Autodesk University 2012

Mechatronics: Taming the Beast

Randy Brunette – Autodesk, Inc.

MA2923 In many companies, the mechanical and electrical design are combined, creating a single mechatronic design process. Do your mechanical designs include electrical wires, harnesses, and components? When completing your electrical designs, would you like to use tools especially designed for electrical drawing creation? Autodesk® Inventor® is the design tool of choice for the physical components, AutoCAD® Electrical for the electrical components. You can create a seamless interface between the two using some basic guidelines. We'll study these rules and how they work. During the class, we will use AutoCAD Electrical to create a circuit representing a wire harness and export the data into Autodesk Inventor. In Autodesk Inventor, we will create wires, cables, and ribbon cables, and export the data into AutoCAD Electrical.

Learning Objectives

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

- Create and export a harness from AutoCAD Electrical
- Create electrical components in Inventor
- Import, route, edit, and export harness data in Inventor
- Export edited harness data from Inventor and import it into AutoCAD Electrical

About the Speaker

Randy Brunette has joined Autodesk as an Electrical Subject Matter Expert. Randy's duties on the CIA team 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 27 years, with experience across multiple segments of the Manufacturing industry. He has been in an Application Engineer role for 18 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

AutoCAD Electrical Overview

AutoCAD Electrical is a purpose built software built on top of the AutoCAD engine. In addition to all of the AutoCAD functionality, AutoCAD Electrical has a complete set of tools to create and modify electrical control systems. Specialized features such as Trim Wire, Copy Component, Copy Circuit, Scoot, and Align, make it much easier to create drawings quickly.

Comprehensive symbol libraries for electrical components aid productivity and consistency. Libraries are even included for creating Pneumatic, Hydraulic, and P&ID schematic drawings. A simple, icon-menu-driven system provides an interface for you to quickly build schematic drawings using simple pick-and-place methods. Links can be established between all of the different types of drawings and components, so that changes to one will automatically update the others.

Many types of error checking capabilities are built into the software to catch problems early in the design process, or eliminate them completely. Duplication of pin assignments and tag names, coil and pin cross referencing errors, and so on are just some of the real-time checking built into the software.

Once the design is complete, the documentation process begins, including BOM, component, and wire connection reports. All design data from the drawings can be quickly and easily extracted, and configured into report styles that match your company's standards.

Top 10 Reasons to Use AutoCAD Electrical

1. Comprehensive symbol libraries
2. Automatic wire numbering and component tagging
3. Automatic project reports
4. Real-time error checking
5. Real-time coil and contact cross-referencing
6. Smart panel layout drawings
7. Electrical-specific drafting features
8. Ability to automatically create PLC I/O drawings from spreadsheets
9. Ability to share drawings with customers and suppliers and track their changes
10. Reuse existing drawings

Autodesk Inventor Overview

Autodesk Inventor is the industry leading software for 3D Mechanical design, simulation, and design communication. The Inventor model is a digital prototype used to validate form, fit, and function of a design as you work. Using digital designs is more efficient and cost effective than the old processes of build, test, and rebuild.

The Inventor parametric design environment automates advanced geometry creation of components such as plastic parts, steel frames, rotating machinery, tube and pipe runs, and electrical cable and wire harnesses. The built-in intelligence of the software helps to reduce errors and improve design accuracy.

Tools are built into the software to provide motion simulation and stress analysis, allowing you to optimize and validate your design before the product is built.

Creating drawings and reports directly from the 3D model is a quick and easy process, enabling you to effectively communicate your 3D digital design information to everyone in manufacturing process. Inventor is tightly integrated with Autodesk data management applications, enabling the efficient and secure exchange of design data and promoting earlier collaboration between design and manufacturing workgroups.

Top 10 Reasons to Use Autodesk Inventor

1. Digital Prototyping
2. 3D Mechanical Design
3. AutoCAD and DWG Compatibility
4. Automatic Drawing Updates and Views
5. Design Automation
6. Integrated Data Management
7. Technical Documentation
8. State-of-the-Art Rendering
9. Integrated Stress Analysis and Simulation
10. Pipe and Cable Routing

AutoCAD Electrical and Autodesk Inventor Inter-workability

A series of demos is included to show the inter-workability of AutoCAD Electrical and Autodesk Inventor Professional. You add connectors and wiring to an AutoCAD Electrical (ACE) drawing. Then you run a report to extract the wiring data for use in Autodesk Inventor Professional (AIP). In Inventor you create a new component and configure an Inventor assembly to prepare it for the importing of the wiring information from ACE.

After importing the wiring information, you route the wires through the 3D assembly, creating the wiring harness, replacing some of the imported wires with a cable.

Lastly you export the wiring harness information as it was created and modified in Inventor. You import the data into ACE, and use the data to automatically insert connectors and re-create the original wiring diagram. This completes the “round-trip” of the wiring information from ACE to AIP, and back to ACE.

Guidelines:

Follow these guidelines to enable transfer of data between AutoCAD Electrical and Autodesk Inventor.

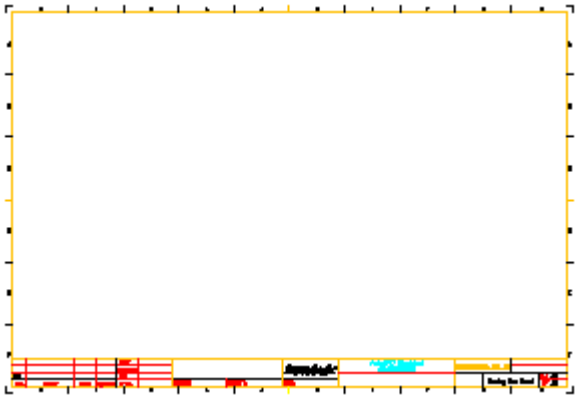
1. RefDes in Autodesk Inventor Professional (AIP) must have the same value as the TAG attribute in AutoCAD Electrical (ACE).
2. Pin in AIP must have the same value as the TermXX attribute in ACE.
3. Wiretype used in ACE must be available in the AIP library.
4. All wires in ACE must have wire numbers.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

*ACE-Insert Connectors

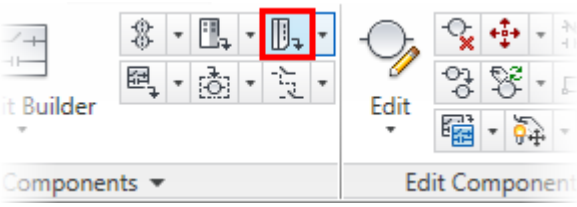
In this demonstration, you add connectors to an AutoCAD Electrical drawing. Any type of component could be inserted, as long as the guidelines are followed.

1. In the Project Manager, right-click in an empty area. Click Open Project.
2. In the Select Project File dialog box, do the following:
- Browse to the dataset folder.
- Select Mechatronics.wdp.
- Click Open.

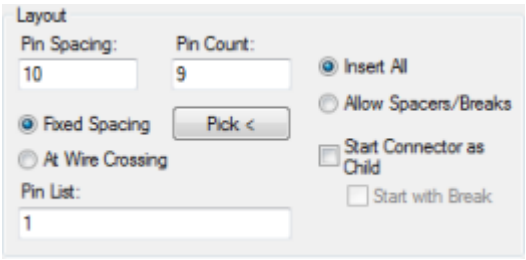
3. In the Project Manager, do the following:
- Double-click the icon for the Mechatronics project.
- Right-click Mechatronics_IEC_022.dwg.
- Click Open.



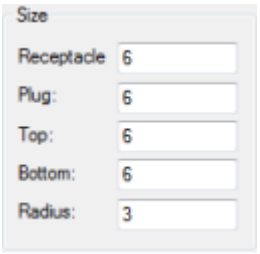
4. On the Schematic tab, Insert Components panel, click Insert Connector.

