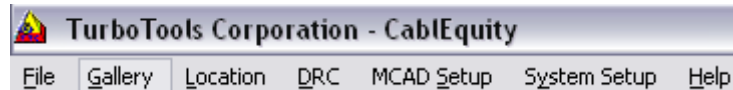


Design Flow Highlights

- [Components Selection](#)
- [turboConfigurator](#)
- [Output Documentation](#)
- [Special Functionality](#)
- [Library](#)
- [Control System Schematic Capture](#)
- [Transfer 3D Models](#)
- [Export and Import designs](#) (Important during updating CablEquity™)
- [Backup and Restore databases](#)
- [User DB Utility](#) (Export and Import Components during updating CablEquity™)
- [Functionalities](#)

Components Selection

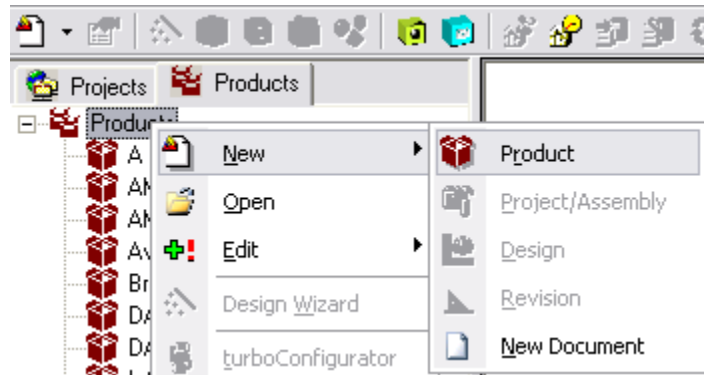
- A. Browsing database
1. Go to the Gallery on the menu bar





2. Select Category, from the pull-down menu by left mouse click

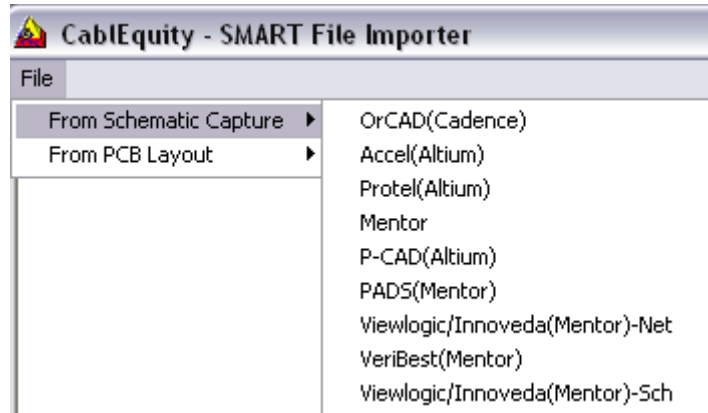


3. Select Type, by left mouse click on the Type picture on the main pane
 4. Select component part number, by left mouse click on the lower pane
 5. Select as many components as you want side-by-side
- B. Setup Project
1. Setup Product
 - a. Click on the Products tab, than right mouse click on the Products on the tree to open the Menu. Click on the New => Product on the menu to open the "New Product" dialog box.



- b. Enter all info on dialog box and click OK. New product will appear on the Products tree immediately.
 2. Setup Project
 - a. Click on the Projects tab, than right mouse click on the Projects on the tree to open the Menu. Click on the New => Project/Assembly on the menu to open the "New Project/Assembly" dialog box.
 - b. Enter all info on dialog box and assign new project to the previously created product, than click OK. New project will appear on the Projects tree immediately.
 3. Setup Design
 - a. Right mouse click on the new created project to open the Menu. Click on the New => Design on the menu to open the "New Design" dialog box.
 - b. Enter all info on dialog box than click OK. New design with automatically created Rev.1 will appear on the tree under the new project immediately.
- C. Design Wizard
 1. Right mouse click on the revision to open the Menu, and than Design Wizard
 2. First screen "Models" will open. Select number of Sources and Destinations required for the design by clicking  
 3. Click button "Next" to continue design wizard
 4. Second screen "Locations" will open
 - a. Left mouse click on the Source 1 location on the upper part of the screen, all controls on the lower part of the screen will appear
 - b. Selection connector for this location can be done manually, by the netfile, or by the MFG/IPN part number
 - i. Manual
 - 1st) Left mouse click on the desire location
 - 2nd) Click on the button "Manual"
 - 3rd) Select radio-button "Terminated"
 - 4th) From "Gender" pull-down select contacts gender
 - 5th) Select "Contact Qty"
 - 6th) Optionally, type-in Ref Des/Alias
 - 7th) Click button "Next" to continue design wizard
 - ii. Netfile
 - 1st) Left mouse click on the desire location
 - 2nd) Click on the button "Netfile"

- 3rd) On the "SMART File Importer" screen select netfile format from the CAD application where it was created and click OK

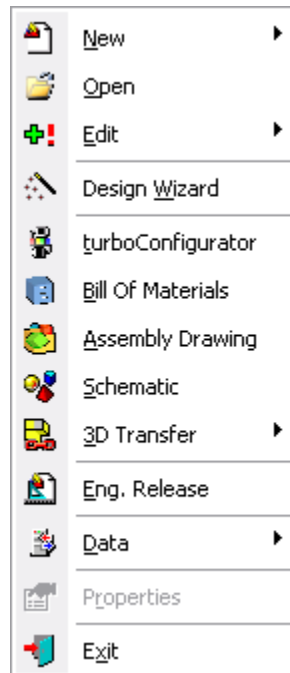


- 4th) Navigate to the location of the file on the proper drive and click Open
- 5th) Select Reference designator for the connector from the netfile and click OK
- 6th) All info from the netfile automatically loads onto "Locations" screen
- 7th) Click button "Next" to continue design wizard
- iii. Part#
- 1st) Left mouse click on the desire location
 - 2nd) Click on the button "Manual"
 - 3rd) Select radio-button "Terminated"
 - 4th) Left mouse click on the text box "MFG/IPN"
 - 5th) Type-in part number, for example 205841-2
 - 6th) Click button "Part#"
 - 7th) Text box "MFG/IPN" becomes gray-out; text boxes "Gender", "Ref Des/Alias" for Ref Des, and "Contact Qty" automatically filled-in with necessary data
 - 8th) Click button "Next" to continue design wizard
- c. After connector (termination, gender, and contacts quantity) selected, alias designator can be assign, than click Next
5. Third screen "Categories" will open
 - a. Click on the Connectors picture
 - b. Click button "Next" to continue design wizard
 6. Forth screen "Connectors" will open
 - a. Click on the Type connector picture
 - b. Click button "Next" to continue design wizard
 7. Fifth screen "Component Specifics" opens
 - a. Expend component's tree with the proper branch to the required level
 - b. Highlight desired node
 - c. Click button "Next" to continue design wizard
 8. Sixth screen "Component Part Number" opens
 - a. Make component selection by Manufacturer, Attributes or Part Number

