins 3 and 4 appear with warnings since they do not exist.* ***Note:*** *On a standard terminal pin 2 would not exist either. Unless a terminal is multi-level, there should be only one pin number per terminal. Pin 2 was added to the terminals for purposes of this exercise.*	

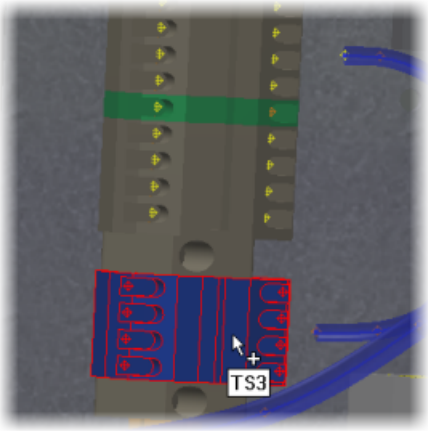
9. Expand the four TS2 terminals. *Notice all four individual pins have warnings. This is because the pin numbers listed for the imported terminals do not match the pin numbers on the existing 3D terminals.*	
10. Right-click over pin 1204:1. Click Issue Description. *Notice the description that the pin does not exist on the component.* **11.** In the Imported Harness Data (Issue Description) dialog box, click Close. **12.** Right-click over pin 1204:1. Click Rename Pin on Electrical Part. **13.** In the assembly, select the right pin on TS2:1, labeled 1:01. ***Note:*** *You can only select pins on the TS2:1 terminal.*	
14. In the Select Pin dialog box, click OK. **15.** Right-click over pin 1205:1. Click Rename Pin on Electrical Part. **16.** In the assembly, select the left pin on TS2:2, labeled 2:01. **17.** In the Select Pin dialog box, click OK. **18.** *Notice how the pins are renamed to 1:01 and 2:01.*	

19. In the Imported Harness Dialog box, right-click TS3:1. Click Issue Description.

Notice the description stating that the RefDes was not found.

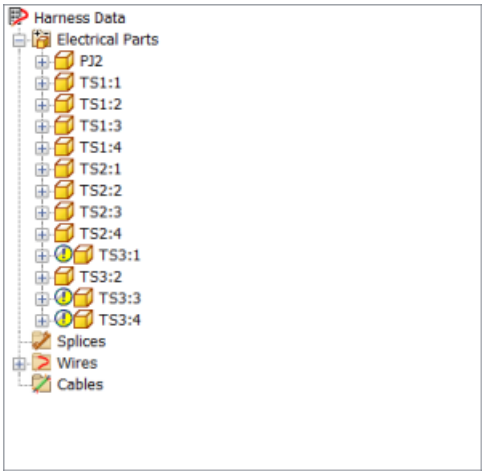
20. In the Imported Harness Data (Issue Description) dialog box, click Close.
21. Right-click over TS3:1. Click Assign to an Existing Electrical Part.
22. In the assembly, select TS3.
23. In the Select Electrical Part dialog box, click OK.

Notice the warning is removed from TS3:1.



24. Right-click over TS3:2. Click Assign to an Existing Electrical Part.
25. In the assembly, select TS3:1.
26. In the Select Electrical Part dialog box, click OK.

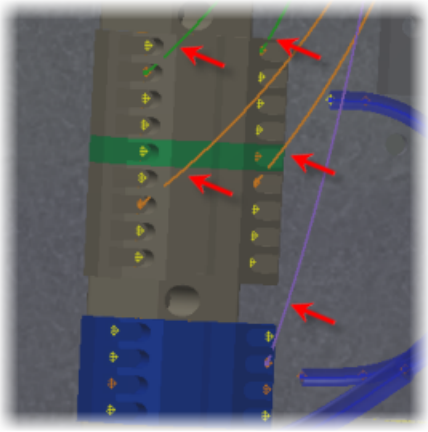
Notice the warning is removed from TS3:2, but now appears again on TS3:1. There is only one TS3 component in the model.



27. In the Imported Harness Data dialog box, click OK.
28. In the Imported Harness Data dialog box, click Close.

Note: *Due the pin numbering discrepancy in the XML file, only five wires are imported into the model.*

29. On the Quick Access toolbar, click Undo as required to remove the wires.



AIP - Import Strip Harness

In this exercise, you import the Terminal harness that includes wiring for terminals that were exported from AutoCAD Electrical as a single terminal strip.

