

NX Routing

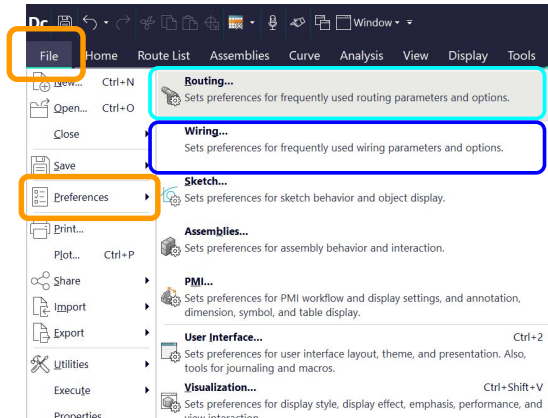
with Artifact

Application Settings

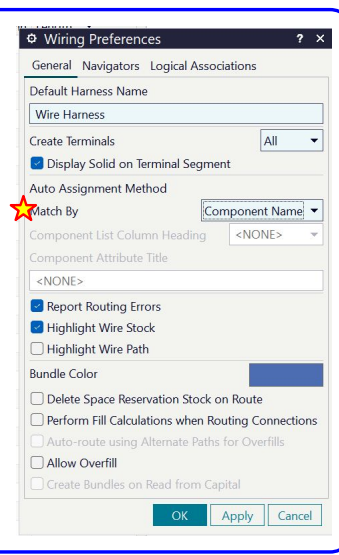
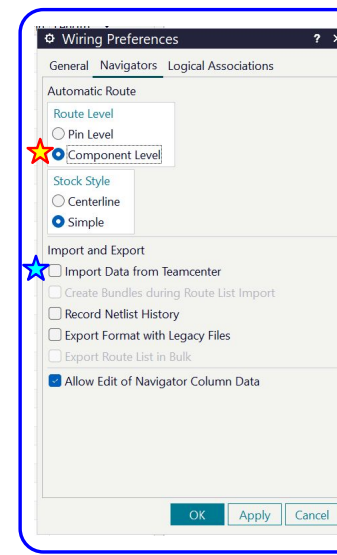
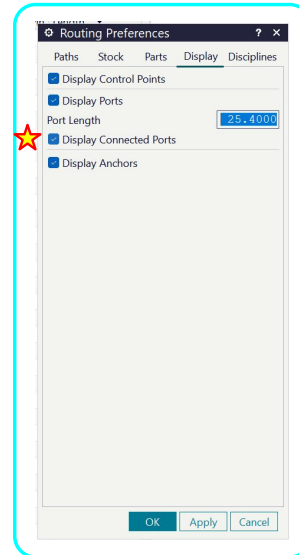
Basic Usability Settings

NX's default settings are not conducive to easy use. There are a few recommended settings to change to improve overall behavior within the Routing Electrical application.

1. **File** → **Preferences** → **Routing**
2. **File** → **Preferences** → **Wiring**



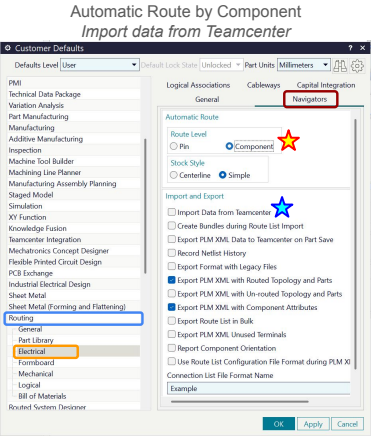
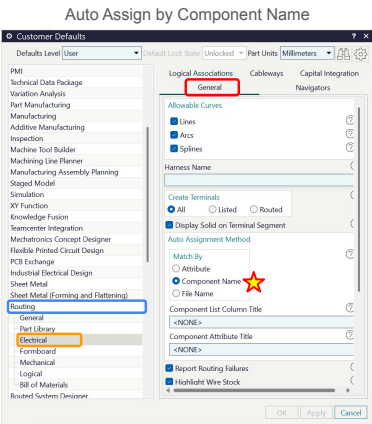
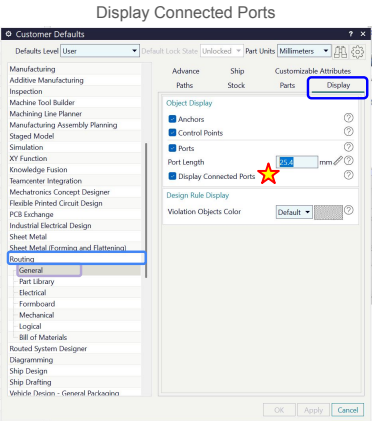
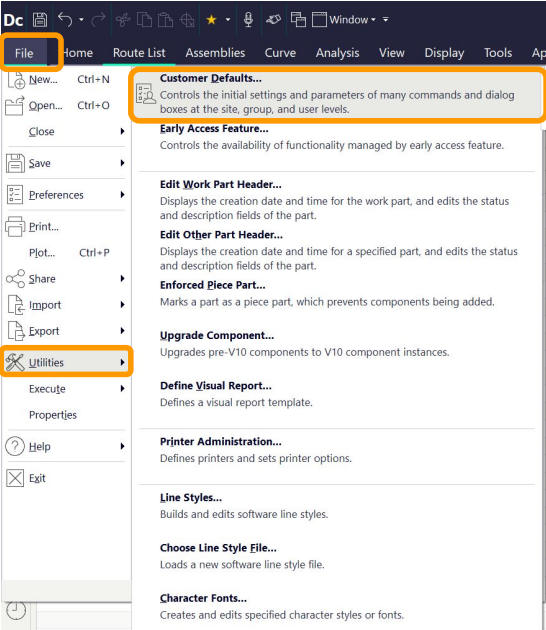
1. **Routing Preferences** → **Display Connected Ports**
When this is disabled, ports are hidden after the first routing connection is made. Routing does not behave reliably when routing to hidden ports.
2. **Wiring Preferences** → **Automatic Route by Component Level**
Pin Level routing requires that the user explicitly defines all of the pins while qualifying a connector, but does not meaningfully impact the design in most cases.
3. **Wiring Preferences** → **Auto Assignment Match by Component Name**
Component Name is an instance-level attribute in NX that is used for Reference Designators in standard practice. This is the most unambiguous way to associate parts in NX with routing parts from any application.
4. **Wiring Preferences** → **Import Data from Teamcenter**
Turn this ON if you're controlling your PLMXML files through teamcenter. Otherwise, turn this OFF.



Setting persistent settings

NX's *Preferences* settings only persist until you exit NX. When you reopen NX, preferences are *always* set to the **Customer Defaults**. Changing these Customer Defaults will make NX persist settings through a restart.

(There are a lot of useful settings in here; Customer Defaults is a good thing to explore in NX if you are a regular user.)