- b. Select Quantity if required
 - c. Click button "Save" to continue design wizard
9. Seventh screen "Internal Part Number" opens
 - a. Component will be listed only it does not have IPN
 - b. To enter IPN, click on the listed component and type-in IPN in the pop-up text box. To finish entering IPN click outside pop-up text box
 - c. Click button "Next" to continue design wizard
 - d. If IPN is not available at this point, skip steps a, b, c and click button "Next" to continue design wizard
10. Eighth screen "Required Components" opens
 - a. By checking appropriate check box next to the required components, make selection of the component, using the same sequence
 - b. After all components selected (by designer decision), check box "Location Completed"
 - c. Click button "Continue" to continue design wizard
11. Second screen "Locations" will open
 - a. Continue components selection for all locations
 - b. The last location to be completed is Cable Material. After it is completed successful completion components selection message will appear
12. Next step in design process is turboCnfigurator

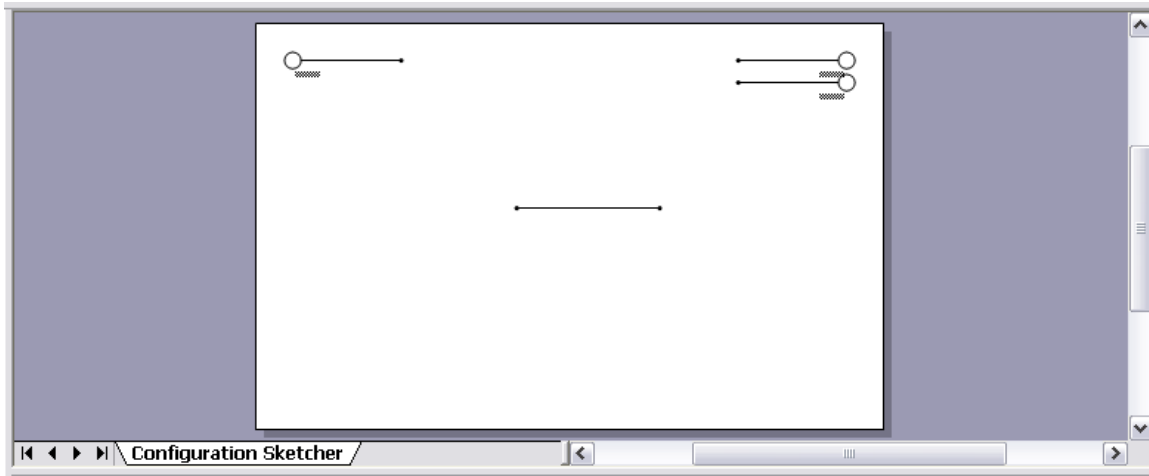
turboConfigurator

- A. turboConfigurator screen
 1. Right mouse click on the revision to open the Menu, and than turboConfigurator to open the screen

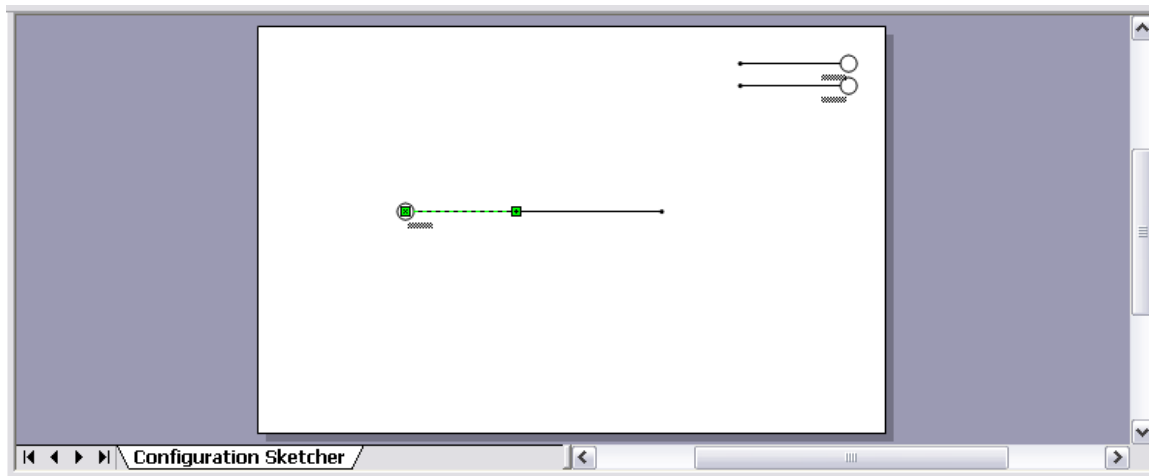


B. Sketcher

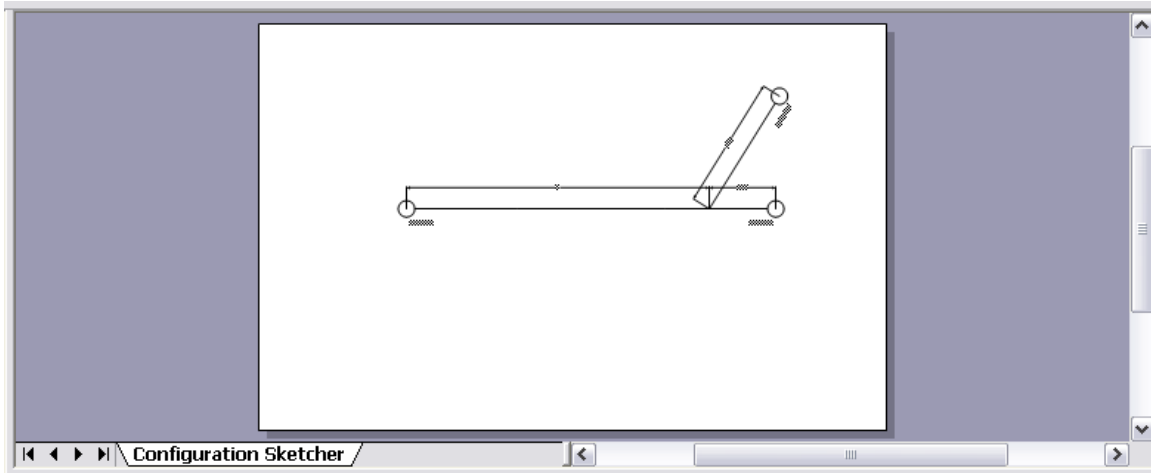
1. Following picture shows default sketcher mode for the 1Sx2D cable model