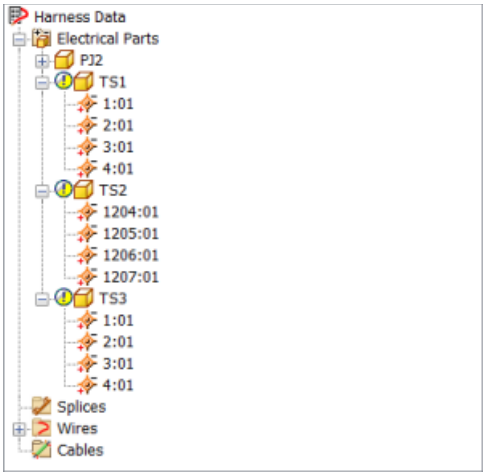
You will discover the challenges produced in this scenario. You will explore various options and methods to correct the errors caused by importing to different types of terminal assemblies in Autodesk Inventor Professional.

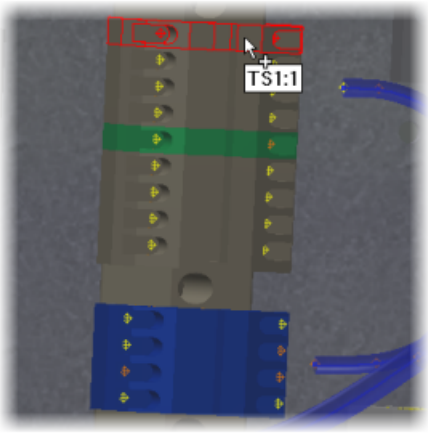
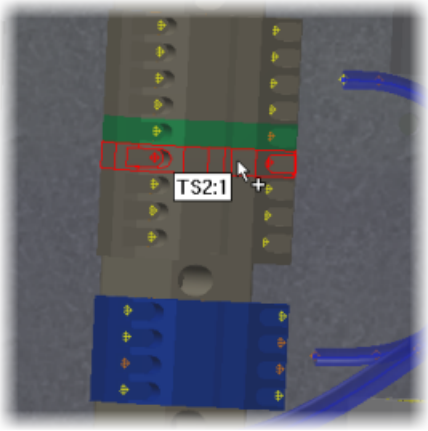
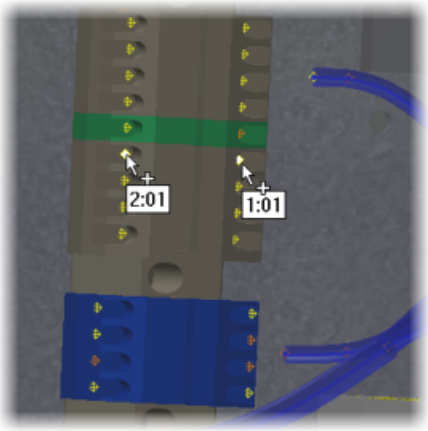
1. On the Cable and Harness tab, Manage panel, click Import Harness Data.
2. In the Import Harness Data dialog box, for Harness Data File, click Browse.
3. In the Select Wire List Data File dialog box, browse to the My Documents folder.
4. Select *ACE to AIP Strip.xml.* Click Open. Click OK.

5. In the Imported Harness Data dialog box, expand Electrical Parts.
6. Expand the three terminal strips.

Note: *All terminals appear as a single component with four pins.*

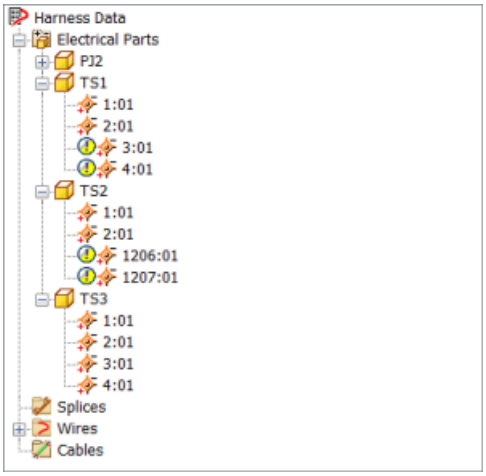
Also note that Warning markers appear on all three terminal strips. This is due to naming convention changes between terminal and strip XML files.