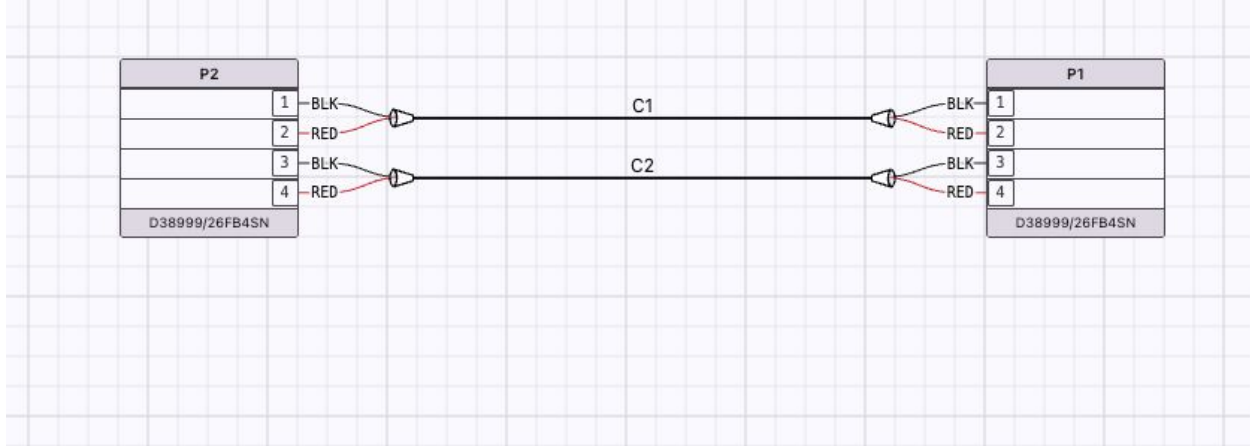


Exporting NX-ready Harness Data

Harness Setup (Diagram)

Before exporting a harness to NX, it must meet the following prereqs:

- All cables need reference names
- All discrete wires need reference names
- All connectors need reference names
- All cables need an outer diameter
- All discrete wires need a gauge, or an outer diameter assigned



Item Reference Names

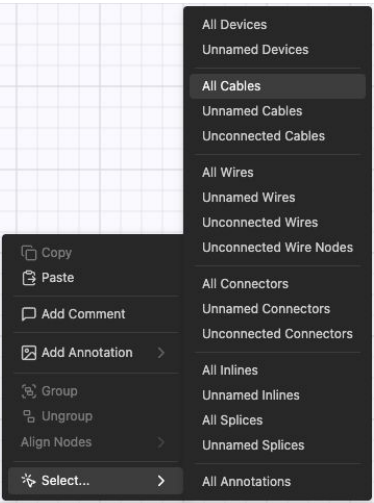
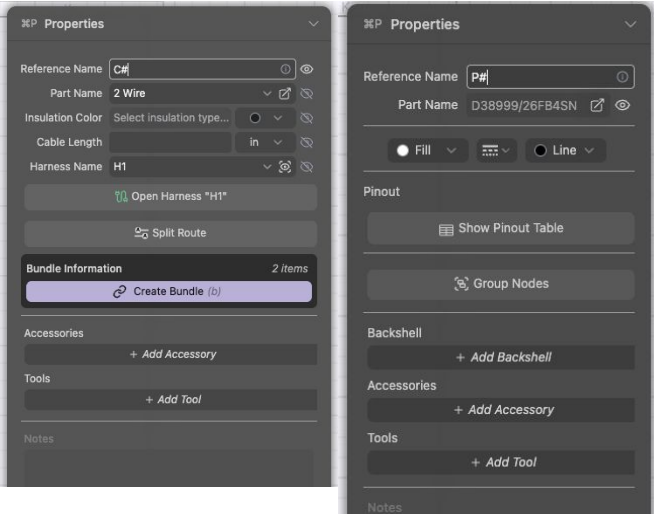
Reference Names

Cable reference names usually follow simple indexes (i.e. C1, C2, C3...). You can add an auto-index to any reference name in Artifact by adding a # to the name, and you can do this to an arbitrary number of items at once, if all items are of the same type.

A convenient way to batch name all of the cables in a diagram is to right click -> Select... -> Select All Cables, and then set the reference name to "C#".

To set reference names on connectors, select the connector and enter the desired reference name. (Reference Naming rules are

Wires inside cables do not need reference names manually assigned. During an export, if a wire has not had a reference name assigned, it will inherit the reference name of the cable (i.e. "C8"), and use its color (i.e. blue, "BLU") to construct a reference name ("C8.BLU")

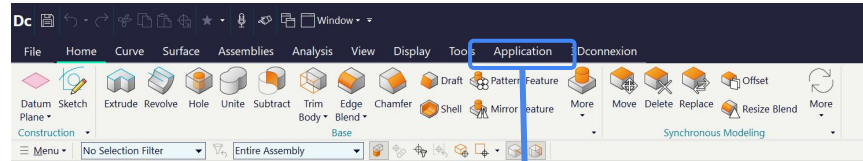


Launching Routing Electrical and Importing Route Lists

Launching the *Routing Electrical* Application

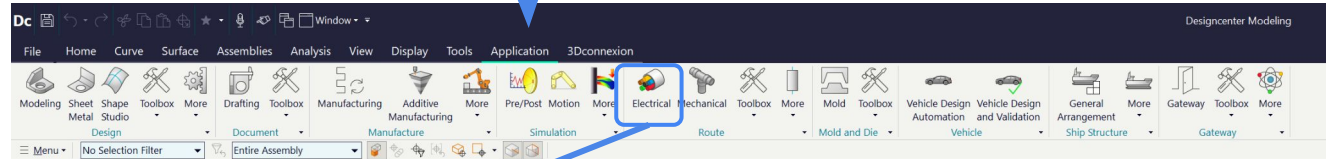
1- Initial Launch

Typically, NX will default to the modeling application on launch, and the Ribbon + Toolbar look like this.



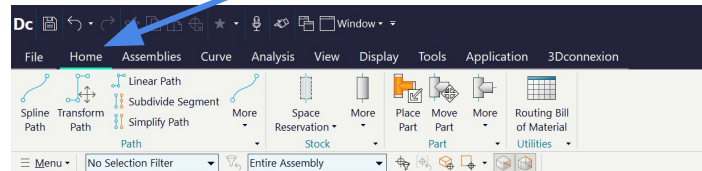
2- Application Picker

Select the “Application” tab on the ribbon to change applications (tool availability) for different technical practices.



3- Basic Electrical Routing

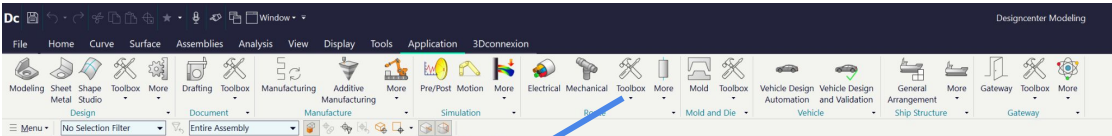
The basic electrical routing application has path, stock, and part management tools in the Home section. It cannot manage Route Lists until you activate the Route List toolbar.