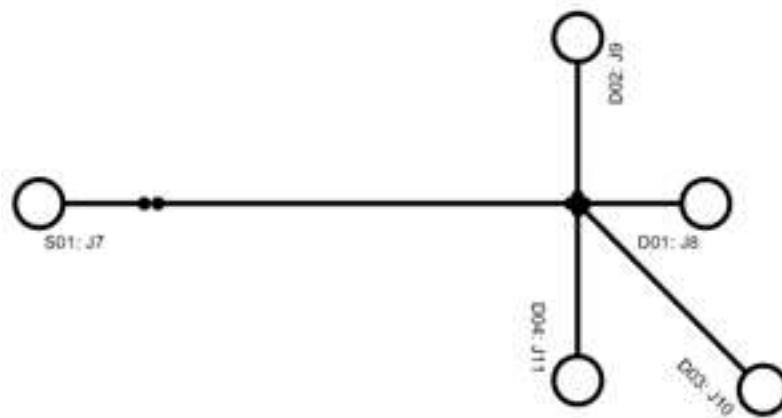
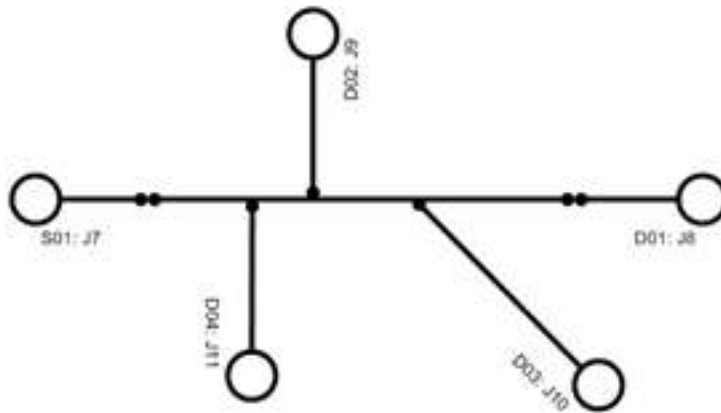
2. To make connection between source, destination and central segments do following:
 - a. Click left mouse on the S01 segment, segment will highlighted and green handles will appear. Position the pointer over the S01 segment. When the pointer changes to a four-headed arrow, hold down the mouse button and drag S01 segment to the central segment until they glued together
 - b. Do the same with other source and destination segments



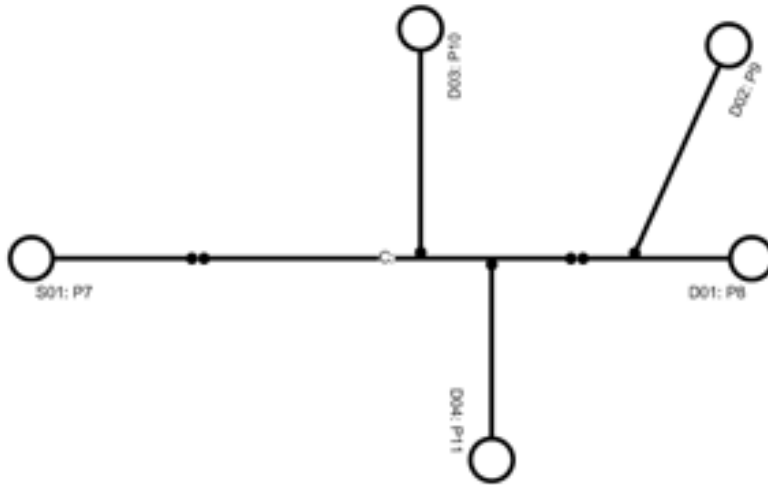
- c. After configuration of the cable assembly completed, click Set radio-button on Dimension panel



d. There are two pictures of the wrong placement of D2, D3 and D4

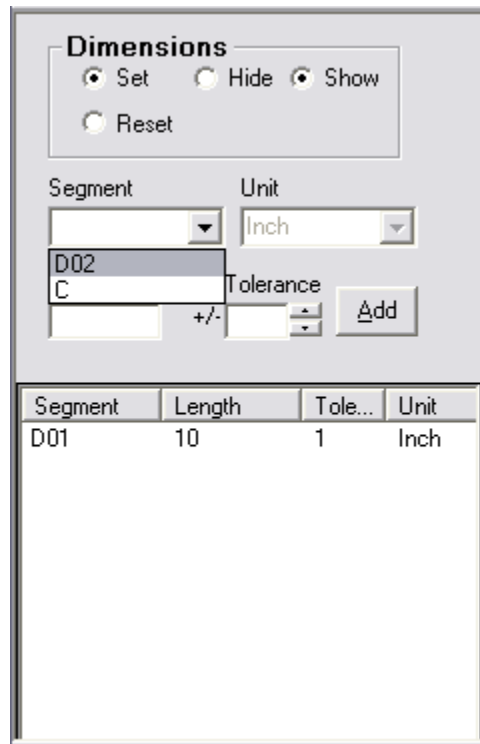


e. There is a picture of the right placement of D2, D3 and D4



C. Dimensions

1. From the pull-down Unit select proper dimension units

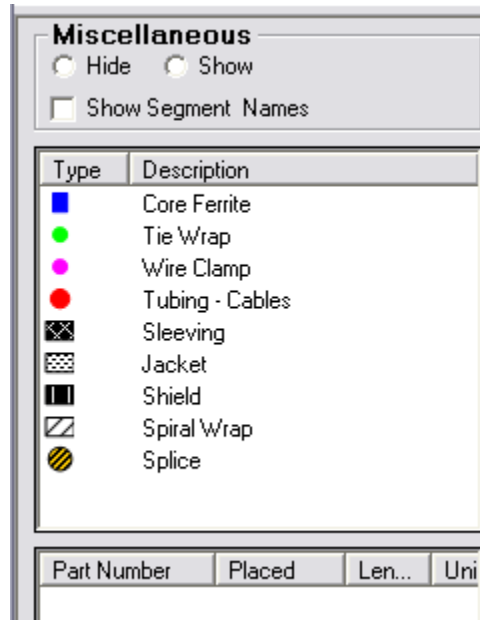


2. Select segment that needs to be assigned with dimension
3. Enter length and tolerance
4. Click button Add
5. Assign all segments with dimensions

6. When all segments are done, Proportional Layout drawing generates automatically on the Sketcher panel
7. To delete any previously entered segment's length, double-click on the corresponded segment on the Dimensions panel
8. To re-enter length for the segment, open Segment pull-down menu and enter length again