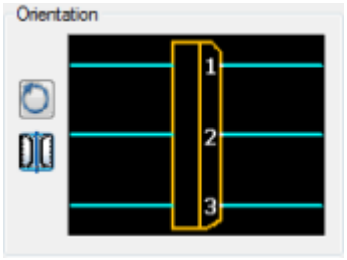
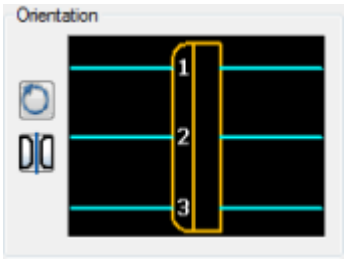
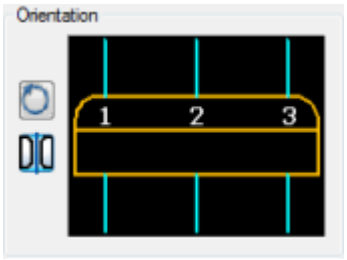
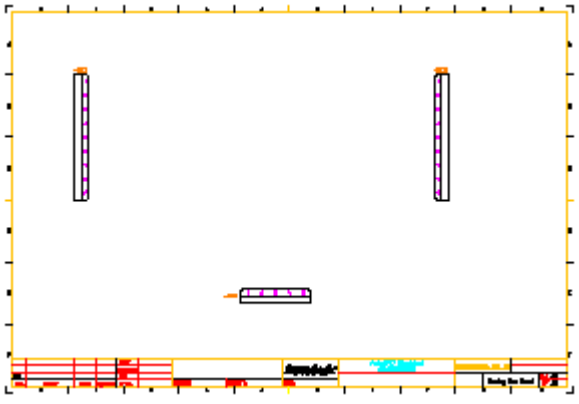


5. In the Insert Connector dialog box, do the following:
- For Pin Spacing, enter **10.**
- For Pin Count, enter **9**.
- Select Fixed Spacing.
- Select Insert All.
- If necessary, click Details >> to expand dialog box.



- Under Size,
  - For all sizes, enter **6**.
  - For Radius, enter **3**.

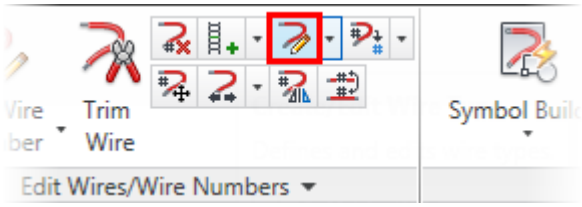


6. Under Orientation, click Rotate and Flip to orient connector with plug to the right as shown. **7.** Click Insert. **8.** In the drawing, select an insertion point at **60.0,230.0**.	
9. On the Schematic tab, Insert Components panel, click Insert Connector. **10.** In the Insert Connector dialog box, do the following: • Click Rotate and Flip to orient connector with plug to the left as shown. • Click Insert. **11.** In the drawing, select an insertion point at **320.0,230.0**.	
12. On the Schematic tab, Insert Components panel, click Insert Connector. **13.** In the Insert Connector dialog box, do the following: • For Pin Count, enter **5**. • Click Rotate and Flip to orient connector with plug to the top as shown. • Click Insert.	
14. In the drawing, select an insertion point at **180.0,75.0**.	

*ACE-Insert Wires

In this demonstration, 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 Create/Edit Wire Type.



2. In the Create/Edit Wire Type dialog box, for Layer Name Format, enter **%S-%C**.

3. In the Create/Edit Wire Type dialog box, do the following:

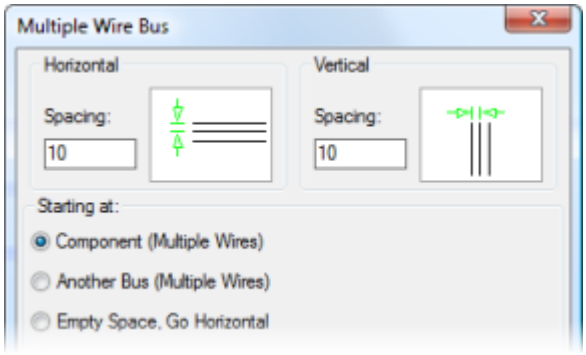
- Create wire layers named,
  - **16AWG-RED**
  - **18AWG-BLU**
  - **14AWG-GRN**
- Set layer colors to match.
- Click OK.

RD	6_mm^2	RD_6_mm^2	Yes
WH	2.5_mm^2	WH_2.5_mm^2	Yes
		WIRES	Yes
RED	16AWG	16AWG-RED	Yes
BLU	18AWG	18AWG-BLU	Yes
GRN	16AWG	16AWG-GRN	Yes

4. On the Schematic tab, Insert Wires/Wire Numbers panel, click Multiple Bus.

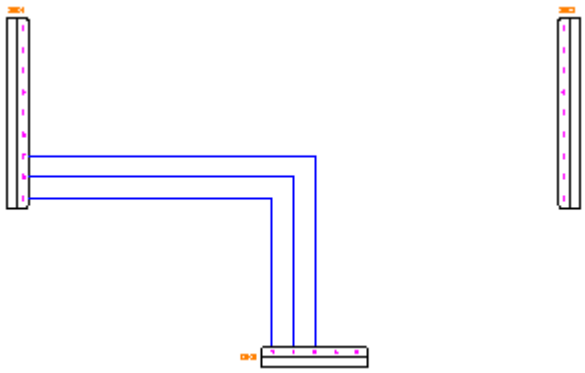
5. In the Multiple Wire Bus dialog box, do the following:

- For Horizontal Spacing, enter **10**.
- For Vertical Spacing, enter **10**.
- Select Component (Multiple Wires).
- Click OK.



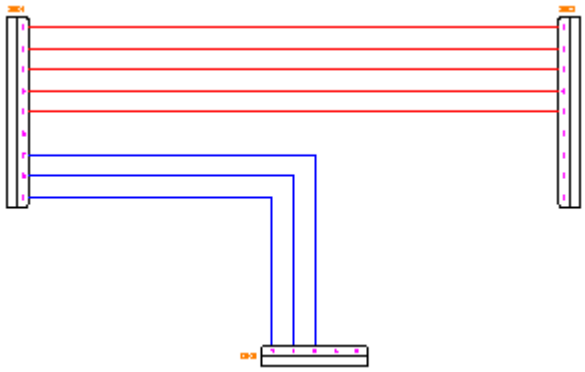
6. In the drawing, on 22X3, window and select pins 1, 2, and 3. Press ENTER.
7. At the command line, enter **T**. Press ENTER.
8. In the Set Wire Type dialog box, select 18AWG-BLU. Click OK.
9. Drag the cursor up and to the left.
10. At the command line, enter **F**. Press ENTER.
11. In the drawing, on 22X1, select pin 7.

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



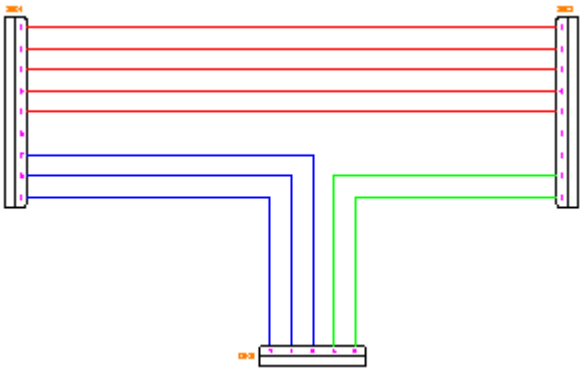
12. On the Schematic tab, Insert Wires/Wire Numbers panel, click Multiple Bus.
13. In the Multiple Wire Bus dialog box, do the following:
- For Horizontal Spacing, enter **10**.
- For Vertical Spacing, enter **10**.
- Select Component (Multiple Wires).
- Click OK.

14. In the drawing, on 22X1, window and select pins 1, 2, 3, 4, and 5. Press ENTER.
15. At the command line, enter **T**. Press ENTER.
16. In the Set Wire Type dialog box, select 16AWG-RED. Click OK.
17. Connect to 22X2, pins 1, 2, 3, 4, and 5.

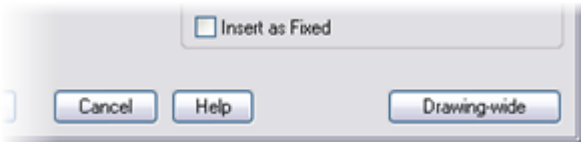


18. On the Schematic tab, Insert Wires/Wire Numbers panel, click Multiple Bus.
19. In the Multiple Wire Bus dialog box, do the following:
- For Horizontal Spacing, enter **10**.
- For Vertical Spacing, enter **10**.
- Select Component (Multiple Wires).
- Click OK.