Activating the *Route List* Toolbar

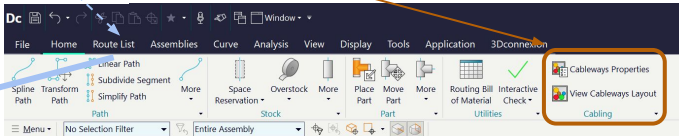
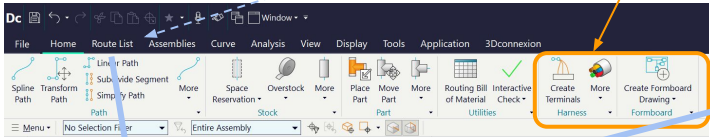
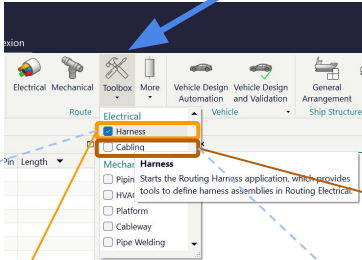
4- Routing Toolbox

In the Applications tab, in the Routing section, Open the Routing Toolbox to access the additional tools



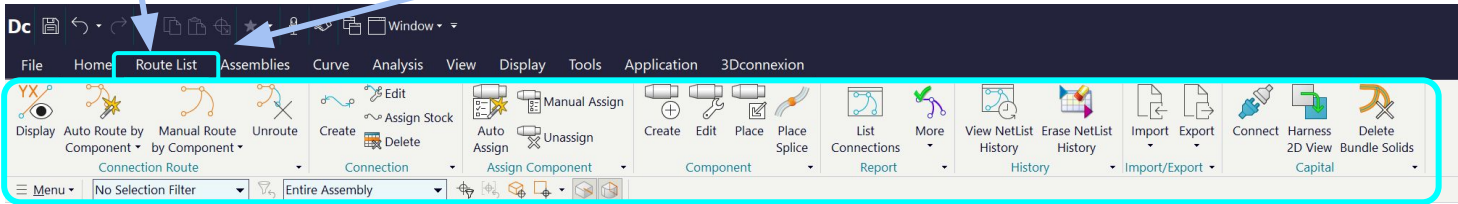
5- Activate additional tools

Activate either the harness or cabling (or both) based on your needs and license availability.
The Harness toolset provides formboard tools in your Home toolbar, while the Cabling toolset provides cableways tools.
If you only have license capacity for one, the Harness tool is usually what you want.



6- Route List

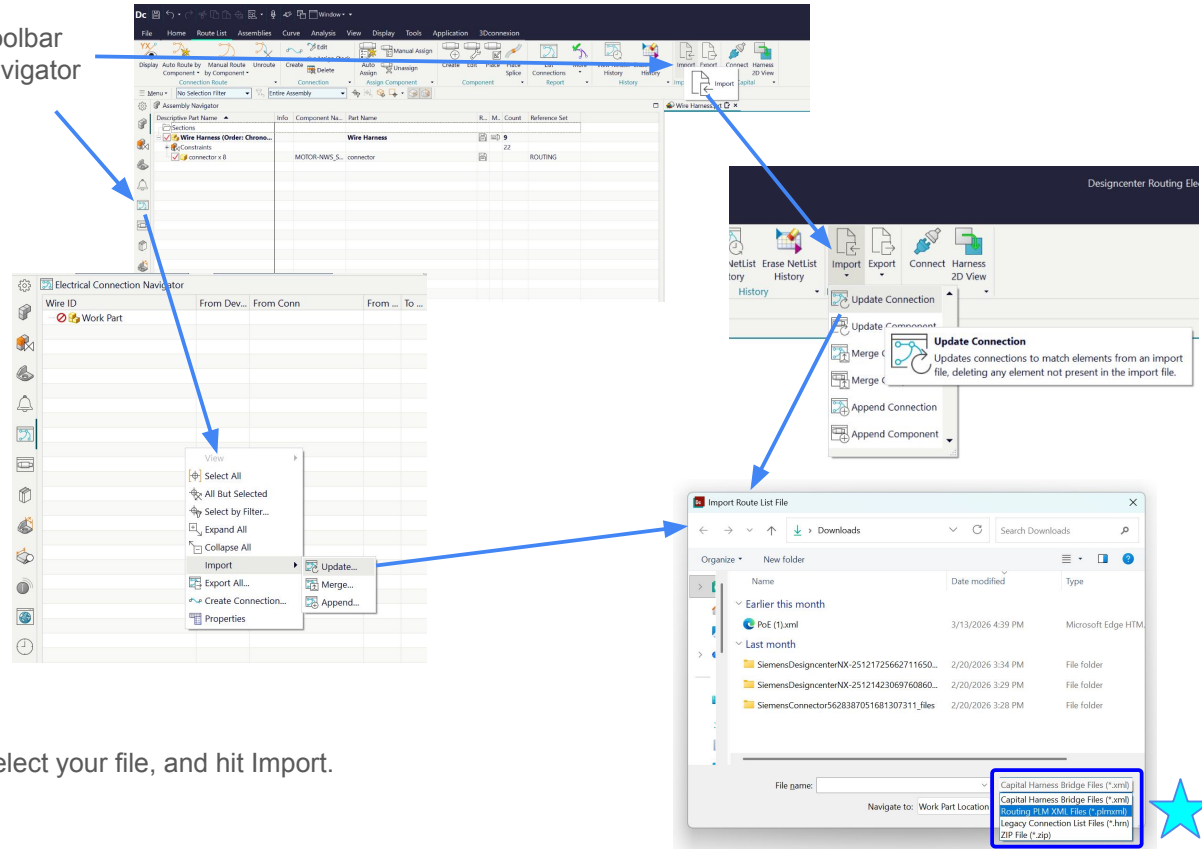
Either option (or both) provides the Route List toolbar, which provides the capability to manage routing parts and connections.