D. Miscellaneous

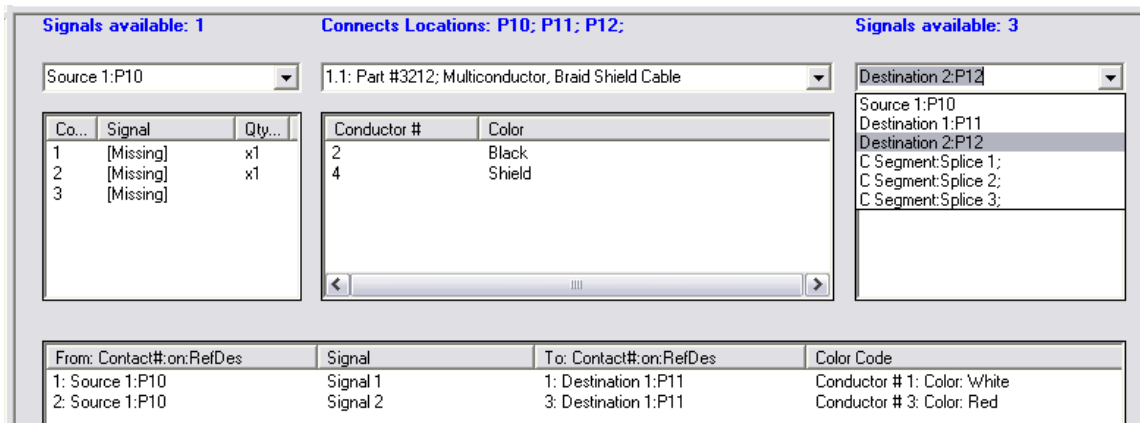
1. Miscellaneous for the cable/harness assembly can be selected from the miscellaneous legend



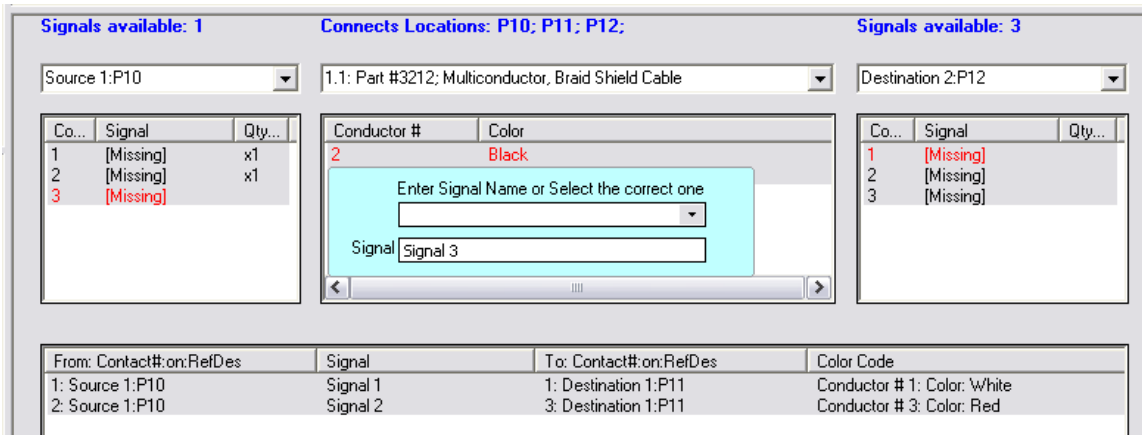
2. To select any miscellaneous, click on the desire type and select the segment where component should be installed
3. Enter all required info to proceed with selection

E. Continuity

1. Continuity is the functionality where all electrical connections between locations can be entered and stored. There is a view of the continuity panel



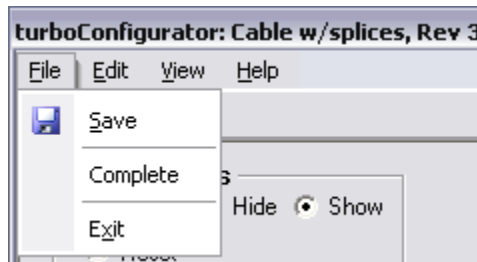
2. To enter electrical connection do following:
 - a. From the pull-down menu "Select Cable Material" select a cable
 - b. From the left pull-down menu select Source location
 - c. From the right pull-down menu select Destination location
 - d. Select contact on the source, conductor on the cable material, and contact on the destination – popup with signal name request will appear



- e. Type-in signal name, and click outside the popup box
- f. Make electrical connections between all necessary locations' contacts

F. Complete turboConfigurator

1. Go to File and select Save



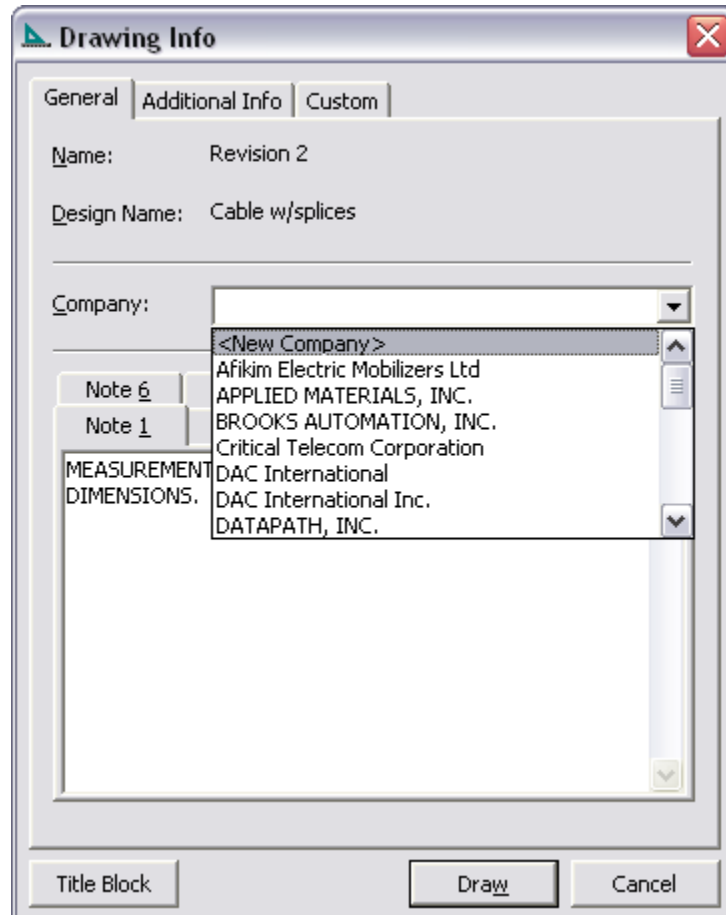
2. Then Complete
3. Close turboConfigurator
4. Those completes revision, and changes revision status to "Completed"
5. CablEquity is ready to generate output documentation