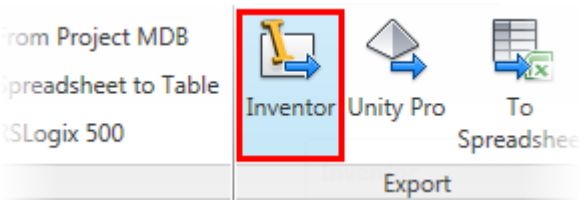
20. In the drawing, on 22X3, window and select pins 4 and 5. Press ENTER.
21. At the command line, enter **T**. Press ENTER.
22. In the Set Wire Type dialog box, select 16AWG-GRN. Click OK.
23. Connect to 22X2, pins 8 and 9.



24. On the Schematic tab, Insert Wires/Wire Numbers panel, Wire Numbers.
25. In the Wire Tagging dialog box, click Drawing Wide.

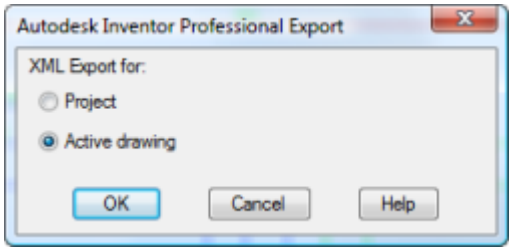


26. On the Import/Export Data tab, Export panel, click Inventor.
27. In the Autodesk Inventor Professional Export dialog box, select Active Drawing. Click OK.
28. If prompted, in the Qsave dialog box, click Always Qsave.



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

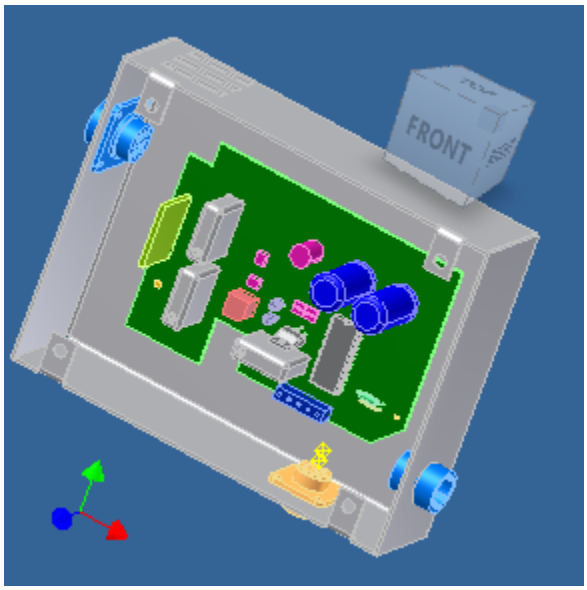
30. On the Quick Access toolbar, click Save.



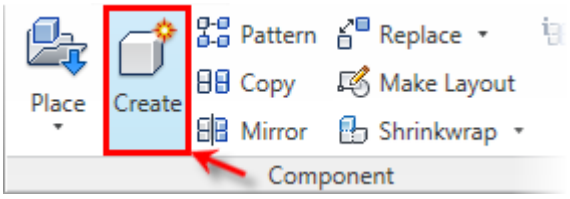
*Inventor-Create Connector

In this demonstration, you open a cabinet file 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.

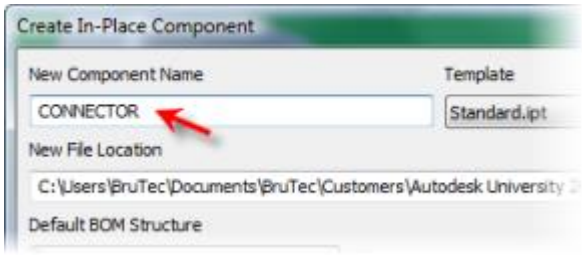
1. Move to Inventor.
2. On the Get Started tab, Launch panel click Projects.
3. In the Projects dialog box, click Browse.
4. In the Choose Project File dialog box, do the following:
   - Browse to the class files Mechatronics Dataset folder.
   - Select Mechatronics.ipj.
   - Click Open
5. In the Projects dialog box, click Done.
6. On the Get Started tab, Launch panel click Open.
7. In the Open dialog box, select Mechatronics.iam. Click Open.



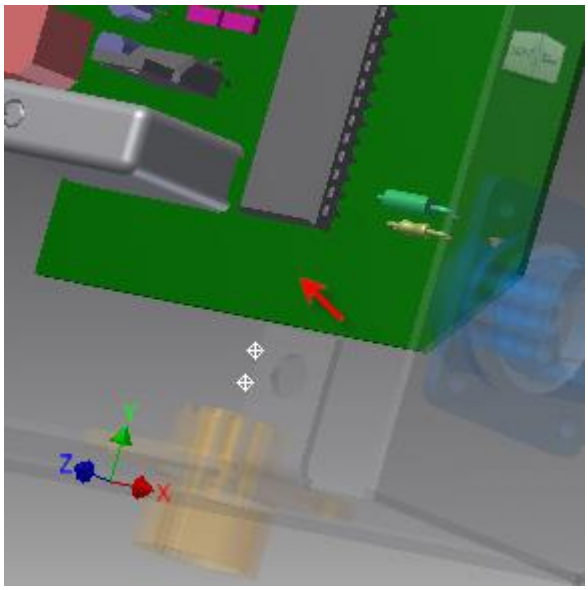
8. In the Browser, double-click PCB:1 to activate it for editing.
9. On the Assemble tab, Component panel, click Create.



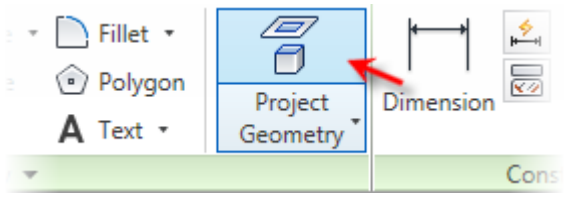
10. In the Create In-Place Component dialog box, do the following:
    - For New Component Name, enter **CONNECTOR**.
    - Click OK.
11. In the drawing, select the back PCB board (green).



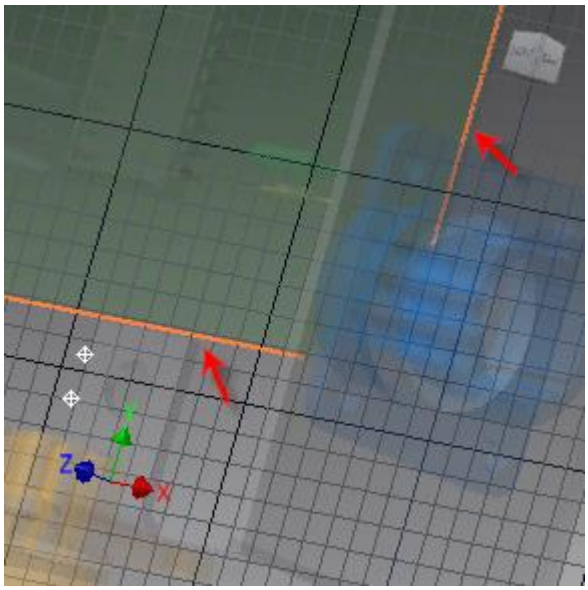
12. If necessary, on the Tools tab, Options panel, Document Settings, Units tab, under Units, set Length to Millimeters. Click OK.
13. On the 3D Model tab, Sketch panel, click Create 2D Sketch.
14. In the drawing, zoom into the lower right corner of the green back panel.
15. Select the top surface of the green back panel.