Importing a Route List

1. Import the route list in one of two ways:

- Click the Import button on the Route List Toolbar
- Right Click on the Electrical Connection Navigator



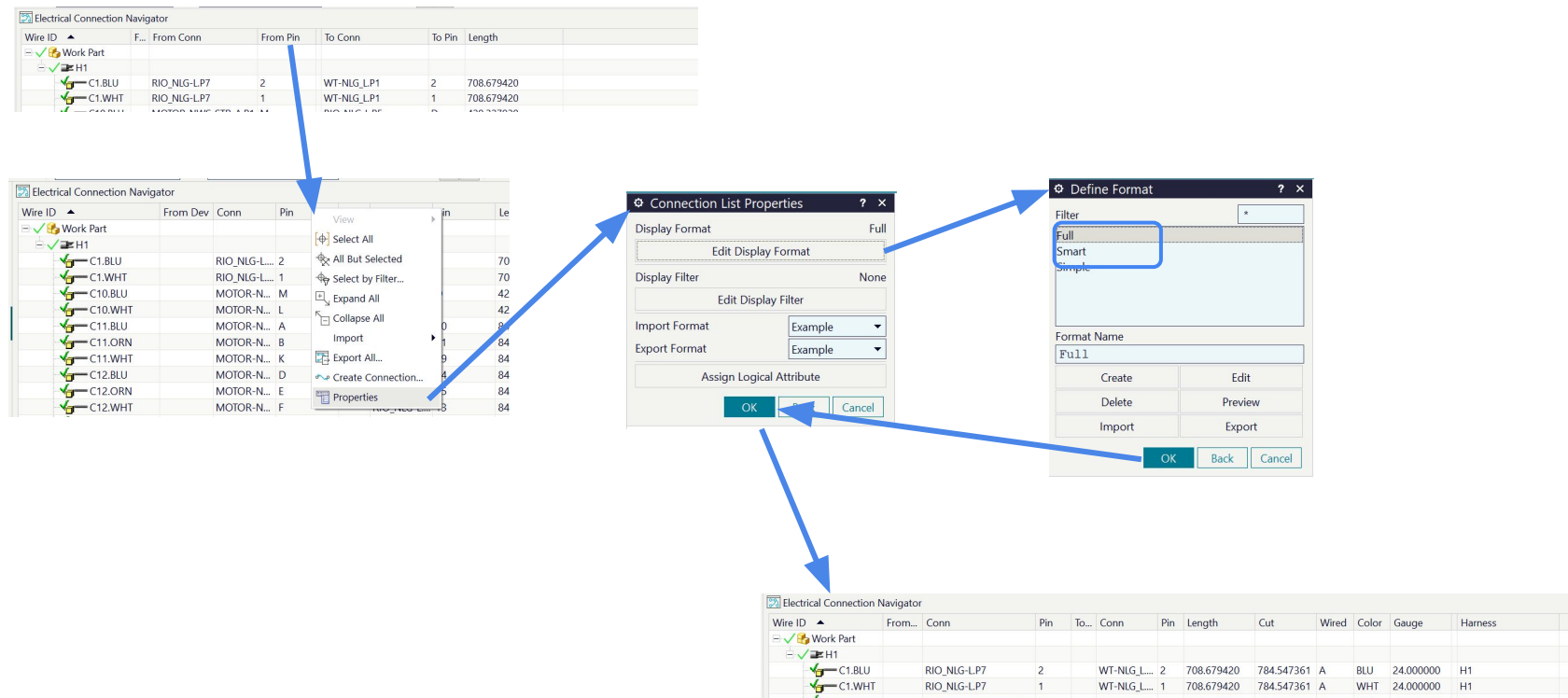
2. “Update Connection” will clear the connection and component lists and replace them with the imported list. It will generally preserves routed connections.

(Merge will attempt to merge by attribute, and Append will create additional entries; “Update” is always recommended.)

3. Ensure the filetype is “*.plmxml”, browse to and select your file, and hit Import.

Changing the Route List Display Format

NX's default 'Simple' display format may not show enough information to be useful in all cases. Right clicking on the top bay allows you to open the connection list properties and switch formats to one with more information included. ('Full' and 'Smart' are two Siemens-provided defaults, or you can create your own)



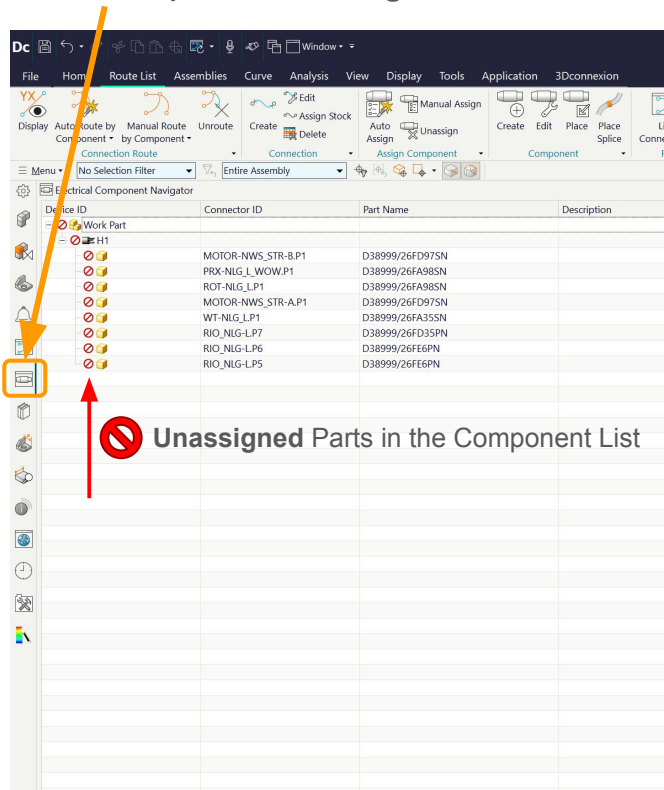
Newly Imported Route List

A newly imported route list will populate the connection navigator and component navigator with component and routing information.

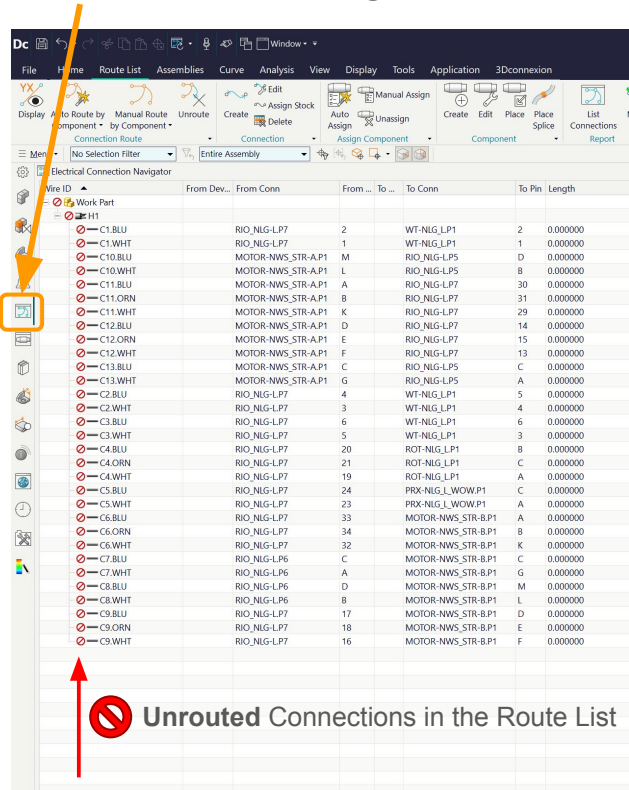
In order to route a connection:

- 1. Components must be assigned.
- 2. A routing path must be drawn.