Output Documentation

A. Assembly Drawing

1. Right mouse click on the revision to open the Menu, and than ASSY Drawing to open the "Drawing Info" dialog box

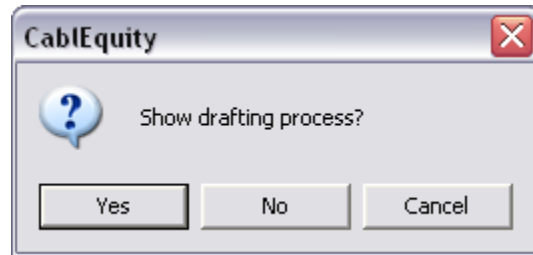
2. On "Drawing Info" dialog box company profile can be entered and than selected
 - a. From pull-down "Company" select company name (profile)
 - b. To create company profile, from pull-down Company, select <New Company>



- c. "Company Info" dialog box opens. Enter all appropriate info for the company profile

- d. If State is not a US state, you can type Country name, City name and State name in the "City" text box
- e. If zip code longer then 5 characters, type it in the "City" text box as well
- f. Before assigning company logo please do following
- i. Create company logo and save it in the BMP format, other file formats will not accepted
 - ii. Copy company logo file into "C:\Program Files\CablEquity\Templates" folder
- g. To assign company logo, click "Logo" button and navigate to the "C:\Program Files\CablEquity\Templates" folder
- h. Highlight company logo file and click OK
- i. If company logo is not selected, TurboTools logo will be assign automatically

- j. After all selections completed, click OK, to return to the "Drawing Info" dialog box
3. If needed enter additional info, use tabs "Additional Info" and "Custom"
4. Click "Title Block" button to select appropriate title block format
5. Click "Draw" button to start generating assembly drawings
6. Next popup message asks if you would like to see drafting process, or generate drawings silently



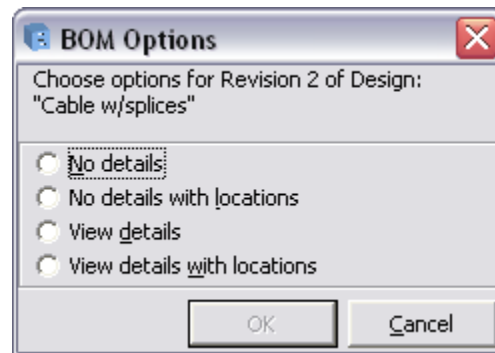
- a. To be able speed up drafting process for a big designs always use silent mode, by clicking "No" button

B. Schematic

1. Right mouse click on the revision to open the Menu, and than Schematic to generate schematic
2. Next popup message asks if you would like to see drafting process, or generate schematic silently

C. BOM

1. Right mouse click on the revision to open the Menu, and than Bill of Materials to open "BOM Options" popup



2. Select BOM option by clicking on the appropriate radio-button, then OK

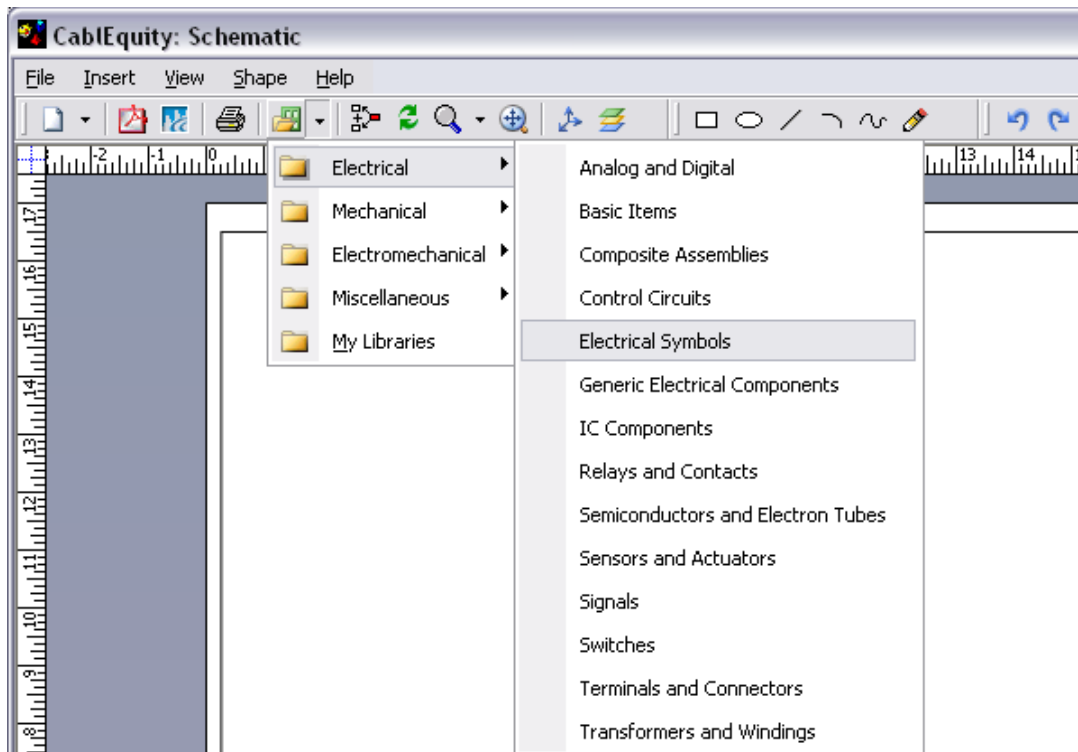
Special Functionality - Listing

- A. Changing Alias name
- B. Editing ASSY Drawing or Schematic
- C. Save ASSY Drawing or Schematic
- D. Conversion ASSY Drawing or Schematic
- E. Save BOM
- F. Exporting BOM

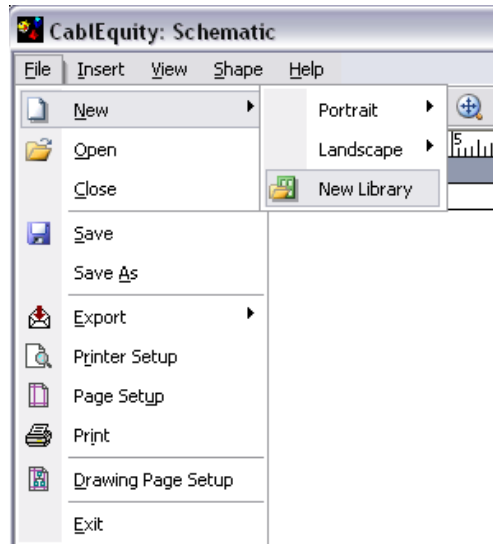
- G. DRC
- H. Location menu
- I. Edit menu
- J. Engineering Release menu
- K. Archive
- L. Search