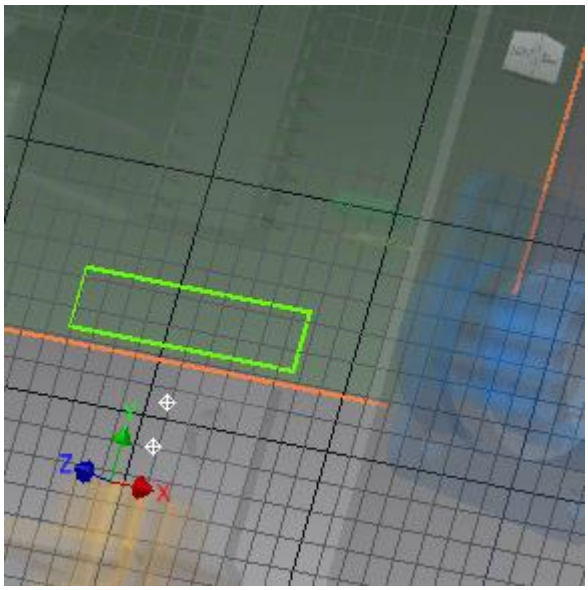
16. On the Sketch tab, Draw panel, click Project Geometry.



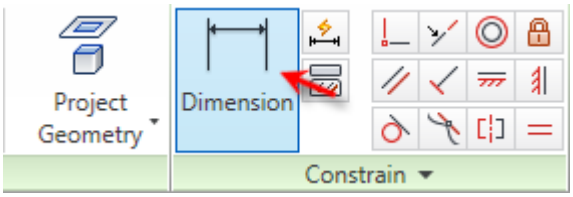
17. In the drawing, select the lower edge of the panel.
18. On the Sketch tab, Draw panel, click Project Geometry.
19. In the drawing, select the right edge of the panel.



20. On the Sketch tab, Draw panel, click Rectangle.
21. In the drawing, select two points to create a rectangle aligned with the lower edge of the panel.
22. Right-click in an empty area. Click OK.

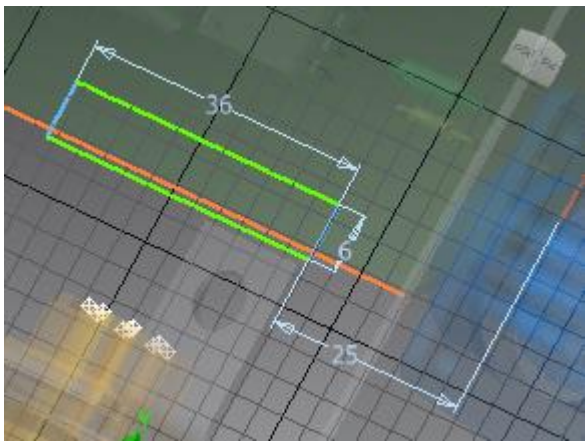


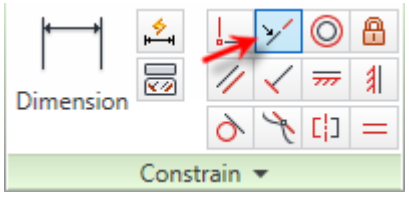
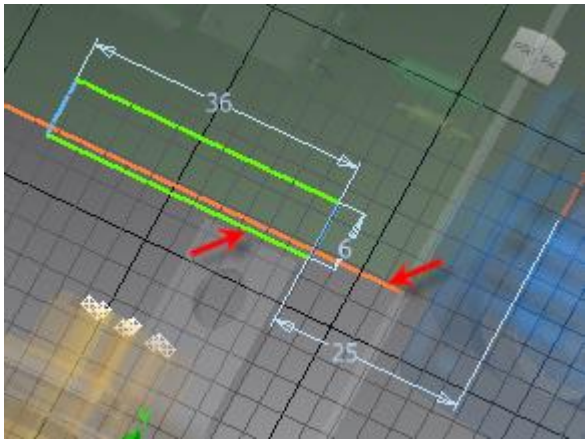
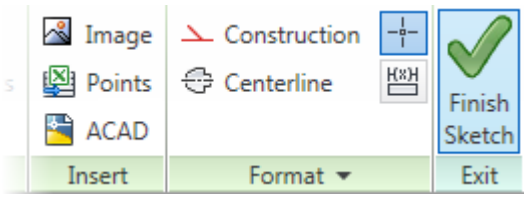
23. On the Sketch tab, Constrain panel, click Dimension.



24. In the drawing, do the following:

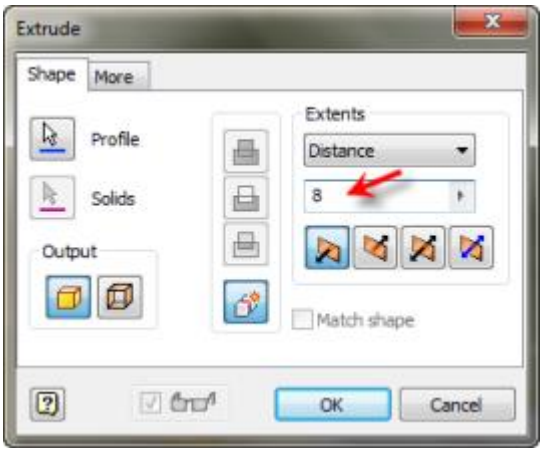
- Select the right vertical line on the rectangle.
- Locate dimension to the right of the rectangle.
- For dimension value, enter **6.0**.
- Select the upper horizontal line on the rectangle.
- Locate dimension above the rectangle.
- For dimension value, enter **36.0**.
- Select the right vertical projected line.
- Select the right vertical line on the rectangle.
- Locate dimension below the rectangle.
- For dimension value, enter **25.0**.



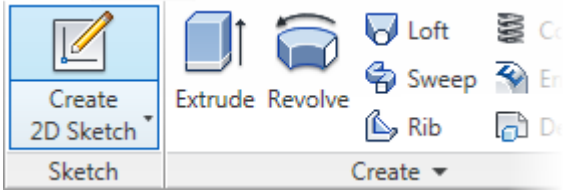
25. On the Sketch tab, Constrain panel, click Collinear Constraint.	
26. In the drawing, do the following: • Select the lower projected line. • Select the lower edge of the rectangle.	
27. On the Sketch tab, Exit panel, click Finish Sketch.	
28. On the 3D Model tab, Create panel, click Extrude.	

29. In the Extrude dialog box, for Distance, enter **8.0**.
30. Click OK.

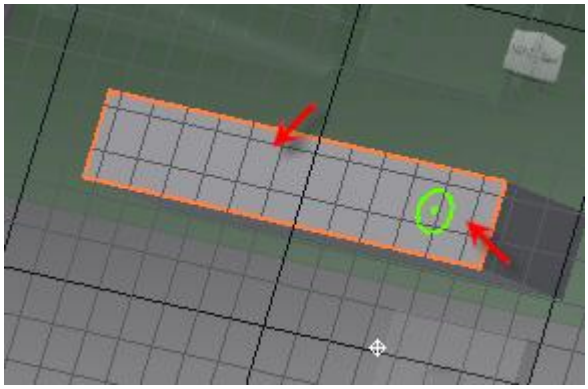
The connector body is created.



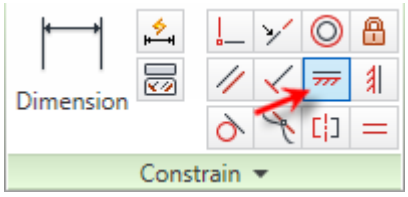
31. On the 3D Model tab, Sketch panel, click Create 2D Sketch.



32. In the drawing, select the top surface of the connector.
33. On the Sketch tab, Draw panel, click Circle.
34. In the drawing, draw a circle on the top of the connector, near the right side.