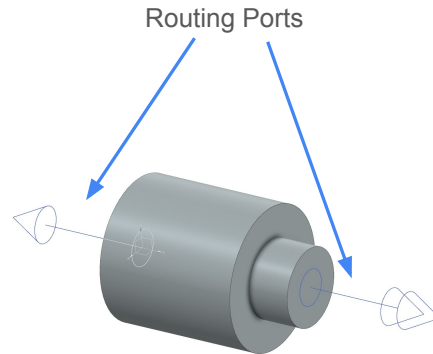
Component Navigator



Connection Navigator



Qualifying and Configuring Parts

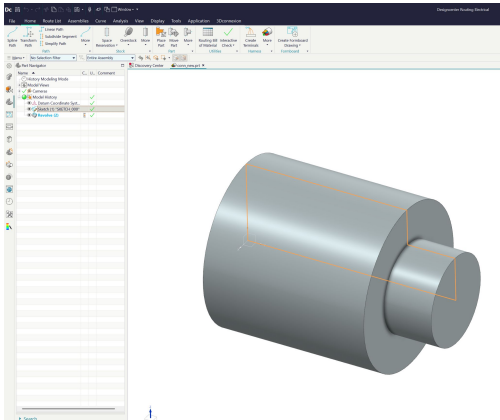


Making connector models usable in Routing

To route to a component in NX, the component must have at least one routing port available. Routing ports are added using the Qualify Part tool in Routing Electrical.

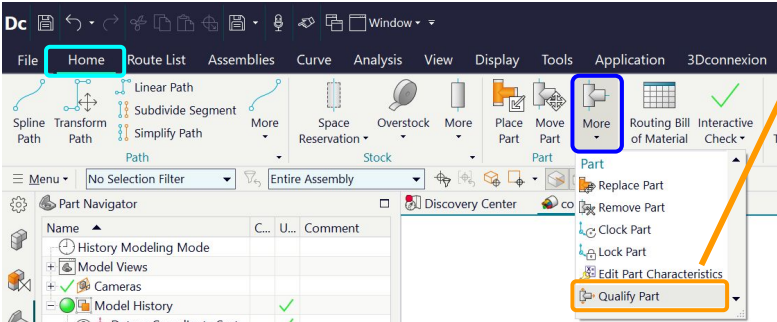
1. Make a new part in NX Modelling.

For routing parts, it's recommended to keep geometry as minimal as possible. This part is just a revolved sketch.



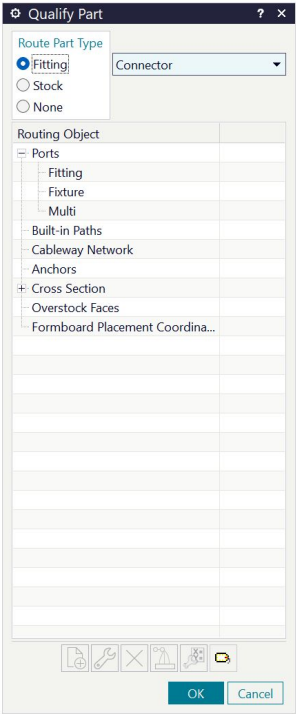
2. Launch the *Qualify Part* modal

Home → More [Part] → Qualify Part



**Qualify Part**

Adds routing specific intelligence to a part file which dictates how the part places into a routing assembly, relates to other parts, and behaves during routing operations.



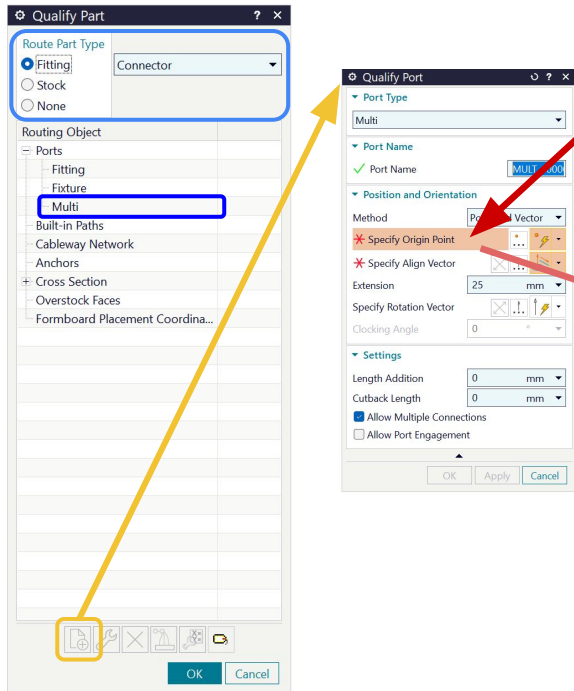
Adding a Routing Port

3. Add a new Routing Multi Port

The Multi Port supports both electrical and mechanical connections.
In Qualify Part, set:

Route Part Type -> Fitting
Dropdown Selector -> Connector

Click on Ports -> Multi, and then click the "New Port" button
The "Qualify Port" modal will open to define the new port.

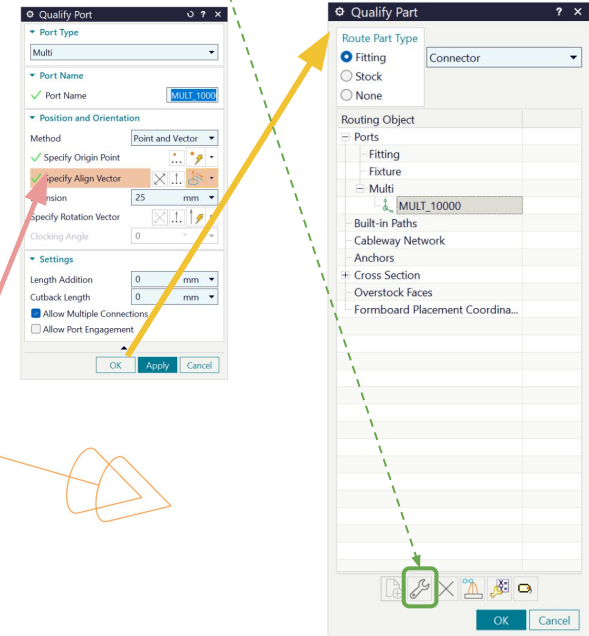
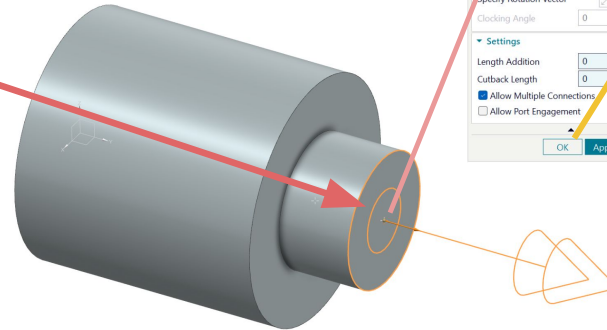


4. Place the port on the part geometry

With 'Specify Origin Point' selected, position the port in the desired location. Here, the origin point is the centroid of the edge of the 'back' face of this simple connector.