Library

- A. Use existing Libraries
 - 1. CablEquity has extensive electrical, mechanical and other libraries



- 2. Every drawing or schematic can be edited using libraries' components
 - 3. To bring shape from the library, open the library and then D&D shape on the drawing
- B. Create a new Library
 - 1. New library can be created under My Libraries
 - 2. To create a new library go to "File => New => New Library"



3. D&D shape that should be stored in the library from the drawing window into the library window
4. Type-in a proper name for the new shape
5. Right mouse click on the "New Library" tab in library window, and select Save As
6. Type-in proper name for the new library
7. Navigate to the "C:\Program Files\CablEquity\Templates" folder and click OK

Control System Schematic Capture

- A. Create a new document
- B. Copy drawing from AutoCAD

Transfer 3D Models

- A. Integration with 3D MCAD
- B. Transfer models

Export and Import designs

1. Exporting existing Projects/Designs/Revisions
 - a. Run CablEquity
 - b. Highlight Project/Design/Revision by clicking left mouse
 - i. Make sure to "Export" all new created components used in exporting designs in order to import them together with designs into the new database
 - c. Go to the "Export" button on the toolbar and left mouse click on it
 - d. Follow the instructions to export selected design into the XML file
 - e. Remember the folder where you exported selected design

2. Importing Projects/Designs/Revisions
 - a. Run CablEquity
 - b. Highlight the highest node on the Project Manager tree "Projects" in order to import entire product or top level project. For the sub-project, design or revision, please select proper level on the Project Manager tree
 - c. Go to the "Import" button on the toolbar and left mouse click on it
 - d. Navigate to the folder where all exported/received designs are stored in XML file format
 - e. Click on the XML file and follow the instructions to Import it into the database
 - i. Make sure to "Import" all new created components used in importing designs in order to use designs properly

Backup and Restore databases

Important! Restore any database is possible only after all databases are backup

Backup procedure:

1. Run CablEquity from the desktop. Before doing anything else, follow the steps for the Backup procedure
2. Click on the "Backup Database" icon on the toolbar
3. Click on the "Projects/Designs" radio button
4. Click the "Backup to" button, and in the "Select the destination file to back up" dialog box click "Save". Default directory is
C:\TurboTools\CablEquity\Data Files\Backup\
5. When backup of "Projects" is completed, do the same for "Components"
6. Click on the "Components" radio button
7. Click the "Backup to" button, and in the "Select the destination file to back up" dialog box click "Save". Default directory is
C:\TurboTools\CablEquity\Data Files\Backup\
8. When backup of "Components" is completed, do the same for "Documents"
9. Click on the "Documents" radio button
10. Click the "Backup to" button, and in the "Select the destination file to back up" dialog box click "Save". Default directory is
C:\TurboTools\CablEquity\Data Files\Backup\
11. When backup of "Documents" is completed, click the "Cancel" button
12. "3D Components" is available only if "Integration with 3D MCAD" package was installed
13. Close CablEquity

Restore procedure:

1. In "C:\TurboTools\CablEquity\Data Files\Backup" directory manually rename "Components.bak" to "Components-install.bak", "Projects.bak" to "Projects-install.bak", and "Documents.bak" to "Documents-install.bak"
2. Copy or uncompress "Projects.zip", and/or "Components.zip", and/or "Documents.zip" into "C:\TurboTools\CablEquity\Data Files\Backup" directory

3. Run CablEquity from the desktop. Before doing anything else, execute entire Backup procedure
4. Click on the "Restore Database" icon on the toolbar
5. Click "OK" to restore
6. Click on the "Components" radio button
7. Click the "Restore from" button, and in the "Select the source back up file to restore" dialog box double-click on "Components.bak"
8. In the "Restore Database" dialog box click "OK"
9. Click on the "Projects/Designs" radio button
10. Click the "Restore from" button, and in the "Select the source back up file to restore" dialog box double-click on "Projects.bak"
11. In the "Restore Database" dialog box click "OK"
12. Click on the "Documents" radio button
13. Click the "Restore from" button, and in the "Select the source back up file to restore" dialog box double-click on "Documents.bak"
14. In the "Restore Database" dialog box click "OK"
15. "3D Components" is available only if "Integration with 3D MCAD" package was installed
16. After the restore is completed click "Cancel", CablEquity will be closed automatically

User DB Utility

General – Attributes

1. If Attribute has a definitive value (not a range), this value should be in the "FROM" box. "TO" box keep empty.
2. Value with decimal point should be in the following format "0.15" (".15" is a wrong format).
3. If Attributes have value "No", do not enter it, unless it is absolutely necessary for the search or DRC (Design Rule Checking).
4. For wire size in AWG, box "FROM" should have lower physical value (higher numerical number) and box "TO" should have higher physical value (lower numerical number).

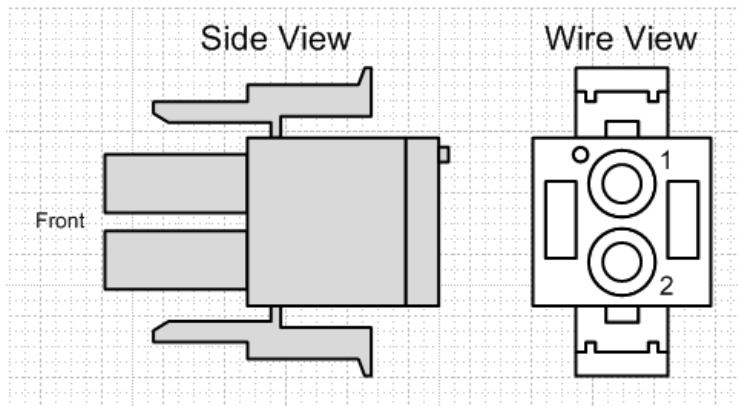
General – Shapes

1. Use "Size & Position", and all necessary tools from the toolbar
2. Use all necessary tools from the "Shape" menu
3. All shapes should be designed in the 1:1 scale
4. All shapes should be designed in the left-top view
5. All sub-shapes must be highlighted and Grouped into one shape (go to Shape, Grouping, Group)

Create NEW Connector in database

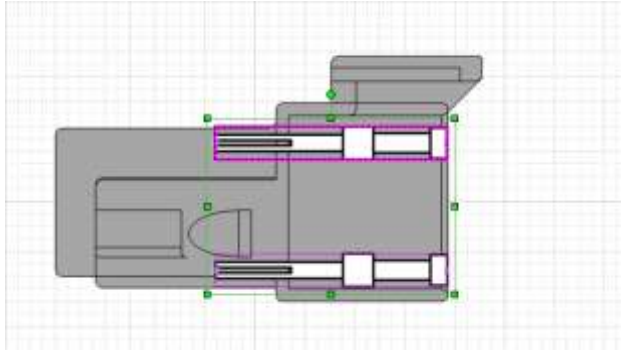
1. Open User DB Utility. "Connector types" is the default view.
2. Select the desired connector type by clicking on the appropriate picture.