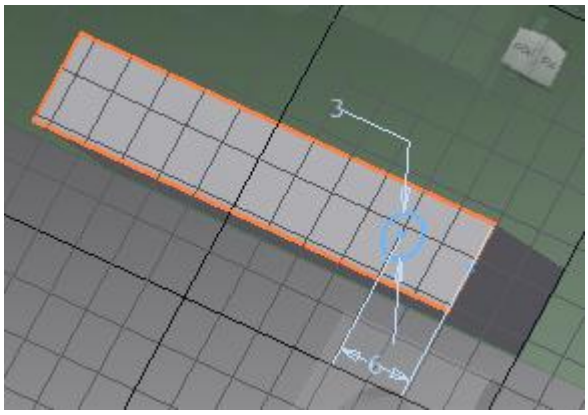


35. On the Sketch tab, Constrain panel, click Horizontal Constraint.

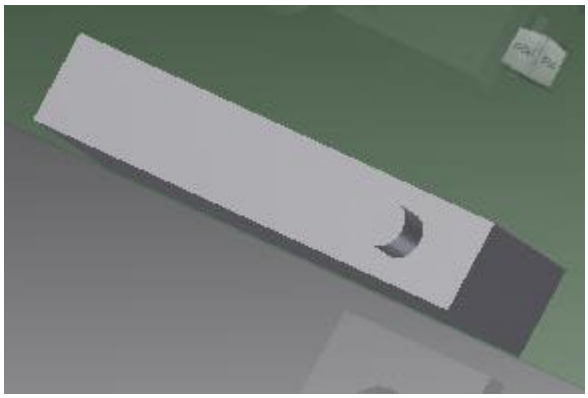


36. In the drawing, do the following:
- Select the midpoint of the right side of the rectangle.
- Select the center of the circle.

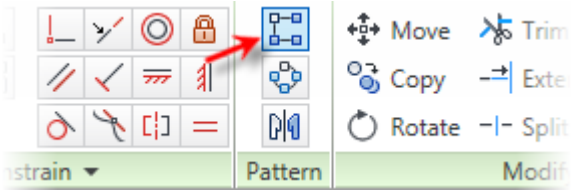
37. On the Sketch tab, Constrain panel, click Dimension.
38. In the drawing, do the following:
39. Select the right edge of the top of the connector.
40. Select the center of the circle.
41. Place the dimension below the connector.
42. For dimension value, enter **6.0**.
43. Select the circle.
44. Place the dimension above the connector.
45. For the dimension value, enter **3.0**.
46. On the Sketch tab, Exit panel, click Finish Sketch.



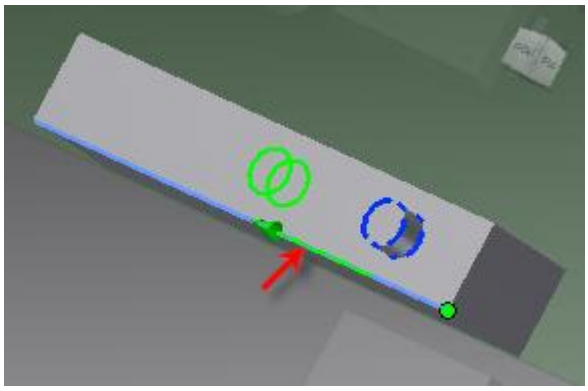
47. On the Model tab, Create panel, click Extrude.
48. In the Extrude dialog box, verify that Profile is selected.
49. In the drawing, select the center of the circle.
50. For the dimension value, enter **2.0**. Click Accept.



51. On the 3D Model tab, Pattern panel, click Rectangular Pattern.



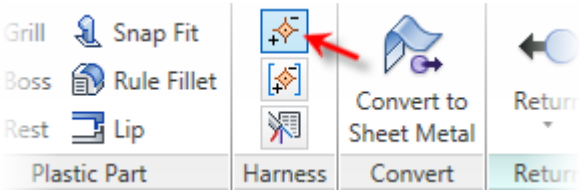
52. In the drawing, select the pin feature you just created.
53. In the Rectangular Pattern dialog box, select Direction 1.
54. In the drawing, select the lower edge of the top surface of the connector.
55. If necessary, flip direction as shown.



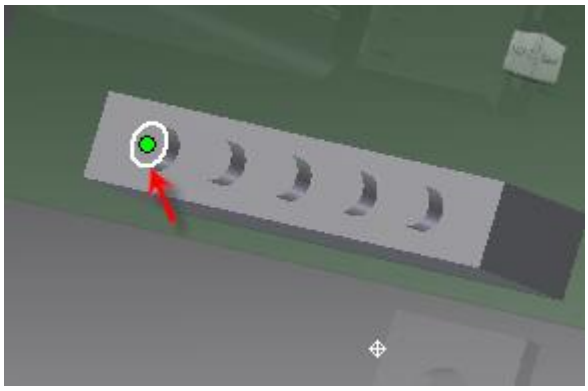
56. In the Rectangular Pattern dialog box, do the following:
57. For Number, enter **5.0.**
58. For Distance, enter **6.0**.
59. Click OK.

Next you insert the pin connections.

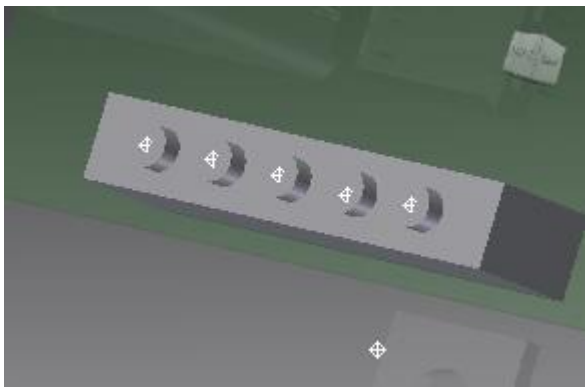
60. On the Model tab, Harness panel, click Place Pin.



61. In the drawing, select the top center of the left most pin.
62. In the Place Pin dialog box, enter 1.

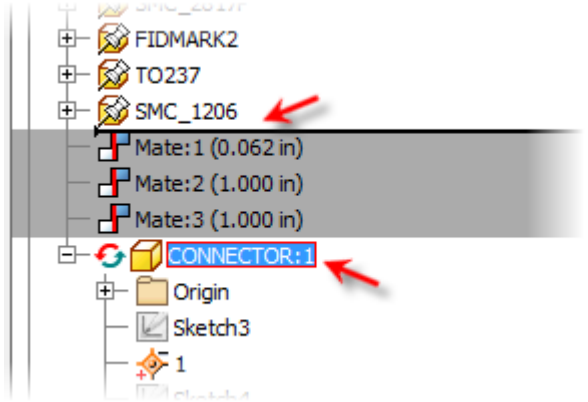


63. Working from left to right, repeat the above steps, placing in the remaining pins, entering values **2, 3, 4, and 5.**
64. On the Manage tab, Author panel, Component flyout, click Connector.
65. In the Connector Authoring dialog box, for Termination Type, select Crimp.
66. Select a top surface on the Connector. Click OK.
67. In the Authoring Result dialog box, click OK.
68. Right-click in the drawing area. Click Finish Edit.



69. (Optional) In the Browser, relocate Connector above the three Mate constraints, by dragging and dropping.

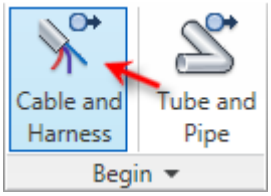
This completes the creation of an electrical part in AIP. The part can be saved and reused in other assemblies.



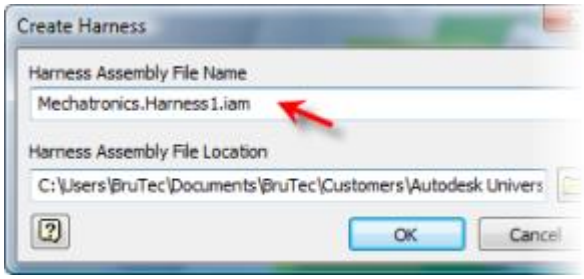
*Inventor-Create Harness and Import Wires

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

1. In the Model pane, double-click Mechatronics.iam.
2. On the Assemble tab, Begin panel, click Cable and Harness.



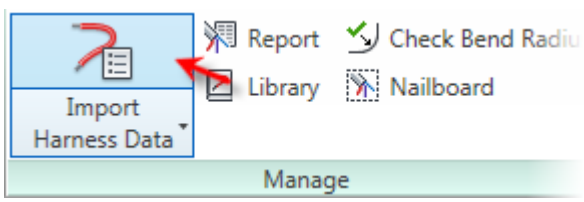
3. In Create Harness dialog, for New Harness Assembly File Name, enter **Mechatronics.Harness1.iam**.
4. Click OK.