The 'Align Vector' should set automatically, or can be manually set
The rest of the settings can typically be left at their defaults
Press 'OK' to close the modal, or 'Apply' to add more ports.

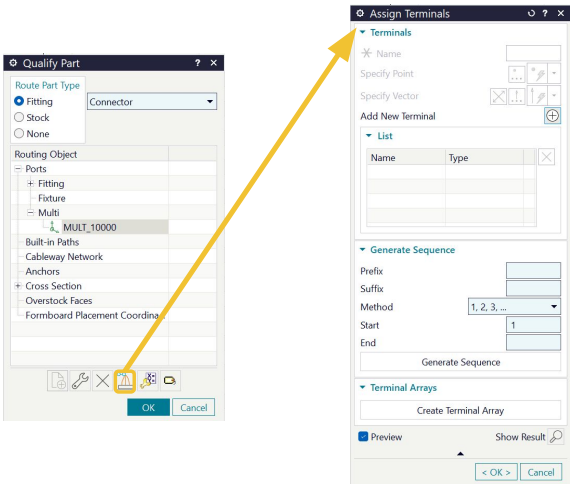
To edit the port in the future, you MUST reopen the Qualify Part modal, select the port and use the "Edit Port" button



Additional Part Qualification

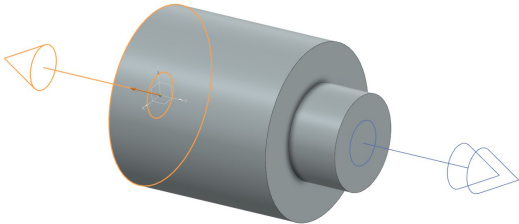
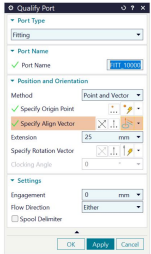
Assigning Terminals (Pins)

NX Allows you to specify pin-level detail on connectors by assigning and naming terminals. Terminals can be located on one point, or positioned using a 3D point array or points on a sketch. This is time consuming to manage, **This is not required in most cases.** Pin-level routing does improve visual fidelity and aesthetics, but carries a significant performance penalty with minimal practical manufacturing impact for large harnesses and standard connectors. **Use NX Component-Level Routing** to avoid the overhead and performance penalties; Component-level routing preserves pin-to-pin data, and cut length padding typically makes up for



Other Port Types

You can also create "Fitting" ports to define mechanical connection points, and "Fixture" points to define wiring fixtures such as clamps or tiedowns. Here, we are defining a Fitting port on the front face of the connector. These are useful for positioning connectors, but not required for routing.



Built In Paths

For components that route wires *through* them, such as backshells, you can construct built-in paths that carry the route through the part.

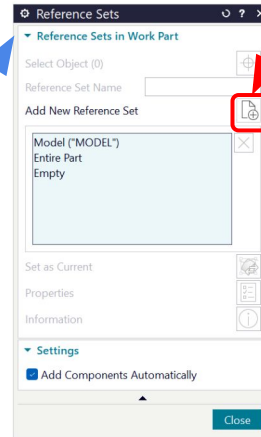
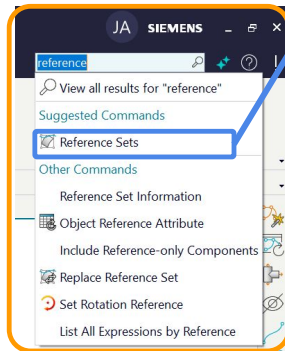
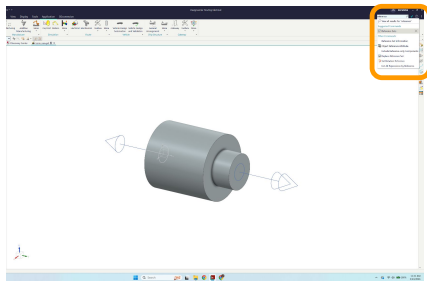
Configuring the Routing Reference Set

NX uses **reference sets** to configure how parts are displayed in higher-level assemblies. By default, routing ports are hidden in the MODEL reference set.

The best and recommended practice is to create a "ROUTING" reference set for parts with ports, and showing the ROUTING reference set while performing routing work.

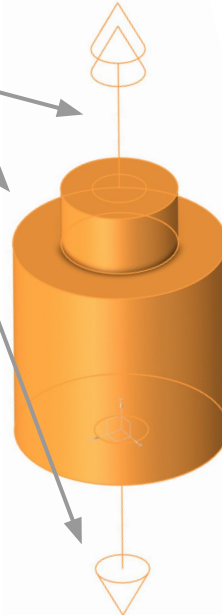
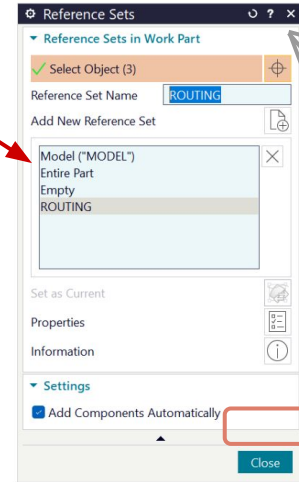
1. Open the Reference Sets modal

Use the **search bar** in the upper right hand corner to find the "Reference Sets" tool, and open it



2. Create a new "ROUTING" Reference Set

Hit the "Add New Reference Set" button create a new reference set.
Name it "ROUTING" (or pick a preferred consistent name)
With the "ROUTING" set selected, select the model body and the routing ports. This will be the geometry shown with this reference set enabled.
Press "Close" on the modal (No "OK" - changes are immediate)

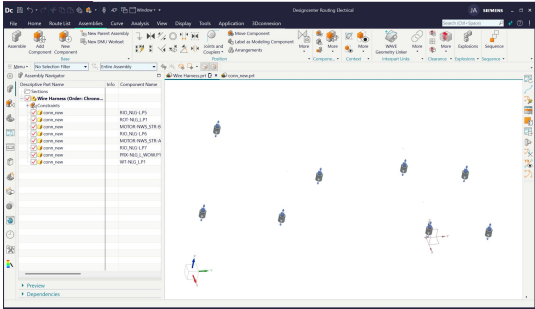


Building the Harness Assembly

Adding Connector Parts to a Harness Assembly

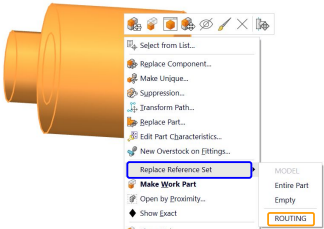
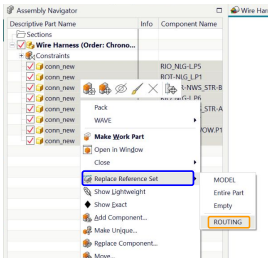
Wire Harness Assembly:

In NX, a wire harness is an assembly that only contains harness parts (Connectors, backshells, splices), route splines, and harness 'stocks' (bundles). To get started building a harness, make a new assembly and add electrical connectors to it.