3. On the pop-up screen select MFG and click on either "Edit/Create from Existing" or "Create from Scratch" button.
4. Enter or edit an attribute's information by clicking on it. Follow the instructions on the subsequent pop-up screens.
5. For proper compatibility of newly added connectors with newly added contacts, the following rules must apply:
 - a. The connector's attribute "Class" value must match the contact's attribute "Connector Specific" value exactly
 - b. The connector's attribute "Contact Size Accept" value must match the contact's attribute "Contact Size" value exactly
 - c. The connector's attribute "Related Type 1" value must match the contact's attribute "Related Type 1" value exactly
6. On "Step 5" design and orient connector shape is similar to the connector in this picture. Please notice:
 - a. All connectors should be designed on a 1:1 scale
 - b. All connectors from one FAMILY should have the same width, as pictured in the "side view"
 - c. All connectors should be designed and oriented so that the front edge is on the left, as pictured in the "side view"

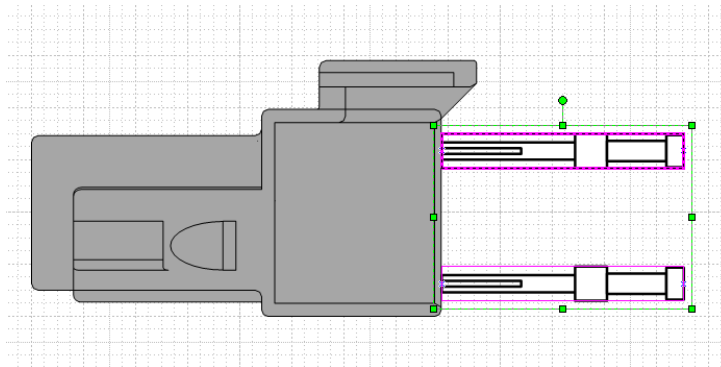


7. After the connector is created and saved, click on the "Linkage" (used to be Drawing) tab.
8. The "Part Number" text box should display the part number of the newly created connector, if it is not displayed, manually type it in. Click on "Fetch" button to open the side view for the connector.
9. Type in the part number of the contact that is associated with this connector and click "Fetch." If the contact has not been designed yet, steps 7 to 13 must be completed when the contact is available
 - a. For a connector with INTEGRATED contacts (one that does not require contacts at all), please use any contact part number, for example 794606-1, to be able to create connection points
10. A message displaying: "Please select contacts on upper and lower boundaries, and align left ends with the proper horizontal position" will appear. Click OK.

- 11.2 contacts must be placed in their proper horizontal and vertical positions with an acceptable distance between them. Please use your mouse and keyboard to highlight contacts (click on a contact to highlight it; press and hold the "Ctrl" key and click on each component to highlight multiple components). Use your keyboard's arrow keys for precise placement, then click on the "Save" button.
- This picture shows the alignment of contacts on a connector with INSERTABLE contacts



- This picture shows the alignment of contacts on a connector with INTEGRATED contacts

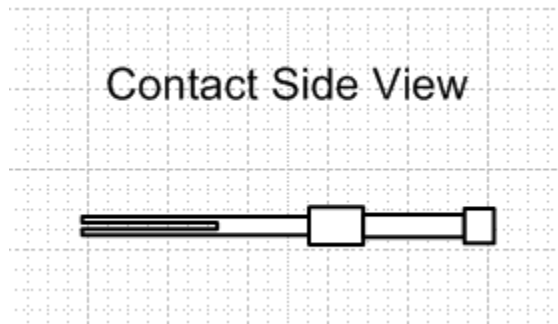


- A message displaying: "Connection points have been added on Connector for Contacts. Changes are saved." will appear. Click OK.
- A message displaying: "You can verify created connection points by activating 'Connection Points' menu under the 'View' menu. If any changes in creating or placement connection points are needed you have to do this process again." will appear. Click OK.

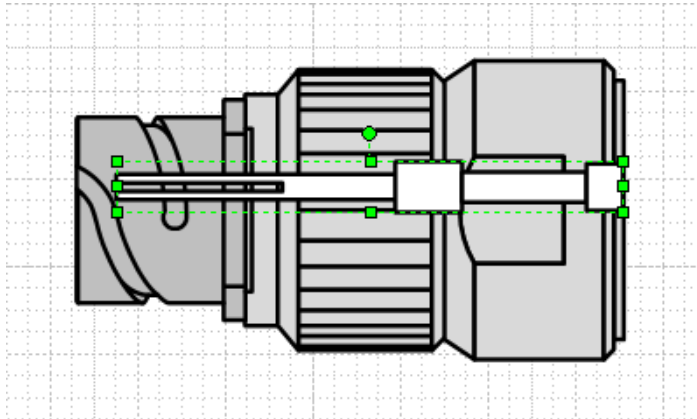
Create NEW Contact in database

- Open User DB Utility. "Connector types" is the default view.
- Select the "Contacts" category, and then select the desired contact type by clicking on the appropriate picture.