7. Right-click over TS1. Click Assign to an Existing Electrical Part. **8.** In the assembly, select the top terminal on TS1. **9.** In the Select Electrical Part dialog box, click OK. *Notice the warning is removed from TS1 and from pins 1 and 2. (Pins 3 and 4 will always produce errors.)*	
10. Right-click over TS2. Click Assign to an Existing Electrical Part. **11.** In the assembly, select the top terminal on TS2. **12.** In the Select Electrical Part dialog box, click OK. *Notice the warning is removed from TS2, but the four pins now have warnings assigned to them.*	
13. Right-click over pin 1204:1. Click Rename Pin on Electrical Part. **14.** In the assembly, select the right pin on TS2, labeled 1:01. **15.** In the Select Pin dialog box, click OK. **16.** Right-click over pin 1205:1. Click Rename Pin on Electrical Part. **17.** In the assembly, select the left pin on TS2, labeled 2:01. **18.** In the Select Pin dialog box, click OK. ***Note:*** *You can only select pins on the TS2 terminal you selected in previous steps.*	

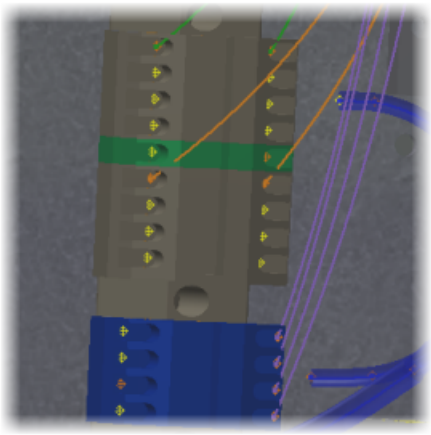
19. In the Imported Harness Data dialog box, right-click over TS3. Click Assign to an Existing Electrical Part.
20. In the assembly, select TS3:1.
21. In the Select Electrical Part dialog box, click OK.

Notice the warning is removed from TS3, and the four pins also appear without warnings. This component has the four terminals from AutoCAD Electrical combined into a single component in Autodesk Inventor Professional.



22. In the Imported Harness Data dialog box, click OK.
23. In the Imported Harness Data dialog box, click Close.

This concludes the exercise steps.

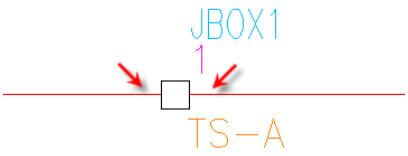


Section 5: More Tips and Tricks

Our customers have provided feedback on a couple of places where they sometimes get stuck. Below, you'll see examples of the challenges, and our suggested solutions.

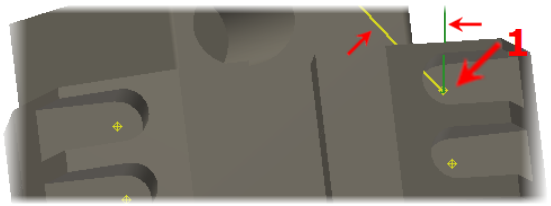
One Physical Location per Terminal

In AutoCAD Electrical, a terminal is typically shown with one wire entering the terminal and another wire leaving the terminal as shown in the image.



This represents the two sides of the physical terminal that would have two wire connection points.

The challenge is that the XML output file from AutoCAD Electrical contains only a single wire connection point, in this case labeled as pin #1. This means that both wires will attach to the same wire connection point (pin) on the physical model in Autodesk Inventor Professional as shown in image.



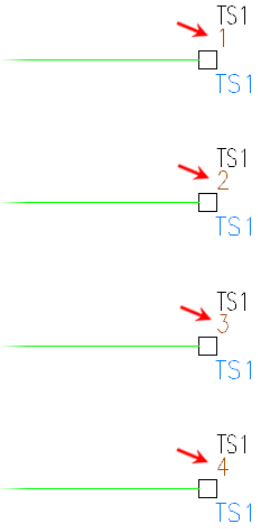
Solution

Ideally, in Autodesk Inventor Professional one wire should attach to each pin on the component model. In the AutoCAD Electrical XML output, only a single pin number is listed for a terminal. While this has been reported, and *may* appear in a hotfix or new release soon, currently there is not a verified work around.

Terminal # Exported as Pin

In AutoCAD Electrical, the unique identifier for the terminal is also used as the pin number on the terminal. This is true for either terminals labeled with numbers, or terminals labeled with wire numbers. For the physical model, the pin numbers on each similar part number should be consistent.

In the schematic example, shown on left, the terminals are identified with the numbers, 1, 2, 3, and 4.



In the XML file, the pin definitions are also labeled with this numbering. In each case the Pin Definition for the terminal *should* be the same, “1:01”.