After adding a part to an assembly, you can change the reference set of one (or many) components at once by right clicking on the component in the model or in the Assemblies list.

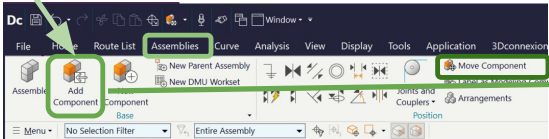
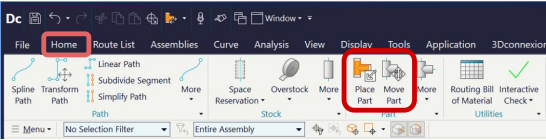
Right Click on part(s) → **Replace Reference Set** → **ROUTING**



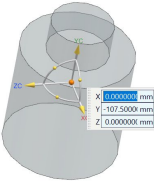
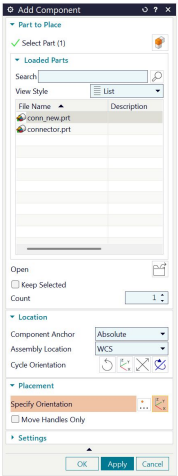
Routing vs Assemblies part handling

Routing Electrical offers **Place Part** and **Move Part** commands on the **Home** tab that provide a routing-centric interface to handle parts.

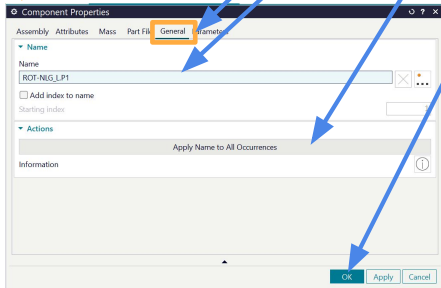
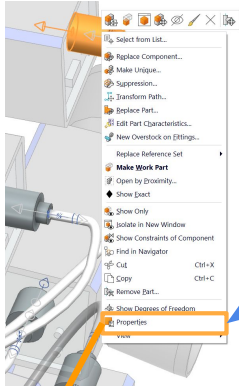
However, the familiar **Assemblies Add Component** and **Move Component** commands are functionally identical and are generally more polished. These are also available in Routing.



In practice, the Assemblies commands are usually faster and preferred by users.

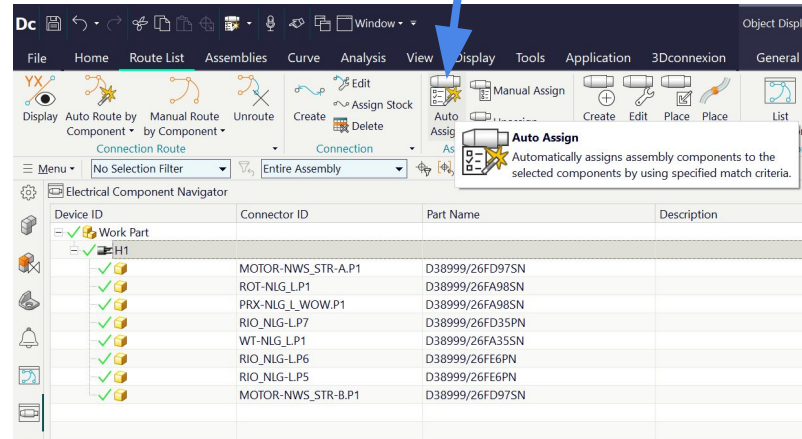


Assigning Connector Reference Names, and Auto Assigning Connectors



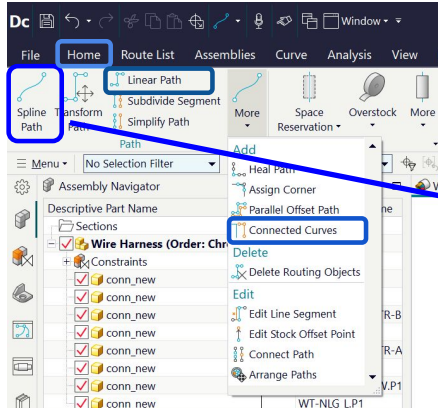
1. In the model, right click on the connector to be named, and select “Properties”
2. Go to the “General” tab
3. Enter the EXACT Reference Name of the connector
4. Hit “Apply Name to All Occurrences” to ensure NX fully propagates the change. (This name will only apply to the selected part instance.)
5. Hit “OK” to close the properties pane.

Once all connectors have reference names assigned, the “Auto Assign” Components button will match the items in the Component Navigator with the assembly parts, based on reference names.



**(This requires “Match By Component Name” settings configured properly)*

Creating Routing Paths



Route Creation

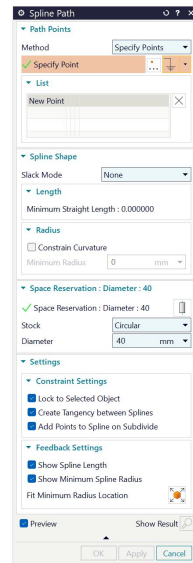
Harness routes are created from port to port, typically using the **Spline Path** tool, or sometimes the **Linear Path** tool.

The **Connected Curves** tool can be used to convert other curves (such as the Spiral Curve) to a routing path.

Spline Creation

Splines should be created with the fewest control points possible; NX has no limitations on spline creation, but >5 points in one spline can become computationally expensive.