3. On the pop-up screen select MFG and click on either "Edit/Create from Existing" or "Create from Scratch" button.
4. Enter or edit an attribute's information by clicking on it. Follow the instructions on the subsequent pop-up screens.
5. For proper compatibility of newly added connectors with newly added contacts, the following rules must apply:
 - a. The connector's attribute "Class" value must match the contact's attribute "Connector Specific" value exactly
 - a. The connector's attribute "Contact Size Accept" value must match the contact's attribute "Contact Size" value exactly
 - b. The connector's attribute "Related Type 1" value must match the contact's attribute "Related Type 1" value exactly
6. On "Step 5":
 - a. From the toolbar, select Libraries => Miscellaneous => Tooling, to open the Tooling library.
 - b. Select "Pin" or "Socket" and drag and drop onto the page.
 - c. If the new contact requires a special shape, it must be designed and oriented similar to the contact in this picture:



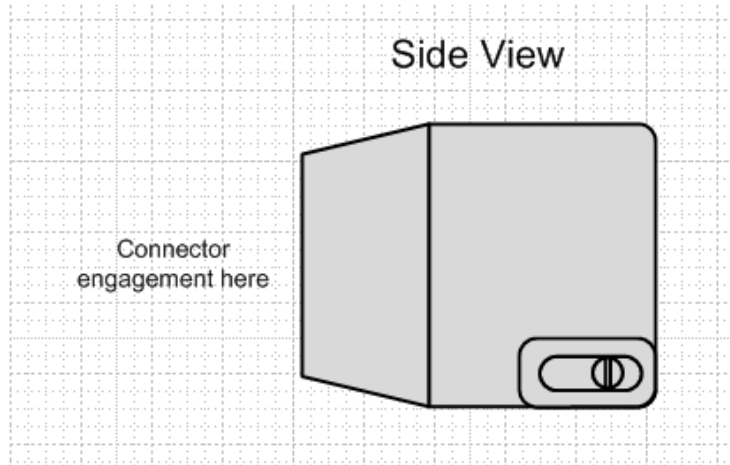
- d. After the shape of the contact is set, it must be sized proportionally to the FAMILY of connectors with which it is meant to work. To do so select Shape => Insert Reference from the toolbar. Enter the connector part number in the pop-up text box representing the FAMILY of connectors that the newly created contact matches. Click OK.
- e. Click on the contact shape. From the toolbar, select Shape => Order => Bring to Front, (the same commands are available from the right mouse click menu) then position the contact shape over the connector shape. Align the right edge of the contact with right edge of the connector.
- f. Re-size the contact to the proper size.
- g. Click the Save button.



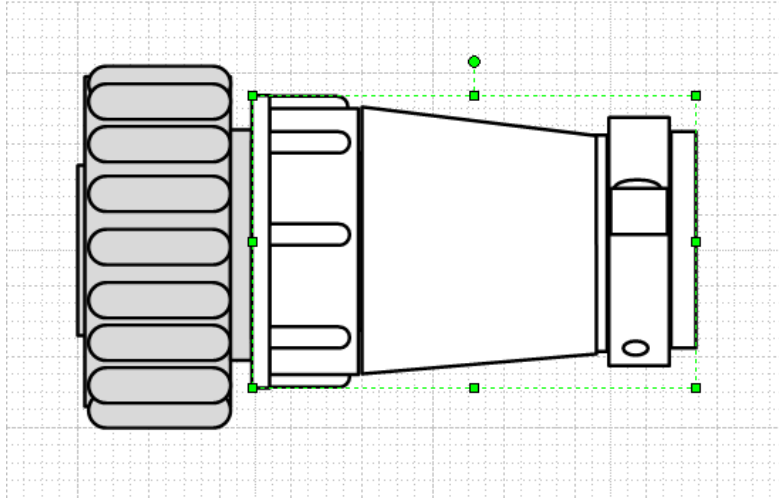
Create NEW Backshell in database

1. Open User DB Utility. "Connector types" is the default view.
2. Select the "Backshells" category, and then select the desired backshell type by clicking on the appropriate picture.
3. On the pop-up screen select MFG and click on either "Edit/Create from Existing" or "Create from Scratch" button.
4. Enter or edit an attribute's information by clicking on it. Follow the instructions on the subsequent pop-up screens.
5. For compatibility of newly added backshells with newly added connectors, the following rules must apply:
 - a. The connector's attribute "Class" value must match the backshell's attribute "Connector Specific" value exactly
 - b. The connector's attribute "Contact Count" value must match the backshell's attribute "Housing Contact Count" value exactly
 - c. The connector's attribute "Contacts Layout - # of Positions per Row" value must match the backshell's attribute "Housing Layout - # of Positions per Row" value exactly
 - d. The connector's attribute "Contacts Layout - # Rows" value must match the backshell's attribute "Housing Layout - # Rows" value exactly
 - e. The connector's attribute "Housing Shell Size" value must match the backshell's attribute "Housing Shell Size" value exactly
 - f. Steps 'c' and 'd' together are interchangeable with step 'e' or vice versa
 - g. The connector's attribute "Related Type 1" value must match the backshell's attribute "Related Type 1" value exactly
6. On "Step 5" design and orient backshell shape is similar to the backshell in this picture. Please notice:
 - a. All backshells should be designed on a 1:1 scale

- b. All backshells from one FAMILY should have identical dimensions, as pictured in the "side view"
- c. All backshells should be designed and oriented so that the connector engagement edge is on the left, as pictured in the "side view"



7. On "Step 5":
 - a. After the shape of the backshell is set, it must be sized proportionally to the connector with which it will work. To do so select Shape => Insert Reference from the toolbar. Enter the connector part number in the pop-up text box that the newly created backshell will match to. Click OK.
 - b. Click on the backshell shape. From the toolbar, select Shape => Order => Bring to Front, (the same commands are available from the right mouse click menu) then position the backshell shape to overlap with the connector shape.
 - c. Re-size the backshell to the proper size. Align the backshell with the connector to the proper engagement position.
 - d. Click on the "Save" button.
8. After the backshell is created and saved, click on the "Linkage" (used to be Drawing) tab.
 - a. The "Part Number" text box should display the part number of the newly created backshell. If it is not displayed, manually type it in; click on the "Fetch" button to open the side view of the backshell.
 - b. Type in the part number of the connector that is associated with this backshell and click "Fetch." If the connector has not been designed yet, step 7 must be completed when the connector is available.
 - c. Click on the backshell shape. From the toolbar, select Shape => Order => Bring to Front, (the same commands are available from the right mouse click menu) then position the backshell shape to overlap the connector shape.
 - d. Align the connector and backshell to the proper engagement position. Click on the "Save" button.



Import and Export components

1. Import
 - a. Go to File=>Import, left mouse click on Import to open dialog box "Open"
 - b. Navigate to the directory where you have component's XML file, highlight it and click button "Open"
 - c. Click OK on confirmation message
2. Export
 - a. Go to File=>Export, left mouse click on Export to open screen "Export Components"
 - b. Select Category, Type and Manufacturer
 - c. Only components created by designer, or previously imported into database will be shown
 - d. Check all check boxes for components that have to be exported
 - e. Click on "Export" button to open dialog box "Save As"
 - f. Navigate to the folder where you want to export XML file, type-in file name and click on "Save" button
 - g. Click OK on confirmation message

Functionalities - Listing

- A. Loading IPN
- B. System Setup
- C. MCAD Setup
- D. Barcode
- E. Netlist options
- F. Wires connectivity Import/Export