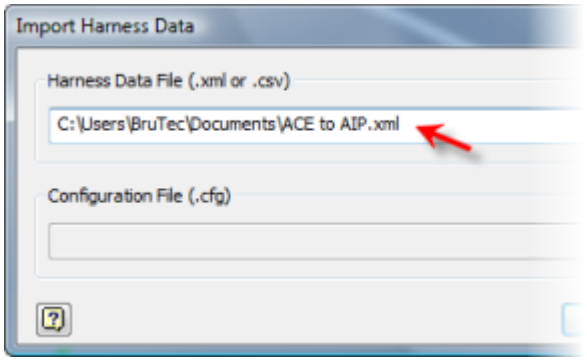
5. With Harness1 selected for editing, in the Browser, do the following:
- Right-click 360124:1 and select Harness Properties.
- In the Part Properties dialog box, for RefDes, enter **22X1**. Click OK.
- Right-click 360575:1 and select Harness Properties.
- In the Part Properties dialog box, for RefDes, enter **22X2**. Click OK.

Note: *22X1 and 22X2 now match the TAG values from the ACE components. The Connector component will be labeled in a later step.*

6. On the Cable and Harness tab, Manage panel, click Import Harness Data.



7. 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.
- Select *ACE to AIP.xml.* Click Open.
- Click OK.

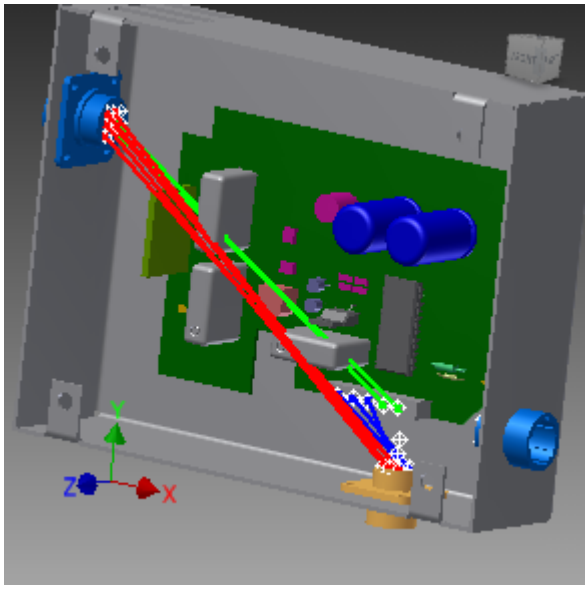


8. In the Imported Harness Data dialog box, expand the Electrical Parts category.

Note: *22X3 shows an error.*

9. Right-click 22X3, click Assign to An Existing Electrical Part.
10. In the drawing, select the connector you created. Click OK.
11. Wires category. Check for errors. Click OK.
12. In the Imported Harness Data dialog box, click Close.

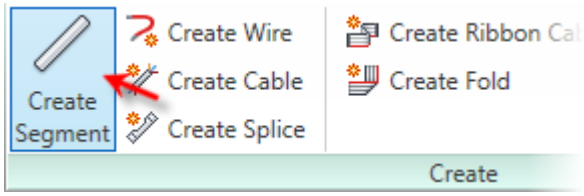
Wires are added to the assembly.



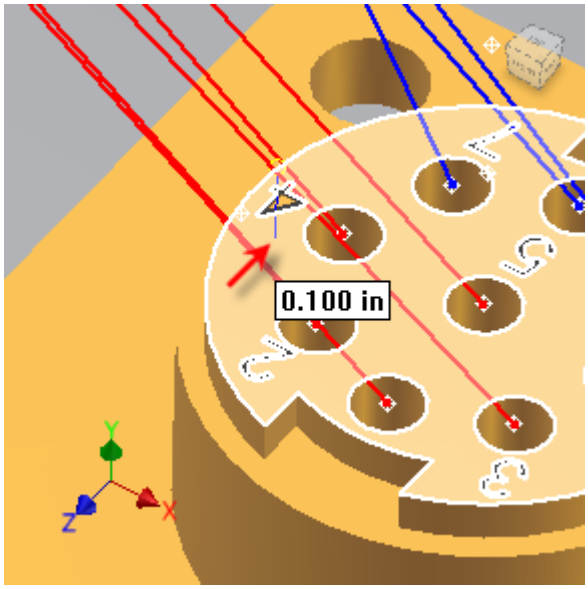
*Inventor-Create Segments

In this demonstration, you create segments. In AIP, 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.

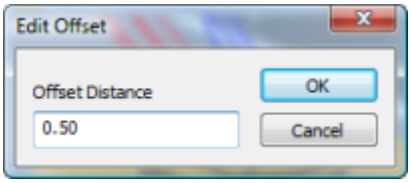


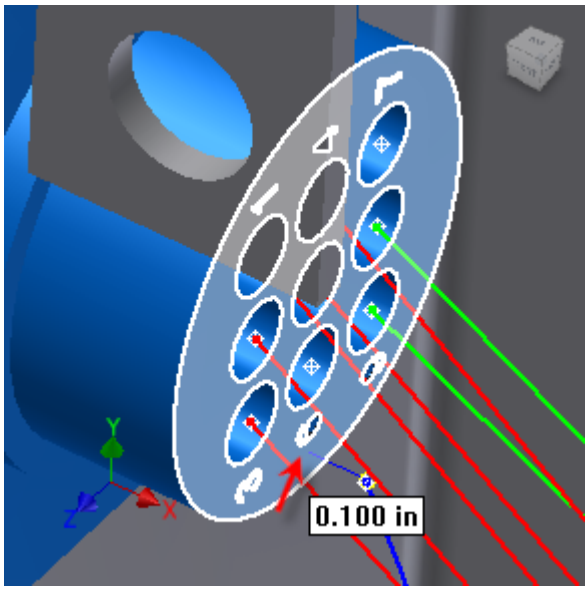
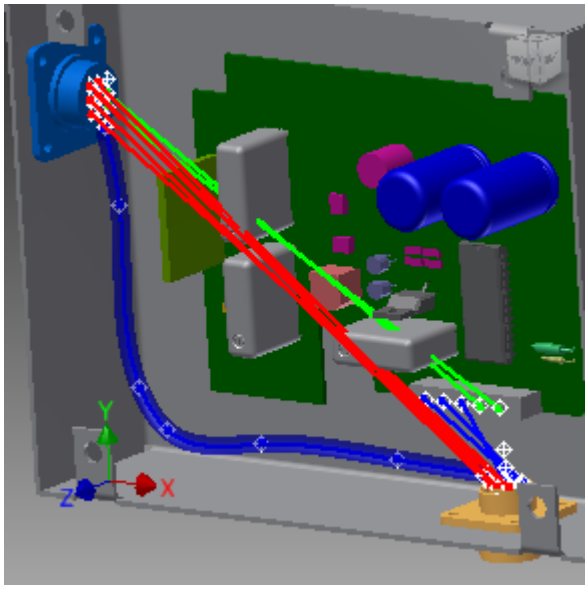
2. In drawing, select the top surface of the 22X1 connector.



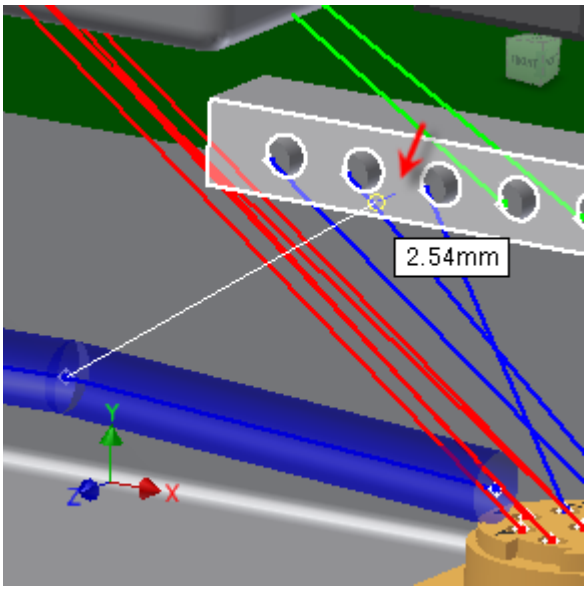
3. Right-click anywhere. Select Edit Offset.