The best practice is to create one spline between each fixed point (connector, clamp, etc)



Place Control Points

- Place, move, and orient spline control points

List of control points

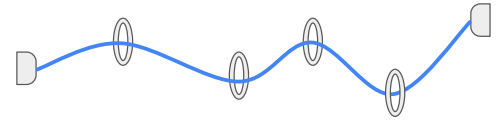
- Select, re-order, and delete control points

Space Reservation

- Creates semi-transparent placeholder bundles, which get replaced with solid bodies after routing the harness

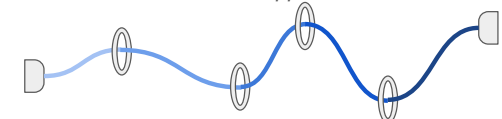
Single Spline Route

Computationally heavy, slow, unreliable
Not recommended



One spline per segment

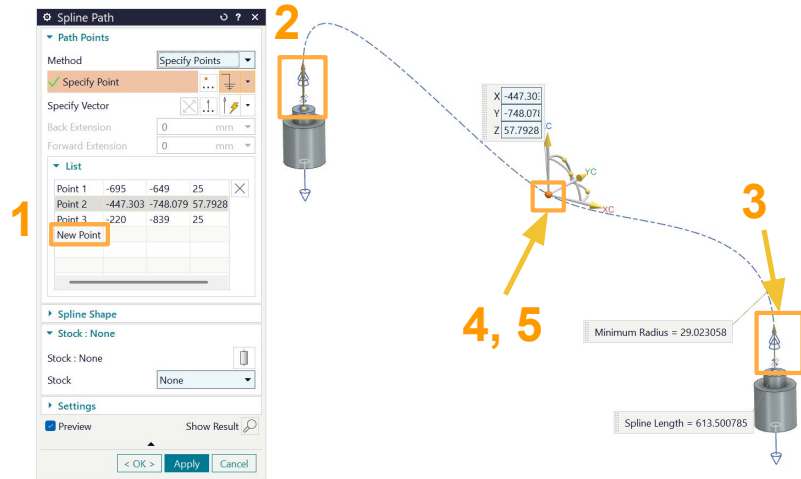
Faster creating, adjusting, and routing
Recommended approach



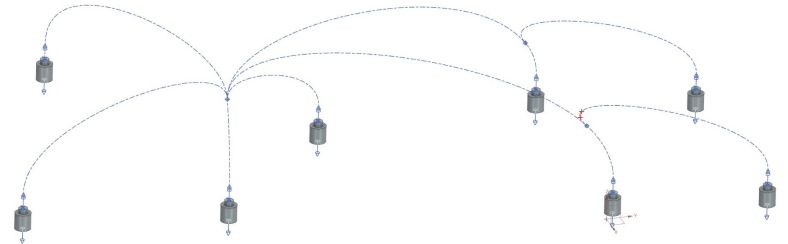
Drawing Route Splines

Typical Route Spline Creation

1. Select "New Point" in the point list to place a new point
2. Select the routing port in the model for the start point of the spline.
3. Select the routing port for the end point of the spline to place the last point.
4. Place additional points by clicking on the length of the spline
5. Move these points by selecting them, and adjusting the model handles.



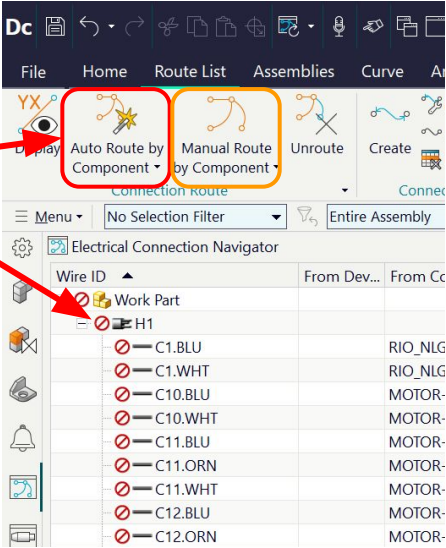
When completed, a harness with connectors and route splines will look something like this:



Routing a connection list

Automatic Route

Press the 'Auto Route' or the  symbol to route a connection



Auto vs Manual Routing

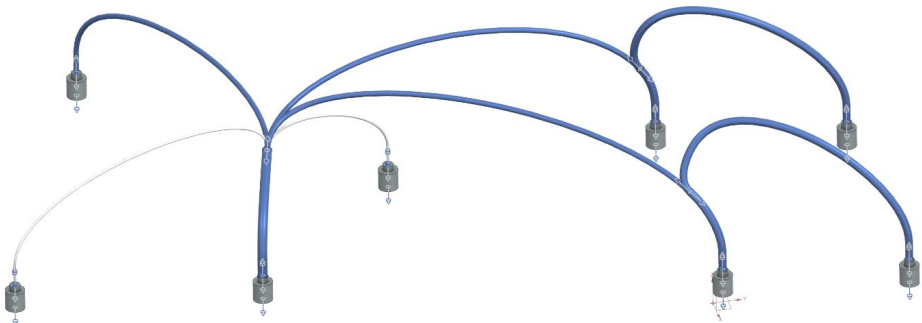
If there are multiple paths for a specific connection, Manual routing will allow you to select the preferred path. In all other ways, Auto Route and Manual Route function identically.



Wire ID	
Work Part	
H1	
C1.BLU	
C1.WHT	
C10.BLU	
C10.WHT	
C11.BLU	
C11.ORN	

Routed Connections

When routed, the  symbol is replaced with a .



The routed harness shows accurate volumes for individual cables (using provided cable diameters), and bundles (using industry-standard bundle diameter calculations)

Exporting a Route List

The screenshot displays the Designcenter Routing Electrical software interface. The top ribbon includes tabs for File, Home, Route List, Assemblies, Curve, Analysis, View, Display, Tools, Application, and 3Dconnexion. The Route List tab is active, showing a table of routing data. Below the ribbon, the Electrical Connection Navigator is visible, showing a tree view of the assembly structure. On the right side, the 'All Connections' dialog is open, indicating that a route list file will be created for all elements in the Connection List.

Wire ID	From Device	From Conn	From Pin	To Device	To Conn	To Pin	Length
Work Part							
H1							
C8....		RIO_NLG-LP6	B	MOTOR-NWS_STR-...	L		483.320523
C4.B...		RIO_NLG-LP7	20	ROT-NLG_LP1	B		455.902068
C11....		MOTOR-NWS_STR-...	A	RIO_NLG-LP7	30		847.115598
C6.B...		RIO_NLG-LP7	33	MOTOR-NWS_STR-...	A		922.781224
C7.B...		RIO_NLG-LP6	C	MOTOR-NWS_STR-...	C		483.320523
C1.B...		RIO_NLG-LP7	2	WT-NLG_LP1	2		708.679420
C6....		RIO_NLG-LP7	32	MOTOR-NWS_STR-...	K		922.781224
C9.O...		RIO_NLG-LP7	18	MOTOR-NWS_STR-...	E		922.781224
C12....		MOTOR-NWS_STR-...	F	RIO_NLG-LP7	13		847.115598
C12....		MOTOR-NWS_STR-...	E	RIO_NLG-LP7	15		847.115598
C12....		MOTOR-NWS_STR-...	D	RIO_NLG-LP7	14		847.115598
C4.O...		RIO_NLG-LP7	21	ROT-NLG_LP1	C		455.902068
C9....		RIO_NLG-LP7	16	MOTOR-NWS_STR-...	F		922.781224
C11....		MOTOR-NWS_STR-...	K	RIO_NLG-LP7	29		847.115598
C11....		MOTOR-NWS_STR-...	B	RIO_NLG-LP7	31		847.115598

Route List Exporting

After routing, the list can be exported and a PLMXML file will be produced with all of the cable lengths. Additionally, NX will produce a JSON file with route geometry encoded in it.