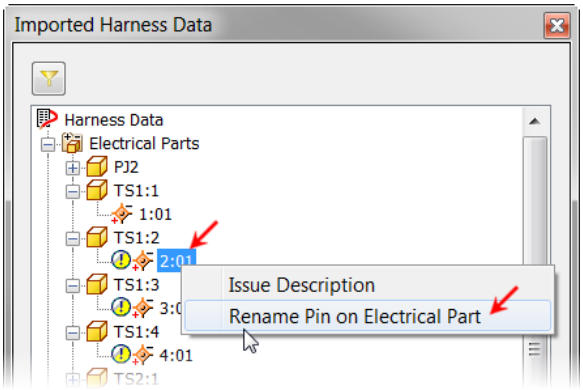
```
</Connector><Connector RefDes="TS1:1"
Definition="HT0001"><Property Name="CAT" Value="TERM-
12"/><Property Name="MFG" Value="ACME"/><Property
Name="WDBLKNAM" Value="HT0001"/><Pin Definition="1:01"/>

</Connector><Connector RefDes="TS1:2"
Definition="HT0001"><Property Name="CAT" Value="TERM-
12"/><Property Name="MFG" Value="ACME"/><Property
Name="WDBLKNAM" Value="HT0001"/><Pin Definition="2:01"/>

</Connector><Connector RefDes="TS1:3"
Definition="HT0001"><Property Name="CAT" Value="TERM-
12"/><Property Name="MFG" Value="ACME"/><Property
Name="WDBLKNAM" Value="HT0001"/><Pin Definition="3:01"/>

</Connector><Connector RefDes="TS1:4"
Definition="HT0001"><Property Name="CAT" Value="TERM-
12"/><Property Name="MFG" Value="ACME"/><Property
Name="WDBLKNAM" Value="HT0001"/><Pin Definition="4:01"/>
```

Solution

When exporting the terminals as individual terminals, export from AutoCAD Electrical following standard workflow.

In Autodesk Inventor Professional, on the component model remove or rename the Pin 2 name reference on the component so that an error is caused in the Imported Harness Data dialog box.

(In the example dataset only terminal number 2 on TS1 needs to be edited.)



Import the XML file. In the Imported Harness Data dialog box, right-click over each pin error. Select Rename Pin on Electrical Part.

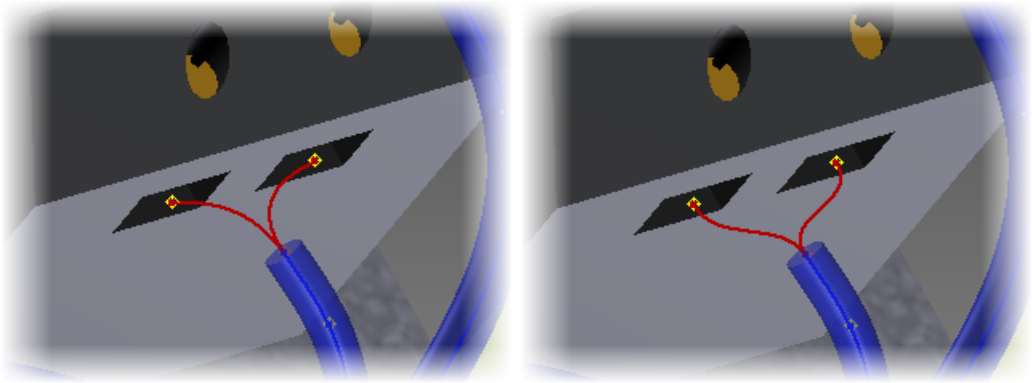
In the canvas, select Pin #1 on the component. Repeat for each terminal in the terminal strip, renaming all pin numbers to match the AutoCAD Electrical drawing.

This process will also work for terminals labeled as wire numbers.

Only One Authored Wire Direction

In Autodesk Inventor Professional, you author a connector to create an electrical component for the content center library. You are given several choices for Termination Type including Crimp and Insulation Displacement. Setting the Outward Direction determines the initial direction of the path of the wire as it leaves the pin.

Only one direction can be specified. This means the physical models that have wires entering and leaving the component from two different directions, such a relay, or terminals have wire routing path discrepancies.



Solution

Ideally, Autodesk Inventor Professional should create a wire direction property for each pin on the component. Currently only one direction is established per a component. While this has been reported, and *may* appear in a hotfix or new release soon, currently there is not a verified work around.

Additional Learning Resources

Product	Description	Link
AutoCAD Electrical	Tutorials	http://docs.autodesk.com/ACAD_E/2013/ENU/ → Tutorials → Interoperability: Inventor and AutoCAD Electrical Help
Inventor	Tutorials	http://help.autodesk.com/view/INVNTOR/2014/ENU/ → Inventor Help Topics → Cable & Harness