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

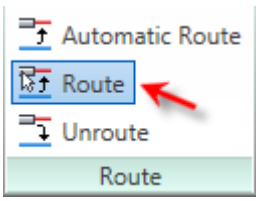


5. Insert workpoints for the segment from X1 to X2, selecting the sides of the enclosure for placement of the workpoints. **6.** For last point, change offset back to **2.54**. Select top surface of 22X2.	
7. Right-click and select Finish.	

8. On the Cable and Harness tab, Create panel, click Create Segment.
9. On the newly created segment, select first workpoint. (closest to 22X2)
10. Right-click anywhere. Select Edit Offset.
11. In the Edit Offset dialog box, for Offset, enter **2.54**. Click OK.
12. Select near the middle of the top surface of 22X3.
13. Right-click and select Finish.



14. On the Cable and Harness tab, Route panel, click Route.

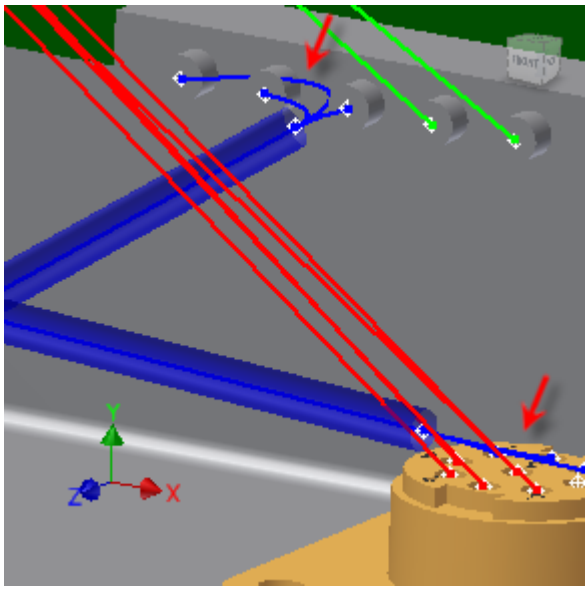


15. In the Route dialog box, select Wires.
16. In the drawing, select the blue wires.
17. In the Route dialog box, select First Segment. In the drawing select the segment nearest the 22X1 plug.

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

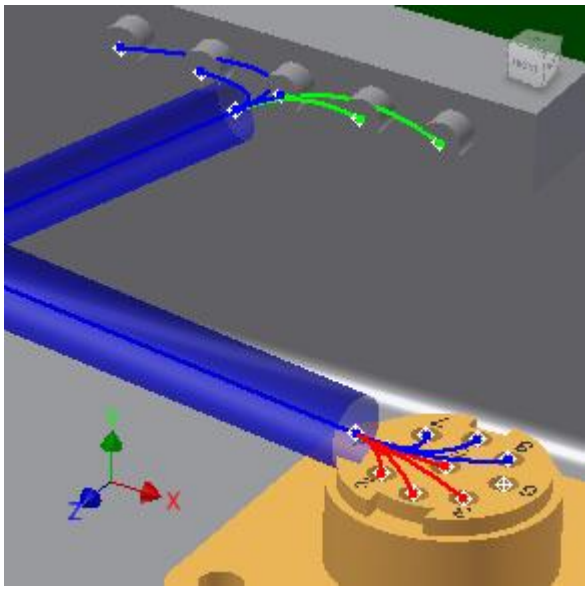
18. In the drawing, select the second segment nearest the 22X3 connector.
19. Click OK.

Notice how the wires are routed through the selected segments and the segments resize to match the wires.



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

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

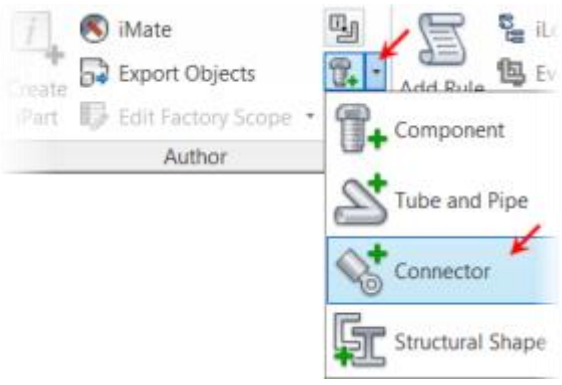


*Inventor-Author Connector Part

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

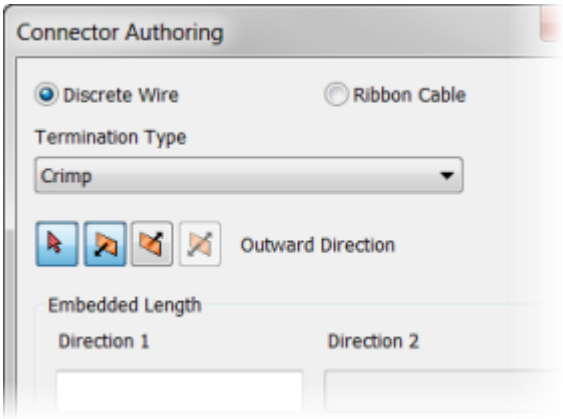
In this demonstration, you author the connector you created establishing the wire connection direction.

1. In the Browser, double-click PCB:1.
2. Double-click Connector to activate it for editing.
3. On the Manage tab, Author panel, on the Component flyout, click Connector.
4.



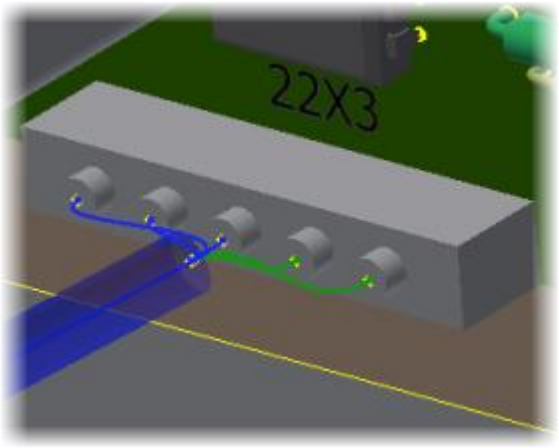
5. In the Connector Authoring dialog box, do the following:
- Select Discrete Wire
- For Termination Type, from the drop down list, select Crimp.

6. In the drawing, select the top face of the Connector part.
7. Verify that the arrow points out from the top face. If not flip its direction.
8. In the Connector Authoring dialog box, click OK.
9. In the Authoring Result dialog box, click OK.



10. Activate Harness1 for editing.

Notice how the flow of the wires is now perpendicular to the pin face.

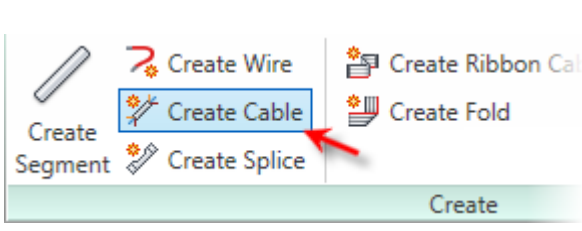


*Inventor-Create Cable

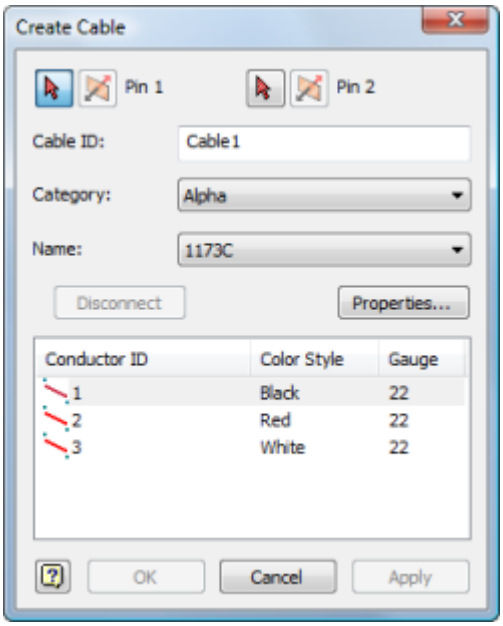
In ACE, only individual wires were created, and exported to the XML file.

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

11. In the drawing, zoom up to parts 22X1 and 22X3.
12. Right-click over one of the three blue wires. Click Delete.
13. Repeat step 2 for the other two blue wires.
14. On the Cable and Harness tab, Create panel, click Create Cable.



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

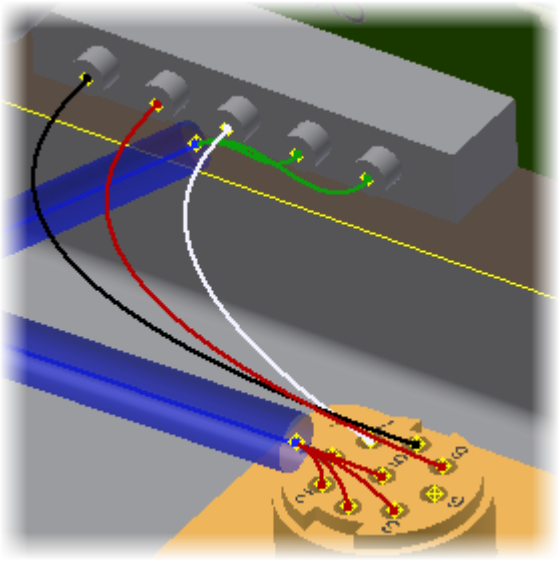


16. Select the Black color style, and in the drawing, do the following:
- For Pin 1, select 22X3:Pin 1.
- For Pin 2, select 22X1:Pin 9.

17. Select the Red color style, and in the drawing, do the following:
- For Pin 1, select 22X3:Pin 2.
- For Pin 2, select 22X1:Pin 8.

18. Select the White color style, and in the drawing, do the following:
- For Pin 1, select 22X3:Pin 3.
- For Pin 2, select 22X1:Pin 7.

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



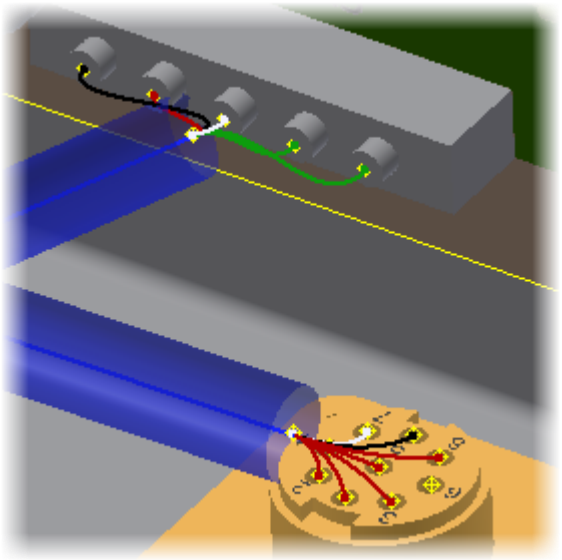
20. On the Cable and Harness tab, Route panel, click Route.
21. In the Route dialog box, select Wires.
22. In the drawing, select any one of the cable wires.

Notice how all cable wires are selected.

23. In the Route dialog box, select First Segment.
24. In the drawing select the segment nearest the 22X1 plug.

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

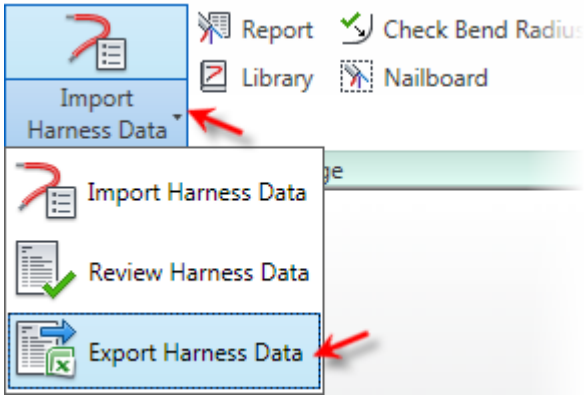
25. In the drawing, select the second segment nearest the 22X3 connector.
26. In the Route dialog box, click OK.



*Inventor-Export Report to ACE

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

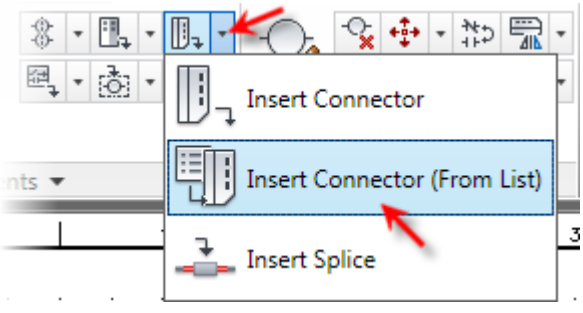
1. On the Cable and Harness tab, Manage panel, click Import Harness Data > Export Harness Data.



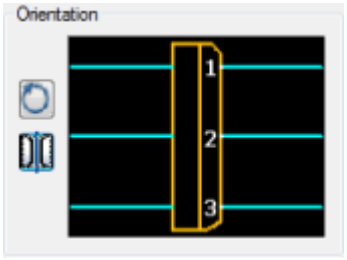
2. In the Export Harness Data dialog box, browse to the *My Documents* folder. **3.** For Name, enter **AIP to ACE.xml**. Click Save. **4.** In the Autodesk Inventor warning dialog box, click Yes. **5.** In the Cable & Harness dialog box, click OK. **6.** On the Quick Access toolbar, click Save. **7.**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

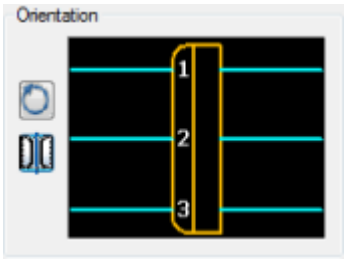
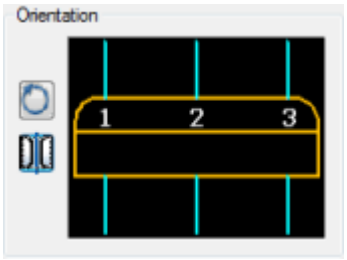
*ACE-Import Inventor Data and Insert Connectors and Wires

In this demonstration, 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 the Project Manager, click New Drawing. **3.** In the Create New Drawing dialog box, do the following: • For Name, enter **AIP-Connectors**. • For Template, click Browse and select *Acad_Electrical_IEC.dwt*. • Click Open. • Click OK. **4.** In the Apply Project Defaults to Drawing Settings dialog box, click Yes.	
5. On the Schematic tab, Insert Components panel, click Insert Connector > Insert Connector From List.	

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

Note: *The Place column is showing that these components already exist in the project.*
 	
7. In the Connector Selection dialog box, do the following:
• Select 22X1.
• For Pin Spacing, enter **10.**
• Click Rotate and Flip to orient connector with plug to the right as shown.
• If necessary, click Details>> to expand the dialog box.
• Under Size,
  • For all sizes, enter **6**.
  • For Radius, enter **3**.
• Click Insert.
8. In the drawing select an insertion point at **60.0,230.0.**
	

9. In the Connector Selection dialog box, do the following: • Select 22X2. • Click Rotate and Flip to orient connector with plug to the left as shown. • Click Insert. **10.** In the drawing select an insertion point at **320.0,230.0.**	
11. In the Connector Selection dialog box, do the following: • Select 22X3. • Click Rotate and Flip to orient connector with plug to the top as shown. • Click Insert. **12.** In the drawing select an insertion point at **180.0,75.0.**	

13. In the Connector Selection dialog box, click Wire It.
14. Click Cancel

15. On the Home tab, Layers panel, click Layer Properties.
16. In the Layer Properties Manager dialog box, change the colors of new layers:
- 14AWG-GRN to Green
- 16AWG-RED to RED.
- 22AWG-BLK to 253.
- 22AWG-RED to RED.

17. Close the Layer Property dialog box.

***Note**: Wire numbers are light blue because they are set as Fixed wire numbers.*

18. On the Quick Access toolbar, click Previous Project Drawing.

Compare the two drawings.

This completes the demonstration.